MD Nastran 2006

Quick Reference Guide

MSC Software

Corporate
MSC.Software Corporation
2 MacArthur Place
Santa Ana, CA 92707 USA
Telephone: (800) 345-2078
Fax: (714) 784-4056

Europe
MSC.Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) (89) 43 19 87 0
Fax: (49) (89) 43 61 71 6

Asia Pacific
MSC.Software Japan Ltd.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo 160-0023, JAPAN
Telephone: (81) (3)-6911-1200
Fax: (81) (3)-6911-1201

Worldwide Web
www.mscsoftware.com

Disclaimer

MDNA:V2006:Z:Z:Z:DC-QRG-PDF

C O N T E N T S

MD Nastran Quick Reference Guide

2

File Management Statements

3 Executive Control Statements

4 Case Control Commands

X-Y PLOT Commands

5
Parameters

Parameter Applicability Tables

6 Item Codes

7 Degree-of-Freedom Sets

8 Bulk Data Entries

C O N T E N T S

MD Nastran Quick Reference Guide

C O N T E N T S

MD Nastran Quick Reference Guide

Preface

- List of MSC.Nastran Books
- Technical Support
- Internet Resources

List of Nastran Books

Below is a list of some of the Nastran documents. You may order any of these documents from the MSC.Software BooksMart site at www.engineering-e.com.

Installation and Release Guides

- ❑ Installation and Operations Guide
- ❑ Release Guide

Reference Books

- ❑ Quick Reference Guide
- ❑ DMAP Programmer's Guide
- ❑ Reference Manual

User's Guides

- ❑ Getting Started
- ❑ Linear Static Analysis
- ❑ Basic Dynamic Analysis
- ❑ Advanced Dynamic Analysis
- ❑ Design Sensitivity and Optimization
- ❑ Thermal Analysis
- ❑ Numerical Methods
- ❑ Implicit Nonlinear (SOL 600)
- ❑ Explicit Nonlinear (SOL 700)
- ❑ Aeroelastic Analysis
- ❑ Superelement
- ❑ User Modifiable
- ❑ Toolkit

Technical Support

For help with installing or using an MSC.Software product, contact your local technical support services. Our technical support provides the following services:

- Resolution of installation problems
- Advice on specific analysis capabilities
- Advice on modeling techniques
- Resolution of specific analysis problems (e.g., fatal messages)
- Verification of code error.

If you have concerns about an analysis, we suggest that you contact us at an early stage.

You can reach technical support services on the web, by telephone, or e-mail:

Web

Go to the MSC.Software website at **www.mscsoftware.com**, and click on **Support**. Here, you can find a wide variety of support resources including application examples, technical application notes, available training courses, and documentation updates at the MSC.Software Training, Technical Support, and Documentation web page.

Phone and Fax

United States
Telephone: (800) 732-7284
Fax: (714) 784-4343

Munich, Germany
Telephone: (49) (89) 43 19 87 0
Fax: (49) (89) 43 61 71 6

Rome, Italy
Telephone: (390) (6) 5 91 64 50
Fax: (390) (6) 5 91 25 05

Moscow, Russia
Telephone: (7) (095) 236 6177
Fax: (7) (095) 236 9762

Frimley, Camberley
Surrey, United Kingdom
Telephone: (44) (1276) 60 19 00
Fax: (44) (1276) 69 11 11

Tokyo, Japan
Telephone: (81) (3) 3505 02 66
Fax: (81) (3) 3505 09 14

Paris, France
Telephone: (33) (1) 69 36 69 36
Fax: (33) (1) 69 36 45 17

Gouda, The Netherlands
Telephone: (31) (18) 2543700
Fax: (31) (18) 2543707

Madrid, Spain
Telephone: (34) (91) 5560919
Fax: (34) (91) 5567280

Email

Send a detailed description of the problem to the email address below that corresponds to the product you are using. You should receive an acknowledgement

that your message was received, followed by an email from one of our Technical Support Engineers.

MSC.Patran Support	mscpatran.support@mscsoftware.com
MSC.Nastran Support	mscnastran.support@mscsoftware.com
MSC.Nastran for Windows Support	vn4w.support@mscsoftware.com
MSC.visualNastran Desktop 2D Support	vn2d.support@mscsoftware.com
MSC.visualNastran Desktop 4D Support	vndesktop.support@mscsoftware.com
MSC.Abaqus Support	mscabaqus.support@mscsoftware.com
MSC.Dytran Support	mscdytran.support@mscsoftware.com
MSC.Fatigue Support	mscfatigue.support@mscsoftware.com
MSC.Interactive Physics Support	ip.support@mscsoftware.com
MSC.Marc Support	mscmarc.support@mscsoftware.com
MSC.Mvision Support	mscmvision.support@mscsoftware.com
MSC.SuperForge Support	mscsuperforge.support@mscsoftware.com
MSC Institute Course Information	msctraining.support@mscsoftware.com

Training

The MSC Institute of Technology is the world's largest global supplier of CAD / CAM / CAE / PDM training products and services for the product design, analysis and manufacturing market. We offer over 100 courses through a global network of education centers. The Institute is uniquely positioned to optimize your investment in design and simulation software tools.

Our industry experienced expert staff is available to customize our course offerings to meet your unique training requirements. For the most effective training, The Institute also offers many of our courses at our customer's facilities.

The MSC Institute of Technology is located at:

2 MacArthur Place
Santa Ana, CA 92707
Phone: (800) 732-7211
Fax: (714) 784-4028

The Institute maintains state-of-the-art classroom facilities and individual computer graphics laboratories at training centers throughout the world. All of our courses emphasize hands-on computer laboratory work to facility skills development.

We specialize in customized training based on our evaluation of your design and simulation processes, which yields courses that are geared to your business.

In addition to traditional instructor-led classes, we also offer video and DVD courses, interactive multimedia training, web-based training, and a specialized instructor's program.

Course Information and Registration. For detailed course descriptions, schedule information, and registration call the Training Specialist at (800) 732-7211 or visit www.mscsoftware.com.

Internet Resources

MSC.Software (www.mscsoftware.com)

MSC.Software corporate site with information on the latest events, products and services for the CAD/CAE/CAM marketplace.

Engineering-e.com (www.engineering-e.com)

Engineering-e.com is the first virtual marketplace where clients can find engineering expertise, and engineers can find the goods and services they need to do their job

CATIASOURCE (plm.mscsoftware.com)

Your SOURCE for Total Product Lifecycle Management Solutions.

CHAPTER
1 nastran Command and NASTRAN Statement

- The nastran Command
- The NASTRAN Statement

1.1 The nastran Command

MD Nastran is executed with a command called nastran. (Your system manager may assign a different name to the command.) The nastran command permits the specification of keywords used to request options affecting MD Nastran job execution. The format of the nastran command is:

```
nastran input_data_file [keyword1 = value1  keyword2 = value2 ...]
```

where input_data_file is the name of the file containing the input data and $keyword_i$=$value_i$ is one or more optional keyword assignment arguments. For example, to run an a job using the data file example1.dat, enter the following command:

```
nastran example1
```

Keyword assignments can be specified as command line arguments and / or included in RC files.

There are two RC files controlled by you:

- The user RC file is in your home (or login) directory. This file should be used to define parameters that are applied to all jobs run by you.
- The local RC file is .nast2005rc, in the same directory as the input data file. If the “rcf” keyword is used, this local RC file is ignored. This file should be used to define parameters that are applied to all jobs contained in the input data file directory.

1. The tilde (~) character is not recognized within RC files.
2. Environment variables are only recognized when used in the context of a logical symbol (see “**Using Filenames and Logical Symbols**” on page 99 of the *MD Nastran 2006 Installation and Operations Guide*).
3. When a keyword is specified on the command line, embedded spaces or special characters that are significant to the shell must be properly quoted; quotes should not be used within RC files.

The keywords listed below are the most common for various computers, but are not available on all computers. Also, the defaults may be site dependent. Please consult your “**Keywords and Environment Variables**” on page 287 of the *MD Nastran 2006 Installation and Operations Guide* for keyword applicability, specialized keywords, and further discussion of the keywords and their defaults. Keywords that use yes / no values accept partial specification and case-independent values. For example, “yes” may be specified as “y”, “ye”, or “yes” using uppercase or lowercase letters. The examples assume the jobs are run under a UNIX operating system.

after — after=*time* — Default: *None*

(UNIX) .Holds the job's execution until the time specified by *time*. See the description of the "at" command in your system documentation for the format of *time*

Example: `nastran example after=10:00`

The job is held until 10:00 AM.

append — append={yes | no} — Default: no

Combines the F04, F06, and LOG files into a single file after the run completes. If "no" is specified, the files are not combined. If "yes" is specified, the files are combined into one file with the suffix ".out".

Example: `nastran example append=yes`

The F04, F06, and LOG files are combined into a file named example.out.

batch — batch={yes | no} — Default: yes

(UNIX) Indicates how the job is to be run. If "yes" is specified, the job is run as a background process. If "no" is specified, the job is run in the foreground. If the "aft" or "queue" keywords are specified the batch keyword is ignored. Jobs submitted with "batch=yes" will run under nice(1).

Note: If the job is already running in an NQS or NQE batch job, the default is "no".

Example: `nastran example batch=no`

The job is run in the foreground.

dbs — dbs=*pathname* — Default= . {Current directory}

Creates database files (see "**Default DBsets and Their Default Attributes**" on page 114 of the *MSC.Nastran 2005 r2 Installation and Operations Guide* using an alternate file prefix. If "dbs" is not specified, database files are created in the current directory using the basename of the input data file as the prefix. If the "dbs" value is a directory, database files are created in the specified directory using the basename of the input data file as the filename.

Note: If "dbs" is specified and "scratch=yes" is specified, a warning will be issued and "scratch=no" assumed.

IIn the following examples, assume the current directory includes sub-directories “mydir” and “other”, and that an “example.dat” exists in both the current directory and “other”. That is, ./example.dat, ./mydir, ./other, and ./other/example.dat exist on UNIX; and .\example.dat, .\mydir, .\other, and .\other\example.dat exist on Windows.

Example: msc2005 nastran example

Database files are created in the current directory with the name “example”, e.g., ./example.DBALL on UNIX; and .\example.DBALL on Windows.

Example: msc2005 nastran other/example

Database files are created in the “other” directory with the name “example”, e.g., ../other/example.DBALL on UNIX and .\other\example.DBALL on Windows.

Example: msc2005 nastran example dbs=myfile

Database files are created in the current directory with the name “myfile”, e.g., ./myfile.DBALL on UNIX and .\myfile.DBALL on Windows.

Example: msc2005 nastran example dbs=mydir

Database files are created in the mydir directory with the name “example”, e.g., ./mydir/example.DBALL on UNIX and .\mydir\example.DBALL on Windows.

Example: msc2005 nastran example dbs=mydir/myfile

Database files are created in the mydir directory with the name “myfile”, e.g., ./mydir/myfile.DBALL on UNIX and .\mydir\myfile.DBALL on Windows

Example: msc2005 nastran example dmp=4 host=a:b:c:d dbs=/aa:/bb:/cc:/dd

This example will set the “dbs” directory to “/aa” on host a, “/bb” on host b, “/cc” on host c, and finally “/dd” on host d.

Note: The use of distinct per-task database directories can have a significant impact on elapsed time performance of DMP jobs on SMP and NUMA systems.

memory memory=*memory_size* Default=estimate

Specifies the amount of open core memory to allocate. The *memory_size* can be specified either as anumber of wordsor as a number followed by one of the following modifiers:

G or Gw	Multiply *memory_size* by 1024**3.
Gb	Multiply *memory_size* by (1024**3) / *bytes_per_word*.
M or Mw	Multiply *memory_size* by 1024**2.
Mb	Multiply *memory_size* by (1024**2) / *bytes_per_word*.
K or Kw	Multiply *memory_size* by 1024.
Kb	Multiply *memory_size* by 1024 / *bytes_per_word*.
w	Use *memory_size* as is.
b	Divide *memory_size* by *bytes_per_word*.

where *bytes_per_word* is 8 on CRAY UNICOS and NEC; 4 on all others. The modifier may be specified using any case combination.

Note: MD Nastran uses standard computer units for K, M, and G.

Example: `nastran example memory=25mw`

The job is run using an open core memory size of 25 megawords, or 25 600 kilowords, or 26 214 400 words.

The maximum *memory_size* is limited as shown in **Table 1-1** (less the size of the executable and I/O buffers).

For a more detailed description see the "**memory**" on page 310 of the *MD Nastran 2006 Installation and Operations Guide*.

news news={yes | no | auto} Default=yes

Displays the news file (*install_dir* / msc2005 / nast / news.txt on UNIX and *install_dir* \ msc2005 \ nast \ news.txt on Windows) in the F06 file. If "auto" is specified, the news file is only displayed if it has been modified since the last time it was displayed for you. If "yes" is specified, the news file is displayed in the F06 file regardless of when it was last changed. If "no" is specified, the news file is not displayed in the F06 file.

Example: `nastran example news=yes`

The news file is displayed in the F06 file after the title page block.

notify — notify={yes | no} — Default=yes

Sends notification when the job is completed. See the “ncmd” keyword to define an alternate notification command.

Note: If the job is queued using the queue keyword, or the job is already running in an NQS batch job, the default is “no”.

Example: `nastran example notify=yes`

old — old={yes | no} — Default=yes

Saves previous copies of the F04, F06, LOG, OP2, OUT, PCH, and PLT output files using sequence numbers (additional user-specified file types can be versioned with the “oldtypes” keyword). Sequence numbers are appended to the keyword filename and are separated by a period.

If “yes” is specified, the highest sequence number of each of the output files is determined. The highest sequence number found is incremented by one to become the new sequence number. Then, all current output files that do not include sequence numbers are renamed using the new sequence number as a type.

Example: `msc2005 nastran example old=yes`

For example, assume your current working directory contains the following files:

v2401.datv2401.f04.1v2401.f06v2401.logv2401.log.1
v2401.f04v2401.f04.2v2401.f06.1v2401.log.1v2401.log.3

Apparently, the user ran the job four times, but deleted some of the files, e.g., v2401.f04.3, v2401.f06.2, and v2401.f06.3. When the job is run again with “old=yes”, the files are renamed as follows: v2401.f04 is renamed to v2401.f04.4, v2401.f06 is renamed to v2401.f06.4, and v2401.log is renamed to v2401.log.4. The sequence number 4 is used because it is one greater than the highest sequence number of all of the selected files (the highest being v2401.log.3).

out out=*pathname* Default= .

Saves the output files using a different file prefix or in a different directory. If “out” is not specified, the output files are saved in the current directory using the basename of the input data file as a prefix. If the “out” value is a directory, output files are created in the specified directory using the basename of the input data file as the filename.

In the following examples, assume the current directory includes sub-directories “mydir” and “other”, and that an “example.dat” exists in both the current directory and “other”. That is, ./example.dat, ./mydir, ./other, and ./other/example.dat exist on UNIX; and .\example.dat, .\mydir, .\other, and .\other\example.dat exist on Windows.

Example: `msc2005 nastran example`

or: `msc2005 nastran other/example`

Output files are created in the current directory with the name “example”, e.g., ./example.f06 on UNIX and .\example.f06 on Windows.

Example: `msc2005 nastran example out=myfile`

Output files are created in the current directory with the name “myfile”, e.g., ./myfile.f06 on UNIX and .\myfile.f06 on Windows.

Example: `msc2005 nastran example out=mydir`

Output files are created in the mydir directory with the name “example”, e.g., ./mydir/example.f06 on UNIX and .\mydir\example.f06 on Windows.

Example: `msc2005 nastran example out=mydir/myfile`

Output files are created in the mydir directory with the name “myfile”, e.g., ./mydir/myfile.f06 on UNIX and .\mydir\myfile.f06 on Windows.

rcf rcf=pathname Default=no

Specifies the name of the local RC file. If this keyword is not specified, the .nast2004rc file on UNIX and nast2004.rcf on Windows located in the input data file’s directory is used

Example: `$ nastran example rcf=nast.rc`

The nastran command will process ./nast.rcf on UNIX, or .\nast.rcf on Windows in lieu of the default local RC file ./.nast2004rc on UNIX and .\nast2004.rcf on Windows.

scratch scratch={yes | no | mini} Default=no

Deletes the database files at the end of the run. If the database files are not required, “scratch=yes” can be used to remove them; preventing cluttering of the directory with unwanted files. If “mini” is specified, a reduced size database (that can only be used for data recovery restarts) will be created. See “**Database Concepts**” on page 513 of the *MSC.Nastran Reference Guide* for further details on the “mini” database.

Example: `nastran example scratch=yes`

All database files created by the run are deleted at the end of the job in the same way as the FMS statement INIT MASTER(S).

sdirectory sdirectory=*directory* Default: *See the description below.*

See “**Determining Resource Requirements**” on page 107 of the *MD Nastran 2006 Installation and Operations Guide* for information on estimating a job’s total disk space requirements.

Specifies the directory to use for temporary scratch files created during the run. MD Nastran can create very large scratch files, the scratch directory should contain sufficient space to store any scratch files created during a run. You must have read, write, and execute privileges to the directory.

UNIX: The default value is taken from the TMPDIR environment variable if it is set to a non-null value. Otherwise the computer’s default temporary file directory is chosen; this is usually /tmp, but on IRIX64 systems, it is /var/tmp.

Windows: The default value is taken from the TEMP environment variable.

UNIX Example: `nastran example sdir=/scratch`

Scratch files are created in the directory /scratch.

Windows example: `msc2005 nastran example sdir=d:\scratch`

Scratch files are created in the d:\scratch directory

If a DMP run was selected with *dmparallel* ≥ 1, unique task-specific scratch directories may be set for each host using the standard PATH separator, i.e, ":" on UNIX and ";" on Windows, to separate entries. The directories will be paired with each host in a round-robin order, that is, the list will be reused if more tasks than directories are specified.

See "**Running Distributed Memory Parallel (DMP) Jobs**" on page 163 for additional information.

UNIX example:

```
msc2005 nastran example dmp=4 \
   sdir=/scratch1:/scratch2
```

In this example, /scratch1 will be used for the first and third tasks, while /scratch2 will be used for the second and fourth tasks.

smemory smemory=*value* Default: 0 (UNICOS and SUPER-UX); 100 (all others)

.Specifies the amount of space in open core to reserve for scratch memory.

The size is specified as the number of blocks (BUFFSIZE words long) or the number of words or bytes follwed by one of the modifiers: "G", "GW", "GB", "M", "MW", "MB", "K", "KW", "KB", "W", "B". See "**Specifying Memory Sizes**" on page 104 of the *MD Nastran 2006 Installation and Operations Guide* for a description of these modifiers. The value specified by this keyword may be overridden by the FMS statement ASSIGN SCRATCH(MEM=*value*).

Example:

```
nastran example smem=200
```

This example reserves 200 GINO blocks for scratch memory. The amount of scratch memory may also be specified in terms of words or bytes if followed by a unit modifier. Please see the memory keyword for valid unit modifiers.

Example:

```
msc2005 nastran example smem=4mw
```

This example reserves 4,194,304 words for scratch memory.

symbol symbol=*name*=*string* Default: *None*

Defines a symbolic (or logical) name used in ASSIGN and INCLUDE statements and in command line arguments. This keyword may be specified in initialization or RC files and on the command line. The symbol definition may include references to previously defined symbols or environment variables using the standard "$*name*" or "${*name*}" syntax on UNIX or %*name*% syntax on Windows. For convenience, the character separating the "symbol" and "*name*" specification and the "*name*" and "*string*" specification may be either an equal sign ("=") or a hash mark ("#"). The use of a hash mark allows this keyword to be specified as an argument to a Windows .bat file.

If "node" is specified, symbolic names defined using this keyword are not used on the local system. Instead the specified values are passed to the remote system. This means that any pathnames must be valid on the remote system. Use the "lsymbol" keyword to specify symbolic names for the local system.

If "node" is not specified, symbolic names defined using the "lsymbol" keyword are processed as if they were defined using the "symbol" keyword.

Symbolic names are processed in the order they are encountered while processing the initialization and RC files and the command line. If a duplicate symbolic name is encountered, the new value replaces the previously specified value.

Symbolic names must be 16 characters or less, the value assigned to the symbolic name must be 256 characters or less. If the symbolic name used in an ASSIGN or INCLUDE statement or in command line arguments is not defined, it is left in the filename specification as is.

For example, many of the TPL and DEMO input data files have ASSIGN statements such as the following:

ASSIGN 'MASTER=DBSDIR:abc.master'

The string "DBSDIR:" specifies a symbolic name that is to be replaced by another string. The replaced string is defined by the "symbol=" keyword (or "lsymbol=" keyword if "node" was not specified) in an initialization or RC file, on the command line, or as environment variable. For example,

(UNIX) `symbol=DBSDIR=/dbs`

(Windows) `symbol=DBSDIR=d:\dbs`

When the previous ASSIGN statement is processed, the filename assigned to the logical name MASTER is `/dbs/abc.master` on UNIX and `d:\dbs\abc.master` on Windows. An alternate way of defining symbolic names is through the use of environment variables. For example, typing the following command

```
export DBSDIR=/dbs
```

at a Korn shell prompt, or

```
setenv DBSDIR /dbs
```

at a C-shell prompt, or

at a Windows shell prompt, is equivalent to the "symbol" keyword definition.

Note: If a symbolic name is defined by both a symbol statement in an RC file and by an environment variable, the symbol statement value will be used.

The section titled "**Environment Variables**" on page 345 of the *MD Nastran 2006 Installation and Operations Guide* contains a list of environment variables that are automatically created by the nastran command. Of particular interest to the logical symbol feature are the OUTDIR and DBSDIR variables. These variables refer to the directory that will contain the output files (set using the "out" keyword) and the directory that will contain the permanent database files (set using the "dbs" keyword), respectively.

xmonast xmonast={yes|no|kill} Default: No

Indicates if XMONAST is to be run to monitor the MD Nastran job. If "xmonast=yes" is specified, XMONAST will be automatically started; you must manually exit XMONAST when the MD Nastran job has completed. If "xmonast=kill" is specified, XMONAST will start and will automatically exit when the MD Nastran job has completed.

Example: `nastran example xmon=kill`

This example runs the XMONITOR utility while the MD Nastran job is running. Once the job completes, the XMONITOR program is automatically terminated.

1.2 The NASTRAN Statement

The NASTRAN statement is used to specify values for certain Executive System operational parameters. These parameters are also called system cells. The NASTRAN statement is used for exceptional circumstances and is therefore not needed in most runs. The NASTRAN statement may also be specified in the Runtime Configuration (RC) files at the system, user, and job level as described in the ***MSC.Nastran 2005 r2 Installation and Operations Guide***.

NASTRAN Executive System Parameter Modification

Specifies values for certain Executive System operational parameters called system cells.

Format:

NASTRAN cellnamei=expressioni, ..., cellname*n*=expression*n*
or
NASTRAN SYSTEM(i)=expressioni, ..., SYSTEM(n)=expression*n*

Describer	Meaning
cellnamei	System cell names from **Table 1-1**.
SYSTEM	Specifies the system cell number.
expression	See DEFINE statement for description.
i	System cell number from **Table 1-1** or from the SYSTEM common block described in the ***User Modifiable MSC.Nastran User's Guide***.

Remarks:

1. The NASTRAN statements may appear anywhere in the File Management Section. The NASTRAN statement may also be specified in Runtime Configuration (RC) files. See "**Customizing Command Initialization and Runtime Configuration Files**" on page 68 of the *MSC.Nastran 2005 r3 Installation and Operations Guide.*
2. System cell values and their associated cell names may also be set with the DEFINE statement. They may also be set or values returned with the DMAP PUTSYS and GETSYS functions and the PARAM module. See "**PUTSYS, GETSYS**" on page 31 of the *MD Nastran 2006 DMAP Programmer's Guide.*
3. More than one NASTRAN statement and / or DEFINE statement may be present and, if a system cell is specified more than once among these statements, then the last specification takes precedence.
4. The expression will use type conversion rules based on the type (i.e., integer, real, or logical) of the cellname, as defined on a previous DEFINE statement (see the DEFINE statement for conversion rules).
5. If expression is omitted, the system cell associated with the cellname will be assigned the value as set on a previous DEFINE statement.

Examples:

1. Either of the following statements could be used to change the default value for block size.

```
NASTRAN SYSTEM (1) = 4097
or
NASTRAN BUFFSIZE = 4097
```

or if a prior DEFINE statement had defined a keyword MY_SYSBUF to the value 4097 then the following code could be used:

```
NASTRAN SYSTEM(1)=MY_SYSBUF
or
NASTRAN BUFFSIZE=MY_SYSBUF
```

The following statement is used to request execution of MSGMESH:

```
NASTRAN MESH
```

2. **Table 1-1** gives a summary of the recommended system cells.

Table 1-1 System Cell Summary

System Cell Name (Number)	Function and Reference
BUFFSIZE (1)	Specifies the number of words in a physical record. Also called block length.
F06 (2)	Specifies FORTRAN unit number for standard output file (.f06). (Integer≥0 (a value of 0 sends the results to the log file); Default=6)
NLINES (9)	Specifies the number of lines printed per page of output. "**LINE**" on page 349.
MAXLINES (14)	"**MAXLINES**" on page 358.
ECHO (19)	"**ECHO**" on page 278.
METIME (20)	Minimum time for execution summary table message. "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide*.
APP (21)	Approach Flag. See the "**APP**" on page 114 Executive Control statement. If APP HEAT is specified, then this system cell is set to 1.
MACHTYPE (22)	Machine type.
DIAGA (25)	Alternate method to set DIAGs 1 through 32. "**DIAG**" on page 122.
CONFIG (28)	Machine subtype.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
MESH (31)	Requests execution of MSGMESH.
ADUMi (46 - 54)	Dummy element flag., i=1 through 9.
IPREC (55)	Machine precision. 1=long word, 2=short word.
HEAT (56)	"**APP**" on page 114.
	0 Structural analysis. (Default)
	1 Heat transfer.
HICORE (57)	Working Memory. "**Managing Memory**" on page 138 of the *MD Nastran 2006 Installation and Operations Guide*
DIAGB (61)	Alternate method to set diagnostics 33 through 64. "**DIAG**" on page 122.
PUNCH (64)	Specifies FORTRAN unit number for PUNCH file (.f07). (Default=7)
MPYAD (66)	Selects / deselects multiplication methods. "**MPYAD**" on page 1447 of the *MD Nastran 2006 DMAP Programmer's Guide*.
SOLVE (69)	Controls matrix decomposition. Same as "**DECOMP**" on page 1000 of the *MD Nastran 2006 DMAP Programmer's Guide* and the "**Option Selection**" on page 69 of the *MSC.Nastran Numerical Methods User's Guide*.
	0 or -1 Print up to 50 messages for null columns and zero diagonals. (Default=-1)
	1 Terminates execution when first null column is encountered.
	2 Suppress printing of message when a null column is encountered.
	4 Terminates execution when first zero diagonal term is encountered.
	16 Place 1.0 in diagonal position for all null columns and proceed with the decomposition.
	32 Terminates execution on zero diagonal term.
	64 Exit after execution of preface for symmetric decomposition.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
FBSOPT (70)	Selects forward-backward substitution methods. "**FBS**" on page 1173 of the *MD Nastran 2006 DMAP Programmer's Guide* and the "**FBS Method Selection**" on page 68 of the *MSC.Nastran Numerical Methods User's Guide*.
DELFF (77)	Deletes form feeds.
REAL (81)	Specifies the amount of open core memory that certain numerical modules will be restricted to.
DBSET	Database neutral file set. "**SubDMAP DBFETCH**" on page 976 of the *MD Nastran 2006 DMAP Programmer's Guide*.
DMAP (82)	Allows NOGO to operate. See "**Processing of User Errors**" on page 41 of the *MD Nastran 2006 DMAP Programmer's Guide*.
IORATE (84)	Input/output rate. "**Time Estimates**" on page 15 of the *MSC.Nastran Numerical Methods User's Guide*.
F04 (86)	Specifies FORTRAN unit number for Execution Summary Table (.f04). "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide*. (Integer≥0 (a value of 0 sends the results to the log file); Default=4)
RADMTX (87)	Type of radiation exchange coefficients, "**RADMTX**" on page 2307. =1 Direct input of a symmetric SCRIPT-AF matrix on RADMTX and RADLST entries is allowed. Due to the symmetry, only one-half of the RADMTX may be entered. =2 Direct input of a unsymmetric SCRIPT-AF matrix on RADMTX and RADLST entries is allowed. Due to the symmetry, the full matrix must be specified on the RADMTX entries. =3 If you are running a view factor calculation in an opened enclosure, NASTRAN assumes that the radiation will be lost to space at absolute zero degree Kelvins. You can set the SYSTEM(87)=3 so that radiation will not be lost to space.
RADLST (88)	Print radiation area summary. "**RADLST**" on page 2302.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
PARALLEL (107)	Parallel processing deselection (deactivation) for matrix operations. "**Keywords**" on page 288 of the *MD Nastran 2006 Installation and Operations Guide.* 0: Deactivate parallel processing. 1 through 1023: Number of processors. The desired number of processors is summed with the values below in order to deactivate parallel processing methods in the following matrix operations: 1024: Forward-backward substitution. 2048: Decomposition. 4096: Multiplication. 8192: Householder in eigenvalue extraction. 65536: Element matrix assembly. 262144: Sparse decomposition. 524288: Sparse forward-backward substitution. If PARALLEL is set simply to the number of processors, then parallel processing is selected in all operations above.
NEWHESS (108)	Request complex eigenvalue method. Please see the "**EIGC**" on page 1509 entry. *MSC.Nastran Numerical Methods User's Guide.*
(109)	Controls DMAP execution: 0: Do not execute DMAP instruction if all outputs are previously computed. 1: Always execute DMAP instruction. (Default)
OS (111)	Operating system.
OSLEVEL (112)	Operating system level.
MMODEL (113)	Machine submodel.
BUFFPOOL (114)	Bufferpool size. "**Keywords**" on page 288 of the *MD Nastran 2006 Installation and Operations Guide.*

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
ATTDEL (124)	Controls the automatic assignment of the delivery database. "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*. See also "**Creating and Attaching Alternate Delivery Databases**" on page 204 of the *MD Nastran 2006 Installation and Operations Guide* 0 Enables automatic assigning. (Default) -1 Disables automatic assigning.
NOKEEP (125)	Controls NOKEEP option of the RESTART File Management statement. 0 Disable NOKEEP. 1 Enable NOKEEP.
SPARSE (126)	Sparse matrix method selection. For unsymmetric sparse matrix decomposition method selection, see cell number 209. The following values may be summed in order to select sparse matrix methods in the operations listed below: 0 Deactivate sparse methods. 1 Multiplication. 8 Symmetric decomposition. 16 Forward-backward substitution. The default is 25, which is the sum of all values.
UPDTTIM (128)	Specifies database directory update time. "**DBUPDATE**" on page 85 FMS statement. 0 Do not update. >0 Time, in minutes, between database directory updates.
SMPYAD67 (129)	Select pre-Version 67 method in the SMPYAD module. "**SMPYAD**" on page 1712 of the *MD Nastran 2006 DMAP Programmer's Guide*. 0 Use current method. (Default) 1 Use pre-Version 67 method.
MAXDBSET	The maximum number of online DBsets attached to the run.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
AUTOASGN (133)	Controls auto-assigning of dbsets. Sum the desired values. (Default=7). "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide.* 0 No databases are automatically assigned. 1 Only the primary database is automatically assigned. 2 Only the delivery database is automatically assigned. 4 Only located databases are automatically assigned.
TSTAMP (135)	Controls timestamp checking of DBsets. 0 Do not check. 1 Check. (Default) 2 Same as 1 and print diagnostics.
QUADINT (141)	Specifies quadratic or linear interpolation for the line search method in nonlinear analysis. 0 Quadratic interpolation. (Default) 1 Linear interpolation.
SCR300 (142)	Requests creation of SCR300 partition on SCRATCH DBset. "**INIT**" on page 95 FMS statement. 1 Do not create SCR300 partition. 2 Create SCR300 partition. (Default)
LOCBULK (143)	LOCBULK=1 or 2 specifies that Bulk Data is being obtained via the DBLOCATE FMS statement. NASTRAN LOCBULK=2 is specified when no Bulk Data entries, except for PARAM entries, are to be deleted or added. All PARAM entries must be respecified. All other entries will be ignored and, if present, may increase CPU times in XSORT and IFP. With LOCBULK=2, the XSORT and IFP modules will not reprocess the Bulk Data Section stored in the SEMAP run. Also, GP1, TASNP2, SEP1 and SEP1X modules will be skipped. 0 is the default, which assumes the RESTART FMS statement. This system cell is recognized only in SOLs 101 through 200.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
(144)	RESTART FMS statement existence flag. Set to 1 if RESTART statement is present.
BFGS (145)	Selects strategies of BFGS updates for the arc-length methods in nonlinear analysis. Please see the *MSC.Nastran Nonlinear Handbook*. 0 Update Δu_R and Δu_p at every iteration with 2. (Default) 1 Update Δu_R only with 2. 2 Update Δu_R only with 2*.
FBSMEM (146)	Reserves (n x BUFFSIZE x 3) memory for faster solution in the Lanczos method of eigenvalue extraction. Default=1. A value of 2 increases the memory reserved by 200%, a value of 3 increases the memory reserved by 300%, etc. See the "**EIGRL**" on page 1522. See system cell 229 for disk space saving feature.
UWM (147)	SYSTEM(147)=1 issues User Warning Message for a DMAP parameter appearing on a CALL statement that has an inconsistent authorization in the called subDMAP. 0 is the default, which means no message is issued.
DBVERCHK (148)	In general, databases are not compatible between major releases, thus a check is performed in MD Nastran to ensure that the major version which created the database is the same as that being executed. Since specific data on the database may be compatible, SYSTEM(148) allows this check to be circumvented. However, circumventing the check may lead to problems later in the run. 0 Check is performed. (Default) 1 Check is not performed
SCR300DEL (150)	Sets minimum number of blocks of SCR300 partition of SCRATCH DBset at which it is deleted. "**INIT**" on page 95 FMS statement. (Default=100)
(151)	Requests spill or no spill of the SCR300 partition of SCRATCH DBset. "**INIT**" on page 95 FMS statement. (Default=0)
DBLAMKD (155)	Differential stiffness formulation for CBEAM and CTETRA elements.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
	0 Current formulation. (Default) 1 Pre-Version 67 formulation.
(166)	Controls sparse symmetric decomposition. Sum the desired values. (Default=0) 0 No action. 1 If insufficient core is encountered, then switch to conventional decomposition and continue. (Default) 2 Print diagnostics. 4 Do not issue fatal message if maximum ratios are exceeded. Although high maximum ratios may be printed, they will not cause job termination. This applies to the DCMP, DECOMP, REIGL, and LANCZOS modules.
LDQRKD (170)	Selects the differential stiffness method for CQUAD4 and CTRIA3 elements: 0 Version 68, improved method. (Default) 1 Pre-Version 68 method.
OLDQ4K (173)	Requests the pre-Version 68 CQUAD4 element stiffness formulation. No value is required after the keyword. Equivalent to SYSTEM(173)=1. 0 Default. 1 Requests pre-V68 QUAD4 Formulation. 2 Requests V68 - V70.5 QUAD4 Formulation.
Q4TAPER (189)	Specifies the maximum allowable value of taper for CQUAD4 element. Taper is computed by connecting opposite grid points and computing the area of the enclosed triangles. Another way to think of taper is the ratio of the areas on the two sides of a diagonal. (Real $\geq$ 0.0; Default=0.5)
Q4SKEW (190)	Specifies the minimum allowable value of skew for the CQUAD4 element. Skew is the angle measured in degrees between the lines that join opposite midsides. (Real $\geq$ 0.0; Default=30.0)

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
TETRAAR (191)	Specifies the maximum allowable aspect ratio of the longest edge to the shortest altitude for the CTETRA element. (Real ≥ 0.0; Default=100.0)
DISKSAVE (193)	Lanczos High Performance Option: Controls whether the matrix/vector multiply is saved in a scratch file or recomputed at every iteration. CRAY and NEC system set the value automatically for optimum performance: 0 Save (Default for all machines except CRAY and NEC). 1 No save (Ignored on CRAY and NEC). 2 Save (Overrides auto-logic on CRAY and NEC). 3 No Save (Overrides auto-logic on CRAY and NEC).
FASTIO (194)	Lanczos High Performance Option: Controls input/output in orthogonalization/normalization routines: 0 UNPACK/PACK (Default) 1 GINO READ/WRITE
FRQSEQ (195)	Lanczos High Performance Option: 100<Exponent for the rational function used to determine segment boundaries. See also the ALPH field on the "**EIGRL**" on page 1522 Bulk Data entry. (Integer; Default=0, which means equal segments.)
SCRSAVE (196)	Lanczos High Performance Option: Controls reuse of scratch files in segment logic. 0 Do not reuse. (Default) 1 Reuse.
NUMSEG (197)	Lanczos High Performance Option: Number of segments. See also the NUMS field on the "**EIGRL**" on page 1522 Bulk Data entry. (Default=1)
MINFRONT (198)	Lanczos High Performance Option: Minimum front size. (The default value is machine dependent.)

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
MASSBUF (199)	Lanczos High Performance Option: Half the number of buffers to set aside for storing the mass matrix in core. (Default=1, which means 2 buffers will be used.)
NSEGADD (200)	Number of segments in the element error table that is generated in adaptive analysis. (Default=2)
CORDM (204)	Specifies the default value for CORDM field on the PSOLID entry. (Integer ≥-1; Default=0)
(205)	Rank to use for real symmetric sparse decomposition high rank update. Default is hardware dependent.
DCMPSEQ (206)	Selects ordering method for sparse matrix decomposition. 0 Method selected automatically in symbolic factoring phase. (Default for NEC) 1 Minimum degree ordering. 2 Modified minimum degree ordering for indefinite matrices. 3 No ordering (uses given sequence). 4 Extreme ordering.(Default for all other machines) 8 METIS ordering. Metis was developed by George Karpis and Vipin Kumar at the University of Minnesota. More information may be found at http://www.cs.umn.edu/~karypis/metis. 9 Selects better of METIS and MMD. 68 This option reduces the number of non-zero factors in the sparse decompensation method for all machines other than the NEC.
USPARSE (209)	Unsymmetric sparse matrix method selection for the decomposition and forward-backward substitution operations. 0 Deactivate 1 Activate (Default)

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
PUNCHTYPE (210)	Used to control punch formula. 0 "Old" punch, default in MSC.Nastran 2001 and earlier versions. 1 "New" punch, default in MSC.Nastran 2004 and uses the NDDL. 2 Same as 1 except the line numbers are eliminated.
CHEXAINT (212)	Specifies CHEXA element's integration rule for p-adaptive analysis and p=2x2x2 (only). 0 Reduced. (Default) 1 Full.
DISTORT (213)	Element distortion fatal termination override. Applies to all p-elements and the TETRA h-elements. 0 Terminate run. (Default) 1 Do not terminate run.
T3SKEW (218)	Allows the user to control the minimum vertex angle for TRIA3 elements at which USER WARNING MESSAGE 5491 is issued. See the description of the CTRIA3.
(219)	Rank to use for complex symmetric sparse decomposition high rank update. (Default = 1)
(220)	Rank to use for real unsymmetric sparse decomposition high rank update. (Default = 1)
(221)	Rank to use for complex unsymmetric sparse decomposition high rank update. (Default = 1)
MEMSAVE (229)	Specifies space-saving method for the old Lanczos method of eigenvalue extraction (system(273)=1). "**EIGRL**" on page 1522. 0 No space savings. (Default) 1 Do not write factors to disk which reduces scratch space usage 67%. However, CPU costs will increase. Controls module BEGN and END messages in .f04 file.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
	0 Print everything. (Default) 1 Print major modules only. 2 Print sub-modules only. 3 No printing.
(253 - 262)	SYSTEM(252) to (262) have been set aside for user DMAPS. MSC will not use these values in its code in present or future versions. The SSSAlter library may use this range.
MAXSET (263)	Controls the default number of vectors in block or set for Lanczos Eigenvalue extraction. See "**EIGRL**" on page 1522. Default is 7 for most machines but it is machine dependent.
QUARTICDLM (270)	A value of 1 selects the new quartic formulation of the doublet lattic kernel (N5KQ) while 0 selects the original quadratic form. (Default=0)
(273)	A value of 1 selects the old Lanczos shift logic from Version 70 and prior systems. (Default=0)
(275)	Specifies the timeout for ISHELL in seconds. Values greater than 2,678,400 (31 days) will be set to 31 days.
MINDEF (303)	Indefinite Mass Matrix Check, the Default=1 does not perform the check. If MINDEF>0, then check is not performed. If MINDEF<0, then epsilon is set to -1.E(MINDEF). If MINDEF=0, then MINDEF defaults to -6.
MPERTURB (304)	Perturbation factor for indefinite mass matrix. The default=1 does not perturb the mass. If MPERTURB>0, then the mass is not perturbed. If MPERTURB<0, then the mass 1.E(MPERTURB) is added to the diagonal terms of the mass matrix. If MPERTURB=0, then MPERTURB defaults to -6. The perturbed mass matrix is used in the subsequent eigenvalue analysis.
(309)	If set to 1, requests the pre-Version 70.7 HEXA8 element stiffness formulation. (Default=0)

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
OLDRBE3 (310)	If set to 1, requests the pre-Version 70.7 RBE3 formulation. (Default=0)
TBCMAG (311)	Changed the stiffness to 1.0E2 if using thermal conductivity in Btu / sec / in.F. See the Bulk Data entry, "**TEMPBC**" on page 2549 for more information. The default stiffness is 1.0E10.
XMSG (319)	If set to 1, gives extended error messages. (Default=0)
OLDDAREA (320)	Do not convert DAREA Bulk Data entries for grid and scalar points to equivalent FORCE / MOMENT / SLOAD Bulk Data entries (equivalent to SYSTEM(320)=-1) Controls the conversion of DAREA Bulk Data entries for grid and scalar points to equivalent FORCE / MOMENT / SLOAD Bulk Data entries as appropriate. 0 Perform the conversion, but do not give details of the conversion. (Default) N Perform the conversion and give details of the first N such conversions. -1 Do not perform the conversion.
RSEQCONT (357)	Default = 0 Setting this system cell to 1 causes all continuation fields to be ignored and treated as blank. If set to 1, the continuation entries must immediately follow the parent.
QLHOUL (359)	=0 Use the user-requested eigensolution method. <>0 When LAN is requested, switch to AHOU if the number of DOFs sent to the eigensolver is ≤ "nswitch", an input parameter to the READ module. This parameter has an MPL default of 20. It may be set to other values in the solution sequences, depending on the context. When HOU, MHOU, or AHOU is selected, switch to the new Householder-QL solution. (Default=1)
PRTPCOMP (361)	If set to 1, then the equivalent PSHELLs and MAT2s from PCOMPs are printed to the .F06 file. (Default = 0, suppresses this printout)

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
STRICTUAI (363)	A value of 1 accepts strict UAI/Nastran Bulk Data entries. (Default=0)
STPFLG (366)	Selects the SUBCASE or STEP layout when there is a number of SUBCASE commands and no STEP command in a Case Control file for SOL 400. (Default=0) 0 Keep all SUBCASE commands in the Case Control file and insert a "STEP 1" for each SUBCASE. 1 Convert all the SUBCASE IDs into STEP IDs, and then insert a "SUBCASE 1" before the first STEP.
QRMETH (370)	Selects the formulation for the QUADR/TRIAR and QUAD4/TRIA3 elements. The default for the QUADR/TRIAR uses the new formulation. The default for the QUAD4/TRIA3 uses the standard formulation. (Default = 0) For QRMETH=4 or 5, MD Nastran converts the QUAD4/TRIA3 formulation into the QUADR/TRIAR formulation without the drilling stiffness or the QUADR/TRIAR formulation. Some of the output requests for QUAD4/TRIA3 are not available for QUADR/TRIAR, for example, the CUBIC option on the STRESS Case Control command. In this case, the equivalent QUADR/TRIAR options are used. 0 Selects the new formulation for QUADR/TRIAR. 1 Selects the old formulation for QUADR/TRIAR. 2 Converts QUADR/TRIAR into QUAD4/TRIA3 using the alternate QUAD4/TRAI3 formulation. 3 Converts QUADR/TRIAR into QUAD4/TRIA3 using the standard QUAD4/TRIA3 formulation. 4 Selects the alternate formulation for QUAD4/TRIA3. 5 Converts QUAD4/TRIA3 into QUADR/TRIAR using the new QUADR/TRIAR formulation.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
PARAMCHK (372)	DMAP parameter initialization check If PARAMCHK=0, then issue User Fatal message for an input parameter that is not utilized in a type statement in the subDMAP argument. If PARAMCHK=1, then issue User Fatal message for the initialized parameter. (Default=0)
TZEROMAX (373)	Controls time step adjustment in nonlinear transient. >0 Maximum number of times to return to time zero. =0 No initial time step adjustment (identical to V2001), default for SOL 400 with GAP elements. <0 No limit on DT adjustment. =4 Default for SOL 400 without GAP element or for SOL 129.
(382)	Disconnects the external response server(s) =0 Keep the connection as in the existing scheme. (Default) =1 Disconnect the server(s)
(383)	Sets timer for the external response server. =0 Use default timeout value: 100,000. (Default) >0 New timeout value.
(384)	Sets timer for the client (MD Nastran) communication with the DR3 server. =0 Use default timeout value: 10,000. (Default) >0 New timeout value.
SPCHLSKY (385)	Control Sparse Cholesky DCOMP. If SPCHLSKY=0, then do not execute Sparse Cholesky. If SPCHLSKY>0, then execute Sparse Cholesky. (Default=0)
NOLIN (386)	Scale factor for controlling adaptive behavior for the nonlin entries. 0 Bisection is suppressed (same as MSC.Nastran 2001). .001 Increase accuracy slightly.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
	0.01 Increase accuracy a little more.
	1.0 Allow full adaptive bisection. (Default)
KRYLOV1 (387)	Fast direct frequency response option.
	-1 Yes.
	0 No. (Default)
KRYLOV2 (388)	Options related to fast direct frequency response analysis Selects subspace generation method.
	1 Lanczos. (Default)
	2 Arnold.
KRYLOV3 (389)	Options related to fast direct frequency response analysis. Defines exponent of relative accuracy.
	-4 Error<1.0E-4 (default).
	-6 Error<1.0E-6.
KRYLOV4 (390)	Options related to fast direct frequency response analysis Defines pole selection distance
	0 Next pole is next unconverged frequency. (Default)
	2 2*next frequency distance.
	-2 1/2*next frequency distance.
KRYLOV5 (391)	Options related to fast direct frequency response analysis. Selects decomp/fbs trade-off parameter.
	1 fbs time = decomp time (default).
	2 fbs time - 2*decomp time.
	-2 fbs time - 1/2*decomp time.

Table 1-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
BARMASS (398)	Allows the user to select the Bar Torsional Mass Moment of Inertia. If set to 0, request the pre-MSC.Nastran 2004. (Default = 0) If set to greater than 0, the Torsional Mass Moment of Inertia Term is included in the Mass Matrix Formulation of Bar elements. For both values of COUPMASS, the torsional inertia is added. For COUPMASS = 1, the Axial Mass will be consistent rather than coupled.
ITRFMT (401)	Select the convergence parameter computation method and the divergence solution checking method for SOL 400. (Default=0) 0 Use the SOL 400 method. -1 Use the method similar to SOL 106.
DPBLKTOL (402)	Specifies Bulk Data tolerance value for GRID, CORD2C, CORD2R, and CORD2S entries. If DPBLKTOL<0.0, then do not remove duplicate entries. If DPBLKTOL=0.0, then check specified Bulk Data entries for exact physical match and remove duplicates. If DPBLKTOL>0.0, then perform the DPBLKTOL=0.0 check and additionally GRID entry as duplicate if $\{\|x1(i) - x2(i)\| \le \text{DPBLKTOL}; i = 1, 2, 3$ and $(cp1 \ne cp2$ and $cp1 \cdot cp2 = 0)$ and $(cd1 \ne cd2$ and $cd1 \cdot cd2 = 0)$ and $(ps1 = ps2)$ and $(seid1 = seid2)\}$ using entry with $cpi \ne 0$ and $cdi \ne 0$ if possible. (Default=0)
OP2NEW (403)	Selects the additional version information in the OUTPUT2 file. If OP2NEW=0, then leave alone and unidentified, pre-2004 convention. If OP2NEW=1, then add Version Major, Minor, Special to Tape Label and change IFP Datablock Locate Code word 3. (Default=0)
DEF_DENS (408)	Set DEFAULT value for the MODEL_CHECK executive statement MAT_DENSITY=DEFAULT operation. (Default = 0.0)
DEF_TECO (410)	Set DEFAULT value for the MODEL_CHECK executive statement MAT_TECO=DEFAULT operation. (Default = 0.0)

Table 1-1 System Cell Summary (continued)

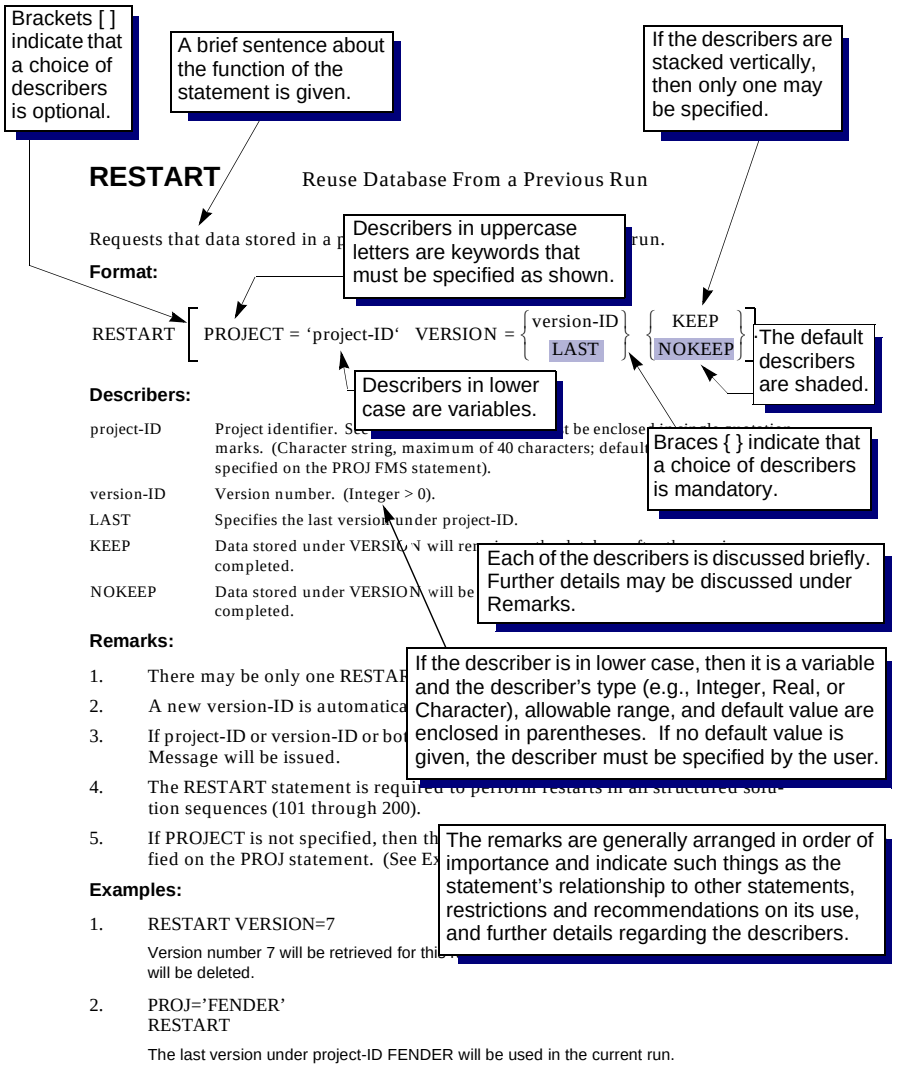
System Cell Name (Number)	Function and Reference
DEF_TEIJ (411)	Set DEFAULKT value for the MODEL_CHECK executive statement MAT_TEIJ=DEFAULT operation. (Default = 0.0)
DEF_DAMP (412)	Set DEFAULKT value for the MODEL_CHECK executive statement MAT_DAMP=DEFAULT operation. (Default = 0.0)
OPTCOD (413)	Specifies which optimization code to be used in SOL 200 (Default = 0). 0 MSCADS (Design Optimization Option) and BIGDOT (Topology Optimization Option) 1 DOT Optimizer 2 BIGDOT Optimizer
(428)	If nonzero, requests the pre-version 2005 r3 method for computing thermal expansion in CHEXA, CPENTA, and CTETRA elements.
NLRGAP (431)	Scale factor for controlling adaptive bisection behavior for the NLRGAP entries. New adaptive bisection will be activated if the contact force magnitude changes more than the preset value. = 0.0 The old bisection method of NLRGAP is reactivated (same as MSC.Nastran 2005, also see System Cell 386) < 0.0 The present value is 1.E+9 (Default) > 0.0 The preset value is computed as 1000.0 / NONLRGAP

CHAPTER

2 File Management Statements

- Key to Descriptions
- File Management Statement Descriptions

2.1 Key to Descriptions



The File Management Section (FMS)

The File Management Section (FMS) is primarily intended for the attachment and initialization of Database sets (DBsets) and FORTRAN files. The initialization of DBsets includes specification of their maximum size, member names, and physical filenames. The initialization of FORTRAN files includes the specification of their filenames, FORTRAN unit numbers, and FORTRAN attributes.

In most classes of problems that use MD Nastran solution sequences (SOL), no File Management statements are required because a default File Management Section is executed at the beginning of every run. The default File Management Section is described in the "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*. If a restart is desired, then the RESTART statement is required. All other solutions may not be restarted. If the problem is large in terms of requiring significant amounts of memory or disk space, then the INIT, ASSIGN, and EXPAND statements may be required. If any FORTRAN files are required, then the ASSIGN statement is required; for example, the OUTPUT2 DMAP module. The ASSIGN statement is also required to assign databases for DBLOCATE, DBLOAD, and DBUNLOAD. Special database operations are performed by the DBLOCATE, DBLOAD, DBUNLOAD, DBLCLEAN, ACQUIRE, DBDICT, DBFIX, DBSETDEL, DBUPDATE, and PROJECT statements.

File Management Statement Summary

The following is a summary of all File Management statement:

$	Comment statement.
ACQUIRE	Selects NDDL schema and MD Nastran Delivery Database.
ASSIGN	Assigns physical files to DBset members or special FORTRAN files.
CONNECT	Group geometry data by evaluator and database.
DBCLEAN	Deletes selected database version(s) and/or projects.
DBDICT	Prints the database directory in user-defined format.
DBFIX	Identifies and optionally corrects errors found in the database.
DBLOAD	Loads a database previously unloaded by DBUNLOAD.
DBLOCATE	Obtains data blocks and parameters from databases.
DBSETDEL	Deletes DBsets.
DBUNLOAD	Unloads a database for compression, transfer, or archival storage.

DBUPDATE	Specifies the time between updates of the database directory.
ENDJOB	Terminates a job upon completion of FMS statements.
EXPAND	Concatenates additional DBset members to an existing DBset.
INCLUDE	Inserts an external file in the input file.
INIT	Creates a temporary or permanent DBset.
NASTRAN	Specifies values for system cells.
PROJ	Defines the current or default project identifier.

The FMS statements are executed in the following order regardless of their order of appearance in the input file:

NASTRAN, DEFINE
RFINCLUDE, INCLUDE
ASSIGN, INIT, EXPAND, DBUPDATE
PROJECT
DBCLEAN
DBFIX
DBDICT(1)
DBSETDEL
ACQUIRE
RESTART
DBLOCATE
DBUNLOAD
DBLOAD
DBDIR (2), DBDICT(2)
ENDJOB

If DBDICT is specified before any of the FMS statements DBSETDEL through DBLOAD, then the directory printout will reflect the processing of DBCLEAN and DBFIX only. If DBDICT is specified after DBSETDEL through DBLOAD, then the directory printout will reflect the processing of all statements in the FMS Section. We recommend that the DBDICT statements be specified last in the FMS Section. Multiple DBLOCATE, DBLOAD, or DBUNLOAD statements are processed in the order in which they appear. If the ENDJOB statement is specified, then only the File Management Section is processed and the Executive Control, Case Control, and Bulk Data Sections are ignored.

2.2 File Management Statement Descriptions

File Management statements may be abbreviated down to the first four characters as long as the abbreviation is unique relative to all other statements. Each statement is described as follows:

Description

A brief sentence about the function of the statement is given.

Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the statement line is longer than 72 columns, then it may be continued to the next line with a comma as long as the comma is preceded by one or more spaces and no keyword is split across lines. For example:

```
DBLOCATE DATABLK=(KAA)  ,
   WHERE(PROJECT='FRONT BUMPER' AND  ,
   SEID>0 AND VERSION=4)  ,
   LOGI=MASTER3
```

However, if a filename is to be continued on the next line, no space must precede the comma, and the continuation line must have no leading spaces.

Example

```
ASSIGN     SDB='/jw/johannes/Projects/secret/Aero/Tests/wing/,
Modes/wing_modal.MASTER'
```

Note that all quote marks shown under formats and examples are right-handed single quotation marks and must be entered as such. For example:

PROJ='MYJOB'

Example

A typical example is given.

Describers and Meaning

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

Remarks

The remarks are generally arranged in order of importance and indicate such things as the FMS statement's relationship to other commands, restrictions and recommendations on its use, and further descriptions of the describers.

WHERE and CONVERT Clauses

The WHERE clause is used in the selection of items (data blocks and parameters) on the DBDICT, DBLOCATE, DBLOAD, and DBUNLOAD statements. The CONVERT clause modifies qualifier values of items selected by the WHERE clause on the DBLOCATE and DBLOAD statements.

The WHERE and CONVERT clauses specify values for PROJECT, VERSION, qualifiers, and DBSET. PROJECT specifies the project-ID that is originally defined on the PROJECT FMS statement at the time the project is created. VERSION specifies the desired version-ID under the project-ID. Qualifiers are used to uniquely identify items on the database with the same name. For example, data block KAA has SEID as one of its qualifiers, which is the superelement ID. An item may have more than one qualifier and the collection of all qualifiers assigned to an item is called a path. All data blocks and parameters with qualifiers are defined in the NDDL Sequence (NASTRAN Data Definition Language), see *MD Nastran 2006 DMAP Programmer's Guide*. Data blocks and parameters are defined on the DATABLK and PARAM NDDL statements. The DATABLK and PARAM statements specify the name of the data block, parameter, and also its pathname. The pathnames are defined on the PATH NDDL statement, which lists the qualifiers assigned to the path. Qualifiers are defined on the QUAL NDDL statement. DBSET specifies the desired DBset. The DBset of an item is specified after the LOCATION keyword on the DATABLK and PARAM NDDL statement.

The format of the WHERE clause is:

WHERE (where-expr)

where-expr is a logical expression that specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. If the result of the logical expression is TRUE for an item on the database then the item is selected. For example, WHERE(VERSlON=4 AND SElD<>2 AND SElD>0) selects all items under version 4 for all values of SEID greater than 0 except 2.

A simple where-expr is a comparison using the following relational operators =, >, <, ≤, ≥, >< or <>. For example, SElD>0 means if SEID is greater than zero, then the logical expression is true. Several simple where expressions may be joined into one where expression by the following logical operators: AND, OR, XOR, and EQV. The NOT operator may be used to negate a where expression. For example, NOT(SEID>0) is the same as SEID≤0. Arithmetic operations and DMAP functions may also be specified in the where expression (see the *MD Nastran 2006 DMAP Programmer's Guide.*)

If a qualifier in a where-expr is not a qualifier in the path of a specified item, then the where-expr is set to FALSE. If the where-expr does not contain a specification for all qualifiers in the path of an item, then the unspecified qualifiers will be wildcarded (i.e., quali=*, all values will be selected.) The default values of qualifiers, PROJECT, VERSION, and DBSET are described under the statement in which the WHERE clause is specified.

Examples of the WHERE clause are:

1. Select all items in the database for all superelements except 10 and 30 from Version 1.

   WHERE (VERSION=1 AND SEID>=0 AND NOT(SEID=10 OR SEID=30))

2. Select all entries in database on DBSET=DBALL from all projects and versions.

   WHERE(PROJECT=PROJECT AND VERSlON>0 AND DBSET='DBALL')

The CONVERT clause modifies project- and version-ID, DBset-name (see INIT statement), and qualifier values of items selected by the WHERE clause on the DBLOCATE and DBLOAD statements. It contains one or more assignment statements separated by semicolons. The format of CONVERT clause is:

CONVERT(PROJECT=project-expr; VERSION=version-expr; ,
DBSET=DBset-expr;quali=qual-expri[;...])

The PROJECT and VERSION statements modify the project-ID (see PROJECT FMS statement) and version-ID. The DBSET statement modifies the DBset-name. The value of quali will be replaced by qual-expri for selected items that have quali in their path. qual-expri is any valid expression (see "**Expressions and Operators**" on page 9 of the *MD Nastran 2006 DMAP Programmer's Guide*) containing constants or any qualifier

name defined in the path of the item. If qual-expri contains names of qualifiers not in the path of the selected item, then a fatal message is issued. If project-expr and/or version-expr produces a project- or version-ID which does not exist, then one will be created. Also, all version-IDs less than version-expr that do not exist will be created; but they will be "empty."

Examples of the CONVERT clause are:

1. Set qualifiers SEID, PEID, and SPC to constants 10, 20, 102 respectively.

   CONVERT(SEID=10;PEID=20;SPC=102)

   If more than one value of a qualifier is found for an item by the WHERE clause, then each value is processed in qual-expri to define the new qualifier value for each of the selected items. In the example below, if the original values of PEID were 1, 2, and 3; then the new values for the SElD qualifier will be 2, 4, and 6.

2. Set all values of qualifier SElD to be twice the value of the PEID qualifier.

   CONVERT(SElD=2*PElD)

$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Format:

$ followed by any characters out to column 80.

Example:

```
$ TEST FIXTURE-THIRD MODE
```

Remarks:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

ACQUIRE Selects NDDL Schema

Selects the NDDL schema and MD Nastran delivery database to be used for primary database creation.

Format:

ACQUIRE NDDL = { NDDL / nddl-name }

Describer	Meaning
NDDL	MD Nastran NDDL schema.
nddl-name	Name of a user NDDL schema specified on a COMPILE NDDL statement when the user NDDL was stored.

Remark:

This statement is used to specify the delivery database when the user wishes to create his or her own solution sequence, yet use the subDMAP objects or NDDL schema or both from the MSC-supplied delivery database.

Example:

The following requests the MD Nastran NDDL schema to be used when creating a new database.

```
ACQUIRE NDDL
SOL MYDMAP
COMPILE DMAP=MYDMAP,SOUOUT=USROBJ
.
.
.
LINK MYDMAP,SOLOUT=USROBJ
```

ASSIGN Assigns Physical File

Assigns physical file names or other properties to DBset members or special FORTRAN files that are used by other FMS statements or DMAP modules. Also, assigns physical name and / or other properties to Modal Neutral Files (MNF) for MD Nastran / ADAMS interface.

Format 1: Assign a DBset member name

```
ASSIGN  log-name=[= 'filename1'] [TEMP] [ DELETE] [ SYS='sys-spec' ]
                 [= *          ]
                 [= '*'        ]
```

Format 2: Assign a FORTRAN file

```
ASSIGN  logical-key [= 'filename2'] [UNIT = u]
                    [= *          ]
                    [= '*'        ]

   [[STATUS =] { NEW     }]
               { OLD     }
               { UNKNOWN }

   [[FORM =] { FORMATTED   }] [RECL = 1] [SIZE = s]
             { UNFORMATTED }
             { BIGENDIAN   }
             { LITTLEENDIAN}
             { LTLEND      }
             { <ostype>    }

[DEFER ] [TEMP   ] [ DELETE] [SYS = 'sys-spec']
         [DELZERO]
```

Examples:

1. Assign the DBALL DBset:

```
ASSIGN DB1='filename of member DB1'
INIT DBALL LOGI=(DB1)
```

2. Assign FORTRAN file 12 to the OUTPUT4 module using the ASCII option:

```
ASSIGN OUTPUT4='filename of FORTRAN file'
UNIT=12, FORM=FORMATTED
```

3. Assign FORTRAN file to the OPCASE using the ASCII option:

```
ASSIGN OPCASE='Filename of FORTRAN file', STATUS=NEW
```

4. Define SYS parameters for the SCR300 DBset file using the default file name.

```
ASSIGN SCR300 SYS='...'
```

5. Set the default OP2 file format to BIGENDIAN and assign two OP2 files, one to unit 12 with the file name "test_op2.12' and one to unit 35 with file name 'test_op2.35' in ASCII mode.

```
ASSIGN OP2 BIGENDIAN
...
ASSIGN OP2='test_op2.12' UNIT=12
ASSIGN OP2='test_op2.35' UNIT=35 FORM=FORMATTED
```

Describer	Meaning
log-name	The name of a DBset member name. log-name may also be referenced on an INIT statement after the LOGICAL keyword.
filename1	The physical filename assigned to the DBset member. If the default filename (if there is one) is to be used, filename1 may be omitted or specified as * or '*'. See Remark **6.**
logical-key	Specifies defaults for STATUS, UNIT, and FORM of FORTRAN files for other FMS statements, DMAP modules, punching and plotting operations.
filename2	The physical file name assigned to the FORTRAN file. If the default filename is to be used, filename2 may be omitted or specified as * or '*". See Remark **7.**
UNIT=u	u is the FORTRAN unit number of the FORTRAN file. If this describer is omitted and if filename2 is omitted, this ASSIGN statement will update the defaults for subsequent ASSIGN statements for the same logical-key value. See Remark **7.**
TEMP	Requests that the file associated with log-name or logical-key / UNIT be deleted at the end of the run.
DELETE	Requests that the file associated with logical-key / UNIT, if it exists before the start of the run, be deleted.

Describer	Meaning
DELZERO	Requests that the file associated with logical-key / UNIT be deleted at the end of the run if it is zero-length, that is, if it does not contain any data.
STATUS	Specifies whether the FORTRAN file is being created (STATUS=NEW) or has been created prior to the run (STATUS=OLD). If its status is not known, then STATUS=UNKNOWN is specified.
FORM	Indicates whether the FORTRAN file is written in ASCII (FORM=FORMATTED) or binary (FORM=UNFORMATTED, BIGENDIAN, LITTLEENDIAN, LTLEND, <ostype>) format. See Remark **10.**, **11.**, **12.**, **13.** and **18.**
DEFER	Defers opening / creating the specified file. That is, the file will not be opened / created during MD Nastran initialization. The file must be explicitly opened by the module or DMAP accessing the file, using, for example, FORTIO, before it can be used.
sys-spec	System specific or machine-dependent controls. For DBset files, these control I / O performance. For FORTRAN files, these are controls for IBM / MVS-type computers only. See Remark **14.**
RECL = l	The size of a block of input / output information specified in words. See Remark **15.**
SIZE = s	The number of blocks allocated to the DBC database. See Remark **16.**

Remarks:

1. The ASSIGN statement and its applications are discussed further in the "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide.*
2. The log-name or logical-key describer must be the first describer on the ASSIGN statement. All other describers may appear in any order. With the exception of log-name, logical-key, filename1, filename2, and sys-spec, describers and values longer than four characters may be abbreviated to four characters.
3. For FORTRAN files, the logical-key names and their default attributes are listed in **Table 2-1**. If a logical-key name is identified as "Assignable YES", then the defaults may be overridden on the ASSIGN statement.

4. Certain reserved names may not be used for log-names or logical-key names. These names are the logical names listed in **Table 2-1** that are identified as "Assignable NO". This list includes: SEMTRN, LNKSWH, MESHFL, LOGFL, INPUT, PRINT, INCLD1, and CNTFL. If they are used, then a fatal message is issued. Also unit numbers 1 through 10, 14, 16, 18, 19 and 21 should not be assigned. PUNCH and PLOT may be used but are not recommended.
5. If one of the logical-key names indicated in the Remarks **3.** and **4.** is not specified on this statement, then it is assumed to be a DBset member name log-name as shown in Format 1.
6. If the same log-name is used on more than one DBset ASSIGN statement, the following rules apply:
   a. If there is no current entry for the specified log-name, a new entry in the DBset tables will be created. If there is an existing entry for the specified log-name, the ASSIGN parameters will modify that entry instead of creating a new one.
   b. If filename1 is omitted or is specified as * or '*', the default file name or, if this is a second or subsequent ASSIGN statement for the same log-name, the previously specified file name (or default name if none was previously specified) will be used.
7. If the same logical-key is used on more than one FORTRAN file ASSIGN statement, the following rules apply:
   a. If filename2 is omitted (or specified as * or '*') and if the UNIT describer is omitted, the ASSIGN parameters will modify the system default entry for the logical-key, establishing the new defaults for any subsequent ASSIGN entry for the logical-key. Note, however, that any entries previously created with the same logical-key will not be modified by the new parameters specified on this ASSIGN statement.
   b. If the value specified by the UNIT describer matches the value for an entry created by a previous ASSIGN statement with a UNIT describer, then:
      - if the logical-key values are different, a UFM will be generated,
      - if the logical-key values are the same, the previous entry will be updated instead of having a new entry created.
   c. If the value specified by the UNIT describer does not match the value for an entry created by a previous ASSIGN statement with a UNIT describer, then a new entry will be created in the FORTRAN unit tables.

d. If the file name is omitted or specified as * or '*', the default file name or, if this is a second or subsequent ASSIGN statement for the same logical-key / UNIT combination, on previously specified file name (or default name if none was previously specified) will be used.

8. If it is necessary to execute the INPUTT4 and OUTPUT4 modules on the same unit, then specify ASSIGN OUTPUT4 only. The same is recommended for the INPUTT2 and OUTPUT2 modules.
9. STATUS, UNIT, and FORM are ignored if assigning a log-name (DBset member name).
10. FORM=FORMATTED must be specified for a unit when:
    - ASCII output is desired from the OUTPUT4 DMAP modules that processes the unit and, for Cray UNICOS, when ASCII input is supplied to the INPUTT4 DMAP module that processes the unit. See the ***MD Nastran 2006 DMAP Programmer's Guide***.
    - FORMAT=NEUTRAL is selected on the DBUNLOAD and DBLOAD FMS statements that process the unit. See the "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*.
    - The neutral file format is desired for the OUTPUT2 module and, for Cray UNICOS, when ASCII input is supplied to the INPUTT2 module.
11. For the DBUNLOAD, OUTPUT2 and OUTPUT4 modules, binary format may be requested using FORM=UNFORMATTED and, for all platforms except Cray UNICOS, using FORM=BIGENDIAN, FORM=LITTLEENDIAN, FORM=LTLEND or FORM=<ostype>. The FORM=BIGENDIAN, FORM=LITTLEENDIAN, FORM=LTLEND and FORM=<ostype> specifications are used when the generated output file is to be processed on a platform other than current platform. The format appropriate for the platform on which the file is to be processed (the target platform) must be specified. FORM=LTLEND is equivalent to FORM=LITTLEENDIAN. The FORM=<ostype> specification can by used as a convenience, allowing the desired output format to be specified using the target platform OS name or vendor (if there can be no ambiguity) instead of its actual binary file format. <ostype> can be one of the following:
    - AIX, FUJITSU, HPUX, IRIX, PRIMEPOWER, SOLARIS, SUPERUX or UXPV. These are equivalent to BIGENDIAN.
    - ALPHA, LINUX or WINDOWS. These are equivalent to LITTLEENDIAN.

See the *MSC.Nastran 2004 r3 Installation and Operations Guide* for further information on binary file formats.

12. For all platforms except Cray UNICOS, the FORM= describer is ignored for the DBLOAD, INPUTT2 and INPUTT4 modules. MD Nastran determines the actual file format when it accesses the specified file. If the FORM= describer is specified on an ASSIGN statement for these logical-keys, the syntax of the describer will be validated but will otherwise be ignored. Note, however, that the DBLOAD and INPUTT2 modules cannot process input files in other than the native binary format. That is, a binary file in BIGENDIAN format cannot be processed on a LITTLEENDIAN platform and vice versa. For MD Nastran on Cray UNICOS, the FORM= describer is required for the DBLOAD, INPUTT2 and INPUTT4 modules if the file does not have the default format.
13. For the DBUNLOAD and OUTPUT2 modules, if FORM is other than UNFORMATTED (or equivalent, e.g., BIGENDIAN on an AIX or HPUX platform and LITTLEENDIAN on a Linux or Windows platform), then only data blocks with an NDDL description are processed. (See the ***MD Nastran 2006 DMAP Programmer's Guide*** under the DATABLK statement.) An NDDL description is required for TYPE=TABLE and none is required for TYPE=MATRIX. The data block must be processed with FORM=UNFORMATTED if TYPE=UNSTRUCTURED, KDICT or KELM.
14. See the *MSC.Nastran 2004 r3 Installation and Operations Guide* for further information on sys-spec controls and on machine-dependent aspects of the ASSIGN statement. Also, if there are SYS specifications on more than one ASSIGN statement specifying the same log-name or logical-key / UNIT combination, the second and subsequent specifications will appended to the current SYS specification with a comma separator.
15. Currently the RECL keyword is used by the DBC module and has a default minimum of 1024 words. The maximum allowed is 65536 words and is used to increase the database capacity.
16. The SIZE keyword is used by the DBC module and has a default of 16777215. The maximum allowed is 2147483647 and is used to increase the database capacity. MSC.Patran releases before 2001 should use the defaults for RECL and SIZE or database verification failures will occur.
17. logical-key name MNF does not utilize UNIT or FORM.

18. For logical-key DBC, if the XDB file is new, the desired binary format may be specified in the same way as for the OUTPUT2 and OUTPUT4 modules, as described in Remark 11., except that FORM=FORMATTED is not valid. If the XDB file is not new, the FORM= describer is ignored and MD Nastran determines the format of the existing XDB file. For all platforms except Cray UNICOS, MD Nastran can read and or update an XDB file in any valid format except Cray UNICOS format.

Table 2-1 FORTRAN Files and Their Default Attributes

Logical Key Name	Physical Name	Unit No.	Form	Status	Assignable	Open	Access	Description/ Application
SEMTRN	sdir / data.f01	1	FORMATTED	NEW	NO	YES	SEQ.	Input Data Copy Unit
LNKSWH	sdir / data.f02	2	UNFORMATTED	NEW	NO	YES	SEQ.	Link Switch Unit
MESHFL	*sdir/data*.f03	3	FORMATTED	NEW	NO	YES	SEQ.	Input Data Copy Unit
LOGFl	*out*.f04	4	FORMATTED	NEW	NO	YES	SEQ.	Execution Summary Unit
INPUT	*data*.dat	5	FORMATTED	OLD	NO	YES	SEQ.	Input File Unit
PRINT	*out*.f06	6	FORMATTED	NEW	NO	YES	SEQ.	Main Print Output Unit
PUNCH	*out*.pch	7	FORMATTED	NEW	YES	YES	SEQ.	Default Punch Output Uniit
	authorize.dat	8	FORMATTED	OLD	NO	YES	SEQ.	Authorization File
INCLD1					NO			Available for Use
CNTFL					NO			Available for Use
INPUTT2	REQ	REQ	++	OLD	YES	NO	SEQ.	INPUTT2 Unit
OUTPUT2+	*out*.op2	12	UNFORMATTED*	NEW	YES	YES	SEQ.	OUTPUT2 Unit
INPUTT4	REQ	REQ	++	OLD	YES	NO	SEQ.	INPUTT4 Unit
OUTPUT4	REQ	REQ	UNFORMATTED*	NEW	YES	NO	SEQ.	OUTPUT4 Unit
PLOT	*out*.plt	14	UNFORMATTED	NEW	YES	YES	SEQ.	Plotter Output Unit
BULKECHO	*out*.becho	18	FORMATTED	NEW	YES	YES	SEQ.	Plotter Output Unit
OUTPUT2F	out	19	UNFORMATTED	NEW	YES		SEQ.	Named OUTPUT2 Pattern
OPCASE	REQ	22	FORMATED	NEW	YES		SEQ.	Available for Use
TOPDES	*out*.des	21	FORMATTED	NEW	YES	YES	SEQ.	Topology Optimization
DBC	*out*.xdb	40	UNFORMATTED	NEW	YES	YES	DIRECT	Database Converter Unit

Table 2-1 FORTRAN Files and Their Default Attributes (continued)

Logical Key Name	Physical Name	Unit No.	Form	Status	Assignable	Open	Access	Description/ Application
DBUNLOAD	REQ	50	UNFORMATTED*	NEW	YES	NO	SEQ.	DBUNLOAD FMS statement
DBLOAD	REQ	51	++	OLD	YES	NO	SEQ.	DBLOAD FMS statement
MNF	*out*.mnf	none	none	NEW	YES	NO	SEQ.	Interface for ADAMS/Flex
A502LU								Available for Use
DBMIG								Available for Use
USER FILE	REQ	REQ	REQ	REQ	YES	NO	SEQ.	Any User-Defined File

where:

Logical Key Name — specifies the logical-key NAME used on the ASSIGN statement.

Physical Name — specifies the default name used to open the file, i.e., the default filename2 name.
"REQ" means that this parameter is required in the ASSIGN statement from the user.

Unit No. — specifies the default FORTRAN unit number used by MD Nastran. "REQ" means that this parameter is required in the ASSIGN statement from the user.

Form — specifies the default FORM used when the file is opened.

Status — specifies the default STATUS used when the file is opened. "REQ" means that this parameter is required in the ASSIGN statement from the user.

Assignable — If "YES", the user may assign a physical file to this logical name.
If "NO", the unit (if any) and logical name are reserved by MD Nastran.

Open — If "YES", the file is opened by default.
If "NO", the file must be explicitly opened.

Access — If "SEQ", the file is opened for sequential access.
If "DIRECT", the file is opened for direct access.

sdir	The scratch directory specified using the “sdirectory” keyword.
data	The name of the input data file with all directory and extensions removed.
out	The directory and file prefix specified using the “out” keyword or taken by default.

Notes:

\+ The actual logical-key name for this is “OP2”. If you use “OUTPUT2” (even though this is still the logical-key name put out by MSC.Patran) you will get a user fatal message from MD Nastran.

* FORMATTED is required for neutral-format OUTPUT2 files and ASCII-format OUTPUT4 files.

++ For Cray Unicos, the default Form is UNFORMATTED. For all other platforms, the Form is ignored. See Remark **12.**

CONNECT Group Evaluator Data

To define a group of external geometric or beam cross-section entities. These entities should belong to the same evaluator-class (set of routines that process them), and in the case of geometric data, should reside on the same database. The GMCURV, GMSURF, PBARL and PBEAML Bulk Data entries refer to the groups defined here.

Format:

CONNECT [GEOMEVAL / BEAMEVAL / DRESP3] group evaluator 'path' 'data'

Describer	Meaning
group	Group name referenced by the GROUP field on DRESP3, GMCURV, GMSURF, PBARL and PBEAML Bulk Data entries.
evaluator	Identifies the particular class of evaluator to which the geometric, beam cross-section, or external response entities belong. Entities belonging to one evaluator-class are handled by the same set of (either MSC provided or user-provided) routines. for geometry, two classes of evaluators are provided internally with MD Nastran. They are MSCRPC (Rational Parametric Cubic) and MSCEQN (Generic Equation). For beam cross-sections, the class MSCBML (MSC Beam-Library) is provided internally. Users may develop custom evaluator libraries for geometry, beam cross-sections, or external responses and configure them for use with MD Nastran. See Remarks **4.**, **5.**, and **7.**
path	Optional pathname or filename used be evaluator. Path must be enclosed by single quotation marks if it contains lowercase characters.
data	Optional character string passed to the evaluator. Data must be enclosed by single quotation marks if it contains lowercase characters or embedded blanks.

Remarks:

1. CONNECT requests:
   - an external data base or evaluator, or
   - a user defined grouping for geometric data defined by GMCURV and GMSURF entries or beam cross-section data defined by PBARL and PBEAML entries.

2. Two reserved group names, MSCGRP0 and MSCGRP1, have been predefined for geometric entities. These names may be used in the GMCURV and GMSURF entries, without being defined explicitly by means of a CONNECT FMS statement. The group MSCGRP0 corresponds to the MSCRPC (Rational Parametric Cubic) evaluator and the group MSCGRP1 corresponds to the MSCEQN (Generic Equation) evaluator.
3. A single reserved group name, MSCBML0 has been predefined for beam cross-section entities. It may be used in the PBARL and PBEAML entries, without being defined explicitly by means of a CONNECT FMS statement. It corresponds to the MSCBML (MSC Beam-Library) evaluator.
4. Custom geometric evaluator libraries developed by users should comply with the *MSC.Nastran Geometry Evaluator Developer's Guide*.
5. Custom beam cross-section evaluator libraries developed by users should comply with the guidelines in the *MSC.Nastran V69 Release Guide*, Section 3.1 Beam Cross-Section Library and Appendix C: Adding Your Own Beam Cross-Section Library.
6. Custom responses developed by users should comply with the procedures and guidelines in "Support of External Response in SOL 200" on page 55 of the *MSC.Nastran 2004 Release Guide*.
7. Once developed, an evaluator may be configured as:
    - internal, where the evaluator routines are linked with the rest of the MD Nastran object modules to comprise the MD Nastran executable program, or
    - external, where the evaluator routines are linked with an MSC-provided server program to constitute an independent geometry server.

Examples:

1. CONNECT GEOMEVAL FENDER, CATIA, ' '/u/kiz/adp', 'Version=6 as of 1/31/93'

    In this case the user is requesting that all calculations on GMCURV and GMSURF Bulk Data entries that are grouped as FENDER use the CATIA database/evaluator. For each GMCURV and GMSURF entry where the group parameter is set to FENDER, appropriate evaluator routines will be called to initialize and perform computations on the curve or surface.

2. CONNECT GEOMEVAL HOOD, MSCRPC

   In this case the user is requesting that all calculations on GMCURV and GMSURF Bulk Data entries that are grouped as HOOD use the MD Nastran RPC database/evaluator. There is no need for additional routines to be supplied by the user since the MSCRPC and MSCEQN evaluator libraries are included in the standard MD Nastran delivery.

3. CONNECT GEOMEVAL DOOR, MSCEQN

   In this case the user is requesting that all calculations on GMCURV and GMSURF Bulk Data entries that are grouped as DOOR use the MD Nastran EQUATION database/evaluator. There is no need for additional routines to be supplied by the user since the MSCRPC and MSCEQN evaluator libraries are included in the MD Nastran standard delivery.

4. CONNECT BEAMEVAL HOIST, NEWBEAMS

   In this case the user is requesting that all calculations on PBARL and PBEAML Bulk Data entries that are grouped as HOIST use the NEWBEAMS evaluator. In this case, the user must supply the NEWBEAMS beam cross-section evaluator library, and configure it to function with the MD Nastran executable program.

5. CONNECT DRESP3 TAILWING, EXTRESP

   In this case the user is requesting that all calculations on DRESP3 Bulk Data entries, that are grouped as TAILWING, use the EXTRESP evaluator. Thus, the user must create the EXTRESP external response server program, and configure it to function with the MD Nastran executable.

DBCLEAN Deletes Database Versions and / or Projects

Deletes one or more versions and / or projects from the database.

Format:

DBCLEAN VERSION = {version-ID,*} [PROJECT={'project-ID',*}]

Describer	Meaning
version-ID	Version identifier of the database to be deleted.
*	Wildcard. All versions or projects to be deleted.
project-ID	Project identifier of the project to be deleted. (See the FMS statement, "**PROJ**" on page 101.)

Remarks:

1. There may be up to ten DBCLEAN statements in the FMS Section.
2. If no project-ID is given, the current project-ID is assumed.

Example:

```
DBCLEAN VERS = 7 PROJ = 'OUTER WING - LEFT'
```

The above example would delete from the database all data blocks and parameters stored under Version 7 of the project identified as OUTER WING - LEFT.

DBDICT Prints Database Directory Tables

Prints the following database directory tables:

- Data blocks described by an NDDL DATABLK statement.
- Parameters described by an NDDL PARAM statement.
- All unique paths (KEYs) and their qualifiers values.
- Qualifiers and their current values.
- Data blocks not described by an NDDL DATABLK statement.
- Parameters not described by an NDDL PARAM statement.
- Project and version information.

Basic Format:

The basic format of DBDICT specifies which tables to print and prints all items (data blocks and parameters) found in the directory. Also, the attributes (colnames) to be printed and the print format are predefined. Note that more than one table may be specified on the same DBDICT statement.

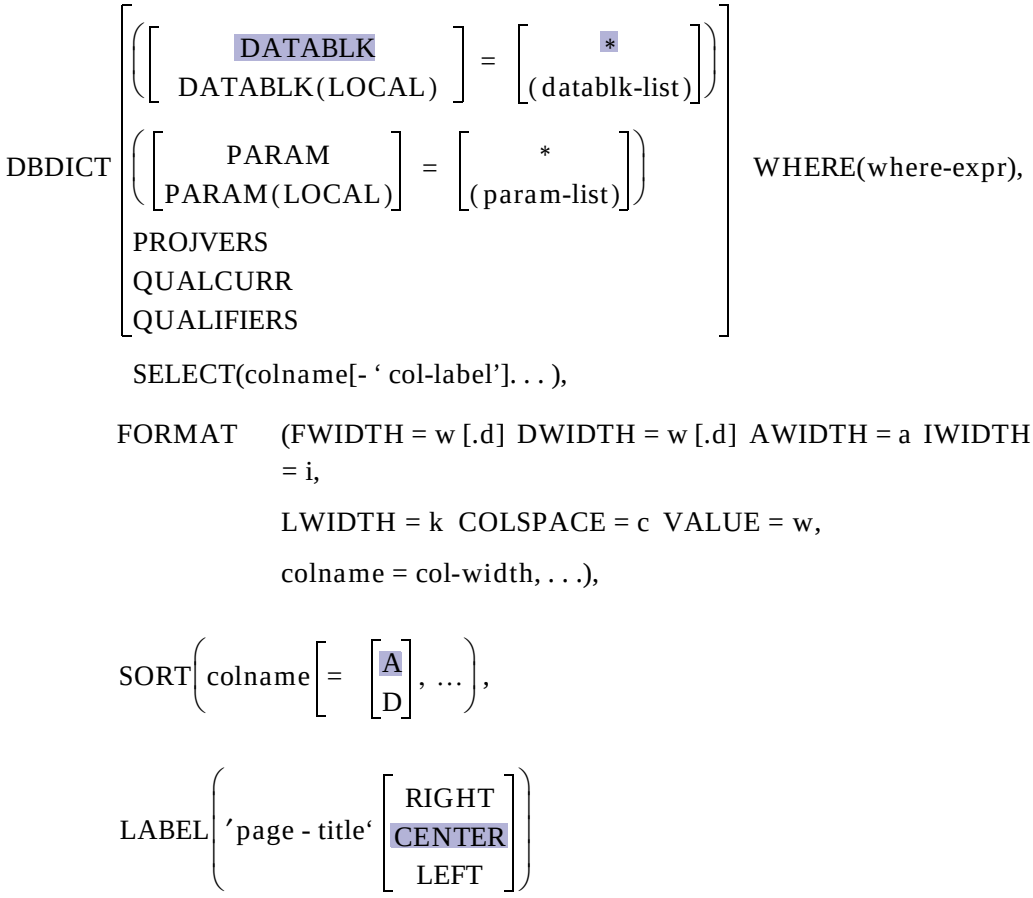
```
DBDICT [DATABLK PARAM PROJVERS QUALCURR QUALIFIERS]
```

Examples:

```
DBDICT
DBDICT PARAM PROJVERS
```

Full Format:

The full format permits the selection of items by name and/or by the WHERE describer. The full format also permits the attributes to be printed using the SELECT describer. In addition, the print format can be specified with the SORT, FORMAT, and LABEL describers. Note that the full format only allows the specification of a single table on a DBDICT statement.



Describer	Meaning
DATABLK	Print the data blocks. datablk-list specifies a list of NDDL-defined data blocks separated by commas. If LOCAL is specified, the non-NDDL-defined data blocks are printed.
PARAM	Print the parameter table. param-list specifies a list of parameters separated by commas. If LOCAL is specified, the non-NDDL-defined parameters are printed.
PROJVERS	Print the project-version table.
QUALIFIERS	Print the qualifier table.
QUALCURR	Print the current values of the qualifiers. SORT is ignored.

Describer	Meaning
where-expr	Logical expression that specifies the desired values of colnames described below. For example, WHERE(VERSION=4 AND SEID <>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section for a further description. The default for VERSION is the last version, and PROJECT is the current project. The default for qual is *, which is all qualifier values found on the database. See also Remark 12.
SELECT	Specifies a list of column names to be printed. The order of the specified colnames will be printed from left to right. If colname is not specified, then all columns will be printed.
colname	Column name. Colname specifies a particular attribute of the database item; such as, data block name (NAME), creation date (CDATE), number of blocks (SIZE), or qualifier name (SEID, SPC, etc.). The allowable colnames are given in the Remarks.
col-label	The label to printed above the column identified by colname. The default for col-label is the colname. col-label may not be specified for colnames: QUALSET, QUALALL, and TRAILER.
FWIDTH=w.d	Specifies the default width for single-precision real numbers in real and complex qualifiers. (Integers: w>0 and d>0, Default=12.5)
DWIDTH=w.d	Specifies the default width for double-precision real numbers in real and complex qualifiers. (Integers: w>0 and d>0, Default=17.10)
AWIDTH=a	Specifies the default width for character string qualifiers. Character strings are printed with enclosing single quotation marks, even if the string is blank. (Integer>0, Default=8)
IWIDTH=i	Specifies the default width for integer qualifiers. (Integer> 0, see Remarks for defaults.)
LWIDTH=k	Specifies the default width for logical qualifiers. Logical values are printed as either “T” for TRUE or “F” for FALSE. (Integer> 0, Default=1)
COLSPACE=c	Specifies the default number of spaces between columns. (Integer> 0, see Remarks for defaults.)
VALUE=w	Specifies the default width for parameter values. The values are printed as character strings with left justification. (Integer>0, Default=40)

Describer	Meaning
col-width	The print width of the data under colname or qual-name. For real numbers, specify w.d where w is the width of the field and d is the number of digits in the mantissa. For integers and character strings, specify w where w is the width of the field. col-width may not be specified for colnames: QUALSET, QUALALL, and TRAILER.
SORT	Specifies how the rows are sorted. The sort is performed in order according to each colname specified in the list. A “D” following the colname causes the sort to be in descending order. An “A” following the colname causes the sort to be in ascending order. Colnames QUALSET, QUALALL, and TRAILER may not be specified under SORT. Each colname specified in SORT must be separated by commas.
page-title	A title to be printed on each page of the directory output.
RIGHT, CENTER, LEFT	Print justification of the page title.

Remarks:

1. DBDICT prints seven different tables according to a default or a user-defined format. The tables are:

Table 2-2 DBDICT Tables

Describer	Description	Default Page-Title	See Remark
DATABLK	Data blocks described by a NDDL DATABLK statement.	NDDL DATABLOCKS	2.
PARAM	Parameters described by a NDDL PARAM statement.	NDDL PARAMETERS	3.
QUALCURR	Current Qualifiers and their values.	CURRENT QUALIFIERS	4.
QUALIFIERS	Qualifiers and their values for each key number.	QUALIFIERS	5.

Table 2-2 DBDICT Tables (continued)

Describer	Description	Default Page-Title	See Remark
DATABLK(LOCAL)	Data blocks not described by a NDDL DATABLK statement.	LOCAL DATABLOCKS	6.
PARAM(LOCAL)	Parameters not described by a NDDL PARAM statement.	LOCAL PARAMETERS	7.
PROJVERS	Project-Version.	PROJECT-VERSION	8.

If DBDICT is specified without any describers then the NDDL Data blocks Table will be printed. See Remark 2.

DATABLK(LOCAL) and PARAM(LOCAL) produce no output, and QUALCURR produces the default values specified on the NDDL QUAL statement.

The defaults and allowable colnames for SELECT, FORMAT, SORT, and LABEL depend on the table. The defaults are described in the following remarks and tables.

2. The default print of the NDDL Data Blocks Table is obtained by:

```
DBDICT
```

or

```
DBDICT DATABLK
```

and is equivalent to:

```
DBDICT DATABLK ,
     SELECT(NAME,DATABASE,DBSET,PROJ,VERS,CDATE,CTIME,
          SIZE,KEY,PURGED='PU',EQUIVD='EQ',
          POINTER='FILE',QUALSET) ,
     FORMAT(NAME=8,DBSET=8,CDATE=6,CTIME=6,SIZE=5,
          KEY=4 ,PURGED=4,EQUIVD=4,POINTER=8,
          IWIDTH=5,COLSPACE=1) ,
     SORT(PROJ=A,VERS=A,DBSET=A,NAME=A) ,
     LABEL('NDDL DATABLOCKS' CENTER)
```

and looks like:

```
* * * *  D I C T I O N A R Y    P R I N T  * * * *

 EXECUTION OF DMAP STATEMENT NUMBER     20
  MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     16

                                        NDDL DATABLOCKS
NAME    DATABASE DBSET   PROJ VERS  CDATE  CTIME  SIZE  KEY  PU  EQ   FILE  SEID  PEID  LOAD   SPC   MPC  METH
-------------------------------------------------------------------------------------------------------------------
-----------------------------------------------------
AGG     MASTER  DBALL    1     1 930805  72340   0    326   1   0  132484   0    0
AXIC    MASTER  DBALL    1     1 930805  72336   0    315   1   0   65764
BGPDTS  MASTER  DBALL    1     1 930805  72338   1    324   0   2  131332        0
BGPDTX  MASTER  DBALL    1     1 930805  72338   1    324   0   1  131332        0
BJJ     MASTER  DBALL    1     1 930805  72341   0    332   1   0  132612        0
BULK    MASTER  DBALL    1     1 930805  72336   2    315   0   0   65700
CASECC  MASTER  DBALL    1     1 930805  72336   1    316   0   2   67428
```

Figure 2-1 DBDICT DATABLK Example

Table 2-3 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-3 DBDICT DATABLK Colnames

colname	Default col-width	Default col-label	Description
PROJECT	40	PROJECT NAME	Project name defined by PROJECT statement
PROJ	4	PROJ NO	Project number associated with PROJECT
VERS	4	VERSION	Version number
CDATE	6	CDATE	Creation date
CTIME	6	CTIME	Creation time
NAME	8	NAME	Parameter name
DATABASE	8	DATABASE	MASTER DBset name
DBSET	8	DBSET	DBset name
RDATE	6	RDATE	Revision date
RTIME	6	RTIME	Revision time
SIZE	5	SIZE	Number of blocks
qual-name	See Note.	qualifier name	Qualifier name
KEY	4	KEY	Key number

Table 2-3 DBDICT DATABLK Colnames (continued)

colname	Default col-width	Default col-label	Description
TRLi	8	TRLi	i-th word in the trailer
TRAILER	8	TRLi	All 10 trailer words
EXTNAME	8	EXTNAME	Extended name
EQUIVD	4	EQ	Equivalenced flag
PURGED	4	PU	Purged flag
EQFLAG	4	EF	Scratch equivalenced flag
SCRFLAG	4	SF	Scratch DBSET flag
POINTER	8	POINTER	Directory pointer
DBENTRY	8	DBENTRY	Database entry pointer
FEQCHAIN	8	FEQCHAIN	Forward equivalence chain
BEQCHAIN	8	BEQCHAIN	Backward equivalence chain
DBDIR20	9	DBDIR(20)	Directory word 20
QUALALL	See Note.	qualifier name	All qualifiers
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers

Note: Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, AWIDTH=8, and FWIDTH=12.5.

3. The default print of the NDDL Parameter Table is obtained by:

```
DBDICT PARAM
```

and is equivalent to:

```
DBDICT PARAM,
     SELECT(NAME,DATABASE,DBSET,PROJ,VERS,CDATE,CTIME,
          KEY,VALUE,QUALSET),
     FORMAT(NAME=8,DATABASE=8,DBSET=8,CDATE=6,CTIME=6,
          KEY=4,VALUE=40,IWIDTH=5,COLSPACE=1),
     SORT(PROJ=A,VERS=A,DBSET=A,NAME=A),
     LABEL('NDDL PARAMETERS' CENTER)
```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *

EXECUTION OF DMAP STATEMENT NUMBER     21
 MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     17

                                                NDDL PARAMETERS
NAME    DATABASE   DBSET  PROJ VERS  CDATE  CTIME  KEY VALUE               SEID  PEID  LOAD   SPC   MPC  METH

-------------------------------------------------------------------------------------------------
ACOUSTIC MASTER   MASTER   1    1   930805  72338  323 0                    0     0
ALTRED   MASTER   MASTER   1    1   930805  72338  319 NO
BCHNG    MASTER   MASTER   1    1   930805  72337  325 FALSE                      0
DBALLX   MASTER   MASTER   1    1   930805  72336  318 DBALL               -1    -1
EPSBIG   MASTER   MASTER   1    1   930805  72339  323 1.000000E+12         0     0
ERROR    MASTER   MASTER   1    1   930805  72338  319 -1
FIXEDB   MASTER   MASTER   1    1   930805  72338  323 0                    0     0
```

Figure 2-2 DBDICT PARAM Example

Table 2-4 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-4 DBDICT PARAM Colnames

colname	Default col-width	Default col-label	Description
PROJECT	40	PROJECT NAME	Project name defined by PROJECT statement
PROJ	5	PROJ	Project number associated with PROJECT
VERS	4	VERS	Version number
CDATE	6	CDATE	Creation date
CTIME	6	CTIME	Creation time
NAME	8	NAME	Parameter name
DATABASE	8	DATABASE	MASTER DBset name
DBSET	8	DBSET	DBset name
RDATE	6	RDATE	Revision date
RTIME	6	RTIME	Revision time
POINTER	8	POINTER	Directory pointer
VALUE	40	VALUE	Parameter value
KEY	4	KEY	Key number

Table 2-4 DBDICT PARAM Colnames (continued)

colname	Default col-width	Default col-label	Description
qual-name	See Note.	qualifier name	Qualifier name
QUALALL	See Note.	qualifier name	All qualifiers
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers

Note: Default widths for qualifiers are DWIDTH=17.10, AWIDTH=8, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5.

4. The default print of the Qualifier Table is obtained by:

```
DBDICT QUALIFIERS
```

and is equivalent to:

```
DBDICT QUALIFIERS ,
     SELECT(KEY QUALALL) ,
     FORMAT(DWIDTH=17.10 AWIDTH=8 IWIDTH=5 LWIDTH=1 ,
          FWIDTH=12.5  COLSPACE=2) SORT(KEY=A) ,
     LABEL('QUALIFIERS' CENTER )
```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *

EXECUTION OF DMAP STATEMENT NUMBER     22

MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     18
                                        QUALIFIERS
KEY APRCH  B2GG   B2PP   BMETH  CMETH  CONFIG  DEFORM  DELTA  DESITER  DLOAD  DRMM  DYRD  EXTRCV  FMETH  FREQ
   FSCOUP GUST HIGHQUAL HINDEX IC IKBAR  IMACHNO  IPANEL  IQ    ISA  ISOLAPP  K2GG        K2PP         LOAD
   M2GG    M2PP  MACHINE METH  METHF  MFLUID MODEL   MPC  MTEMP  NCASE   NL99  NLOAD  NLOOP  NOQUAL  OPERALEV
   OPERASYS P2G   PEID  PVALID  SDAMP  SEDWN   SEID  SOLAPP SOLID SPC STATSUB   SUBDMAP      SUBMODEL SUPORT
   TEMPLD TFL  TSTEP  ZNAME       ZUZR1  ZUZR2  ZUZR3
-------------------------------------------------------------------------------------------------------------
335  '     '                                              F   0
                    0                                                ,       ,
     '     '                                                0
           '       '   0      0                 0

-------------------------------------------------------------------------------------------------------------
336  '     '                                                  0
                    0                                                ,       ,
                                                    0      0                  -1
                      0      0                   0
```

Figure 2-3 DBDICT QUALIFIERS Example

QUALALL selects all qualifiers to be printed. The qualifiers will be printed in alphabetic order. QUALSET selects the only the qualifiers SEID, PEID, SPC, MPC, LOAD, and METH to be printed.

Table 2-5 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers. QUALALL and QUALSET may not be specified in the FORMAT or SORT describers. The qualifier names and values are not printed one per row, but rather from left to right as one logical line that is allowed to wrap after 132 columns.

Table 2-5 DBDICT QUALIFIERS Colnames

colname	Default col-width	Default col-label	Description
KEY	5	KEY	Key number
qual-name	See Note.	qualifier name	Qualifier name
QUALALL	See Note.	qualifier name	All qualifiers
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers

Note: Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5. AWIDTH defaults to the length specified on the QUAL statement in the NDDL sequence.

5. The default print of the Current Qualifier Table is obtained by:

```
DBDICT QUALCURR
```

and is equivalent to:

```
DBDICT QUALCURR SELECT(QUALALL),
     FORMAT(AWIDTH=8,IWIDTH=5,LWIDTH=1,COLSPACE=2),
     LABEL=('CURRENT QUALIFIERS' CENTER)
```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *

EXECUTION OF DMAP STATEMENT NUMBER     24

MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     20
                                           CURRENT QUALIFIERS
APRCH  B2GG     B2PP     BMETH CMETH CONFIG DEFORM DELTA DESITER DLOAD DRMM DYRD EXTRCV  FMETH   FREQ  FSCOUP
       GUST  HIGHQUAL HINDEX  IC IKBAR IMACHNO IPANEL IQ  ISA ISOLAPP K2GG   K2PP        LOAD  M2GG
       M2PP  MACHINE  METH  METHF MFLUID MODEL  MPC MTEMP  NCASE NL99 NLOAD NLOOP NOQUAL  OPERALEV  OPERASYS
       P2G      PEID  PVALID SDAMP SEDWN SEID SOLAPP SOLID    SPC  STATSUB  SUBDMAP         SUBMODEL  SUPORT
       TEMPLD TFL TSTEP  ZNAME     ZUZR1 ZUZR2 ZUZR3
-------------------------------------------------------------------------------------------------------------
'    '  '     '  '       '   0     0     0     0    F        0    0     F     0       0      0      0       F
          0     0     0    0     0     0     0    0       0      1  '       '  '      '    300  '        '
        '     '      0    0     0     0     0   100    0       0    0     0     -1      0       0        0
        '     '     0    0     0     0     0  '      '   0     400      0  '           '        0       0
           0     0      0  '  '     0     0    0
-------------------------------------------------------------------------------------------------------------
```

Figure 2-4 DBDICT QUALCURR Example

Table 2-6 giv es the allowable colnames along with a description that may be specified in the SELECT describers.

Table 2-6 DBDICT QUALCURR Colnames

colname	Default col-width	Default col-label	Description
qual-name	See Note.	qualifier name	Qualifier name
QUALALL	See Note.	qualifier name	All qualifiers
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers

Note: Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5. AWIDTH defaults to the length specified on the QUAL statement in the NDDL sequence.

6. The default print of the Local Data Block Table is obtained by:

```
DBDICT DATABLK(LOCAL)
```

and is equivalent to:

```
DBDICT DATABLK(LOCAL),
     SELECT(NAME,SUBDMAP,SIZE='BLOCKS',PURGED='PU',
          EQUIVD='EQ',POINTER,TRL1,TRL2,TRL3,TRL4,
          TRL5,TRL6,TRL7),
     FORMAT(NAME=8,SUBDMAP=8,IWIDTH=8,COLSPACE=2),
     SORT(NAME=A) LABEL('LOCAL DATABLOCKS' CENTER)
```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *

 EXECUTION OF DMAP STATEMENT NUMBER     23

  MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     19
                                LOCAL DATABLOCKS
NAME    SUBDMAP     BLOCKS   PU   EQ   POINTER   TRL1   TRL2   TRL3   TRL4   TRL5   TRL6   TRL7
-------------------------------------------------------------------------------------------------
CASEW   PHASE1DR      1      0    0    131780     201    4       0     308     0     0       0
```

Figure 2-5 DBDICT DATABLK(LOCAL) Example

TRLi specifies the data block trailer word i where $1 \leq i \leq 10$. TRAILER selects all 10 data block trailer words.

Table 2-7 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-7 DBDICT DATABLK(LOCAL) Colnames

colname	Default col-width	Default col-label	Description
NAME	8	NAME	Parameter name
SUBDMAP	8	SUBDMAP	SubDMAP name
SIZE	8	BLOCKS	Number of blocks
EQUIVD	8	EQ	Equivalenced flag
PURGED	8	PU	Scratch flag
POINTER	8	POINTER	Directory pointer
TRLi	8	TRLi	i-th word in the trailer
TRAILER	8	TRLi	All 10 trailer words
EXTNAME	8	EXTNAME	Extended name

7. The default print of the Local Parameter Table is obtained by:

   ```
   DBDICT PARAM(LOCAL)
   ```

   and is equivalent to:

   ```
   DBDICT PARAM(LOCAL) SELECT(NAME,SUBDMAP,VALUE),
        FORMAT(COLSPACE=4,VALUE=40,AWIDTH=8),
        SORT(NAME=A) LABEL(' LOCAL PARAMETERS' CENTER)
   ```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *

 EXECUTION OF DMAP STATEMENT NUMBER     24
  MODULE NAME = DBDICT   , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     20
                                                        LOCAL PARAMETERS
NAME        SUBDMAP     VALUE
-----------------------------------------------------------------
AERO        SESTATIC    FALSE
AERO        PHASE1DR    FALSE
ALTRED      SESTATIC    NO
ALTRED      PHASE1DR    NO
ALTSHAPE    SESTATIC    0
ALWAYS      PHASE1DR    -1
ALWAYS      PHASE1C     -1
ALWAYS      SEKRRS      -1
ALWAYS      SESTATIC    -1
APP         PHASE1DR    STATICS
APP         PHASE1C     STATICS
APP         SESTATIC    STATICS
APRCH       SESTATIC
ASING       PHASE1DR    0
ASING       SEKRRS      0
ASING       PHASE1C     0
ASING       SESTATIC    0
```

Figure 2-6 DBDICT PARAM(LOCAL) Example

Table 2-8 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-8 DBDICT PARAM(LOCAL) Colnames

colname	Default col-width	Default col-label	Description
NAME	8	NAME	Parameter name
SUBDMAP	8	SUBDMAP	SubDMAP name
VALUE	40	VALUE	Parameter name

8. The default print of Project-Version Table is obtained by:

   ```
   DBDICT PROJVERS
   ```

   and is equivalent to:

   ```
   DBDICT PROJVERS ,
        SELECT(PROJECT='PROJECT NAME',PROJ='PROJ NO.' ,
             VERS='VERSION',DELFLG='DELETED' ,
             CDATE='CREATION DATE' CTIME='CREATION
             TIME') ,
        FORMAT(PROJECT=40,PROJ=10,VERS=10,DELFLG=7,
             COLSPACE=1 ,CDATE=13,CTIME=13) ,
        LABEL('PROJECT-VERSION',CENTER) ,
             SORT(PROJ=A,VERS=A)
   ```

and looks like:

```
* * * *  D I C T I O N A R Y   P R I N T  * * * *
EXECUTION OF DMAP STATEMENT NUMBER     19
 MODULE NAME = DBDICT  , SUBDMAP SEKRRS  , OSCAR RECORD NUMBER     15
                                                PROJECT-VERSION
PROJECT NAME                                 PROJ NO.    VERSION DELETED CREATION DATE CREATION TIME
-------------------------------------------------------------------------------------------------------
'LEFT FENDER                              '       1          1               930805         72319
```

Figure 2-7 DBDICT PROJVERS Example

Table 2-9 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT describers.

Table 2-9 DBDICT PROJVERS Colnames

colname	Default col-width	Default col-label	Description
PROJECT	40	PROJECT NAME	Project name defined by PROJECT statement
PROJ	10	PROJ NO	Project number associated with PROJECT
VERS	10	VERSION	Version number
DELFLG	7	DELETED	Flag indicating whether this project/version has been deleted by the RESTART NOKEEP or DBCLEAN statements.
CDATE	13	CREATION DATE	Creation date
CTIME	13	CREATION TIME	Creation time

CDATE is printed as YYMMDD where YY, MM, and DD are the year, month, and date, respectively. CTIME is HHMMSS where HH, MM, and SS are the hour, minute, and second, respectively.

9. If a parameter or qualifier value is defined to be character string, then the value will be printed with enclosing single quotation marks. Blank strings will also be printed with single quotation marks.
10. If a given qualifier is not in the path of a given data block or parameter, then blank spaces will be printed.

11. A line will wrap if additional columns need to be printed and not enough space is available on the output (assumed to be 132). The first column of each additional line is to be indented by the width of the first column printed for the entry.
12. The where-expr has the following rules:
    - If the where-expr specifies a colname that is not assigned to the data block or parameter then no directory information will be printed for that data block or parameter. For example, given that SPC is not a qualifier for KGG, then the following DBDICT statement will produce no output:

      ```
      DBDICT DATABLK=KGG WHERE(SPC=10)
      ```

    - If the where-expr does not specify a colname that is assigned to the data block (or parameter), then the qualifier is wildcarded. For example, given that SEID is a qualifier for KAA, then the following DBDICT statements are equivalent:

      ```
      DBDICT DATABLK=KAA
      DBDICT DATABLK=KAA WHERE(SEID = *)
      ```

13. A colname specified in the where-expr must be specified in the SELECT clause if the SELECT clause is also specified.

Examples:

1. Print the Project Version Table with a title.

   ```
   DBDICT  PROJVERS SORT(PROJ,VERSION) LABEL('PROJECT
        VERSION TABLE' LEFT)
   ```

2. Print a directory of all data blocks qualified with PEID=10 or SEID=10. Prit columns for the NAME and DBSET, and the qualifiers SPC, MPC, and LOAD.

   ```
   DBDICT  DATABLK SELECT(NAME,SPC,MPC,LOAD,DBSET,SIZE,
             SEID,PEID) ,
        SORT(NAME,SIZE=D)  WHERE( SEID=10 OR PEID=10)
   ```

DBDIR Prints Database Directory Tables

Obsolete. See the **DBDICT** statement.

DBFIX Database Directory Error Detection

Detects and optionally corrects errors in the database directory.

Format: DBFIX [{ LIST / NOLIST } { CORRECT / NOCORRECT }]

Describer	Meaning
LIST	Requests a debug listing of the database directory pointers.
NOLlST	Suppresses a debug listing of the database directory.
CORRECT	Corrects the database if any errors are found.
NOCORRECT	Suppresses the correction of the database.

Remarks:

1. It is recommended that a backup copy of the database be made before this statement is used since corrections of the database are achieved through the deletion of data. Data blocks and parameters are deleted from the database if they have (1) incorrect paths (different than listed in the NDDL), (2) incorrect names (two or more names that are not equivalenced and reference the same data), or (3) incorrect directory pointers.
2. NOLlST does not suppress the listing of any corrections made to the database.

Example:

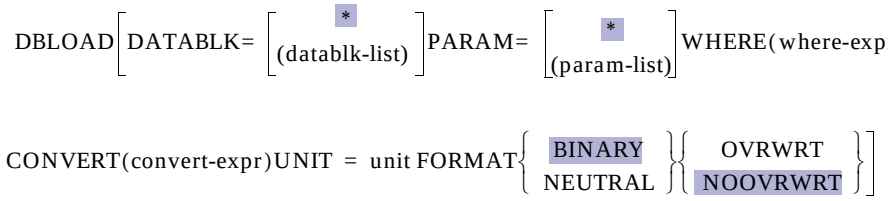
```
DBFIX LIST,NOCORRECT
```

The example above requests a printout of the directory pointers and any errors, but not the corrections.

DBLOAD Loads a Database from a FORTRAN File

Recovers data blocks or parameters from a database created by the DBUNLOAD statement.

Format:



Example:

1. Load the database stored in ASClI format on FORTRAN unit 12.

```
DBLOAD      UNIT=12 FORMAT=NEUTRAL

ASSIGN      DBLOAD='physical file name of unloaded database'
            UNIT=12 FORMATTED
```

2. Load version 1 of KAA under project FRONT BUMPER and store it on the primary database under version 5 and project BUMPER. Overwrite duplicates found on the primary database.

```
DBLOAD      DATABLK=(KAA) WHERE(PROJECT='FRONT BUMPER'
            AND,SEID=10 AND VERSION=1) CONVERT(VERSION=5;
            , PROJECT='BUMPER') OVRWRT

ASSIGN      DBLOAD='physical file name of unloaded database'
```

Describer	Meaning
datablk-list	Specifies a list of data blocks separated by commas. The default is * which selects all data blocks. The loaded data block may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if KLL is to be renamed to KLL1, then DATABLK=(KLL/KLL1) is specified.
param-list	Specifies a list of parameters separated by commas. The default is *, which selects all parameters. The loaded parameter may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if LUSETS is to be renamed to LUSET, then PARAM=(LUSETS/LUSET) is specified.

Describer	Meaning
where-expr	A logical expression that specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. For example, WHERE(VERSlON=4 AND SElD<>2 AND SEID>0) selects all items under version 4 for all values of SElD greater than 0 except 2. See the beginning of this section on WHERE and CONVERT clauses. The default for VERSION is * for all versions; PROJECT is * for all projects; and DBSET is* for all DBsets. The default for qual is *, which is all qualifier values found on the loaded database. See also Remark **8.**
convert-expr	Modifies the values for PROJECT, VERSION, DBSET, and qualifiers selected by the where-expr. The format of convert-expr is: PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name; quali=qual-expri[;..] For example, CONVERT (SEID=100+SElD; SPC=102). See the beginning of this section on WHERE and CONVERT clauses. The default action for VERSION and PROJECT is to use the same version lDs and project IDs; i.e., CONVERT(PROJECT=PROJECT; VERSlON=VERSlON). But if either PROJECT or VERSION is specified in the convert-expr, then both must be specified. The default action for qualifiers and DBSET is to use the same values as long as they are defined in both databases. If not, see Remark **8.**
unit	Specifies the FORTRAN unit number of the database to be loaded. The unit must be specified on an ASSIGN statement that references the physical filename of the loaded database. The default is 51.
OVRWRT NOOVRWRT	By default, if duplicate data blocks or parameters exist on the loaded and primary databases, then a fatal message is issued. A duplicate means that a data block or parameter has not only the same name but also the same qualifier values, PROJECT, VERSION, and DBSET as a data block or parameter on the primary database.
NEUTRAL BINARY	The database to be loaded may be in BINARY or NEUTRAL format. BlNARY indicates the database to be loaded is in binary or FORTRAN unformatted format. NEUTRAL indicates the database to be loaded is in ASClI format. The default is BlNARY.

Remarks:

1. The DBLOAD statement and its applications are discussed further in "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*.
2. If the DATABLK keyword is specified and PARAM is not specified, then only data blocks may be loaded. If the PARAM keyword is specified and DATABLK is not specified, then only parameters may be loaded. If neither DATABLK nor PARAM is specified, then all data blocks and parameters may be loaded.
3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.
4. The database to be loaded is attached as read-only. In other words, items can only be fetched and not stored on this database.
5. If more than one DBLOAD statement is specified, then they will be processed in the order in which they appear. If a duplicate data block or parameter is found on two or more different DBLOAD statements, then the last duplicate will be used.
6. If NEUTRAL is specified, then the FORMATTED keyword must be specified on the corresponding ASSIGN statement.
7. If a data block or parameter is being renamed, then the new name must be defined in the NDDL of the primary database.
8. If the database to be loaded and the primary database have different NDDL schemes and CONVERT is not used, then the following is performed:
    - If a qualifier in the NDDL of the database to be loaded is not in the NDDL of the primary database, then all of its values are converted to the null value corresponding to its type. For example, if the qualifier is integer, real, double-precision, complex or character, then the value is converted to 0, 0., 0.D0, (0.,0.), or blank, respectively. If this conversion results in a duplicate data block(s) or parameter(s), then a User Warning Message is printed and the duplicates are not loaded.
    - If a DBset-name in the NDDL of the database to be loaded is not in the NDDL of the primary database, then its values will be converted to the PARAM default value in the NDDL of the database to be loaded.
9. Data blocks that are equivalenced on the database to be loaded remain equivalenced as long as they are loaded in the same DBLOAD statement or in consecutive DBLOAD statements with the same unit number. Otherwise, a separate copy for the secondary data block is produced.

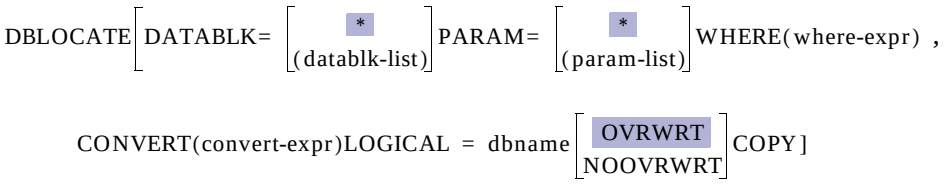
10. It is not possible to restart from a database created by DBLOAD in the same run.
11. SOL 190 (or DBTRANS) is also required with DBLOAD if:
    - The database to be loaded has a different BUFFSlZE.
    - The database to be loaded is in neutral format or is being transferred between different machine types.

    See also the “**Database Concepts**” on page 513 of the *MSC.Nastran Reference Guide*.

DBLOCATE Attaches Secondary Databases

Obtains data blocks or parameters from prior versions of the primary database, or other databases. DBLOCATE may also be used to compress the primary database and to migrate databases created in prior MD Nastran versions.

Format:



Example:

1. Locate in version 4 of MASTER3 all data blocks named KAA for all superelements with lDs greater than 0.

```
DBLOCATE   DATABLK=(KAA) WHERE(PROJECT='FRONT BUMPER'
           ,AND SEID>0 AND VERSION=4) LOGI=MASTER3
ASSIGN     MASTER3='physical file name of master DBset'
```

2. Copy all data blocks and parameters from the last version of MASTER3 to the primary database. For all items with the qualifier SElD, change the SElD to twice the old ID number.

```
DBLOCATE    CONVERT(SEID=2*SEID) COPY LOGI=MASTER3
ASSIGN      MASTER3='physical file name of master DBset'
```

3. Compress a database with multiple versions. All versions under the current project-ID (see PROJ statement) will be copied from the database OLDDB to NEWDB.

```
ASSIGN     MASTER3='physical filename of new master DBset'
ASSIGN     OLDDB='physical filename of old master DBset'
DBLOCATE   LOGI=OLDDB COPY WHERE(VERSION=*) ,
           CONVERT(VERSION=VERSION;PROJECT=PROJECT)
```

Describer	Meaning
datablk-list	Specifies a list of data blocks separated by commas. The default is *, which selects all data blocks. The located data block may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if KLL is to be renamed to KLL1, then DATABLK=(KLL/KLL1) is specified.
param-list	Specifies a list of parameters separated by commas. The default is *, which selects all parameters. The located parameter may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if LUSETS is to be renamed to LUSET, then PARAM=(LUSETS/LUSET) is specified.
where-expr	A logical expression that specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. For example, WHERE(VERSION=4 AND SEID<>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the the beginning of this section on WHERE and CONVERT clauses. The default for VERSION is the last version-ID and PROJECT is the current project-ID. The default for qual is *, which is all qualifier values found on the located database. See also Remark **9.**
convert-expr	Modifies the values for PROJECT, VERSION, DBSET, and qualifiers selected by the where-expr. The format of convert-expr is: PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name; quali=qual-expri[;...] For example, CONVERT (SEID=100+SEID; SPC=102). See the beginning of this section on WHERE and CONVERT clauses. The default action for VERSION and PROJECT is to convert to the current version-ID and current project-ID. But if either PROJECT or VERSION is specified in the convert-expr, then both must be specified. See Example **3.** The default action for qualifiers and DBSET is to use the same values as long both databases have the same NDDL scheme. If not, see Remark **9.**

Describer	Meaning
dbname	Specifies the logical name of the master directory DBset of the located database. dbname must be specified on an ASSIGN statement, which references the physical file name. By default, the located database is also the primary database. (If dbname is specified for the primary database, then dbname must be MASTER.)
OVRWRT NOOVRWRT	By default, duplicate data blocks or parameters on the located database will take precedence over those on the primary database. A duplicate means that a data block or parameter has not only the same name but also the same qualifier values, PROJECT, VERSION, and DBSET as the data block or parameter on the primary database. If NOOVRWRT is specified, then a fatal message is issued.
COPY	Requests that the located data blocks or parameters be copied to the primary database.

Remarks:

1. The DBLOCATE statement and its applications are discussed further in the "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*.
2. If the DATABLK keyword is specified and PARAM is not specified, then only data blocks may be located. If the PARAM keyword is specified and DATABLK is not specified, then only parameters may be located. If neither DATABLK nor PARAM is specified, then all data blocks and parameters may be located.
3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.
4. If more than one DBLOCATE statement is specified, then they will be processed in the order in which they appear. If a duplicate data block or parameter is found on two or more different DBLOCATE statements, then the last duplicate will be used.
5. If the located database is not the primary database, then it is attached for read-only purposes. In other words, items can only be fetched and not stored on the located database.

6. If the RESTART FMS statement is also specified, then located data blocks and parameters are treated as if they exist in the restart version. In other words, restart equivalences will be made on located items at the beginning of the run and can be subsequently broken as a result of regeneration and / or NDDL dependencies.
7. If a data block or parameter is being renamed, then the new name must be defined in the NDDL of the primary database.
8. If LOGICAL refers to the primary database and one version is to be copied to another, then the items are equivalenced.
9. If the located database and the primary database have different NDDL schemes and CONVERT is not used, then the following is performed:
    - If a qualifier in the NDDL of the located database is not in the NDDL of the primary database, then all of its values are converted to the null value corresponding to its type. For example, if the qualifier is integer, real, double precision, complex or character then the value is converted to 0, 0., 0.D0, (0.,0.), or blank, respectively. If this conversion results in a duplicate data block(s) or parameter(s), then a User Warning Message is printed and the duplicates are not located.
    - If a dbset-name in the NDDL of the located database is not in the NDDL of the primary database, then its values will be converted to the PARAM default value in the NDDL of the located database.

DBSETDEL Deletes DBsets

Deletes DBsets, all of its members, and associated physical files.

Format:

DBSETDEL dbsetnamei

Describer	Meaning
dbsetnamei	Specifies the name(s) of DBset(s) to be deleted. The DBset names MASTER, OBJSCR, or SCRATCH may not be specified.

Remarks:

1. The DBSETDEL statement and its applications are discussed further in "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide.*
2. If dbsetnamei does not exist, then no action is taken.
3. After a DBset has been deleted with this statement, it may be recreated with the INIT statement in a subsequent run.

Example:

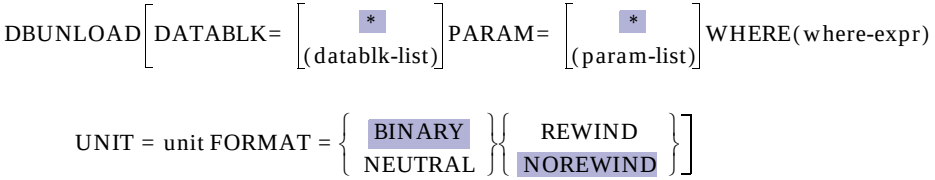
Delete DBset DBUP20 from the database.

```
DBSETDEL DBUP20
```

DBUNLOAD Unloads a Database to a FORTRAN File

Stores data blocks or parameters from the primary database onto a FORTRAN file in a binary or neutral format, for purposes of database compression or database transfer between different computers.

Format:



Example:

1. Unload the database in ASCII format onto FORTRAN unit 12.

```
DBUNLOAD    UNIT=12 FORMAT=NEUTRAL
ASSIGN      DBUNLOAD='physical file name of FORTRAN unit 12' ,
            UNIT=12 FORMATTED
```

2. Unload version 1 of KAA under project FRONT BUMPER.

```
DBUNLOAD    DATABLK=(KAA) WHERE(PROJECT='FRONT BUMPER'
            ,AND SEID=10 AND VERSION=1)
ASSIGN      DBUNLOAD='physical file name of FORTRAN unit 50'
```

Describer	Meaning
datablk-list	Specifies a list of data blocks separated by commas. The default is* which selects all data blocks.
param-list	Specifies a list parameters separated by commas. The default is*, which selects all parameters.
where-expr	Logical expression which specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. For example, WHERE(VERSION=4 AND SEID<>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section on WHERE and CONVERT Clauses.

Describer	Meaning
	The default for VERSION is * for all versions; PROJECT is* for all projects; and DBSET is * for all DBsets. The default for qual is*, which is all qualifier values found on the primary database.
unit	Specifies the FORTRAN unit number to unload the database. The unit must be specified on an ASSIGN statement, which references its physical filename. The default is 50.
NEUTRAL BINARY	The database may be unloaded in BINARY or NEUTRAL format. BlNARY indicates the database is to be unloaded in binary or FORTRAN unformatted. NEUTRAL indicates the database is to be unloaded in ASCII format. The default is BlNARY.
NOREWIND REWIND	By default, if DBUNLOAD is executed more than once for the same unit, then the unit is not rewound. REWlND requests that the unit be rewound prior to unloading.

Remarks:

1. The DBUNLOAD statement and its applications are discussed further in the "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*.
2. If the DATABLK keyword is specified and PARAM is not specified, then only data blocks may be unloaded. If the PARAM keyword is specified and DATABLK is not specified, then only parameters may be unloaded. If neither DATABLK nor PARAM is specified, then all data blocks and parameters may be unloaded.
3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.
4. If more than one DBUNLOAD statement is specified, then they will be processed in the order in which they appear.
5. If NEUTRAL is specified, then the FORMATTED keyword must be specified on the corresponding ASSIGN statement.
6. If NEUTRAL is specified, then only data blocks with an NDDL description are unloaded. (See the *MD Nastran 2006 DMAP Programmer's Guide* under the DATABLK statement.) An NDDL description is required for TYPE=TABLE and none is required for TYPE=MATRlX. The data block must be unloaded in BINARY if TYPE=UNSTRUCTURED, KDlCT, or KELM.

7. Data blocks that are equivalenced on the primary database remain equivalenced as long as they are unloaded in the same DBUNLOAD statement or in consecutive DBUNLOAD statements with the same unit number. Otherwise, a separate copy for the secondary data block is produced.

DBUPDATE Specifies Database Directory Update Interval

Specifies the maximum length of CPU time between database directory updates to the MASTER DBset. This statement is intended to be used if the INIT MASTER(RAM=r) option is specified.

Format:

DBUPDATE [=] update-time

Describer	Meaning
update-time	CPU time interval in minutes (real or integer) between database directory updates.

Remarks:

1. The difference in CPU time from the last update is checked after the execution of each DMAP instruction. The database directory is updated if this difference is greater than update-time. Update-time and CPU time is accurate to the nearest whole second only.
2. If update-time < 0, then database directory updates are only performed at the end of the run.
3. Defaults for update-time are machine dependent and may be found in the *MD Nastran Installation and Operations Guide*.
4. Periodic updates of the directory tables to MASTER DBset increases the integrity of the database during system crashes (for example, crashes due to insufficient time or space).
5. Directory updates are performed automatically at various points in the execution of the DMAP in addition to those specified by DBUPDATE. An asterisk appears after the word “BEGN” in the Executive Summary Table whenever an update occurs. See the “**Output Description**” on page 373 of the *MSC.Nastran Reference Guide*. These updates occur whenever a permanent data block or parameter DMAP equivalence or restart equivalence is broken. Updates also occur upon deletions. Additions to the database do not automatically cause a directory update to take place.
6. This statement is in effect only when INIT MASTER(RAM=r) is being used. INIT MASTER(S) and INIT MASTER(NORAM) disable periodic and automatic updates.

7. Update-time may also be changed with the DMAP instruction PUTSYS(update-time, 128) or the NASTRAN SYSTEM(128)=update-time statement. (The update-time must be a real, single-precision value specified in minutes.)

Example:

```
DBUPDATE = 5.5
```

The above example would call for a database directory update at the end of a DMAP module execution after five and one-half minutes of CPU time have elapsed from the last update.

DEFINE Parameter Definition

Assigns user defined keywords (or cellnames) to NASTRAN system cell. (See the NASTRAN statement for a description of “cellname”). In addition, the DEFINE statement provides a mechanism to set default values for system cells.

Format:

DEFINE keyword [=expression] [LOCATION=SYSTEM(i)] [TYPE=type]

Describer	Meaning
keyword	User defined name, 1 through 24 characters in length. The first character must be alphabetic. The following characters can be used for keywords: A through Z, ’_" , and 0 through 9. Any other characters are invalid.
expression	Expression produces a single value from a set of constant and / or variable parameters separated by operators. The value is assigned to the “keyword” and is also used to set the value for the NASTRAN system cell specified by “LOCATION”. The TYPE determines both the type of the result and the type conversions that will be applied to the constants and variables within the expression--mixed mode expressions are allowed (see Remark **6.**). The parentheses can be used to change the order of precedence. Operations within parentheses are performed first with the usual order of precedence being maintained within the parentheses. The variable parameters within the expression must be keywords previously defined on a DEFINE statement. The following operations are allowed:

Describer **Meaning**

Parameter Type	Operator	Operation
Integer or Real	+	Addition
	-	Subtraction
	*	Multiplication
	/	Division
Logical	+	Bit-wise OR
Logical	-	Bit clear. For example, the result of *a*-*b* is equal to the value of *a* with the bits associated with *b* set to 0.

SYSTEM(I) Specifies the NASTRAN system cell number to be associated with the keyword.

type The type of the expression result and the type conversions that will be applied to the constants and variables within the expression. Allowable data types are as follows:

Description	Type
Integer (default)	I
Real	R
Logical	LOGICAL

Remarks:

1. If TYPE, LOCATION and EXPRESSION is omitted, the default data type is integer and the default value is zero.
2. If expression omitted, an internal default will be assigned to the keyword / cellname based on the LOCATION (See "**NASTRAN**" on page 13 Statement for a list of internal default values).
3. A DEFINE statement that specifies a LOCATION is actually setting the default for a NASTRAN system cell and therefore it is not necessary to also set the system cell value on a subsequent NASTRAN statement, unless the user wishes to override the previous DEFINE statement setting. Also, since more than one DEFINE statement may be present for the same "keyword"

the last specification takes precedence. “Keywords” referenced on a NASTRAN statement or in an expression on the DEFINE statement are automatically substituted by the last specification of the “keyword” prior to the current statement being processed.

4. DEFINE statements may also be specified in Runtime Configuration (RC) files. See the *MSC.Nastran 2005 r2 Installation and Operations Guide.*
5. System cells may also be set with the NASTRAN statement. In addition, they may be set or values returned with the DMAP PUTSYS and GETSYS functions and the PARAM module. See the *MD Nastran 2006 DMAP Programmer’s Guide.*
6. Each operand within the expression will be converted to the result type prior to the arithmetic operation. For example: the statement “DEFINE JJ=2.5 + 3.6 TYPE=I” would result in 2.5 and 3.6 being converted to 2 and 3 respectively and the result of 5 would be assigned to JJ.

Examples:

1. Change the default value for block size.

   DEFINE BUFFSIZE=4097 LOCATION=SYSTEM(1)

2. Set the sparse matrix selection to forward-backward substitution only.

   DEFINE SPARSE=16 LOCATION=SYSTEM(126)

3. Define the system cell keyword and default value for the maximum output line count and then reset it to another value on a NASTRAN statement. Note: The DEFINE statement would typically be placed in an RC file and the NASTRAN statement would be placed in the File Management Section whenever the user wants to override the DEFINE statement default setting.

```
DEFINE    MAXLINES=999999999 LOCATION=SYSTEM(9)
NASTRAN   MAXLINES=100000
```

4. Define system cells that behave like “toggles”, turning some feature on or off.

```
DEFINE    MESH=2 LOCATION=(31)
DEFINE    NOMESH=0 LOCATION=(31)
NASTRAN   MESH
```

Note: Since each subsequent DEFINE statement redefines the default value, the second DEFINE of system cell location 31 sets the default value to 0. A NASTRAN statement can then be inserted in the input file to reset the MESH system cell back to a value of 2. This same technique can be used with any system cell where the user wishes to simply refer to the system cell keyword and have the system cell set to a previous DEFINE statement default.

5. Invalid usage of the DEFINE and NASTRAN statement:

```
DEFINE    BUFFSIZE=4097
NASTRAN   BUFFSIZE=2048
```

Valid usage:

```
DEFINE    BUFFSIZE=4097 LOCATION=SYSTEM(1)
NASTRAN   BUFFSIZE=2048
```

ENDJOB Terminates Job

Terminates the job at a user-specified location in the FMS Section.

Format:

ENDJOB

Remark:

ENDJOB is normally used after a DBDICT or DBDIR statement or after database initialization.

Example:

```
DBDICT
ENDJOB
```

EXPAND Concatenates New DBset Members

Concatenates additional DBset members on an existing permanent DBset previously defined with an INIT statement.

Format:

EXPAND dbset-name LOGlCAL=(log-namei [(max-sizei)]...)

Describer	Meaning
dbset-name	The name of a DBset previously defined with an INIT statement.
log-namei	Specifies the logical name of a DBset member. log-namei may also be referenced on an ASSIGN statement which refers to the physical file name of the DBset member.
max-sizei	Specifies the maximum size, in blocks, words or bytes, of a DBset member. For storage units specified in words or bytes, the size must be followed by one of the unit keywords below

Unit Keyword	Storage Unit
W	Words
B	Bytes
KW, K	Kilowords (1024 words)
KB	Kilobytes (1024 bytes)
MW, M	Megawords (1024^2 words)
MB	Megabytes (1024^2 bytes)
GW, G	Gigawords (1024^3 words)
GB	Gigabytes (1024^3 bytes)
TW, T	Terawords (1024^4 words)
TB	Terabytes (1024^4 bytes)

For example, 100MB is 100 megabytes, 1.5GB is 1.2 gigabytes = 1536 megabytes, 2.5M = 2.5 megawords = 2560 kilowords. The size of a block in words is defined by BUFFSlZE.

Remark:

1. On all computers with dynamic file allocation, the physical filename of a DBset member may be specified on an ASSIGN statement:

   ASSIGN log-name='physical filename'

   If an ASSIGN statement is not specified for the member, then a name is automatically assigned. The naming convention is described in the "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*.

Example:

```
ASSIGN      DBMEM02='physical file name'
EXPAND      DBALL LOGICAL=(DBMEM02)
```

This would create and add the DBset member DBMEM02 to the already existing DBset DBALL.

INCLUDE Inserts External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

Format:

INCLUDE 'filename'

Describer	Meaning
filename	Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks (').

Example:

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
BEGIN BULK
INCLUDE 'MYBULK.DATA'
ENDDATA
```

Remarks:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example the file:

   /dir123/dir456/dir789/filename.dat

   may be included with the following input:

   ```
   INCLUDE '/dir123
             /dir456
           /dir789/filename.dat'
   ```

3. See the *MSC.Nastran 2005 r2 Installation and Operations Guide* for more examples.

INIT Creates a DBset

Creates a temporary or permanent DBset. For the SCRATCH and MASTER DBsets, all or some of their space may be allocated to real memory.

Format 1: Initialize any DBset except MASTER and SCRATCH:

INIT DBset-name [LOGICAL=(log-namei(max-sizei),...) BUFFSIZE=b CLUSTER=c]

Format 2: Initialize the MASTER DBset:

INIT MASTER [(RAM = r , S / NORAM) LOGICAL = (log-name(max-sizei), ...),

BUFFSIZE = b CLUSTER = c]

Format 3: Initialize the SCRATCH DBset:

INIT SCRATCH [(MEM = m / NOMEM) LOGICAL = (log-name(max-sizei), ...),

SCR300 = (log-namei(max-sizei),...) BUFFSIZE = b CLUSTER=c]

Example:

1. Modify the default allocation of the DBALL DBset to 50000 blocks.

```
INIT DBALL LOGI=(DBALL(50000))
```

2. Do not allocate any real memory for the MASTER and SCRATCH DBsets.

```
INIT      MASTER(NORAM)
INIT      SCRATCH(NOMEM)
```

3. Create a new DBset called DBUP with two members DBUP1 and DBUP2.

```
INIT      DBUP LOGI=(DBUP1, DBUP2)
ASSIGN    DBUP1 ='physical filename 1'
ASSIGN    DBUP2='physical filename 2'
```

Describer	Meaning
dbset-name MASTER SCRATCH	The name of a temporary or permanent DBset.
log-namei	Specifies the logical name of a DBset member. log-namei may also be referenced on an ASSIGN statement, which refers to the physical file name of the DBset member. If no log-namei is specified, then the DBset will have one member and the log-name will be the same as the DBset-name. A maximum of twenty log-names may be specified. For the SCRATCH DBset see also Remark **8.** SCR300 is a special keyword that indicates that the log-names are members reserved for DMAP module internal scratch files.
max-sizei	Specifies the maximum size, in blocks, words or bytes, of a DBset member. For storage units specified in words or bytes, the size must be followed by one of the unit keywords below.

Unit Keyword	Storage Unit
W	Words
B	Bytes
KW, K	Kilowords (1024 words)
KB	Kolobytes (1024 bytes)
MW, M	Megawords (1024^2 words)
MB	Megabytes (1024^2 bytes)
GW, G	Gigawords (1024^3 words)
GB	Gigabytes (1024^3 bytes)
TW, T	Terawords (1024^4 words)
TB	Terabytes (1024^4 bytes)

For example, 100MB is 100 megabytes, 1.5GB is 1.5 gigabytes = 1536 megabytes, 2.5M = 2.5 megawords = 2560 kilowords. The size of a block in words is defined by BUFFSlZE. The default for DBALL and SCRATCH may be found in the *MSC.Nastran 2005 r2 Installation and Operations Guide* and ranges from 250,000 blocks to 4,000,000 blocks.

Describer	Meaning
RAM NORAM	RAM=r requests that r words of real memory are to be allocated for the MASTER DBset. See the "**nastran Command**" on page 7 of the *MSC.Nastran Reference Guide*. The default is RAM or RAM=30000. NORAM or RAM=0 specifies no real memory is to be allocated.
S	If the primary database is being created in the run, this option requests that all DBsets in the primary database will be automatically deleted at the end of the run. INIT MASTER(S) is equivalent to specifying scr=yes on the "nastran" command. See the "**nastran Command and NASTRAN Statement**" on page 1. If the run is a restart, then this option is ignored.
MEM NOMEM	MEM=m specifies that m blocks of real memory are to be allocated for the SCRATCH DBset. See "**The NASTRAN Statement (Optional)**" on page 8 of the *MSC.Nastran Reference Guide*. The default m is machine dependent and may be found in the *MSC.Nastran 2005 r2 Installation and Operations Guide*. NOMEM or MEM=0 requests that no real memory is to be allocated.
BUFFSlZE	BUFFSlZE=b specifies the number of words per block in the DBset and will override the value specified by the BUFFSlZE keyword on the NASTRAN statement. The default for b is obtained from the the NASTRAN BUFFSIZE statement. See the *MSC.Nastran 2005 r2 Installation and Operations Guide*.
CLUSTER	CLUSTER=c specifies the number of blocks per cluster in the DBset. The default is 1, and any other value is not recommended.

Remarks:

1. The INIT statement and its applications are discussed further in the "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*.
2. There are four DBsets that are predefined and automatically allocated by the program. Their DBset-names are MASTER, DBALL, SCRATCH, and OBJSCR, and they are described in the "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*.
3. On all computers with dynamic file allocation, the physical filename of a DBset member may specified on an ASSIGN statement:

   ASSIGN log-name='physical filename'

If an ASSIGN statement is not specified for the member, then a name is automatically assigned. The naming convention is described in the "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide*.

4. It is recommended that there is sufficient physical space to hold a DBset member should it reach its maximum size (max-sizei). The max-sizei may be converted to words by multiplying by b. A summary of space usage and allocation is printed at the end of the Execution Summary Table.
5. In restart runs, the INIT statement is ignored for pre-existing permanent DBsets. The INIT statement is intended to be specified only in the run in which the DBset is being created. If more DBset members need to be added to the DBset, then the EXPAND statement is used.
6. If RAM or RAM=r is specified and the run terminates because the computer operating system detects insufficient space or time or the computer halts due to a power outage or operator interruption, then it may not be possible to restart from the database. See the DBUPDATE FMS statement.
7. BUFFSIZE=b and CLUSTER=c must satisfy the following inequality:

$$b \leq \frac{64000}{c} + 5$$

8. By default, the SCRATCH DBset is divided into two partitions: LOGICAL and SCR300. The LOGICAL partition, log-names after the LOGICAL keyword, are reserved for DMAP scratch data blocks and the SCR300 partition for DMAP module internal scratch files.
    - The maximum total number of log-names for LOGICAL and SCR300 is 20. For example, if LOGICAL has 8 log-names, then SCR300 can have no more than 12 log-names.
    - If NASTRAN SYSTEM(142)=1 is specified, then the SCR300 partition is not created and internal scratch files, along with DMAP scratch data blocks will reside on the LOGICAL partition. The default is SYSTEM(142)=2.
    - If NASTRAN SYSTEM(151)=1 is specified and the LOGICAL partition has reached its maximum size, then the SCR300 partition will be used. The default is SYSTEM(151)=0.

- By default, the space specified for the SCR300 partition is released to the operating system after each DMAP module is executed as long as the module used more than 100 blocks for internal scratch files. If 100 blocks is not a desirable threshold, then it may be changed by specifying NASTRAN SYSTEM(150)=t, where t is the number of blocks for the threshold.

9. BUFFSlZE=b is predefined for DBset-names MSCOBJ, OBJSCR, and USROBJ and may not be changed by BUFFSIZE on this statement or the NASTRAN BUFFSlZEbmax statement (see "**The NASTRAN Statement (Optional)**" on page 8 of the *MSC.Nastran Reference Guide*). The default for b is recommended for all except very large problems. bmax must reflect the maximum of b specified for all DBsets attached to the run, including the delivery database. See *MSC.Nastran 2005 r2 Installation and Operations Guide* for the defaults of b and bmax.
10. If INIT MASTER(RAM=r) and INIT SCRATCH(MEM=m) are specified, then BUFFSIZE for these DBsets must be the same. If not, a warning message is issued, and the BUFFSlZE for the SCRATCH DBset is reset to that of the MASTER DBset.
11. Only one INIT statement per dbset-name may be specified in the File Management Section.

MEMLIST Specify Datablocks Eligible for SMEM

Specify a list of scratch datablocks that may reside in scratch memory (SMEM).

Format:

MEMLIST DATABLK = (DBname1, DBname2, ..., DBnamei)

Describer	Meaning
DBnamei	Name of a Nastran datablock.

Remarks:

1. Only NDDL and local scratch datablocks may be included in MEMLIST specification.
2. Datablocks specified will reside in SMEM on a first come, first served basis.
3. Datablocks not specified by this command will not reside in SMEM.
4. Database directories for the SCRATCH DBset reside in SMEM and are not affected by any MEMLIST specification.
5. Continuation lines are allowed.
6. Multiple MEMLIST commands are honored.
7. Scratch I/O activity is reported in the F04 file by including DIAG 42 in the Executive Section.

Example:

```
MEMLIST DATABLK = (KOO, MOO, KQQ, MQQ)
```

If generated, datablocks KOO, MOO, KQQ, and MQQ will reside in scratch memory. All other datablocks will be excluded from scratch memory.

PROJ Defines Database Project-Identifier

Defines the current or default project identifier, project-ID.

Format:

PROJ [=] 'project-ID'

Describer	Meaning
project-ID	Project identifier. Must be enclosed in single quotes. (Character string, maximum of 40 characters; Default=blank)

Remarks:

1. There may be only one PROJECT statement in the File Management Section. The PROJECT statement must be specified before all DBCLEAN, DBDIR, DBDICT, RESTART, DBLOCATE, and DBLOAD statements where project-ID is not specified by the user.
2. This statement is optional and specifies that all data blocks and parameters to be stored on or accessed from the database in the current run shall also be identified by project-ID. Therefore, in subsequent runs that may access this data through other FMS statements such as RESTART, the project-ID must be specified.
3. Project-ID is the default on DBCLEAN, DBDIR, DBDICT, and RESTART FMS statements and in the WHERE and CONVERT clause of the DBLOCATE statement.
4. Leading blanks and trailing blanks enclosed within the single quotes are ignored. All other blanks are considered part of the project-ID.
5. Project-ID is saved with only the first 40 characters specified.

Examples:

1. `PROJ = 'MY JOB'`
2. The following project-ID will be truncated to 40 characters:

   ```
   PROJ 'CAR MODEL 1999 BODY FRAME SYM - PROTYP B RUN'
   ```

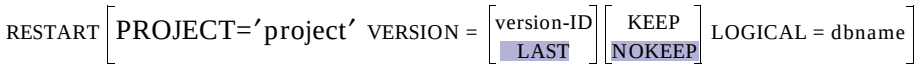
   and all subsequent restarts must have the statement.

   ```
   PROJ 'CAR MODEL 1999 BODY FRAME SYM - PROTYP B'
   ```

RESTART Reuses Database From a Previous Run

Requests that data stored in a previous run be used in the current run.

Format:



Describer	Meaning
project-ID	Project identifier. See PROJ FMS statement. Must be enclosed in single right-hand quotation marks (’). (Character string, maximum of 40 characters; default is the project-ID specified on the PROJ FMS statement).
version-ID	Version number. (Integer>0)
LAST	Specifies the last version under project-ID.
KEEP	Data stored under VERSION will remain on the database after the run is completed.
NOKEEP	Data stored under VERSION will be deleted from the database after the run is completed.
dbname	Specifies the logical name of an existing MASTER (master directory) DBset to be used for restart purposes. This MASTER and its associated database will be opened in a read-only mode to perform the restart, any new data will be written to the database for the current run.

Remarks:

1. There may be only one RESTART statement in the File Management Section.
2. A new version-ID is automatically assigned whenever a restart is performed.
3. If project-ID or version-ID or both are specified and cannot be found, then a fatal message will be issued.
4. The RESTART statement is required to perform restarts in solution sequences 4, and 101 through 200.
5. If PROJECT is not specified, then the run will restart from the project-ID specified on the PROJ statement. (See Example **2.** below.)
6. Databases created in one version typically cannot be directly restarted into a different version. Restrictions are typically documented in the current release guide; however a DBLOCATE type restart might work.

7. Restarts do not work with ACMS and DMP.

Examples:

1. RESTART VERSION=7

   Version number 7 will be retrieved for this run (version 8). At the end of the run version 7 will be deleted.

2. PROJ='FENDER'
   RESTART

   The last version under project-ID FENDER will be used in the current run.

3. ASSIGN RUN1='run1.MASTER'
   RESTART LOGICAL=RUN1

   The run1.MASTER and its associated database will be used (read only) for restart purposes.

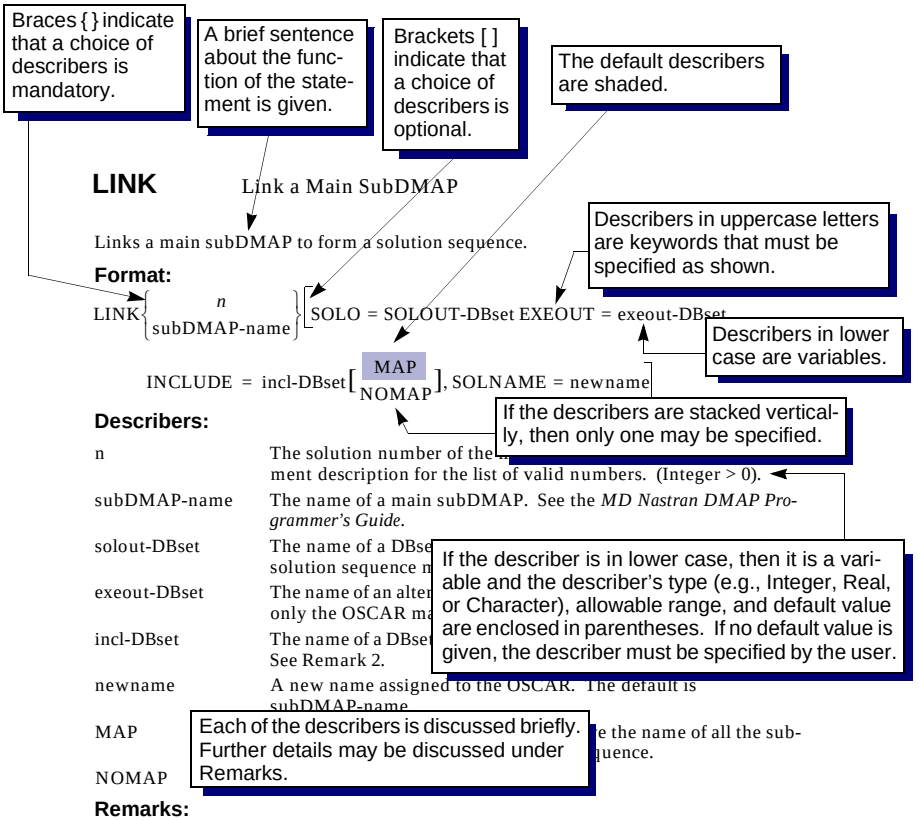
RESTART

Reuses Database From a Previous Run

CHAPTER 3

Executive Control Statements

3.1 Key to Descriptions



LINK Link a Main SubDMAP

Links a main subDMAP to form a solution sequence.

Format:

LINK { *n* / subDMAP-name } [SOLO = SOLOUT-DBset EXEOUT = exeout-DBset

INCLUDE = incl-DBset [MAP / NOMAP], SOLNAME = newname

Describers:

n	The solution number of the ... ment description for the list of valid numbers. (Integer > 0).
subDMAP-name	The name of a main subDMAP. See the *MD Nastran DMAP Programmer's Guide*.
solout-DBset	The name of a DBse... solution sequence n...
exeout-DBset	The name of an alter... only the OSCAR ma...
incl-DBset	The name of a DBse... See Remark 2.
newname	A new name assigned to the OSCAR. The default is subDMAP-name.
MAP	...e the name of all the sub-...quence.
NOMAP	

Remarks:

1. All DBsets specified on this statement must have the same BUFFSIZE. See the INIT FMS statement.
2. SubDMAP objects are created with the ... run or obtained from previous runs. ... following order:
   a. Objects created with the CON...
   b. Objects residing on the DBset... word. The default is MSCOB...
3. Upon successful linking of a subDMAP, the subDMAP may be executed with the SOL statement.

Executive Control Section

This section describes the Executive Control statements. These statements select a solution sequence and various diagnostics.

Most Executive Control statements are order independent. The exceptions are the COMPILE, COMPILER, ALTER, ENDALTER, and LINK statements. If used, the LINK statement must appear after all COMPILE statements. The COMPILER statement (or equivalent DIAGs) must appear before all COMPILE statements. The COMPILER statement also sets the defaults for subsequent COMPILE statements.

Executive Control Statement Summary

The Executive Control statements are summarized as follows:

ALTER	Specifies deletion and / or insertion of following DMAP statements.
APP	Specifies an approach in a solution sequence.
CEND	Designates the end of the Executive Control statements.
COMPILE	Requests compilation of specified subDMAPs or the NDDL file.
COMPILER	Specifies DMAP compilation diagnostics.
DIAG	Requests diagnostic output or modifies operational parameters.
DOMAINSOLVER	Selects domain decomposition solution methods.
ECHO	Controls the echo of Executive Control statements.
ENDALTER	Designates the end of a DMAP sequence headed by an ALTER.
GEOMCHECK	Specifies tolerance values and options for optional finite element geometry tests.
ID	Specifies a comment.
LINK	Requests the link of a main subDMAP.
MODEL_CHECK	Specifies model checkout run options.
SOL	Requests execution of a solution sequence or DMAP program.
TIME	Sets the maximum allowable execution time.

3.2 Executive Control Statement Descriptions

Executive Control statements may be abbreviated down to the first four characters as long as the abbreviation is unique relative to all other statements. Each statement is described as follows:

Description

A brief sentence about the function of the statement is given.

Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the statement line is longer than 72 columns, then it may be continued to the next line with a comma. For example:

```
COMPILE   SEDRCVR   SOUIN=MSCSOU,
          NOREF     NOLIST
```

Example

A typical example is given.

Describers and Meaning

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), its allowable range, and its default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

Remarks

The remarks in the Remarks Section are generally arranged in order of importance and indicate such things as the Executive Control statement's relationship to other statements, restrictions and recommendations on its use, and further descriptions of the describers.

$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Format:

$ followed by any characters out to column 80.

Example:

```
$ TEST FIXTURE-THIRD MODE
```

Remarks:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

ALTER Inserts and / or Deletes DMAP Statements

Inserts and / or deletes DMAP statements in a subDMAP.

Format:

```
ALTER k1 [,k2]
```

or

```
ALTER 'string1'[(occurrence,offset)] ,['string2'[(occurrence,offset)] ]
```

or

```
ALTER k1 , ['string2'[(occurrence,offset)] ]
```

or

```
ALTER 'string1'[(occurrence,offset)] , [k2]
```

Describer	Meaning
k1	If k2 or 'string2' is not specified, the subsequent DMAP statements will be inserted after either the statement number k1 or the 'string1' , [(occurrence,offset)] reference point.
k1, k2	DMAP statements numbered k1 through k2 will be deleted and may be replaced with subsequent DMAP statements.
'string1'	if 'string2' or k2 is not specified, the subsequent DMAP statements will be inserted after the first occurrence of 'string1'.
'string1','string2'	DMAP statements beginning with the first occurrence of 'string1' through DMAP statements containing the first occurrence of 'string2' will be deleted and may be replaced with subsequent DMAP statements.
occurrence	This flag indicates which occurrence of the preceding string is to be used, starting at the beginning of the subDMAP. (Integer>0; Default=1)
offset	This flag indicates the offset from the reference DMAP statement. Depending on the sign the specific DMAP statement may be above (-offset) or below (+offset) the referenced DMAP statement. (Integer; Default=0)

Remarks:

1. The ALTER statement must be used in conjunction with the COMPILE Executive Control statement. Note: ALTER statements cannot be used in conjunction with an MALTER statement, and therefore, should never immediately follow this statement.
2. If an MALTER statement is used in conjunction with the ALTER statement, then the MALTER should be placed above the COMPILE statements. Failure to place the MALTER in this position may cause ALTER statements to be ignored.
3. The ALTERs can reference the DMAP statements in any order within a subDMAP. Two restrictions on ordering are:
   - K2 or 'string2'(occurence, offset) references must refer to a DMAP statement number that is greater than or equal to the k1 or 'string1'(occurrence,offset) reference within a single ALTER statement.
   - K1 or 'string1' and k2 or 'string2' cannot overlap DMAP line positions with another ALTER that references the same subDMAP.
4. The 'string1' or 'string2' used as a search pattern will apply to one complete DMAP statement-i.e., a multiline DMAP statement will be searched for a pattern match as if each 72 character line of the DMAP statement were concatenated together into one string-all blanks and comments (either embedded or immediately preceding the DMAP statement) will be retained. However, comments are ignored for the following type of alter:

   alter '^ *gp0'
5. Within a SUBDMAP, both 'string1' and 'string2' will be used to search for a pattern match starting at the beginning of the subDMAP-not at the current position of the last string match.
6. The special characters (metacharacters) used for string searching are described in Remark 9. The characters <,>, and $, which are common DMAP characters, are also special metacharacters. If they are to be used in the search string as regular characters, then they must be preceded by a backward slash (\). For example, to find the string

   IF(DDRMM >=-1)

   the command is

   ALTER 'IF (DDRMM \>=-1)' $

7. The ALTER statement must not exceed 72 characters (no continuations are allowed).

8. 'string2' (r2,02) can be defaulted to 'string1' (r1,01) by using a null string (''). For example, the alter statement

   ALTER 'string1'(r1,01),''

   is equivalent to

   ALTER 'string1'(r1,01),'string1'(r1,01)

   The defaults for (r2,02) using the null string can be overridden by specifying (r2,02).

   As another example, the alter statement

   ALTER 'string1'(r1,01),''(r2,02)

   is equivalent to

   ALTER 'string1'(r1,01),'string1'(r2,02)

9. Metacharacters*:

.	Matches any *single* character except *newline.*
*	Matches any number (including zero) of the single character (including a character specified by a regular expression) that immediately precedes it. For example, since "." (dot) means any character, ".*" means "match any number of characters".
[...] or < >	Matches any *one* of the characters enclosed between the brackets. For example, "[AB]" matches either "A" or "B". A range of consecutive characters can be specified by separating the first and last characters in the range with a hyphen. For example "[A-Z]" will match any uppercase letter from A to Z and "[0-9]" will match any digit from 0 to 9. Some metacharacters lose special meaning inside brackets. A circumflex (^) as the first character in the bracket tries to match any one character *not* in the list.
^ or ! or .	Requires that the following regular expression be found at the beginning of the line. Note, that these metacharacters may lead to UFM 802 if the preceding line is a comment.

$	Requires that the preceding regular expression be found at the end of the line.
\	Treats the following special character as an ordinary character. For example " \ ." stands for a period and " *" for an asterisk. Also, to search for a tic (’), the search string must be single quotes.
’	Marks the beginning and end of a pattern to be matched.
Note:	Nonportable characters such as [ ] and ^ should be replaced (e.g., ^ ->! and [ ] -><>) if portability is required. However, all the above characters are recognized by MD Nastran.

10. If a string-based alter uses the "!" in the expression (find occurrence at the beginning of line), it is possible MD Nastran will fail with USER FATAL MESSAGE 802.

Examples:

1. The following alter will insert a MATPRN DMAP statement after the first occurrence of the string ’SDR2’ in subDMAP DSASTAT.

```
SOL 101
COMPILE DSASTAT $
ALTER ’SDR2’ $
MATPRN OESDS1//$
CEND
```

2. The following alter will delete the second occurrence of the OUTPUT4 DMAP statement in subDMAP DSASTAT and replace it with a MATPRN DMAP statement.

```
SOL 101
COMPILE DSASTAT $
ALTER ’OUTPUT4’(2),’OUTPUT4’(2) $
$ OR
$ ALTER ’OUTPUT4’(2),’’ $
MATPRN OESDS1//$
CEND
```

APP Specifies Solution Sequence Approach

Selects heat transfer analysis in the linear static SOLs 101 or a coupled analysis combining heat transfer and structural analysis in SOL 153.

Format:

APP approach

Describer	Meaning	
approach	Specifies one of the following:	
	HEAT	Indicates that heat transfer is to be performed in SOLs 101.
	COUPLED	Indicates that a coupled analysis combining heat transfer and structural analysis in SOL 153.

Remarks:

1. The APP statement is optional.
2. The APP HEAT statement applies only to linear static SOLs 101. The APP HEAT statement is not required in SOLs 153 and 159, or in SOL 101 if PARAM,HEATSTAT,YES is specified.
3. The NASTRAN HEAT=1 statement is an alternate specification of APP HEAT. See “**nastran Command and NASTRAN Statement**” on page 1.

Example:

The following requests a heat transfer rather than a structural analysis in Solution Sequence 101.

```
SOL 101
APP HEAT
```

CEND End of Executive Control Delimiter

Designates the end of the Executive Control Section.

Format:

CEND

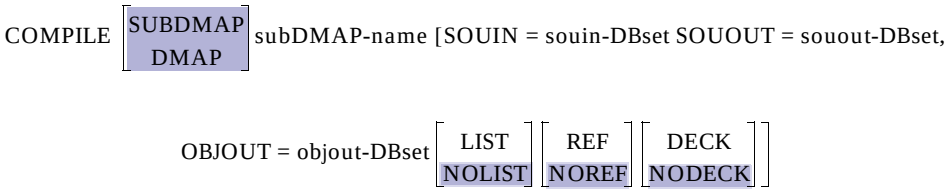
Remark:

1. CEND is an optional statement. If CEND is not specified, then the program will automatically insert one.

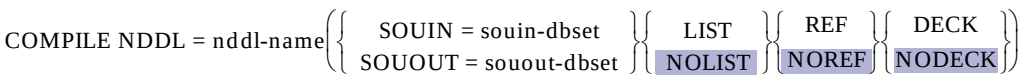
COMPILE Compiles DMAP Statements

Requests the compilation of a subDMAP, subDMAP alter, or NDDL sequence.

Format 1: Compiles a subDMAP or subDMAP alter sequence



Format 2: Compiles an NDDL sequence



Examples:

1. The following compiles an alter in subDMAP PHASEIDR.

```
COMPILE PHASE1DR
ALTER 'CALL PHASE1A'
CEND
```

2. The following compiles a subDMAP called MYDMAP. (SUBDMAP and END are DMAP statements; see *MD Nastran 2006 DMAP Programmer's Guide*.)

```
COMPILE MYDMAP LIST REF
SUBDMAP MYDMAP $
.
.
.
END $
CEND
```

3. The following obtains a listing of the NDDL.

```
ACQUIRE NDDL
COMPILE NDDL=NDDL LIST
CEND
```

Describer	Meaning
subDMAP-name	The name of a subDMAP sequence. SubDMAP-name must be 1 to 8 alphanumeric characters in length and the first character must be alphabetic. The keywords DMAP and SUBDMAP are optional and do not have to be specified.
nddl-name	The name of an NDDL sequence. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.) The keyword NDDL must be specified.
souin-DBset	The name of a DBset from which the subDMAP or NDDL source statements will be retrieved. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.) The default is MSCSOU if the next statement is not a subDMAP statement.
souout-DBset	The name of a DBset on which the subDMAP or NDDL source statements will be stored. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.) The default is the SCRATCH DBset.
objout-DBset	The name of a DBset on which the subDMAP object code will be stored. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.) The default is the OBJSCR DBset.
LIST,NOLIST	LIST requests a compiled listing of the subDMAP or NDDL sequence. NOLIST suppresses the listing. NOLIST is the default.
REF,NOREF	REF requests a compiled cross reference of the subDMAP or NDDL sequence. NOREF suppresses the cross reference. NOREF is the default.
DECK,NODECK	DECK requests the subDMAP or NDDL source statements to be written to the PUNCH file. NODECK suppresses the writing to the PUNCH file. NODECK is the default.

Remarks:

1. SubDMAP names for MD Nastran solution sequences are given in the SOL statement description. The “COMPILER LIST REF” statement may be used to determine the appropriate subDMAP-name.

2. If a subDMAP is being compiled and SOUIN=souin-DBset is specified, then an ALTER Executive Control statement or an INCLUDE statement which contains an ALTER statement as the first non comment line, must appear immediately after this statement. If not, then the SUBDMAP DMAP statement must appear immediately after this statement. See *MD Nastran 2006 DMAP Programmer's Guide*.

3. Starting in Version 69, DBsets USRSOU and USROBJ were no longer automatically created. They must be initialized by the INIT FMS statement and then may be specified for souin-dbset (or souout-dbset) and objout-dbset, respectively. They may be used to store the subDMAP source statements and object code on the primary database for re-execution in a subsequent run. For example:

   In the first run, the following COMPILE statement compiles and stores a subDMAP called MYDMAP.

```
COMPILE MYDMAP SOUOUT=USRSOU OBJOUT=USROBJ
SUBDMAP MYDMAP $
.
.
END $
CEND
```

   In the second run, the SOL statement is used to execute the MYDMAP stored in the previous run. The LINK statement is required to retrieve the object code from the USROBJ DBset.

```
SOL MYDMAP
LINK MYDMAP INCL=USROBJ
CEND
```

   In the third run, the COMPILE statement is used to alter MYDMAP and execute.

```
SOL MYDMAP
COMPILE MYDMAP SOUIN=USRSOU
ALTER...
.
.
.
CEND
```

4. If SOUOUT or OBJOUT is specified and a subDMAP with the same name as subDMAP-name already exists on the database, then its source statements or object code will be replaced.

5. A COMPILE statement is required for each subDMAP to be compiled. If two or more COMPILE statements reference the same subDMAP name, then only the last is used in the linking of the object code. If the COMPILE statement is being used only to alter a subDMAP and two or more COMPILE statements reference the same subDMAP name, then the multiple alters are assembled and the subDMAP is compiled only once.
6. Only one COMPILE statement for an NDDL sequence may be specified in the input file.
   - SOUIN=souin-DBset requests only a compilation of the NDDL sequence stored on souin-DBset for purposes of obtaining a listing or a cross reference, and it cannot be modified with the ALTER statement. See Remark **3.** COMPILE NDDL=NDDL SOUIN=MSCSOU LIST requests a listing of the MD Nastran NDDL sequence. The ACQUIRE FMS statement or the SOL statement must be specified in order to attach the corresponding Delivery Database.
   - To alter the MD Nastran NDDL sequences, the entire modified NDDL sequence is included after the COMPILE statement and SOUIN=souin-DBset is not specified.
   - SOUOUT=souout-DBset requests the storage of the NDDL source statements on the souout-DBset and may not be specified with SOUIN=souin-DBset.
7. The COMPILER statement may be used to override the defaults of NOLIST, NOREF, and NODECK. In other words, if LIST or NOLIST, REF or NOREF, or DECK or NODECK is not specified, then the corresponding option on the COMPILER statement will be used. In the following example, REF on the COMPILER statement will override the default of NOREF on the COMPILE statement.

```
COMPILER REF
COMPILE MYDMAP
```

8. MSCSOU and MSCOBJ, specified with SOUOUT and OBJOUT, are special DBsets similar to USRSOU and USROBJ except that they are used in the creation or modification of a delivery database. For an example application, see the *MSC.Nastran 2005 r2 Installation and Operations Guide.*

COMPILER DMAP Compiler Output Options

Requests compilation of a DMAP sequence and/or overrides defaults on the COMPILE statement.

Format:

COMPILER[=] [LIST / NOLIST] [DECK / NODECK] [REF / NOREF] [GO / NOGO] [SORT / NOSORT]

Describer	Meaning
LIST, NOLIST	LIST requests the compilation listing of the solution sequence. NOLIST suppresses the listing.
DECK, NODECK	DECK requests that the DMAP source statements of the solution sequence be written to the PUNCH file. NODECK suppresses the DECK option.
REF, NOREF	REF requests a compilation cross reference. NOREF suppresses a compilation cross reference.
GO, NOGO	GO requests the execution of the solution sequence following compilation. NOGO requests termination following compilation.
SORT, NOSORT	SORT compiles subDMAPs in alphabetical order. NOSORT compiles subDMAPs in calling sequence order.

Remarks:

1. REF is equivalent to DIAG 4. LIST is equivalent to DIAG 14. DECK is equivalent to DIAG 17.
2. NOGO is an alternative to NOEXE on the SOL statement.
3. This statement provides the user a means of obtaining a compilation or source listing, or both, of a complete solution sequence, including all the component subDMAPs.
4. See the COMPILE statement to compile a single subDMAP.
5. This statement also requests the automatic link of the solution sequence. Therefore, all objects must be created in the current run or obtained from the DBset such as USROBJ. See the COMPILE statement for how to create and store objects.

6. The COMPILER statement may be used to override the defaults of NOLIST, NOREF, NODECK on the COMPILE entry when they are not explicitly specified. However, COMPILER LIST produces a list of the entire solution sequence. If a listing of only specific subdmaps are desired, then COMPILER LIST should not be specified and the LIST request should be made on the COMPILE entry.

```
COMPILER REF
COMPILE MYDMAP
```

Example:

```
COMPILER=LIST
```

DIAG Requests Diagnostic Output

Requests diagnostic output or special options.

Format:

DIAG [=] k1[k2, ..., kn]

Describer	Meaning
ki	A list separated by commas and/or spaces of desired diagnostics.

Remarks:

1. The DIAG statement is optional.
2. Multiple DIAG statements are allowed.
3. The following table lists the possible values for ki and their corresponding actions:

k=1	Dumps memory when a nonpreface fatal message is generated.
k=2	Prints database directory information before and after each DMAP statement. Prints bufferpooling information.
k=3	Prints "DATABASE USAGE STATISTICS" after execution of each functional module. This message is the same as the output that appears after the run terminates. See the "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide.*
k=4	Prints cross-reference tables for compiled sequences. Equivalent to the COMPILER REF statement.
k=5	Prints the BEGIN time on the operator's console for each functional module. See the "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide.*
k=6	Prints the END time for each functional module in the log file or day file and on the operator's console. Modules that consume less time than the threshold set by SYSTEM(20) do not create a message. See the "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide.*
k=7	Prints eigenvalue extraction diagnostics for the Complex Determinate method.

k=8 Prints matrix trailers as the matrices are generated in the Execution Summary Table. See the "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide*.

k=9 Prints a message in the. f04 when EQUIV and EQUIVX perform a successful equivalence; in other words, both the input and output exists.

k=10 Uses alternate nonlinear loading in linear transient analysis. Replaces N_{n+1} with $(N_{n+1} + N_n + N_{n-1})/3$

k=11 DBLOAD, DBUNLOAD, and DBLOCATE diagnostics.

k=12 Prints eigenvalue extraction diagnostics for complex Inverse Power and complex Lanczos methods.

k=13 Prints the open core length (the value of REAL on VAX computers). See the "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide*.

k=14 Prints solution sequence. Equivalent to the COMPILER LIST statement.

k=15 Prints table trailers.

k=16 Traces real inverse power eigenvalue extraction operations

k=17 Punches solution sequences. Equivalent to the COMPILER DECK statement.

k=18 In aeroelastic analysis, prints internal grid points specified on SET2 Bulk Data entries.

k=19 Prints data for MPYAD and FBS method selection in the Execution Summary Table.

k=20 Similar to DIAG 2 except the output appears in the Execution Summary Table and has a briefer and more user-friendly format. However, the .f04 file will be quite large if DIAG 20 is specified with an MD Nastran solution sequence. A DMAP Alter with DIAGON(20) and DIAGOFF(20) is recommended. DIAG 20 also prints DBMGR, DBFETCH, and DBSTORE subDMAP diagnostics. See the *MD Nastran 2006 DMAP Programmer's Guide*.

k=21 Prints diagnostics of DBDIR and DBENTRY table.

k=22 EQUIV and EQUIVX module diagnostics.

k=23 Not used.

k=24 Prints files that are left open at the end of a module execution. Also prints DBVIEW diagnostics.

k=25 Outputs internal plot diagnostics.

k=26 Dynamic file allocation diagnostics on IBM / MVS computers.

k=27 Prints Input File Processor (IFP) table. See the *MSC.Nastran Programmer's Manual*, Section 4.5.9.

k=28 Punches the link specification table. (XBSBD). The Bulk Data and Case Control Sections are ignored, and no analysis is performed.

k=29 Process link specification table update. The Bulk Data and Case Control Sections are ignored, and no analysis is performed. See the *MSC.Nastran Programmer's Manual*, Section 6.10.3.1.

k=30 In link 1, punches the XSEMii data (i.e., set ii via DIAG 1 through 15). The Bulk Data and Case Control Sections are ignored, and no analysis is performed. After link 1, this turns on BUG output. Used also by MATPRN module. See also Remark 5 on the "**TSTEP**" on page 2587 Bulk Data entry.

k=31 Prints link specification table and module properties list (MPL) data. The Bulk Data and Case Control Sections are ignored, and no analysis is performed.

k=32 Prints diagnostics for XSTORE and PVA expansion.

k=33 Not used.

k=34 Turns off plot line optimization.

k=35 Prints diagnostics for 2-D slideline contact analysis in SOLs 106 and 129.

k=36 Prints extensive tables generated by the GP0 module in p-version analysis.

k=37 Disables the superelement congruence test option and ignores User Fatal Messages 4277 and 4278. A better alternative is available with PARAM,CONFAC. See “Parameters” on page 659.

k=38 Prints material angles for CQUAD4, CQUAD8, CTRIA3, and CTRIA6 elements. The angle is printed only for elements that use the MCID option of the connection entry.

k=39 Traces module FA1 operations and aerodynamic splining in SOLs 145 and 146.

k=40 Print constraint override / average information for edges and faces in p-adaptive analysis

k=41 Traces GINO OPEN / CLOSE operations.

k=42 Prints output on F04 file the usage statistic for datablock defined in the FMS command, MEMLIST.

k=43 Not used.

k=44 Prints a mini-dump for fatal errors and suppresses user message exit.

k=45 Prints the same database directory information as DIAG 2 except that it prints only after each DMAP statement.

k=46 Used by MD Nastran development for GINO printout.

k=47 Prints DBMGR, DBFETCH, and DBSTORE subDMAP diagnostics.

k=48 Used by MD Nastran development for GINO printout.

k=49 DIAG 49 is obsolete and should not be used. The utility f04rprt should be used to summarize the f04 Execution Summary instead.

k=50 Traces the nonlinear solution in SOLs 106, 129, 153, and 159. Prints subcase status; echoes NLPARM, NLPCI, and TSTEPNL entry fields; and prints initial arc-length. Prints iteration summary only in SOLs 129, and 159.

In static aeroelastic analysis (SOL 144), prints transformation information associated with the generation of the DJX matrix in the ADG module and intermediate solutions information in the ASG module.

k=51 Prints intermediate displacement, load error vectors, and additional iteration information helpful to debugging in SOLs 106, 129, 153, and 159.

k=52 Disables the printing of errors at each time step in SOLs 129 and 159.

k=53 MESSAGE module output will also be printed in the execution summary table. See the "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide.*

k=54 Linker debug print.

k=55 Performance timing.

k=56 Extended print of Execution Summary table (prints all DMAP statements and RESTART deletions). See the "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide*.

k=57 Executive table (XDIRLD) performance timing and last-time-used (LTU) diagnostics.

k=58 Data block deletion debug and timing constants echo.

k=59 Buffpool debug printout.

k=60 Prints diagnostics for data block cleanup at the end of each module execution in subroutines DBCLN, DBEADD, and DBERPL.

k=61 GINO block allocator diagnostics.

k=62 GINO block manager diagnostics.

k=63 Prints each item checked by the RESTART module and its NDDL description.

k=64 Requests upward compatibility DMAP conversion from Version 65 only. Ignored in Version 70.5 and later systems.

Examples:

```
DIAG 8,53
```

or

```
DIAG 8
DIAG 53
```

DOMAINSOLVER Domain Decomposition Solution Method

Selects domain decomposition solution methods.

Format:

$$\text{DOMAINSOLVER}\left[\begin{matrix}\text{STAT}\\ \text{MODES}\\ \text{FREQ}\\ \text{ACMS}\end{matrix}\right]\left[\left(\left(\text{PARTOPT} = \left[\begin{matrix}\text{DOF}\\ \text{GRID}\\ \text{FREQ}\end{matrix}\right], \text{NUMDOM} = \text{int},\right.\right.$$

$$\text{UPFACT} = \text{real}, \text{TREF} = \left[\begin{matrix}\text{SINGLE}\\ \text{MULTI}\end{matrix}\right], \text{ALLOC} = \left[\begin{matrix}\text{STATIC}\\ \text{DYNAMIC}\end{matrix}\right], \text{PRINT} = \left[\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right],$$

$$\left.\left.\text{NCLUST} = \text{int}, \text{CLUSTSZ} = \text{int}\right)\right]$$

Examples:

DOMAINSOLVER STAT (PARTOPT=DOF)

DOMAINSOLVER ACMS (UPFACT=15,NUMDOM=128)

For DOMAINSOLVER ACMS, there are additional parameters UPFACT, ALLOC, and PRINT. The keywords function as follows:

STAT	Linear Statics
MODES	Normal Modes
FREQ	Frequency Response
ACMS	Automated Component Modal Synthesis

The description of the parameters is as follows:

PARTOPT Partitioning option. Selects which domain is to be decomposed.

DOF	Degree of Freedom domain
GRID	Grid point (Geometric) domain
FREQ	Frequency domain

The default is dependent upon solution sequence. See **Table 3-2** for further descriptions.

NUMDOM Selects the number of domains as follows:

ACMS If NUMDOM = 0 or 1, then the model will not be split. Default depends on the model size and the value of PARTOPT. For PARTOPT=GRID, the default NUMDOM is determined by the number of grid points as follows:

NGRID <= 2,000; NUMDOM = 4
2,000 <= NGRIDS < 10,000; NUMDOM = 16
10,000 <= NGRIDS < 50,000; NUMDOM = 32
50,000 <= NGRIDS < 100,000; NUMDOM = 64
100,000 <= NGRIDS < 300,000; NUMDOM = 128
300,000 <= NGRIDS; NUMDOM = 256

For PARTOPT=DOF, the default NUMDOM is determined by the number of degrees of freedom in the analysis set, as follows:

NDOF <= 5000; NUMDOM = 2
5,000 < NDOF <= 20,000; NUMDOM = 4
20,000 < NDOF <= 50,000; NUMDOM = 8
50,000 < NDOF <= 200,000; NUMDOM = 16
200,000 < NDOF <= 500,000; NUMDOM = 32
500,000 < NDOF <= 1,000,000; NUMDOM = 64
1,000,000 < NDOF <= 2,000,000; NUMDOM = 128
2,000,000 < NDOF <= 4,000,000; NUMDOM = 256
4,000,000 < NDOF <= 8,000,000; NUMDOM = 512
8,000,000 < NDOF; NUMDOM = 1024

The model will be split into NUMDOM domains.

STAT Default = *dmp*; if NUMDOM has any other value, it will automatically be set to *dmp* (= no. of processors used for the run). The model will be divided into NUMDOM domains in either the geometric (grid based) or DOF domains, depending on the value of PARTOPT.

MODES Default = *dmp*; if NUMDOM has any other value, it will be reset to *dmp* (= no. of processors used for the run). The model will be divided into NUMDOM domains in either the frequency, geometric (grid based) or DOF domains, depending on the value of PARTOPT.

FREQ Default = *dmp*; if NUMDOM has any other value, it will automatically be set to *dmp* (= no. of processors used for the run). The frequency range will be divided into NUMDOM regions which are then solved independently.

UPFACT (ACMS)	By default, the frequency range used for upstream component modes is two times larger than the desired range on the EIGR/L entry. To modify this factor, specify the UPFACT parameter (Real; Default=2.0).
TREE (ACMS)	Specifies the type of elimination tree to use: TREE = MULTI: binary multilevel tree (Default) TREE = SINGLE: single level tree A multilevel tree requires that the number of domains be a power of two and be evenly divisible by *dmp*, the number of processors specified for the run; otherwise a single level tree will be used automatically.
ALLOC (ACMS)	Determines how superelements are assigned to processors: ALLOC = STATIC: components are divided evenly at job start (Default) ALLOC = DYNAMIC: components are assigned to processors on a first come, first served basis. This option is effective when one or more processor is heavily loaded and/or has a slower computational rate.
PRINT (ACMS)	Controls intermediate print of upstream and data recovery processing in F06 and F04 files. Default='NO'. If PRINT=NO and an error occurs upstream, the intermediate output is placed in a separate output file named "jid.acms_out" for examination.
NCLUST (MODES)	Specifies the number of frequency segments for hierarchic parallel Lanczos. The frequency range is divided into NLCUST segments, and, if PARTOPT=dof, the stiffness and mass matrices are partitioned into dmp/NCLUST matrix domains; if PARTOPT=grid, or if PARTOPT is not specified, the model geometry is partitioned into dmp/NCLUST domains.
CLUSTSZ (MODES)	Specifies the number of matrix or geometric domains for hierarchic parallel Lanczos. If PARTOPT=dof, the stiffness and mass matrices are partitioned into CLUSTSZ matrix domains; if PARTOPT=grid, or if PARTOPT is not specified, the model geometry is partitioned into CLUSTSZ domains. In either case, the frequency range is divided into dmp/CLUSTSZ frequency segments.

Table 3-1 shows the availability of partitioning methods with each analysis type. An asterisk ("*") indicates a supported implementation.

Table 3-1 Analysis Types and Partitioning Methods

Solution Sequence	DMP Method	Partitioning Methods Available				
		GEOM	DOF	FREQ	G+F	D+F
101	STAT	*				
	ACMS	*				
103	MODES	*	*	*	*	*
	ACMS	*	*			
108	FREQ			*		
111	MODES		*	*		*
	ACMS	*	*	*	*	*
	FREQ			*		
112	MODES		*			
	ACMS	*				
200	MODES		*	*		*
	ACMS	*	*	*	*	*
	FREQ			*		

The DOMAINSOLVER command is optional. If “dmp=” is specified on the command without a DOMAINSOLVER command in the Executive Section, the following actions will result based on solution sequence.

Table 3-2 DOMAINSOLVER Defaults

Solution Number	Default DOMAINSOLVER Options	
	DMP Method	Partitioning Option
101	STAT	GRID
103	MODES	GRID
	ACMS	DOF
108	FREQ	FREQ
111	MODES	FREQ
	FREQ	FREQ
	ACMS	DOF

Table 3-2 DOMAINSOLVER Defaults

Solution Number	Default DOMAINSOLVER Options	
	DMP Method	Partitioning Option
112	MODES	FREQ
200	MODES	DOF
	FREQ	FREQ
	ACMS	DOF

Remarks:

1. A multi-level tree requires the number of tip superelement (i.e., NUMDOM) to be evenly divisible by the number of processors specified for the run; otherwise the program will switch to a single-level tree automatically.
2. If both NLCUST and CLUSTSZ are specified, then the product (NCLUST) X (CLUSTSZ) should equal dmp. If not, the CLUSTSZ parameter is ignored.

ECHO Controls Printed Echo

Controls the echo (printout) of the Executive Control Section.

Formats:

ECHOOFF
ECHOON

Remarks:

1. The ECHO statement is optional.
2. ECHOOFF suppresses the echo of subsequent Executive Control statements. ECHOON reactivates the echo after an ECHOOFF statement.

ENDALTER End of DMAP Alter

Designates the end of an alter.

Format:

ENDALTER

Remark:

1. The ENDALTER statement is required when using an alter unless the alter package ends with the CEND, COMPILE, or LINK statement.

GEOMCHECK Specifies Geometry Check Options

Specifies tolerance values and options for optional finite element geometry tests.

Format:

```
GEOMCHECK test_keyword [= tol_value], [MSGLIMIT = n], [ MSGTYPE = FATAL | INFORM | WARN ],
    [SUMMARY], [NONE]
```

Describer	Meaning
test_keyword	A keyword associated with the particular element geometry test. See Remark 2. for a list of acceptable selections.
tol_value	Tolerance value to be used for the specified test. See Remark 2. for default values of the test tolerances.
n	The minimum number of messages that will be produced. The default is 100 messages for each element type. See Remark 3.
FATAL	Geometry tests that exceed tolerance values produce fatal messages. See Remark 4.
INFORM	Geometry tests that exceed tolerance values produce informative messages. See Remark 4.
WARN	Geometry tests that exceed tolerance values produce warning messages. See Remark 4.
SUMMARY	A summary table of the geometry tests performed is produced. No individual element information messages are output.
NONE	None of the optional element geometry tests will be performed.

Remarks:

1. The GEOMCHECK directive controls the number and severity of certain informational and warning messages produced by element matrix generation geometry checking operations. Controls are currently available for the CQUAD4, CQUADR, CTRIA3, CTRIAR, CHEXA, CPENTA, CTETRA, CBAR and CBEAM elements only. Multiple GEOMCHECK directives may be present. Continuations are acceptable.

2. The following table summarizes the acceptable specifications for test_keyword.

Name	Value Type	Default	Comment
Q4_SKEW	Real≥0.0	30.0	Skew angle in degrees
Q4_TAPER	Real≥0.0	0.50	Taper ratio
Q4_WARP	Real≥0.0	0.05	Surface warping factor
Q4_IAMIN	Real≥0.0	30.0	Minimum Interior Angle in degrees
Q4_IAMAX	Real≥0.0	150.0	Maximum Interior Angle in degrees
T3_SKEW	Real≥0.0	10.0	Skew angle in degrees
T3_IAMAX	Real≥0.0	160.0	Maximum Interior Angle in degrees
TET_AR	Real≥0.0	100.0	Longest edge to shortest edge aspect ratio
TET_EPLR	Real≥0.0	0.50	Edge point length ratio
TET_DETJ	Real	0.0	| J | minimum value
TET_DETG	Real	0.0	| J | minimum value at vertex point
HEX_AR	Real≥0.0	100.0	Longest edge to shortest edge aspect ratio
HEX_EPLR	Real≥0.0	0.50	Edge point length ratio
HEX_DETJ	Real	0.0	| J | minimum value
HEX_WARP	Real≥0.0	0.707	Face warp coefficient
PEN_AR	Real≥0.0	100.0	Longest edge to shortest edge aspect ratio
PEN_EPLR	Real≥0.0	0.50	Edge point length ratio
PEN_DETJ	Real	0.0	| J | minimum value
PEN_WARP	Real≥0.0	0.707	Quadrilateral face warp coefficient
BEAM_OFF	Real≥0.0	0.15	CBEAM element offset length ratio
BAR_OFF	Real≥0.0	0.15	CBAR element offset length ratio

where:

- Test_keyword names starting with the characters Q4 are applicable to CQUAD4 and CQUADR elements. Test_keyword names starting with the characters T3 are applicable to CTRIA3 and CTRIAR elements. Test_keyword names starting with the characters TET_ are

applicable to CTETRA elements. Test_keyword names starting with the characters HEX_ are applicable to CHEXA elements. Test_keyword names starting with the characters PEN_ are applicable to CPENTA elements.

- Skew angle for the quadrilateral element is defined to be the angle between the lines that join midpoints of the opposite sides of the quadrilateral. Skew angle for the triangular element is defined to be the smallest angle at any of the three vertices.
- Interior angles are defined to be the angles formed by the edges that meet at the corner node of an element. There are four for quadrilateral shapes and three for triangular shapes.
- Taper ratio for the quadrilateral element is defined to be the absolute value of [the ratio of the area of the triangle formed at each corner grid point to one half the area of the quadrilateral - 1.0]. The largest of the four ratios is compared against the tolerance value. It may be noted that as the ratio approaches 1.0, the shape approaches a rectangle.
- Surface warping factor for a quadrilateral is defined to be the distance of the corner points of the element to the mean plane of the grid points divided by the average of the element diagonal lengths. For flat elements (all of the grid points lie in a plane), this factor is zero.
- The edge point length ratio test is only performed for the solid elements when edge node points exist. The test evaluates the relative position of the edge node point along a straight line connecting the two vertex nodes of that edge. Ideally, the edge point should be located on this line at a point midway between the two end points. The default tolerance allows the edge node to be positioned anywhere between the two quarter points on this line. In addition, the angles between the lines joining the edge node and the end points are determined. If the angle is greater than 30°, then the edge point length ratio test is considered violated and a diagnostic message will be generated if appropriate.
- The face warp coefficient test tolerance is the cosine of the angle formed between the normal vectors located at diagonally opposite corner points on each face surface. This value is 1.0 for a face where all 4 corners lie in a plane. The default tolerance allows angles of up to 45° before a message is generated.

3. A single line of output summarizing the results of all tests for an element will be output if any of the geometry tests exceeds the test tolerance. Only the first n of these messages will be produced. A summary of the test results indicating the number of tolerances exceeded as well as the element producing the worst violation is also output. If the SUMMARY keyword has been specified, only the summary table is produced and none of the single line element messages will be output.
4. When SUMMARY is not specified, each geometry test that exceeds the tolerance will be identified in the single line output summary by an indicator based on the specification for MSGTYPE. For the FATAL option, the indicator is “FAIL”; for the INFORM option, it is “xxxx”; for the WARN option, it is “WARN”. If the FATAL option is specified and any test fails, the run is terminated.

Examples:

1. Set the tolerance for the CQUAD4 element skew angle test to 15.0 degrees and limit messages to 50.

```
GEOMCHECK   Q4_SKEW=15.0,MSGLIMIT=50
```

2. Limit messages to 500 for each element type.

```
GEOMCHECK  MSGLIMIT=500
```

3. Set the message type to fatal for CQUAD4 element taper tests.

```
GEOMCHECK  Q4_TAPER,MSGTYPE=FATAL
```

4. Request summary table output only using default tolerance values.

```
GEOMCHECK  SUMMARY
```

ID Comment

Specifies a comment.

Format:

ID [=] i1, i2

Describer	Meaning
i1, i2	Character strings (1 to 8 characters in length and the first character must be alphabetic).

Remark:

1. The ID statement is optional and not used by the program.

INCLUDE Inserts External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

Format:

INCLUDE 'filename'

Describer	Meaning
filename	Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks (’).

Example:

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA:

```
SOL 101
INCLUDE 'MYEXEC.DATA'
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
BEGIN BULK
ENDDATA
```

Remarks:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example the file:

   /dir123/dir456/dir789/filename.dat

   may be included with the following input:

   ```
   INCLUDE '/dir123
                 /dir456
               /dir789/filename.dat'
   ```

3. See the *MSC.Nastran 2005 r2 Installation and Operations Guide* for more examples.

LINK Links a Main SubDMAP

Links a main subDMAP to form a solution sequence.

Format:

$$\text{LINK} \left\{ \begin{matrix} n \\ \text{subDMAP-name} \end{matrix} \right\} [\text{SOLOUT} = \text{solout-DBset} \quad \text{EXECOUT - exeout-DBset,}$$

$$\text{INCLUDE - incl-DBset} \left[\begin{matrix} \text{MAP} \\ \text{NOMAP} \end{matrix} \right] \text{SOLNAME} = \text{newname}]$$

Describer	Meaning
n	The solution number of the main subDMAP. See the SOL statement description for the list of valid numbers. (Integer>0)
subDMAP-name	The name of a main subDMAP. See the *MD Nastran 2006 DMAP Programmer's Guide*. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
solout-DBset	The name of a DBset where the solution sequence executable and the link table of the solution sequence may be stored. See Remark **6.** (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
exeout-DBset	The name of an alternate DBset different than solout-DBset where only the solution sequence executable may be stored. See Remark **6.** (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
incl-DBset	The name of a DBset where other subDMAP objects are obtained. See Remark **2.** (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
newname	A new name which is referenced by the SOL statement. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic; default is subDMAP-name.)
MAP	Prints the link map. A link map will give the name of all the subDMAPs that make up the solution sequence.
NOMAP	Suppresses printing of the link map.

Remarks:

1. All DBsets specified on this statement must have the same BUFFSIZE. See the "**INIT**" on page 95 FMS statement.
2. SubDMAP objects are created with the COMPILE statement either in the current run or obtained from previous runs. The LINK statement collects objects in the following order:
   - Objects created with the COMPILE statement in the current run.
   - Objects residing on the DBset-name specified by the INCLUDE keyword. The default is MSCOBJ.
3. Upon successful linking of a subDMAP, the subDMAP may be executed with the SOL statement.
4. The LINK statement must appear after all the COMPILE packages, if any. A compile package begins with the COMPILE statement and is delimited by the ENDALTER, CEND, LINK, or another COMPILE statement.
5. The link table is necessary for COMPILER (or DIAG 4, 14, 17) Executive statement requests and the automatic link process.
6. EXEOUT is useful in building delivery databases where executables are not to be saved. EXEOUT will be defaulted to the same DBset as specified by SOLOUT.

Examples:

1. LINK STATICS

   Links the STATICS main subDMAP. The program links any subDMAPs compiled in this run, with any other subDMAP objects called in STATICS and stored on the MSCOBJ DBset.

2. LINK MYDMAP,SOLNAM=STATICS,SOLOUT=USROBJ,
   NOMAP,INCLUDE=USROBJ

   Links MYDMAP and rename the solution sequence executable to STATICS. The executable will be saved on the USROBJ DBset. The order of search for subDMAP objects is:
   - Compiled subDMAP in this run.
   - USROBJ DBset.

MALTER Inserts and / or Deletes DMAP Statements in Solution Sequences

Inserts or deletes DMAP statements by allowing a global “string” search across all subDMAPs within the current solution sequence.

Format:

MALTER ’string1’[(occurrence,offset)] , [’string2’[(occurrence,offset)]]

or

MALTER ’string1’[(occurrence,offset)] , [k2]

Describer	Meaning
’string1’	If ’string2’ or k2 is not specified, the subsequent DMAP statements will be inserted after the first occurrence of ’string1’.
’string1’,’string2’	DMAP statements beginning with the first occurrence of ’string1’ through DMAP statements containing the first occurrence of ’string2’ will be deleted and may be replaced with subsequence DMAP statements.
k2	If k2 is specified, it is applied to the subDMAP in which ’string1’ was found. (Integer>0)
occurrence	This flag indicates which occurrence of the preceding string is to be used, starting at the beginning of the subDMAP. (Integer>0, Default=1)
offset	This flag indicates the offset from the referenced DMAP statement. Depending on the sign the specific DMAP statement may be above (-offset) or below (+offset) the referenced DMAP statement. (Integer, Default=0)

Remarks:

1. If an MALTER statement is used in conjunction with the ALTER statement, then the MALTER should be placed above the COMPILE statements. Failure to place the MALTER in this position may cause ALTER statements to be ignored.

2. The MALTER statement can reference the DMAP statements in any order within a subDMAP. Two restrictions on ordering are:
   - K2 or 'string2'(occurence,offset) references must refer to a DMAP line number that is greater than or equal to the k1 or 'string1'(occurrence,offset) reference within a single MALTER statement.
   - 'string1' and k2 or 'string2' cannot overlap DMAP line positions with another MALTER that references the same subDMAP.
3. The 'string1' or 'string2' used as a search pattern will apply to one complete DMAP statement-i.e., a multiline DMAP statement will be searched for a pattern match as if each 72 character line of the DMAP statement were concatenated together into one string-all blanks and comments (either embedded or immediately preceding the DMAP statement) will be retained.
4. The special characters used for string searching are described in Remark **9.** The characters <, >, and $, which are common DMAP characters, are also special metacharacters. If they are to be used in the search string as regular characters, then they must be preceded by a backward slash (\). For example, to find the string

   IF (DDRMM >=-1)

   the command is

   ALTER 'IF (DDRMM \>=-1)' $
5. 'string2' (r2,02) can be defaulted to 'string1' (r1,01) by using a null string (''). For example, the alter statement

   MALTER 'string1'(r1,01),''

   is equivalent to

   MALTER 'string1'(r1,01),'string1'(r1,01)

   The defaults for (r2,02) using the null string can be overridden by specifying (r2,02).

   As another example, the alter statement

   MALTER 'string1'(r1,01),''(r2,02)

   is equivalent to

   MALTER 'string1'(r1,01),'string1'(r2,02)

6. The existing COMPILE statement options, such as LIST, XREF, SOUIN, etc., cannot be directly specified on the new MALTER statement. They are obtained as follows:
    - If a COMPILE statement exists for the subDMAP referenced by the MALTER, then options from this COMPILE statement will be used.
    - Else, they will be taken from the COMPILER statement, with the exception that the LIST, and SORT option is always on.
7. The MALTER string search order is as follows:
    - All COMPILE statement references that are part of the existing solution sequence (i.e., SOL=) are searched first.
    - Then, all remaining subDMAPs in the solution sequence are searched in ascending alphabetical order.
    - Within a subDMAP, both 'string1' and 'string2' will be used to search for a pattern match starting at the beginning of the subDMAP-not at the current position of the last string match.
8. The MALTER statement must not exceed 72 characters (no continuations are allowed).
9. Metacharacters:

.	Matches any *single* character except *newline.*
*	Matches any number (including zero) of the single character (including a character specified by a regular expression) that immediately precedes it. For example, since "." (dot) means any character, ".*" means "match any number of characters."
[...] or< >	Matches any *one* of the characters enclosed between the brackets. For example, "[AB]" matches either "A" or "B". A range of consecutive characters can be specified by separating the first and last characters in the range with a hyphen. For example "[A-Z]" will match any uppercase letter from A to Z and "[0-9]" will match any digit from 0 to 9. Some metacharacters lose special meaning inside brackets. A circumflex (^) as the first character in the bracket tries to match any one character *not* in the list.
^ or ! or .	Requires that the following regular expression be found at the beginning of the line.

$	Requires that the preceding regular expression be found at the end of the line.
\	Treats the following special character as an ordinary character. For example "\." stands for a period and "*" for an asterisk. Also, to search for a tic (’), the search string must be "\’".
’	Marks the beginning and end of a pattern to be matched.
Note:	Nonportable characters such as [ ] and ^ should be replaced (e.g., ^ → ! and [ ] → < >) if portability is required. However, all the above characters are recognized by MD Nastran.

10. Labels for use with the MALTER have been included in the solution sequences. See **Table 3-3**. These labels will be maintained in future versions and it is strongly suggested that alters which use the MALTER command take advantage of the unique MALTER labels. Use of the MALTER labels will significantly reduce the time required to convert alters between versions.

Table 3-3 DMAP Labels and Corresponding SubDMAP Positions

DMAP MALTER Labels
$MALTER:AFTER PREFACE MODULES
$MALTER:TOP OF PHASE 1 SUPERELEMENT LOOP, AFTER PARAMETERS AND QUALIFIERS SET
$MALTER:AFTER SUPERELEMENT STIFFNESS, VISCOUS DAMPING, MASS, AND ELEMENT STRUCTURAL DAMPING GENERATION (KJJZ, BJJZ, MJJZ, K4JJ)
$MALTER:AFTER X2GG MATRICES READ (K2JJ, M2JJ, B2JJ)
$MALTER:AFTER TOTAL SUPERELEMENT STIFFNESS, VISCOUS DAMPING, AND MASS FORMULATED, STRUCTURAL + DIRECT INPUT
$MALTER:AFTER SUPERELEMENT LOAD GENERATION (PJ)
$MALTER:AFTER UPSTREAM SUPERELEMENT MATRIX AND LOAD ASSEMBLY (KGG, BGG, MGG, K4GG, PG)
$MALTER:AFTER SUPERELEMENT MATRIX AND LOAD REDUCTION TO A-SET, STATIC AND DYNAMIC (KAA, KLAA, MAA, MLAA, BAA, K4AA, PA)
$MALTER:BOTTOM OF PHASE 1 SUPERELEMENT LOOP
$MALTER:AFTER X2PP MATRICES READ (K2PP, M2PP, B2PP)
$MALTER:AFTER SUPERELEMENT DISPLACEMENT RECOVERY (UG)
$MALTER:AFTER ELEMENT STRESS, STRAIN, ETC. DATA RECOVERY, SORT1 (OUGV1, OES1, OEF1, ETC.)

Table 3-3 DMAP Labels and Corresponding SubDMAP Positions (continued)

$MALTER:AFTER ELEMENT STRESS, STRAIN, ETC. DATA RECOVERY, SORT2 (OUGV2, OES2, OEF2, ETC.)
$MALTER:BOTTOM OF SUPERELEMENT DATA RECOVERY LOOP
$MALTER:USERDMAP - AFTER CALL PREFACE

Examples:

1. The following MALTER will insert a MATPRN DMAP statement to print the KJJ matrix for each superelement.

```
SOL 101
MALTER 'MALTER:AFTER SUPERELEMENT STIFFNESS .* GENERATION'
MESSAGE //'SEID='/SEID $
MATPRN KJJZ/ $
```

2. The following MALTER will add a user DMAP after the PREFACE modules in SOL 100 (USERDMAP).

```
SOL 101
MALTER 'AFTER CALL PREFACE'
.
.
.
```

MODEL_CHECK Specifies Model Check Options

Specifies model checkout run and specifies options to be used.

Format:

$$\text{MODEL\_CHECK}\left[\text{MAT\_DENSITY} = \left\{\begin{array}{c}\text{OFF}\\ \rho\\ \text{DEFAULT}\end{array}\right\}\right]\left[\text{MAT\_TECO} = \left\{\begin{array}{c}\text{OFF}\\ \alpha\\ \text{DEFAULT}\end{array}\right\}\right],$$

$$\left[\text{MAT\_TEIJ} = \left\{\begin{array}{c}\text{OFF}\\ \alpha_{ij}\\ \text{DEFAULT}\end{array}\right\}\right]\left[\text{MAT\_DAMPING} = \left\{\begin{array}{c}\text{OFF}\\ g_e\\ \text{DEFAULT}\end{array}\right\}\right],$$

[CHECKOUT][PRINT = item_list]

Describer	Meaning
MAT_DENSITY	Selects material density processing option.
ρ	Value to be used for the density.
MAT_TECO	Selects material thermal expansion direct coefficient processing option.
α	Value to be used for the thermal expansion direct coefficients.
MAT_TEIJ	Selects material thermal expansion shear coefficient processing option.
α_{ij}	Value to be used for the thermal expansion shear coefficients.
MAT_DAMPING	Selects material structural element damping processing option.
g_e	Value to be used for the structural element damping coefficient.
OFF	Material property value is set to zero.
DEFAULT	Material property value is set to system default value. See Remark 3.
CHECKOUT	Selects model checkout solution option. See Remark 5.

Describer	Meaning
PRINT	Selects items to be printed during model checkout solution.
item_list	List of model data items to be printed during model checkout run. If more than one item is specified, enclose the list in parenthesis. See Remark 6.

Remarks:

1. The MODEL_CHECK statement is ignored in RESTART runs.
2. The values specified for material properties using the MODEL_CHECK statement will be used to temporarily update data for **all** MAT1, MAT2, MAT3, MAT8 and MAT9 bulk data entries only for the duration of the run. These values do NOT replace data specified on the MATi Bulk Data entries. Caution should be used when post-processing results via the param POST options since operations using in-consistent data could be performed.
3. System default values of 0.0 have been defined for each of the properties. The defaults can be changed using the following Nastran statement keywords: DEF_DENS for MAT_DENSITY, DEF_TECO for MAT_TECO, DEF_TEIJ for MAT_TEIJ and DEF_DAMP for MAT_DAMPING.
4. The MAT_TECO describer causes the direct components of the thermal expansion coefficient to be modified. The MAT_TEIJ describer causes the shear components of the thermal expansion coefficient to be modified.
5. The CHECKOUT option has the same effect as a PARAM,CHECKOUT,YES bulk data entry.
6. The following table summarizes the acceptable specifications for the PRINT item_list.

Value	Output Generated	Parameter
CSTM	Coordinate systems	PRTCSTM
BGPDT	Basic Grid point data	PRTBGPDT
GPTT	Grid point temperature data	PRTGPTT
MGG	G-set Mass matrix	PRTMGG
PG	G-set load vectors	PRTPG

See the DMAP parameter descriptions in Section 5 for a discussion of the parameter name in the last column of the table and the output generated. The specification of a print item has the effect of adding a PARAM,parameter,YES entry to the Case Control portion of the file.

Examples:

1. Execute a basic model checkout run. No special output is required.

   MODEL_CHECK CHECKOUT

2. Execute a model checkout run. Print coordinate system and basic grid point data.

   MODEL_CHECK CHECKOUT PRINT=(CSTM,BGPDT)

3. Execute a full solution. Modify the material density temporarily to a value of 0.0.

   MODEL_CHECK MAT_DENSITY=OFF

   or

   MODEL_CHECK MAT_DENSITY=0.0

4. Execute a full solution. Temporarily modify the values for material density and thermal expansion coefficient.

   MODEL_CHECK MAT_DENSITY=0.001 MAT_TECO=1.0 MAT_TEIJ=0.0

SOL Executes a Solution Sequence

Specifies the solution sequence or main subDMAP to be executed.

Format:

$$\text{SOL} \left\{ \begin{matrix} n \\ \text{subDMAP-name} \end{matrix} \right\} [\text{SOLIN} = \text{obj-DBset} \quad \text{NOEXE}]$$

Describer	Meaning
n	Solution number. See Remark **6.** for the list of valid numbers. (Integer>0)
subDMAP-name	The name of a main subDMAP. See the *MD Nastran 2006 DMAP Programmer's Guide*. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
obj-DBset	The character name of a DBset where the OSCAR is stored. See Remarks **1.** and **2.** (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic.)
NOEXE	Suppresses execution after compilation and/or linkage of the solution is complete. Also, the Bulk Data and Case Control Sections are not read or processed.

Remarks:

1. If SOLIN keyword is not given and if there are no LINK statements within the input data, the program will perform an automatic link. The program will first collect the objects created in the current run by the COMPILE statement and the remaining objects stored in the MSCOBJ DBset. The program will then perform an automatic link of the collected objects.
2. If the SOLIN keyword is not given but a LINK statement is provided, the SOLIN default will be obtained from the SOLOUT keyword on the LINK statement.
3. The OSCAR (Operation Sequence Control ARray) defines the problem solution sequence. The OSCAR consists of a sequence of entries with each entry containing all of the information needed to execute one step of the problem solution. The OSCAR is generated from information supplied by the user's entries in the Executive Control Section.
4. The SOLIN keyword will skip the automatic link and execute the OSCAR on the specified DBset.

5. The DOMAINSOLVER may be used in conjunction with Solution Sequences 101, 103, 108, and 111 to select domain decomposition solution methods.
6. The following Solution Sequences are currently available in MD Nastran:

Table 3-4 Solution Sequences

SOL Number	SOL Name	Description
101	SESTATIC	Statics with Options: Linear Steady State Heat Transfer Alternate Reduction Inertia Relief
103	SEMODES	Normal Modes
105	SEBUCKL	Buckling with options: Static Analysis Alternate Reduction Inertia Relief
106	NLSTATIC	Nonlinear or Linear Statics
107	SEDCEIG	Direct Complex Eigenvalues
108	SEDFREQ	Direct Frequency Response
109	SEDTRAN	Direct Transient Response
110	SEMCEIG	Modal Complex Eigenvalues
111	SEMFREQ	Modal Frequency Response
112	SEMTRAN	Modal Transient Response
114	CYCSTATX	Cyclic Statics with Option: Alternate Reduction
115	CYCMODE	Cyclic Normal Modes
116	CYCBUCKL	Cyclic Buckling
118	CYCFREQ	Cyclic Direct Frequency Response
129	NLTRAN	Nonlinear or Linear Transient Response
144	AESTAT	Static Aeroelastic Response
145	SEFLUTTR	Aerodynamic Flutter
146	SEAERO	Aeroelastic Response

Table 3-4 Solution Sequences (continued)

SOL Number	SOL Name	Description
153	NLSCSH	Static Structural and / or Steady State Heat Transfer Analysis with Options: Linear or Nonlinear Analysis
159	NLTCSH	Transient Structural and / or Transient Heat Transfer Analysis with Options: Linear or Nonlinear Analysis
190	DBTRANS	Database Transfer, "**Output Description**" on page 373 of the *MSC.Nastran Reference Guide.*
200	DESOPT	Design Optimization
400	NONLIN	Nonlinear static and transient analysis

Examples:

1. In the following example, SOL 103 is executed from MSCOBJ.

```
SOL 103
```

2. In the following example, the PHASE0 subDMAP is altered, SOL 103 is relinked onto the OBJSCR DBset (which is the default for SOLOUT), and SOL 103 is executed.

```
SOL 103
COMPILE PHASE1
ALTER 'DTIIN'
TABPT SETREE,,,,// $
.
.
.
ENDALTER $
```

3. In the following example, the solution sequence called DYNAMICS is executed from the USROBJ DBset.

```
SOL DYNAMICS SOLIN = USROBJ
```

SOL 600,ID Executes MSC.Marc from Inside MD Nastran

Creates MSC.Marc input and optionally executes MSC.Marc from inside MD Nastran Implicit Nonlinear (SOL 600)

Format:

SOL 600, ID PATH= COPYR= NOERROR OUTR=op2,xdb,pch,f06,eig,dmap,beam, sdrc,pst MARCEXE=SOLVE NOEXIT STOP= CONTINUE= S67OPT=MSGMESH SMEAR

Examples:

SOL 600,106 PATH=/progs/marc2003/tools OUTR=op2,f06
SOL 600,106 PATH=1 STOP=1
SOL 600,106 PATH=1 CONTINUE=1
SOL 600,106 PATH=1 MARCEXE=SOLVE OUTR=OP2

SOL 600,ID is an Executive Control statement similar to SOL. The difference between SOL and SOL 600,ID is that the computations (element matrix formulations, matrix decomposition, etc.) will be performed by MSC.Marc rather than directly within MD Nastran. Inputs and outputs as much as possible will be the same as (or similar to) the familiar MD Nastran inputs and outputs. The SOL 600,ID statement should normally be used only for nonlinear analysis, but it may also be used for certain classes of linear static or dynamic analyses. The recommended form of this command is shown with the options provided above. If entered with "SOL 600,ID" only, it acts just like SOL except a MSC.Marc input data file “jid.marc.dat” will be generated ("jid" is the name of the MD Nastran input file without the extension.) For example, if the MD Nastran input file is named abcd.dat, (or abcd.bdf) then "jid"=abcd.

The required ID represents many valid solution sequence integer or names shown in **Table 3-4** for the SOL statement. Examples are 106, 129, NLSTATIC, NLTRAN. The following solutions are not available: 107, 110, 114, 115, 116, 118, 144, 145, 146, 153, 159, 190, 200, and 400 (and their equivalent names). Solutions specified in **Table 3-4** of the SOL statement may be used except for 7, 10, 14-16 and their equivalent names. In future releases, the heat transfer solutions will be added.

All items on the SOL 600,ID after ID itself may be specified in environmental variables. This may be done any way environmental variables can be set. They may be set by the MD Nastran user at run time or by the system administrator when MD Nastran is installed. Any values specified on the SOL statement override those in the environment. Environmental variables are fully described in the *MSC.Nastran Installation and Operations Guide.* A keywords file is available to describe the format of each variable. The variable is normally set in the system-wide rc file, a user’s rc file, a

local rc file or in a script used to submit MD Nastran. Any string or value listed on the SOL 600,ID statement is also valid as an environmental variable. If the environmental variables are placed in the system-wide rc file, they may be used by a company for all MD Nastran users and even hide the fact that MSC.Marc is being spawned if so desired.

The following environmental variables are available:

Environmental Variable	Item on SOL Statement
NASM_PATH	PATH
NASM_COPYR	COPYR
NASM_OUTR	OUTR
NASM_STOP	STOP
NASM_NOERROR	NOERROR
NASM_STRFILE	Path and name of marcfilt.txt file (see below)

PATH

PATH is an optional item which determines the location of the version of MSC.Marc to be executed. If PATH is omitted, the version of MSC.Marc included with MD Nastran will be used if it can be located. In this case, the run script for MSC.Marc (run_marc or run_marc.bat) will be expected to be in the directory under /MSC_BASE. MSC_BASE is an environmental variable set when MD Nastran first starts execution that defines the base installation directory for MD Nastran. If for some reason MSC_BASE cannot be determined, the commands to spawn MSC.Marc will fail and the user must re-run MD Nastran with one of the PATH options set or the NASM_PATH environmental option set to the desired location of MSC.Marc's tools directory.

PATH=1

If PATH=1 (the recommended option) is specified, MD Nastran will determine the proper command to execute the companion program. To aid MD Nastran in determining the program's location, a file named marcrun.pth must be available in the same directory where the MD Nastran input file resides. The marcrun.pth file must contain one line providing the location (complete path) of the run_marc script. A typical example of the line in the file marcrun.pth would be

```
/mycomputer/marc200x/tools
```

To this path is appended the string "/run_marc -jid name.marc -v no" and possibly other items to form the complete string used to execute MSC.Marc. This complete string looks like the string shown in the following PATH=3 example. Note that on Windows systems, substitute a back slash for the forward slashes shown. Do not terminate the line with a forward slash or back slash.

PATH=2

If PATH=2 is specified, it is expected that the directory with the run_marc script is on the PATH. If PATH=2 is specified, MSC.Marc will be executed from inside MD Nastran using the command:

```
run_marc -jid jid.marc.dat -v no
```


PATH=3

When PATH=3 is specified, the complete command to execute MSC.Marc must be contained in a file named marc.pth (lowercase). This file should typically contain one line of the form:

```
/mycomputer/marc200x/tools/run_marc -jid name.marc -v no
```

COPYR

COPYR is an optional item. If COPYR is specified, MSC.Marc output files will be copied to MD Nastran output files and/or deleted according to the options shown in the following table:

COPYR Option	Copy MSC.Marc Output Files to MD Nastran Output Files	Delete MSC.Marc Input & Output Files
0 (default)	No	No
1 or -1 (see below)	Yes	Yes
2 or -2 (see below)	Yes	No
3	No	Yes

If COPYR is 1 or 2, MSC.Marc's out and log files will be copied exactly as produced by MSC.Marc.

If COPYR is -1 or -2 the actions as shown above for +1 or +2 will occur, and MSC.Marc-type text will be converted to MD Nastran-type text (or any other desired text) using an ASCII file named marcfilt.txt. This file must be located in the same directory where

the MD Nastran input resides or in the same directory where the MSC.Marc executable resides. The marcfilt.txt file can contain as many lines as desired like the one shown below:

"Marc string 1" "Replacement String 1"
"Marc string 2" "Replacement String 2"

That is, each line contains two strings. Each string starts and ends with a double quote sign ("). The MSC.Marc string must match the exact content and case as found in the MSC.Marc .out or .log files. The replacement string may be any string desired and can be the same length, shorter or longer than the MSC.Marc string. The two strings must be separated by at least one space, but more spaces are acceptable. Line lengths for marcfilt.txt, as well as MSC.Marc's .out and .log files are limited to 200 characters for the text replacement option.

The following MSC.Marc files are potentially affected by the COPYR option:

MSC.Marc Output File	MD Nastran Output Copied to	COPYR
name.marc.out	name.f06	1, 2, -1, -2
name.marc.log	name.log	1, 2, -1, -2
name.marc.t16	Not copied, will remain if produced	
name.op2, fort.11 or ftn11	Not copied, will remain if produced	

MARCEXE=SOLVE

MARCEXE=SOLVE is an optional item. If MARCEXE is entered, an existing input file named jid.marc.dat is assumed to exist in the directory where the run was submitted. Nastran will execute Marc using the existing jid.marc.dat file. A new Marc file will not be created. Other options available when MARCEXE is used are PATH and OUTR. Options not available with MARCEXE are COPYR, STOP, NOEXIT, NOERROR and CONTINUE. Beware that the original jid.marc.dat will be renamed to jid.marc.dat.1 automatically by Nastran just like an existing jid.f06 is renamed to jid.f06.1

NOERROR

NOERROR is an optional item. If NOERROR is specified, errors due to features that are available in MD Nastran but not available in MSC.Marc, and/or features not yet supported by the translator will be ignored (see Restrictions and Limitations). If

NOERROR is entered and STOP=2 (or 3) is not specified, MSC.Marc will be executed even though the complete MD Nastran model may not have been completely translated. We recommend that NOERROR only be used by experienced analysts and then only with extreme caution.

NOEXIT

NOEXIT is an optional item. If entered, the DMAP generated "on the fly" to process the OUTR options will not contain EXIT and MD Nastran will proceed. This means in most cases, the MD Nastran solution as well as the MSC.Marc solution will occur. If f06 is specified as one of the OUTR options, this could cause confusing output as both the MSC.Marc and MD Nastran results will be in the f06 file. Confusion could also result from both outputs being in op2, xdb and/or punch files. Therefore, this option should only be used with great care.

Listing of the DMAP generated on the fly for SOL 600 can be suppressed by placing ECHOOFF just after the SOL 600 entry.

OUTR

OUTR is an optional item. If OUTR is specified, MSC.Marc output results will be converted to various types of MD Nastran formats. The OUT option on the Nastran command should not be used with any OUTR options. The type of output to be produced depends on the OUTR options entered as well as any DMAP entered in the executive control. If OUTR is omitted, no MSC.Marc output will be brought back into MD Nastran, but standard MSC.Marc .out, .t16 and/or .t19 as well as an op2 file will be available depending on the options selected with PARAM,MARCT16, PARAM,MARCT19 and other options. The following options are available:

Option 1 -- Specify a String of Desired Output Types (Preferred Method)

OUTR=OP2,F11,F06,PCH,XDB,T16,T19,PST

Use any or all of the above to request the following options:

OP2 Create output2 file named jid.op2 consisting of input model and output results datablocks. This option requires PARAM,POST,-1 or PARAM,POST,-2 in the Bulk Data.

F11 Create output file fort.11 or ftn11 (depending on the computer system) consisting of output results datablocks only.

F06 Put MSC.Marc output results (displacements, stresses, strains) in MD Nastran's jid.f06 file using OFP. The resulting output will look just like any standard MD Nastran run.

PCH Create punch file named jid.pch with MSC.Marc's output in standard MD Nastran punch format.

XDB Create xdb database file named jid.xdb with input model and output results. This option requires PARAM,POST,0 in the Bulk Data. XDB is not available with the eig option and if entered will switch to the OP2 option.

eig The eig option must be specified if op2, xdb, pch, or f06 options are specified and MSC.Marc performs natural frequency or buckling eigenvalue analysis. The reason it must be provided on the SOL entry is to enable MD Nastran to create DMAP on the fly which include the LAMA data block. If the eig option is omitted, eigenvectors will be present in the MD Nastran output but no eigenvalues will be available. The beam and eig options are mutually exclusive (you cannot specify both).

BEAM The BEAM option must be specified if op2, xdb, pch, or f06 options are specified and you want to place internal loads in any of these files.The BEAM option is not available for Windows systems.

SDRC An SDRC op2 file will be produced. PARAM,POST,-2 is also necessary in the Bulk Data for this option. Note: The datablocks might be in a different order than for other solution sequences.

T16 MSC.Marc's results will be saved during the MSC.Marc execution on a binary (or unformatted) file named jid.marc.t16 (this happens by default and does not need to be specified on the SOL 600 line).

T19 MSC.Marc's results will be saved during the MSC.Marc execution on an ASCII file named jid.marc.t19. The t19 file will normally be saved if param,maract19,1 is entered.

PST MD Nastran will be run to output a previous MSC.Marc run's results contained on t16 file in the desired forms (OP2,F11,F06,PCH and/or XDB). The appropriate OUTR T16 options must be selected in addition to PST (specify one or more of OP2,F11,F06,PCH and/or XDB.) MD Nastran will not be run past IFP and is used only to perform the desired output results conversions. A previous MSC.Marc t16 file must be copied to the new jid.marc.t16.1 (you may not process XDB and OP2 in the same run.)

DMAP The user will enter his own DMAP to create whatever type of output that is desired, such as op2, xdb, pch, or .f06. For all other options, DMAP as needed is generated internally by MD Nastran.

cdb 3D Contact will be output in one of the datablocks described below:

0 Store output in OESNLBR and OESNLXR (OESNLXR will be empty like SOL 106)

1 Store output in OESNLBR

2 Store output in OESNLBD

3 Store output in OESNLXD

Note:

1. This type of output was not available prior to the MSC.Nastran 2005 r2 release, only cdb=3 was available. Starting with MSC.Nastran 2005 r3, cdb options 0-3 are available.
2. SOL 106 outputs both OESNLBR and OESNLXR but OESNLXR is empty.
3. SOL 129 only outputs the OESNLXD datablock and it is empty.
4. Case Control BOUTPUT is also required to obtain this type of output.
5. The default is 0 if contact is present in the model and OUTR=op2 (or xdb, punch and/or f06).
6. This option is specified like the example shown:

   OUTR=OP2,F06,CDB=0
7. The datablocks have the same names and type of information whether executing SOL 600,106 or SOL 600,129.

Option 2 -- Specify an Integer to Select Certain Options (Not Recommended)

OUTR=1 or 2 and an op2 file named fort.11 or ftn11 will be produced and DMAP as shown below is required to bring the Marc output results back into the Nastran database.

```
COMPILE NLSTATIC
ALTER 'SUPER1' $
INPUTT2 /OUGV1,OES1,OSTR1,TOL,/-1/11 $
OFP OUGV1,OES1,OSTR1/ /0/1 $
EXIT $
```

The 1 at the end of the OFP statement produces output in the f06 file. If a punch file is also needed, change the 1 to a 5. If an XDB file is also needed, add the following lines just after the OFP line:

```
DBC TOL,CASECC,,,,,,,,,,,,,,,,,,//
   'OL'/'CASECC'////////////////////
   -1/DBCPATH/S,N,CP/''TRAN''//GEOMU/LOADU/POSTU/
   DBCDIAG/DBCCONV/DBCOVWRT $
DBC OUGV1,OES1,,,,,,,,,,,,,,,,,,//
   'OUG'/'OES'////////////////////
   -1/DBCPATH/S,N,CP/''TRAN''//GEOMU/LOADU/POSTU/
   DBCDIAG/DBCCONV/DBCOVWRT $
DBC OSTR1,,,,,,,,,,,,,,,,,,//
   'OES'////////////////////
   -1/DBCPATH/S,N,CP/''TRAN''//GEOMU/LOADU/POSTU/
   DBCDIAG/DBCCONV/DBCOVWRT $
```

OUTR can be set to one of the following values to automatically produce the output in Nastran form without entering any DMAP. In fact, no DMAP should be entered for the options greater than 2 shown:

Table 3-5 Integer Options Available Using SOL 600 OUTR Option -- Nastran Output Results Produced When Marc Exits

OUTR (IO)	OP2 with Input Datablocks	fort.11 or ftn11 Output Datablocks Only	.f06 (Print)	.pch (Punch)	.xdb	Marc File Used
1	N	Y	N	N	N	.t19
2	N	Y	N	N	N	.t16
16	Y	Y	N	N	Y	.t16
166	Y	Y	Y	N	Y	.t16
266	Y	Y	N	Y	Y	.t16
366	Y	Y	Y	Y	Y	.t16
19	Y	Y	N	N	Y	.t19
199	Y	Y	Y	N	Y	.t19
299	Y	Y	N	Y	Y	.t19
399	Y	Y	Y	Y	Y	.t19

If OUTR = –1, -2, –16, -166, -266, -366, -19, -199, -299 or –399 only the output conversion process takes place. An MSC.Marc input file is not produced, MSC.Marc is not spawned from MD Nastran, but op2, xdb, pch and / or f06 results can be produced. For such cases, the Case Control and Bulk Data files can be dummies (for example, they can contain several nodes and one element) or a full file could be used. These options are handy if MSC.Marc is run by modifying the MSC.Marc input file (jid.marc.dat) with an editor or for someone who creates MSC.Marc input and runs MSC.Marc outside the MD Nastran environment, but wants output in one of the Nastran formats (see remark 7).

STOP

STOP is an optional item. STOP is used to prevent execution of MSC.Marc or exit MD Nastran after IFP, if so desired. DO NOT ENTER any of the STOP options if any of the OUTR options are entered as the DMAP generated automatically by MD Nastran will put an EXIT in the proper place. The various options are as follows:

STOP=0

If STOP=0 MD Nastran will not be stopped after MSC.Marc exits. MD Nastran will attempt to obtain its own solution to the problem if possible. Use of this option can lead to confusion because results from both MSC.Marc and MD Nastran will be available. If the MSC.Marc results are placed in the .f06 file and if the MD Nastran results are also available in the .f06 file, it will be difficult to tell which results came from MD Nastran and which results came from MSC.Marc. This also applies to op2 files and xdb files. It is suggested the STOP=0 option be used by extremely experienced SOL 600 users and even then with great care.

STOP=1

If STOP=1 MD Nastran will be gracefully stopped after IFP. This option is used to prevent MD Nastran from performing its own solution (normally used when the solution is performed by the MSC.Marc). STOP=1 should be normally used if OUTR is not specified. STOP=1 is the default if no STOP, CONTINUE or OUTR options are entered.

STOP=2

For STOP=2 MSC.Marc will not be executed. This option is used if you wish to examine the MSC.Marc input file and make changes prior to running MSC.Marc. However, if STOP=2 is entered, the OUTR options will not be available.

STOP=3

STOP=3 is a combination of STOP=1 and STOP=2. MD Nastran is stopped after IFP and MSC.Marc is not executed. This would be the normal STOP option if you want to examine a MSC.Marc input file, then execute MSC.Marc manually. The STOP=2 option is normally used if you want to obtain comparative results between standard MD Nastran solutions and MSC.Marc solutions (in which case, all input options must be fully supported by both programs). If STOP=3 is entered, the OUTR options will not be available.

CONTINUE=

CONTINUE= specifies an option as to how MD Nastran will continue its analysis after MSC.Marc finishes. For this to happen, do not enter any STOP or OUTR options. It is possible to perform more than one of these operations if necessary.

1 MD Nastran will continue the current solution sequence as normal. For example if SOL 600,106 is entered, SOL 106 will continue as normal after MSC.Marc finishes. Of course, no 3D contact or materials not supported by SOL 106 may be used.

2 MD Nastran will switch to SOL 107 to compute complex eigenvalues. MSC.Marc will generate DMIG matrices for friction stiffness (and possibly damping) on a file specified by pram,marcfil1,name and time specified by param,marcstif,time. This is accomplished by making a complete copy of the original MD Nastran input file and spawning off a new job with the SOL entry changed and an include entry for the DMIG file.

3 (Option not presently available.) MD Nastran will switch to SOL 107 to compute complex eigenvalues. MSC.Marc will generate OUTPUT4 matrices for friction stiffness (and possibly damping) on a file specified by pram,marcfil2,name and time specified by param,marcstif,time, This is accomplished by making a complete copy of the original MD Nastran input file and spawning off a new job with the SOL entry changed and an include entry for the DMIG file.

The original MD Nastran file should include CMETHOD=id in the Case Control command and a matching CEIG entry in the Bulk Data.

MD Nastran will switch to SOL 111 to compute modal frequency response. MSC.Marc will generate natural frequencies and mode shapes in (tbd) format which are read into MD Nastran from a file specified by param,marcfil3,name.

4 (Option not presently available.) Same as option 3 except SOL 112 for linear transient response will be used.

5 MD Nastran will switch to the solution sequence given in field 9 of the MDMIOUT entry.

In addition, the DMIG entries specified by MDMIOUT will be included in a separate MD Nastran execution spawned from the original execution. Case Control and Bulk Data will be added to the original input to properly handle these matrices in the spawned MD Nastran execution.

6 Same as option 1 except SOL 110 is run. For this option, the original MD Nastran input file must contain METHOD=ID1 and CMETHOD=ID2 in the Case Control as well as matching EIGRL (or EIGR) and CEIG entries in the Bulk Data.

7 Same as option 1 except SOL 103 is run for real eigenvalues / eigenvectors. The database can be saved to restart into SOL 110 if desired. This should be done on the command line or in a rc file with scratch=no. For this situation, the original MD Nastran input file must include METHOD=id in the Case Control command and a matching EIGRL or EIGR entry in the Bulk Data. (CMETHOD and CEIG can also be included.) The actual restart from SOL 103 to 110 must be performed manually at the present time.

101+ Continue options 101 to 400 are used to convert MSC.Marc's initial contact tying constraints to MPC's and then continue in SOL 101 to 112 as a standard MD Nastran execution. For example, if CONTINUE=101, a SOL 101 run with all the geometry load cases, etc. from the original run would be conducted with the addition of the initial contact MPC determined from MSC.Marc. The continue=101+ options are frequency used to model dissimilar meshes as well as glued contact which does not change throughout the analysis. This option can be used for any standard MD Nastran sequence where the initial contact condition does not change. In order for initial contact to work, the surfaces must be initially touching. If they are separated by a gap, the MPC's will be zero until the gap closes and thus the initial MPC's are zero. This option automatically sets BCPARA INITCON=1.

S67OPT=NO

If S67OPT=NO is entered the following action will be taken for SOL 600 or 700: TA1MCK and EMGPRO will not be disabled (when these routines are disabled, materials used only by SOL 600 or 700 such as MATG, MATF, MATHP, etc. may be in the model and the t16op2 conversion will take place, otherwise the job will fail with a FATAL ERROR). Also, Case Control FATAL error termination will occur at the same place as other MD Nastran Solution Sequences. If S67OPT=YES or S67OPT is omitted entirely, TA1MCK and EMGPRO will be disabled and Case Control FATAL ERRORS will cause job termination immediately. S67OPT=YES is the default for MSC.Nastran 2005 r3 and subsequent.

MSGMESH

Is an optional item and should be omitted unless the Bulk Data contains some entries in the MSGMESH format. If there are any MSGMESH entries present, set MSGMESH=Nhigh. Where Nhigh is an integer equal to the largest node, element, material or property in the model (after MSGMESH entries have been expanded). Optionally this item can be set as an environmental variable S600_M001=Nhigh. If

MSGMESH entries are present and neither S600_M001 nor MSGMESH on the SOL 600 entry are set, the job will terminate with an appropriate message. Nhigh is required for memory allocation of such jobs. For the MSC.Nastran 2005 r3 release only the environmental variable option (S600_M001=nhigh) was available. MSGMESH on the SOL 600,ID entry is available starting with the MD Nastran 2006 release.

SMEAR

If the string SMEAR is entered on the SOL 600,ID command line, composite shell entries using PCOMP will use the smeared approach similar to other Nastran solution sequences. If SMEAR is not entered, the through-the-thickness integration approach will be used. The smeared approach is identical to other Nastran solution sequences where PCOMP entries are converted to PSHEL and MAT2 entries. The through-the-thickness integration approach is more accurate for post-buckling and nonlinear analyses but takes more computer time. OP2.f06 and punch outputs are available and are controlled by the OUTR options OUTR=xxx where xxx is OP2, F06 and / or PCH. If any OUTR options are specified, OP2 must be included. In addition, standard Case Control requests are required.

SMEAR Option Restrictions (Initial MD Nastran Release)

1. For the initial MD Nastran release, the SMEAR option may only be used if the only composite materials in the model are made of shell elements (if there are any composite solid elements, this option may not be used.)
2. Case Control requests for DISP(options)=ALL, the STRESS(options)=ALL must be entered. STRAIN(options)=ALL is optional. (options) consist of any combination of (print,plot,punch)
3. The SMEAR output options may not be controlled using sets.
4. It is suggested that the Marc t16 file be limited to only those output “times” absolutely necessary as composite output can be large and take significant computer time.

Running SOL 600 in Steps -- Modification of the SOL 600 Statement Using Environmental Variables:

For MSC.Nastran 2005 r2 and beyond it is possible to run the main portions of SOL 600 in single steps without changing the Nastran input file. This is accomplished using one of the two environmental variables discussed below. A user can set these variables in a script that runs MD Nastran, from the command line or for Windows

using the control panel. Note that on UNIX and Linux systems, the name of the environmental variable must be in upper case. The string to which it is set can be in upper or lower case and will be converted to upper case.

To Run SOL 600 in Three Steps Without Changing the SOL 600 Statement in the Input File:

First, make sure that your SOL 600 input file has a SOL 600 statement that contains all of the features you would want if all steps were done in a single run. For example, if you wish to make an op2 file and place the results in the f06 file, a typical SOL 600 statement would be as follows:

```
SOL 600,NLSTATIC PATH=1 OUTR=OP2,F06
```

or

```
SOL 600,NLSTATIC OUTR=OP2,F06
```

(if the default path to Marc is to be used).

It is important to have the OUTR options specified at the end of the SOL 600 statement. The following environmental variable can be set as shown to run the three steps (a UNIX Korn shell example is shown):

1. export MARC_RUN=”stop”

   This will tell MD Nastran to run the internal MD Nastran-to-MSC.Marc translator only. The first SOL 600 statement shown would be changed internally just for this run above to the following:

   ```
   SOL 600,NLSTATIC PATH=1 OUTR=OP2,F06 STOP=3
   ```

   This change will be shown in the f06 file.

2. export MARC_RUN=”solv”

   This will tell MD Nastran to run MSC.Marc from inside MD Nastran. The first SOL 600 statement shown would be changed internally just for this run above to the following:

   ```
   SOL 600,NLSTATIC PATH=1 OUTR=OP2,F06 MARCEXE=SOLVE
   ```

   For this run, it is assumed that a file named jid.marc.dat resides in the input file directory created from a previous translator-only run. The MD Nastran script will automatically rename jid.marc.dat to jid.marc.dat.1, but the MD Nastran executive processing will name it back to jid.marc.dat

3. export MARC_RUN=”pst” This will tell MD Nastran to run the t16 to op2 translator inside MD Nastran. The first SOL 600 statement shown would be changed internally just for this run above to the following:

SOL 600,NLSTATIC PATH=1 OUTR=OP2,F06,PST

For this run, it is assumed that a file named jid.marc.t16 as well as the original MD Nastran input file jid.dat resides in the input file. The “t16” file should have been created from a previous MSC.Marc execution using the same computer system (cross-platform support is not available for this step). The MD Nastran script will automatically rename jid.marc.t16 to jid.marc.t16.1, and the MD Nastran t16op2 conversion routines will look for files with names jid.marc.t16, jid.marc.t16.1, jid.marc.t16.2 up to jid.marc.t16.5 in that order. If no such files are found, the t16op2 job will exit with a message.

Method to Completely Modify the SOL 600 Statement:

For maximum versatility without having to modify the MD Nastran input file, the SOL 600 statement can be modified completely using the environmental variable SOL600_CMD. Assuming that the original SOL 600 statement in jid.dat contains the string:

SOL 600,NLSTATIC OUTR=OP2,F06

and the environmental variable is set as follows:

export SOL600_CMD=”SOL 600,NLSTATIC PATH=1 MARCEXE=SOLVE”

the new SOL command line internal to MD Nastran will be

SOL 600,NLSTATIC PATH=1 MARCEXE=SOLVE

and MD Nastran will stop after creating the MSC.Marc input file. This would be the same as if the following SOL 600 statement was entered:

SOL 600,NLSTATIC PATH=1 STOP=3

Any valid SOL 600 statement can be issued using the SOL600_CMD environmental variable without changing the original MD Nastran input file at all.

Caution: Do not set both SOL600_CMD and MARC_RUN environmental variables on the same run. Do not use OUT= or OUTDIR in connection with either of these environmental variables.

Remarks:

1. The function of SOL 600,ID can alternatively be controlled by parameters (marc3d, marcrun, marconly, marccpy, marcver). We suggest that either SOL 600,ID or the parameters be entered, not both. Creation of op2, xdb, pch, etc., output must be controlled by using the OUTR option on SOL 600,ID and applicable Case Control requests (both are required).

2. Only one SOL 600,ID job may be run in a directory at any given time. However, if a previous run was made and output files such as name.marc.t16 were produced, they will be renamed name.marc.t16.1, etc. following the MD Nastran re-naming convention.
3. If OUTR is specified, STOP must not be specified.
4. The COPYR option can be used to delete all files directly created by MSC.Marc if the output desired are MD Nastran files only.
5. When OUTR is specified, the MSC.Marc files such as jid.marc.out, jid.marc.t16 will be renamed to jid.marc.out.1 and jid.marc.t16.1 at the start of the run. This renaming is accounted for when opening the files. However, do not start with files named jid.marc.t16.1 or jid.marc.t16.2 etc. for this option.
6. To generate xdb files, PARAM,POST,0 must be included in the Bulk Data Section. To generate OP2 files with geometry, PARAM,POST,-1 (for MSC.Patran and Femap) or PARAM,POST,-2 (for SDRC) should be included in the Bulk Data.
7. For Windows systems, the Intel Version of Marc is required if OUTR is specified. If OUTR is not specified, either the Intel or Digital Visual Fortran Version of Marc may be used.
8. Material IDs (MID) are restricted to a maximum value of 399,999 on 32-bit systems. This is an MSC.Marc memory restriction, not an MD Nastran restriction.
9. If PARAM,OGEOM,NO is specified, it will be changed to PARAM,OGEOM,YES so that geometry datablocks will always be written when OUTR=OP2 is specified on the SOL 600 statement
10. The ID or the largest GRID number or element number must be equal to or greater than the ID of the largest property or material number.
11. Although SOL 600,ID supports linear analysis (ID=101, 103, 105), not all features are available. For example, Case Control commands, STATSUB, SUBCOM, SUBSEQ, SYMCOM, AUXMODEL, AXISYMMETRIC, CLOAD, DEFORM, HARMONICS, MFLUID, NSM, and REPCASE are not available. For nonlinear analyses (ID=106, 129) Case Control commands, NNLOAD and NONLINEAR are not available.
12. Element IDs and Grid IDs are restricted to maximum values of 9,999,999
13. To output displacements in the jid.marc.out file, do the following:

    In the Case Control, set DISP(PRINT)=ALL or DISP(PRINT,PLOT)=ALL

In the Bulk Data, include the following two parameters:

PARAM,MARCPRNH,1

PARAM,MARCND99,-1

14. All SOL 600 character variable parameters, such as MRAFFLOW, must be left justified in the starting in field 3.
15. If generation options such as EGRID, GRIDG, CGEN, etc., are used, the MSGMESH option on this entry or an environmental variable, S600_M001 must be set with a value equal to the largest grid ID or element ID (whichever is larger). This is needed for initial SOL 600 memory sizing, which happens before these generation entries are converted to standard MD Nastran GRID, CQUAD4, etc. entries. SOL 600 can not read the generation features. This environmental variable needs to be included in the script or batch file that runs SOL 600 jobs (for Windows, it can be set in the environment).
16. Fixed load stepping (or time stepping) is controlled primarily by PARAM,MARCITER and PARAM,MARCAUTO rather than NLPARM or NLAUTO.

Restrictions and Limitations of MD Nastran Implicit Nonlinear (SOL 600)

Because MD Nastran Implicit Nonlinear (SOL 600) is a new product, and since certain features are available in MD Nastran that are not available in MSC.Marc, and vice versa, the following restrictions / limitations are imposed on MSC.Nastran 2005 r3. Those restrictions indicated by (*) will be removed as soon as possible. Items with (**) will issue a FATAL error (for the MD Nastran-to-MSC.Marc translator internal to MD Nastran) and MSC.Marc will not be “spawned” from MD Nastran unless NOERROR is entered on the SOL 600 statement.

- External superelements are supported starting with MSC.Nastran 2005 r2. Other types of superelements are not currently supported.
- DMIG (and other DMI entries) are partially supported for MSC.Nastran 2005.
- Scalar points are not supported (**).
- PBCOMP is not supported (**).
- CCONEAX is not supported (**).
- CBARAO is not supported (**).

- Support of coordinate transformations on 3D contact nodes or MPCs is a new capability in MSC.Marc 2005. Whenever possible, we recommend that field 7 of all GRID entries be blank if 3D contact or MPCs or RBEi elements are included in the model.
- Spherical coordinate transformations in field 7 of the GRID entry are not supported (**).
- Output set definitions that contain grid or element numbers greater than the largest grid or element in the model will produce errors in MSC.Marc.
- Output set definitions may include the word BY as in output plot set definitions for use by MSC.Marc only. MD Nastran must be stopped using STOP=1 or one of the OUTR options since BY is a FATAL ERROR to MD Nastran.
- MD Nastran's CREEP entry must be changed to the new MATVP entry (**).
- For orthotropic materials using MATORT, all shear moduli must be entered.
- SPOINT, SLOAD and other scalar features are not supported (**).
- CELAS3 and CELAS4 are not supported (**).
- CLOAD is not supported (*).
- Fracture Mechanics is not supported (*, **).
- Aerodynamics is not supported (**).
- Bulk data entries with + or * in column 73 must have an actual continuation card for most entries. MD Nastran does not require this, but the internal MSC.Marc translator does. (*).
- RSSCON is not supported. (**).
- Slideline contact is not supported (BLSEG, BWIDTH, BFRIC, BCONP, and BOUTPUT, if entered will cause FATAL ERRORS for MD Nastran Implicit Nonlinear).
- p-elements are not supported.
- Offsets are available for CBAR, CBEAM, CQUAD4 and CTRIA3 in all types of structural analysis (linear or nonlinear). The offsets must be specified in the global coordinate system (displacement output coordinate system) unless PARAM,MAROFSET is 1 (2 or 3, see description in "**Parameter Descriptions**" on page 660). Offsets are available for CQUAD8 and CTRIA6

but only if all 8 or 6 grids are defined for these elements, respectively. If PARAM,MAROFSET,1 (2 or3) is included in the bulk data, the offsets will be incorporated using a new MSC.Marc feature that does not need extra grids or elements. This new feature is the default for 2005 r3.

- SPCs are allowed to change in both static and dynamic analysis between subcases.
- We recommend that displacements be applied using SPCD rather than SPC entry.
- MPCs must be the same for all load cases.
- MAT10 is not presently supported. (**)
- The following Solution Sequences are not presently supported: 7, 8, 10, 11, 107, 108, 110, 111, 114-116, 118, 144-146, 153, 159, 190, 200 and will cause Severe Warnings (FATAL ERRORS) in the internal translator.
- DOMAINSOLVER is not supported. If this executive control statement is entered, and MD Nastran Implicit Nonlinear is requested by the SOL 600, ID statement, the DOMAINSOLVER request will be commented out internally by MD Nastran.
- In some cases, continuation commands that do not have a + or * symbol in column 1 or + or * in column 73 of the parent card will fail. This restriction is being removed gradually.
- CGAP does not completely map to MSC.Marc's gap element. The user should change all MD Nastran gaps to contact before running SOL 600. If the gaps are not changed to contact, some options will fail to translate as indicated by warning messages. Certain simple gaps translate as expected and will produce nearly the same results as standard MD Nastran solution sequences, but the user is responsible for ensuring that any model with gaps gives the behavior he expects when using SOL 600.
- SOL 600 is not supported on Cray, NEC and another similar computers.
- SOL 600 is not supported on computer systems that MSC.Marc does not support. At present, the supported computer systems are IBM AIX 32-bit, 64-bit, Alpha, HPUX 64-bit, HPUX 64-bit (Itanium II), SGI IRIX 64-bit, Linux 32-bit and 64-bit, Windows 32-bit, SUN Solaris.
- Continuation lines for PLOAD4 are not supported and the job will exit with a Severe Warning.
- MD Nastran MATS1 Mohr-Coulomb is mapped to MSC.MARC's Linear Mohr Coulomb option.

- MD Nastran MATS1 Drucker-Prager is mapped to MSC.MARC's Parabolic Mohr Coulomb option.
- Pin flags for beams and bars are now available. The action taken depends on PARAM,MARCPINN.
- Composite output is not presently available using the OUTR options. It must be postprocessed directly using the t16 file - Patran is recommended.
- If layered output for Composite Structures is desired in the t16 file, the following parameters must be included in the Bulk Data:

  param,mroutlay,N
  param,marcslht,N

  where N is the maximum number of layers in any composite PCOMP description.
- The rotational acceleration portion of RFORCE (RACC) is not supported and if entered will generate a Severe Warning and MSC.Marc will not be spawned.
- PARAM,TSTATIC is not supported and will generate a "Severe Warning" if used in SOL 600.
- The following Case Control displacement / velocity / acceleration / spcforce / mpcforce options are not supported and will be ignored if entered:
  - SORT2, REAL, IMAG, PHASE, PSDF, ATOC, CRMS, RALL, RPRINT, RPUNCH, NORPRINT, CID, TM, RM
- Elements with mid-side nodes must have all mid-side nodes. For example CTETRA must either have 4 or 10 nodes.
- For PC systems, if SOL 600 is run from a command prompt (DOS box), if any old DOS programs are used prior to running SOL 600 the path where the job is being run is usually adjusted such that any names longer than 8 characters will be shortened (for example brake-squeal becomes BRAKE-~3). The continue options including brake squeal jobs will not work when this happens. Open a new command prompt and run SOL 600 before any old DOS programs are run in that window.

- The CID field on the RFORCE entry is not completely supported. If entered with a positive integer, the job will abort with a Severe Warning unless PARAM,MARCRCID is entered. PARAM,MARCRCID,1 may be used to ignore this field in which case R1,R2,R3 define the direction cosines of the rotation vector (see Marc Volume C, ROTATION A description) and the magnitude is given by A*sqrt(R1**2+R2**2+R3**2) [see RFORCE description for definitions of CID, A, R1, R2, R3 as well as Remark 16].

SOL 700,ID Executes DYTRAN-LSDYNA from Inside MD Nastran

Format:

SOL 700,ID PATH= OUTR= STOP= NP= NOERROR MEM=

Example:

SOL 700,129 PATH=3 NP=4

(700,129 request nonlinear transient dynamics, path=3 requests use of the dytran-lsdyna script called out in file sol700.pth, np=4 requests that 4 processors be used)

Summary:

SOL 700 is a new Executive Control statement like SOL that activates an explicit nonlinear transient analysis integration scheme using dytran-lsdyna. The calculations will not be performed directly within MD Nastran. Instead, SOL 700 will use a separate solver based on LS-DYNA, spawned from MD Nastran. This client-server approach is similar to SOL 600, using MSC.Marc.

SOL 700 statement will spawn dytran-lsdyna, which uses a Dytran text input interface to LS-DYNA. dytran-lsdyna is a 3D, explicit nonlinear analyses code with SMP and DMP (shared and distributed memory parallel processing domain decomposition) capabilities.

Inputs and outputs will be the same as or similar to the familiar MD Nastran inputs and outputs or, at, your option, certain LS-DYNA type inputs and outputs are available.

For ID=129 or NLTRAN, SOL 700 will generate a dytran-lsdyna input data file, jid.dytran.dat, where “jid” is the name of the MD Nastran input file without the extension). For example, if the MD Nastran input file is named abcd.dat, (or abcd.bdf) then “jid”=abcd).

Unless specified differently using the **STOP=3** option, dytran-lsdyna will be executed from MD Nastran on any computer system capable of doing so (which includes most UNIX systems and Windows systems). For dytran-lsdyna to run, it must be installed, properly licensed, and accessible from the directory where the MD Nastran input data resides, MSC_BASE must be provided in the environment.

Executive Control Parameters:

The required ID may be one of several valid solution sequence integers or names shown in **Table 3-4** for the SOL statement. Examples are 129 and NLTRAN.

The following solutions are available: 101, 106, 109, 129 (and their equivalent names).

All items on the SOL 700,ID after ID itself may be specified by environmental variables. This may be done any way environmental variables can be set. They may be set by the MD Nastran user at run time or by the system administrator when MD Nastran is installed. Any values specified on the SOL statement override those in the environment. Environmental variables are fully described in the *MSC.Nastran Installation and Operations Guide*. A keywords file is available to describe the format of each variable. The variable is normally set in the system-wide rc file, a user's rc file, a local rc file or in a script used to submit MD Nastran.

The following describes the various options for PATH. We suggest that PATH=3 for all computer systems.

PATH=1 (Windows Only)

If PATH=1 is specified, MD Nastran will determine the proper command to execute a serial dytran-lsdyna run. To aid MD Nastran in determining where dytran-lsdyna is located, the dynrun.pth file must be located in the same directory where the MD Nastran input file resides. The dynrun.pth file must contain one line providing the location (complete path) of the dytran-lsdyna run script. A typical example of the line in the file dynrun.pth follows.

Windows c:\sol700\

A string is appended to this path to form the complete command used to execute dytran-lsdyna.
"dytran-lsdyna jid=name.dytr.dat
O=name.dytr,d3hsp G=name.dytr.d3plot D=name.dytr.d3dump
F=name.dytr.d3thdt
A=name.dytr.runrsf B=name.dytr.d3drfl

For Windows, MD Nastran will spawn dytran-lsdyna using the following command assuming the MD Nastran input data is named enf2e.dat. (Although the example appears like it is on multiple lines, it is actually on a single line.)

c:\sol700/dytran-lsdyna i=enf2e.dytr.dat O=enf2e.dytr.d3hsp G=enf2e.dytr.d3plot D=enf2e.dytr.d3dump F=enf2e.dytr.d3thdt A=enf2e.dytr.runrsf B=enf2e.dytr.d3drfl

PATH=3 (All Systems)

If PATH=3 is specified, a script or batch file to execute dytran-lsdyna located in the same directory as the dytran-lsdyna executable will be executed. The name of the script or batch files is run_dytran (or run_dytran.bat). This directory and name of the

script is determined by the first line in a file named sol700.pth which must be in the same directory as the Nastran input file. Options are specified on subsequent lines of the sol700.pth file.

Available PATH=3 options for Windows PC systems are as follows:

exe=	The full path to the executable for dytran-lsdyna that is to be used. Optional -- If exe= is omitted, the directory where the script or batch file resides (first line of sol700.pth) will be used and dytran-lsdyna for UNIX/Linux and dytran-lsdyna.exe for windows will be appended. If exe= is used, it must be the second line in the sol700.pth file.
nproc	Number of processors. (Default is to used NP on the SOL 700 line. If NP and nproc are omitted, the default is 1). For parallel execution, the directory where the MD Nastran input file exists must be shared with read/write privileges. If wdir is used, it must also be shared (see below). The directory where the dytran-lsdyna executable resides must also be shared for parallel execution. In addition, all rules for MPICH must be followed properly, see your system administrator to be sure all computers are properly configured for parallel execution using MPICH. The version of MPICH to use is 1.2.5 as of the initial SOL 700 release. It can be obtained from ftp.mcs.anl.gov if necessary.
bat	Run in background or foreground (default).
memory	Amount of memory. Example: memory=20m.
steps	Number of steps (1 or 2; default is 2). Two steps means that lsdyna is executed twice: once to form the “structured input file” and again to analyze it. Although steps=1 is faster, there are some models that fail using the steps=2 option.
wdir	Working directory. For parallel execution, this directory must be shared with read/write privileges. Default is directory where MD Nastran input resides.
copy	Yes or no. Input and output files are copied from wdir to the input directory. Default is yes.
delete	Yes or no. LS-DYNA scratch files are deleted or not. Default is yes.

machine	Machines and number of processors to use in the form: machine1#2+marchin2#4 (use 2 processors on machine 1 and 4 processors on machine 2)
host	file name. Name of a hostfile containing the same information as "machine" The format of hostfile is as follows for the example for machine: machine1 2 machine1 4

A Windows example of the file sol700.pth for the PATH=3 case follows.

```
e:\sol700\dytran-lsdyna\run_Dytran
exe=f:\latest_dytran-lsdyna\dytran-lsdyna.exe
nproc=4
memory=20m
steps=2
wdir=f:\temp
delete=yes
machine=cp01#2+cp02#2
```

For the above example, MD Nastran will create the following command to spawn dytran-lsdyna assuming your input file is named abcd.dat. (Although the example appears like it is on multiple lines, it is actually on a single line.)

```
e:\sol700\dytran-lsdyna\run_dytran exe=f: \latest_dytran-lsdyna\dytran-lsdyna.exe jid=abcd.dytr nproc=4 memory=20m widr=f:\temp delete=yes marchine=pc01#2+pc02#2
```

Available PATH=3 options for UNIX/Linux systems follows.

exe	The full path to the executable for dytran-lsdyna that is to be used. (Optional -- If exe= is omitted, the directory where the script or batch file resides (first line of sol700.pth) will be used and dytran-lsdyna for UNIX/Linux and dytran-lsdyna.exe for windows will be appended.) If exe= is used, it must be the second line in the sol700.pth file.
nproc	Number of processors. (Default is to use NP on the SOL 700 line. If NP and nproc are omitted, the default is 1.)
bat	Yes or no. Run in background or forground (default). Leave out for steps=2
memory	Amount of memory; example: memory=20m (20 MB).

steps	Number of steps (1 or 2; default is 2). Two steps means that lsdyna is executed twice: once to form the “structured input file” and again to analyze it. Although steps=1 is faster, there are some models that fail using the steps=2 option.
wdir	Working directory. Default is directory where MD Nastran input resides.
copy	Yes or no. Input and output files are copied from wdir to the input directory. Default is yes.
delete	Yes or no. LS-DYNA scratch files are deleted or not. Default is yes.
cluster	Yes or no. If yes is specified, the job will be initiated on the machine that you are logged on to, but the analysis is performed on the cluster nodes that you specify in machinefile. If the default of off is used, the job will run on the local machine and the machines listed in the machine file depending on the number of processors specified. This option is not available for MSC.Nastran 2005 r2 SOL 700. Default is no.
mpipath	The MPI install directory if you wish to used a non-default MPI directory.
mpirun	The MPI run command you want to use. If entered, it overrides the default MPI run command on your machine as well as the command in mpipath.
hlist	Filename. Name of a hostfile containing the name(s) of machine(s) to use for each processor. The format of hostfile.txt for 4 processors is of the form: machine 1 machine 2 machine 3 machine 4

A UNIX/Linux example of the file sol700.pth for the PATH=3 case is as follows:

```
/users/joe/sol700/run_dytran
nproc=4
memory=20m
```

steps=2
wdir=/tmp/dyna
hlist=hostfile.txt

For the above example, MD Nastran will create the a command similar to the following to spawn dytran-lsdyna assuming your input file is named abcd.dat

/users/joe/sol700/run_dytran \
exe=/users/joe/sol700/dytran-lsdyna \
jid=abcd.dytr nproc=4 memory=20m steps=2 widr=/tmp/dyna

If PATH is not specified, a special version of PATH=3 will be used to locate dytran-lsdyan. This version will be located in a subdirectory named dyna/machine below the MD Nastran base directory (MSC_BASE). Not all PATH3 options are available using this default path option.

STOP

STOP is an optional item. STOP is used to prevent execution of dytran-lsdyna or prevent execution of MD Nastran after IFP if so desired. DO NOT ENTER any of the STOP options if any of the OUTR options are entered as the DMAP generated automatically by MD Nastran will put an EXIT in the proper place. The various options are as follows:

STOP=1

If STOP=1 MD Nastran will gracefully stop after IFP. This option is used to prevent MD Nastran from performing its own solution (normally used when the solution is performed by dytran-lsdyna with ID=129).

STOP=3

STOP=3 MD Nastran is stopped after IFP and dytran-lsdyna is not executed. This would be the normal STOP option if the user wants to examine a dytran-lsdyna input file, make some changes and then execute dytran-lsdyna manually.

The following dytran-lsdyna files are potentially affected by the COPYR option:

OUTR=OP2,XDB,F06,PCH (Not Currently Implemented)

OUTR=OP2,XDB,F06,PCH -- choose one or more or omit -- translate dytran-lsdyna jid.dytr.d3plot output to MD Nastran. This option requires the use of the MD Nastran Toolkit. A license is not needed for the Toolkit as it is imbedded in standard SOL 700 licensing. The conversion between LS-DYNA's d3plot and op2.xdb.f06,punch is

made using MSC.Patran's DRA / DAC together with a special version of the toolkit. The special toolkit executable is spawned from the original MD Nastran job after dytran-lsdyna completes and if any of the OUTR options are specified.

NP=the Number of Processors

NP=the number of processors (domains) for parallel processing. The default is one. In order to use more than one domain, MPI, Lam, POE, or whatever parallel program is needed must be properly installed on all computers involved and a hostfile designating which computers are to be used for each domain must have been setup prior to running the job. It is required that if NP>1, PATH=3 be used and a file named sol700.pth be located in the same directory as the MD Nastran input data. The sol700.pth file should contain all commands necessary to run dytran-lsdyna in parallel. This file must have execute permissions.

NOERROR is an Optional Item (Not currently implemented)

If NOERROR is specified, errors due to features that are available in MD Nastran but not available in dytran-lsdyna, and / or features not yet supported by the translator will be ignored. If NOERROR is entered and STOP=2 (or 3) is not specified, dytran-lsdyna will be executed even though the complete MD Nastran model may not have been completely translated. It is recommended that NOERROR only be used by experienced analysts and then only with extreme caution.

MEM= Control how much memory will be used for the dytran-lsdyna portion of the program. This value can also be specified in the sol700.pth file (memory option) if PATH=3 is entered. If the "default path" is used, it must be entered here. A typical entry would be MEM=20M for 20mb.

The following table summarizes the Case Control commands for SOL 700.Only those which are supported or will produce fatal errors are listed, others are ignored. Y = Yes (Supported), N = Not Supported.

Item	Case Control Commands Available in SOL 700
$	Y
ACCELERATION	Y
BCONTACT	Y
BEGIN BULK	Y (Other BEGIN forms are not allowed)
DISPLACEMENT	Y
DLOAD	Y

Item	Case Control Commands Available in SOL 700
ECHO	Y
ELFORCE see FORCE	Y
ENDTIME	Y (new)
FORCE & ELFORCE	Y (automatically produced in d3plot files no user control)
GROUNDCHECK	Y (MD Nastran f06 only)
IC	Y
INCLUDE	Y
LABEL	Y (MD Nastran f06 only)
LINE	Y (MD Nastran f06 only)
LOAD	Y (for dynamic pseudo-statics only)
LOADSET	Y
MAXLINES	Y (MD Nastran f06 only)
MPC	Y
NLPARM	Y (Psuedo static analysis only)
NLSTRESS	Y (Changed to STRESS)
PAGE	Y (In MD Nastran only)
PARAM	Y (Only applicable parms are used)
PRESSURE	Y
SET	Y
SET – OUTPUT(PLOT)	N
SKIP	Y (Required if multiple subcases are present)
SPC	Y
STRAIN	Y
STRESS	Y
SUBCASE	Y (See note)

Item	Case Control Commands Available in SOL 700
Note:	Only one subcase can be selected for a particular SOL 700 analysis. Many subcases may be entered in the input file, but the one to be used must be selected using the SKIP ON and SKIP OFF Case Control commands. If the SKIP ON / OFF commands are not found or are in the wrong place, the first subcase encountered will be used and the others ignored.
SUBTITLE	Y
TITLE	Y
TSTEP	Y (Same as TSTEPNL)
TSTEPNL	Y
VELOCITY	Y
WEIGHTCHECK	Y (In MD Nastran only)

The following summarizes the Bulk Data entries for SOL 700:

Item	Bulk Data Entries Available in SOL 700	Fatal Error
AXIC	N	Y
AXIF	N	Y
AXSLOT	N	Y
BAROR	Y	
BCBPDY	Y	
BCHANGE	N	Y
BSURF	Y	
BCBOX	Y	
BCPROP	Y	
BCMATL	Y	
BCONP	N	
BCTABLE	Y	
BLSEG	N	Y
CBAR	Y	
CBEAM	Y	

Item	Bulk Data Entries Available in SOL 700	Fatal Error
CBEND	N	Y
CBUSH	N	Y
CCONEAX	N	Y
CDAMP1D	Y (New)	
CDAMP2D	Y (New)	
CELAS1D	Y (New)	
CELAS2D	Y (New)	
CFLUID	N	Y
CGAP	N	Y
CHACAB	N	Y
CHEXA	Y (8 Nodes only)	
CONM2	Y	
CONROD	Y	
CORD1C	Y	
CORD1R	Y	
CORD1S	Y	
CORD2C	Y	
CORD2R	Y	
CORD2S	Y	
CORD3G	N	Y
CPENTA	Y (5 Nodes only)	
CQUAD4	Y	
CQUAD8	Y (4 Nodes only)	
CQUADR	Y	
CQUADX	N	Y
CREEP	N	Y
CROD	Y	
CSHEAR	N	Y

Item	Bulk Data Entries Available in SOL 700	Fatal Error
CTETRA	Y (4 Nodes only)	
CTRIA3	Y	
CTRIA6	Y (3 Nodes only)	
CTRIA3R	Y	
CTRIAX	N	Y
CTRIAX6	N	Y
CTUBE	Y	
CVISC	Y	
CWELD	N	Y
CSPOT	Y (New – LS-DYNA Weld)	
CFILLET	Y (New – LS-DYNA Weld)	
CBUTT	Y (New – LS-DYNA Weld)	
CCRSFIL	Y (New – LS-DYNA Weld)	
COMBWLD	Y (New – LS-DYNA Weld)	
DAMPGBL	Y (New for Dynamic Relaxation)	
DAREA	Y	
DEFORM	N	Y
DELAY	N	Y
DMI	N	Y
DMIAX	N	Y
DMIG	N	Y
DPHASE	N	Y
DTI	N	Y
ECHOOFF	Y	
ECHOON	Y	
ENDDATA	Y	
EOSPOL	Y (New – Equation of state)	
FORCE	Y	

Item	Bulk Data Entries Available in SOL 700	Fatal Error
FORCE1	N	Y
FORCE2	Y	
FORCEAX	N	Y
GENEL	N	Y
GRAV	Y	
GRDSET	Y	
GRID	Y	
INCLUDE	Y	
IPSTRAIN	N	Y
ISTRESS	N	Y
LOAD	Y	
LSEQ	Y	
MAT1	Y	
MAT2	Y	
MAT3	Y	
MAT8	Y	
MATDxxx	Y (New LS-DYNA materials)	
MATD20M	Y (New Rigid Material Merge)	
MATEP	N	Y
MATHE	N	Y
MATHED	N	Y
MATF	N	Y
MATHP	Y	
MATS1	Y	
MATVE	N	Y
MATORT	N	Y
MATVORT	N	Y
MATVP	N	Y

Item	Bulk Data Entries Available in SOL 700	Fatal Error
MATG	N	Y
MFLUID	N	Y
MOMAX	N	Y
MPC	Y	
MPCAX	N	Y
NLPARM	Y (For pseudo statics)	
NLRGAP	N	Y
NOLINi	N	Y
NTHICK	N	Y
PANEL	N	Y
PBAR	Y	
PBARL	Y	Y
PBCOMP	N	Y
PBEAM	Y	
PBEAML	N	Y
PBEND	N	Y
PBUSH	N	Y
PCOMP	Y	
PDAMP	Y	
PDAMP5	N	Y
PELAS	Y	
PELAST	N	Y
PGAP	N	Y
PHBDY	N	Y
PINTC	N	Y
PINTS	N	Y
PLOAD	Y	
PLOAD1	N	Y

Item	Bulk Data Entries Available in SOL 700	Fatal Error
PLOAD2	Y	
PLOAD4	Y (Continuation supported)	
PLOADX1	N	Y
PLPLANE	Y	
PLSOLID	Y	
PMASS	N	Y
PRESPT	N	Y
PROD	Y	
PSHEAR	N	
PSHELL	Y	
PSOLID	Y	
PTUBE	Y	
PVISC	Y	
RBAR	Y	
RBE1	N	Y
RBE2	Y	
RBE3	Y (Changed to RBE3D)	
RESTART	Y	Y
RFORCE	(CID, METHOD, continuation line not supported)	
RLOADi	N	Y
RROD	N	Y
RSPLINE	N	Y
RTRPLT	N	Y
SLOAD	N	Y
SPC	Y	
SPC1	Y	
SPCADD	Y	

Item	Bulk Data Entries Available in SOL 700	Fatal Error
SPCAX	N	Y
SPCD	Y	
SUPAX	N	Y
TABLED1	Y	
TABLED2	Y	
TABLED3	Y	
TABLES1	Y	
TEMP	N	Y
TEMPD	N	Y
TIC	Y	
TICD	Y (New with increment options)	
TIC3	Y (New MSC.Dytran type entry)	
TLOAD1	Y	
TLOAD2	Y	
TSTEP	Y (Changed to TSTEPNL)	
TSTEPNL	Y	
WALL	Y (New rigid wall entry)	

Notes and Restrictions:

The following Case Control stress / strain / load options are not supported and will be ignored if entered:

- SORT2, REAL, IMAG, PHASE, MAXS, SHEAR, STRCUR, FIBER, CENTER, CORNER, BILIN, SGAGE, CUBIC, PSDF, ATOC, CRMS, RALL, RPRINT, RPUNCH, NORPRINT

The following Case Control displacement / velocity / acceleration / spcforce / mpcforce options are not supported and will be ignored if entered:

- SORT2, REAL, IMAG, PHASE, PSDF, ATOC, CRMS, RALL, RPRINT, RPUNCH, NORPRINT, CID, TM, RM

On Linus systems, if multiple jobs are to be run by a particular user simultaneously, a manual lamboot from the command line (or a script) must be done prior to running the first job. In addition, PATH=3 must be specified on the SOL 700,ID entry and a line lamboot=no added. A user can determine the proper location of lam by running a smalll job, examining the f06 file and searching for the string “execut”. The command shown in the example below should then be executed.

Example

A typical f06 file shows the following:

```
dytran-lsdyna will be executed using the command
/usr/msc/programs/nastran/dytran-lsdyan/run_dytran
```


For this case the following commands should be placed in a script and the script executed to establish the lamboot:

```
#!/bin/csh
seetenv LAMHOME /usr/msc/programs/nastran/dytran-lsdyna/mpi/bin
$LAMHOME/lamboot
```

To check if lam is running, the user should execute the following command:

```
ps -ef     grep lam
```

As long as the lam processes is not “killed” it will remain running unless the machine is rebooted. If the user wishes to terminate the process he can either kill it from the command line or enter the following command:

```
/usr/msc/programs/nstran/dytran-lsdyna/mpi/bin/lamhalt
```

TIME Sets maximum CPU and I/O time

Sets the maximum CPU and I/O time.

Format:

TIME[=]t1[,t2]

Describer	Meaning
t1	Maximum allowable execution time in CPU minutes. (Real or Integer>0; Default=1.89E9 seconds)
t2	Maximum allowable I/O limit in minutes. (Real or Integer>0; Default is infinity, which is machine dependent.)

Remarks:

1. The TIME statement is optional.
2. If t2 is specified then t1 must be specified.

Examples:

1. The following example designates a runtime of 8 hours:

   TIME 480

2. The following example designates 90 seconds:

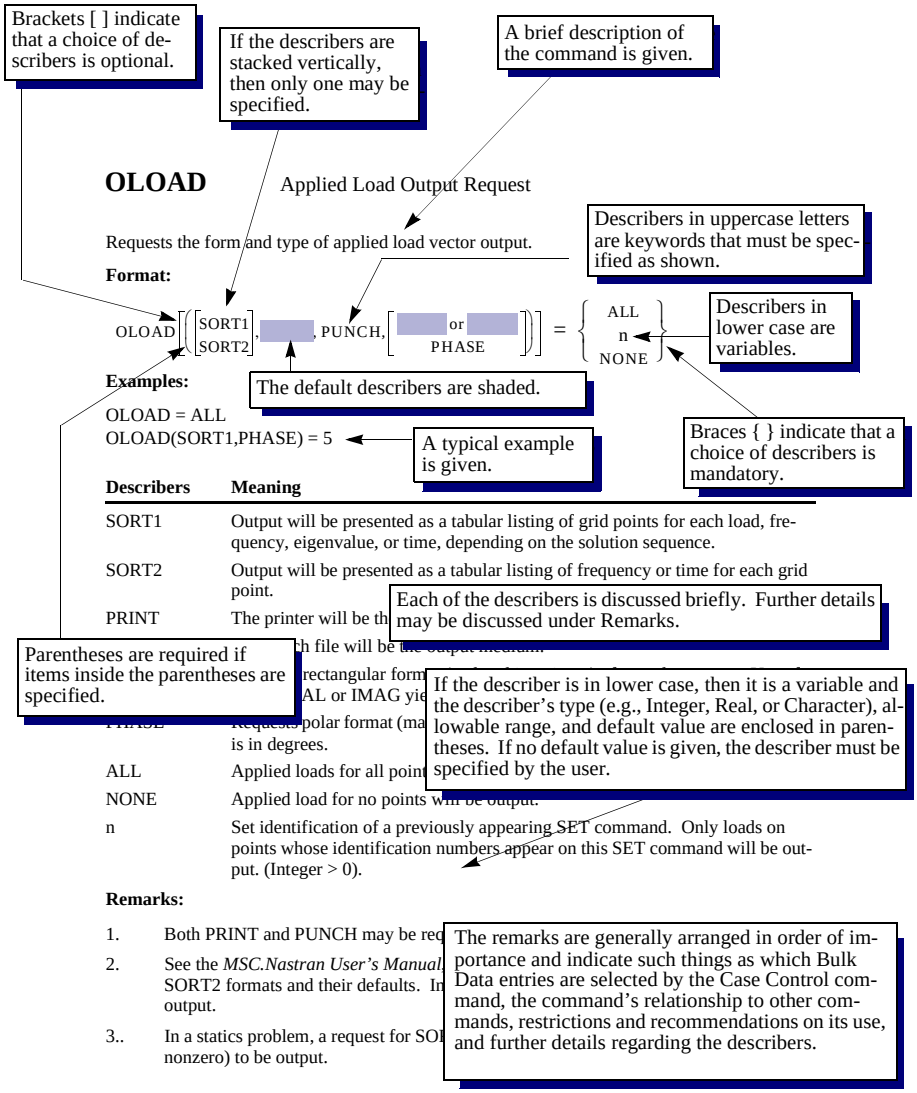
   TIME 1.5

CHAPTER

4 Case Control Commands

- Key to Descriptions
- Case Control Command Descriptions
- Case Control Applicability Tables
- OUTPUT(PLOT) Commands
- X-Y PLOT Commands

4.1 Key to Descriptions



4A CASE

The Case Control Section

The Case Control Section has several basic functions; specifically, it:

- Selects loads and constraints.
- Requests printing, plotting, and/or punching of input and out data (plotter commands are described in "**Plotting**" on page 527 of the *MSC.Nastran Reference Guide*).
- Defines the subcase structure for the analysis.

Table 4-1 through **Table 4-6** at the end of this section indicate the applicability of each command in all solution sequences.

4.2 Case Control Command Descriptions

Case Control commands may be abbreviated down to the first four characters as long as the abbreviation is unique relative to all other commands. If not, the full name of the command (or at least the first eight characters if the name exceeds eight characters) must be specified in order to avoid errors. Each command is described as follows:

Description

A brief sentence about the function of the command is stated.

Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the command line is longer than 72 columns, then it may be continued to the next line with a comma. For example:

```
SET 1 = 5, 6, 7, 8, 9,
        10 THRU 55
```

Example

A typical example is given.

Describers and Meaning

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

Remarks

The Remarks are generally arranged in order of importance and indicate such things as which Bulk Data entries are selected by the Case Control command; the command's relationship to other commands, restrictions and recommendations on its use; and further descriptions of the describers.

Case Control Command Summary

This section contains a summary of all Case Control commands under the following headings:

Subcase Definition

1. Output Request Delimiters

ENDCARDS	Reactivates processing of Bulk Data entries (MSGMESH).
OUTPUT	Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS.
OUTPUT(PLOT)	Beginning of structure plotter output request.
OUTPUT(POST) or SETS DEFINITION	Beginning of grid point stress output requests and/or p-element data recovery set definition.
OUTPUT(XYOUT) or OUTPUT(XYPLOT)	Beginning of curve plotter output request.
OUTPUT(CARDS)	Suppresses processing of Bulk Data entries (MSGMESH).

2. Subcase Delimiters

REPCASE	Delimits and identifies a repeated output request subcase.
SUBCASE	Delimits and identifies a subcase.
SUBCOM	Delimits and identifies a combination subcase.
SYM	Delimits and identifies a symmetry subcase.
SYMCOM	Delimits and identifies a symmetry combination subcase.

3. Subcase Control

MASTER	Allows the redefinition of a MASTER subcase.
MODES	Repeats a subcase.
SUBSEQ	Gives the coefficients for forming a linear combination of the previous subcases.
SYMSEQ	Gives the coefficients for combining the symmetry subcases into the total structure.

Data Selection

1. Static Load Selection

DEFORM	Selects the element deformation set.
CLOAD	Requests a CLOAD Bulk Data entry that defines a list of superelement loads and their scale factors in nonlinear static analysis only.
LOAD	Selects an external static loading set.

2. Dynamic Load Selection

DLOAD	Selects a dynamic load or an acoustic source to be applied in a transient or frequency response problem.
LOADSET	Selects a sequence of static load sets to be applied to the structural model. The load sets may be referenced by dynamic load commands.
NONLINEAR	Selects nonlinear dynamic load set for transient problems.

3. Constraint Selection

AXISYMMETRIC	Selects boundary conditions for an axisymmetric shell problem or specifies the existence of fluid harmonics for hydroelastic problems.
AUTOSPC	Requests that stiffness singularities and near singularities be automatically constrained via single or multipoint constraints.
BC	Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.
DSYM	Provides for either one or two planes of overall symmetry in DIH-type cyclic symmetry problems.
MPC	Selects a multipoint constraint set.
SPC	Selects a single-point constraint set to be applied.
STATSUB	Selects the static solution to use in forming the differential stiffness for buckling analysis, normal modes, complex eigenvalue, frequency response and transient response analysis.
SUPORT1	Selects the fictitious support set (SUPORT1 entries only) to be applied to the model.

4. Thermal Field Selection

TEMPERATURE	Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.
TEMPERATURE (INITIAL)	Selects initial temperature distribution for temperature dependent material properties and heat transfer problems.
TEMPERATURE (LOAD)	Selects temperature set for static thermal load.
TEMPERATURE (MATERIAL)	Selects temperature set for temperature-dependent material properties.
TSTRU	Defines a temperature set ID for a structures run based on a heat transfer subcase.

5. Static Solution Conditions

SMETHOD	Selects iterative solver parameters.

6. Dynamic Solution Conditions

CMETHOD	Selects complex eigenvalue extraction parameters.
FREQUENCY	Selects the set of forcing frequencies to be solved in frequency response problems.
IC	Selects the initial conditions for direct transient analysis (SOLs 109, 129, and 159).
METHOD	Selects the real eigenvalue extraction parameters.
MODESELECT	Requests a set of computed mode shapes for inclusion in dynamic analysis.
NSM	Selects non-structural mass set.
RANDOM	Selects the RANDPS and RANDT1 Bulk Data entries to be used in random analysis.
RESVEC	Specifies options for and the calculation of residual vectors.
RGYRO	Activates gyroscopic effects and selects RGRYO or UNBALNC entries.
SDAMPING	Requests modal damping as a function of natural frequency in modal solutions or viscoelastic materials as a function of frequency in direct frequency response analysis.

SMETHOD	Selects iterative solver override options in frequency response analysis.
TSTEP	Selects integration and output time steps for linear or nonlinear transient analysis.

7. Direct Input Matrix Selection

A2GG	Selects direct input fluid-structure coupling matrix.
B2GG	Selects direct input damping matrices.
B2PP	Selects direct input damping matrices.
K2GG	Selects direct input stiffness matrices.
K2PP	Selects direct input stiffness matrices, which are not included in normal modes.
K42GG	Selects direct input structural damping matrices.
M2GG	Selects direct input mass matrices.
M2PP	Selects direct input mass matrices, which are not included in normal modes.
MFLUID	Selects the MFLUID Bulk Data entries to be used to specify the fluid-structure interface.
P2G	Selects direct input load matrices.
TFL	Selects the transfer function set(s) to be added to the direct input matrices.

8. Nonlinear Analysis

BCONTACT	Requests line contact output
ENDTIME	Specifies final analysis time for SOL 700.
NLPARM	Selects the parameters used for nonlinear static analysis.
SMETHOD	Selects iterative solver override parameters in nonlinear static analysis.
STEP	Defines and identifies a nonlinear analysis for SOL 400.
TSTEPNL	Transient time step set Selection for nonlinear analysis

9. Aerodynamic Analysis

AECONFIG	Assigns the aerodynamic configuration parameter used to locate the associated datablocks on the aerodynamic and aeroelastic databases.
AESYMXY	Aerodynamic XY plane of symmetry flag.
AESYMXZ	Aerodynamic XZ plane of symmetry flag.
AEUXREF	Define the Reference Aerodynamic Extra Point (Controller) Vector)
CSSCHD	Aerodynamic Control Surface Schedule.
DIVERG	Selects the divergence parameters in a static aeroelastic divergence problem.
FMETHOD	Selects the parameters to be used by the aerodynamic flutter analysis.
GUST	Selects the gust field in an aerodynamic response problem.
TRIM	Selects trim variable constraints in static aeroelastic response.

10. Design Optimization and Sensitivity (SOL 200)

ANALYSIS	Specifies the type of analysis being performed for the current subcase.
AUXCASE	Delimits Case Control commands for an auxiliary model in SOL 200.
AUXMODEL	References an auxiliary model for generation of boundary shapes in shape optimization.
DSAPRT	Specifies design sensitivity output parameters.
DESGLB	Selects the design constraints to be applied at the global level in a design optimization task.
DESOBJ	Selects the DRESP1 or DRESP2 entry to be used as the design objective.
DESSUB	Selects the design constraints to be used in a design optimization task for the current subcase.
DESVAR	Selects a set of DESVAR entries for the design set to be used.
DRSPAN	Selects a set of DRESP1 entries for the current subcase that are to be used in a DRESP2 or DRESP3 response that spans subcase.
MODTRAK	Selects mode tracking options in design optimization (SOL 200).

11. Original Design Sensitivity Analysis (DSA) (SOLs 101, 103, and 105)

SENSITY	Requests the generation of the combined constraint / design sensitivity matrix for original design sensitivity analysis (DSA).

12. p-element and Adaptivity Analysis

ADACT	Specifies whether or not the subcase is to participate in the adaptivity process.
ADAPT	Specifies adaptivity control parameters.
DATAREC	Requests form and type of output for p-version elements.
OUTRCV	Selects the output options for p-elements defined on an OUTRCV Bulk Data entry.
SET	Defines a set of element identification numbers only for the SURFACE and VOLUME commands (grid point stress) or the OUTRCV Bulk Data entry (p-element data recovery). This form of the SET command must and can only be specified after the SETS DEFINITION or OUTPUT(POST) command delimiter.
SETS DEFINITION	Delimits the various type of commands under grid point stress and / or p-version element set definitions. This command is synonymous with OUTPUT(POST).
VUGRID	Requests output of view grid and view element entries used in p-version element data recovery.

13. Fluid-Structure Analysis

A2GG	Selects a direct input fluid-structure coupling matrix.
FLSFSEL	Fluid-structure parameter collector for frequency and fluid superelement selection.
FLSPOUT	Fluid-structure parameter collector for mode participation.
FLSTCNT	Fluid-structure parameter collector for symmetry and force request.

14. MD Nastran / MSC.ADAMS Interface

ADAMSMNF	Control for MD Nastran / MSC.ADAMS interface Modal Neutral File (MNF). For MSC.Nastran 2004, to insure compatibility with the MSC.Adams msc2mnf took kit, if the ADAMSMNF Case Control command has the keyword ADMOUT=YES, the MD Nastran SYSTEM word OP2NEW is automatically set to OP2NEW=0. This means that any output2 files generated will have a pre-MSC.Nastran 2004 format.

Output Selection

1. Output Control

ECHO	Controls echo (i.e., printout) of the Bulk Data.
ECHOOFF	Suppresses echo of Case Control.
ECHOON	Reactivates echo of Case Control.
LABEL	Defines a character string that will appear on the third heading line of each page of printer output.
LINE	Defines the maximum number of output lines per printed page.
MAXLINES	Sets the maximum number of output lines.
PAGE	Causes a page eject in the echo of the Case Control Section.
PLOTID	Defines a character string that will appear on the first frame of any plotter output.
POST	Activates post-processor operations for selected output data.
SKIP	Activates or deactivates the execution of subsequent commands in the Case Control (including plot commands).
SKIPON	Defines commands in Case Control that are not to be processed.
SKIPOFF	Resumes processing of commands in Case Control.
SUBTITLE	Defines a subtitle that will appear on the second heading line of each page of printer output.
TITLE	Defines a character string that will appear on the first heading line of each page of MD Nastran printer output.

2. Set Definition

MAXMIN	Specifies options for max/min surveys of certain output data associated with grid points.
OFREQUENCY	Selects a set of frequencies for output requests.
OMODES	Selects a set of modes for output requests.
OTIME	Selects a set of times for output requests.
PARTN	Specifies a list of grid point identification numbers that will be partitioned with the DMAP module MATMOD (Option 17). In SOLs 111 and 200, the PARTN command specifies the points at which modal participation factors are to be computed.
SET	Defines a set of element or grid point numbers to be plotted.
SET2	Lists paired set numbers for design sensitivity. These sets refer to constraint and design variable set identification numbers.
SURFACE	Defines a surface for the calculation of grid point stresses, strains, or mesh stress discontinuities.
VOLUME	Defines a volume for the calculation of grid point stresses, strains, or mesh stress discontinuities.

3. Physical Set Output Requests

ACCELERATION	Requests the form and type of acceleration vector output.
BOUTPUT	Requests line or 3D (SOL 600) contact output.
CMSENERGY	Requests the output of component (superelement) modal strain, kinetic, and damping energies.
DISPLACEMENT, VECTOR, or PRESSURE	Requests the form and type of displacement or pressure vector output. Note: PRESSURE and VECTOR are equivalent commands
EDE	Requests the output of the energy loss per cycle in selected elements.
EKE	Requests the output of the kinetic energy in selected elements.
ELSDCON	Requests mesh stress discontinuities based on element stresses (see STRESS).
ENTHALPY	Requests form of enthalpy vector output in transient heat transfer analysis (SOL 159).

ESE	Requests the output of the strain energy in selected elements.
EQUILIBRIUM	Requests equilibrium force balance output.
FLUX	Requests the form and type of gradient and flux output in heat transfer analysis.
FORCE or ELFORCE	Requests the form and type of element force output or particle velocity output in coupled fluid-structural analysis. Note: ELFORCE is an equivalent command.
GPFORCE	Requests grid point force balance at selected grid points.
GPKE	Requests the output of the kinetic energy at selected grid points in normal modes analysis only.
GPSDCON	Requests mesh stress discontinuities based on grid point stresses (see GPSTRESS).
GPSTRAIN	Requests grid points strains for printing only.
GPSTRESS	Requests grid point stresses for printing only.
HDOT	Requests form of rate of change of enthalpy vector output in transient heat transfer analysis (SOL 159).
MCFRACTION	Requests modal contribution fractions output.
MODALKE	Requests modal kinetic energy output.
MODALSE	Requests modal strain energy output.
MEFFMASS	Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis.
MPCFORCE	Requests the form and type of multipoint force of constraint vector output.
NLSTRESS	Requests the form and type of nonlinear element stress output in SOL 106.
NOUTPUT	Requests physical output in cyclic symmetry problems.
OLOAD	Requests the form and type of applied load vector output.
RCROSS	Requests computation and output of cross-power spectral density and cross-correlation functions in random analysis.
SPCFORCES	Requests the form and type of single-point force of constraint vector output.

STRAIN	Requests the form and type of strain output.
STRESS or ELSTRESS	Requests the form and type of element stress output. Note: ELSTRESS is an equivalent command.
STRFIELD	Requests the computation of grid point stresses for graphical postprocessing and mesh stress discontinuities.
SVECTOR	Requests the form and type of solution set eigenvector output.
THERMAL	Requests the form and type of temperature output.
VELOCITY	Requests the form and type of velocity vector output.

4. Solution Set Output Requests

AEROF	Requests the aerodynamic loads on aerodynamic control points.
APRESSURE	Requests the aerodynamic pressures in static aeroelastic response.
HARMONICS	Controls the number of harmonics output in axisymmetric shell or axisymmetric fluid problems; controls the number of harmonics to be used for analysis in cyclic symmetry problems.
HOUTPUT	Requests harmonic output in cyclic symmetry problems.
MPRES	Requests the pressure for selected wetted surface elements when virtual mass (MFLUID) is used.
NLLOAD	Requests the form and type of nonlinear load output for transient problems.
SACCELERATION	Requests the form and type of solution set acceleration output.
SDISPLACEMENT	Requests the form and type of solution set displacement output.
SVELOCITY	Requests the form and type of solution set velocity output.

5. Model Checkout

ELSUM	Requests a summary of element properties for output
GROUNDCHECK	Perform grounding check analysis on stiffness matrix to expose unintentional constraints by moving the model rigidly.
WEIGHTCHECK	At each stage of the mass matrix reduction, compute rigid body mass and compare with the rigid body mass t the g-set.

Superelement Control

EXTSEOUT	Specifies the data to be saved for an external superelement as well as the medium on which the data is to be saved.
SEALL	Specifies the superelement identification numbers of Phase 1 processing in which all matrices and loads are generated and assembled. Controls execution of the solution sequence.
SEDR	Specifies the superelement identification numbers for which data recovery will be performed.
SEDV	Specifies the superelement identification numbers for which the design variables will be processed.
SEEXCLUDE	Specifies the superelement identification numbers for which all matrices and loads will not be assembled into the downstream superelement.
SEFINAL	Specifies the superelement identification number of the final superelement to be assembled.
SEKREDUCE	Specifies the superelement identification numbers for which stiffness matrices are assembled and reduced.
SELGENERATE	Specifies the superelement identification numbers for which static loads will be generated.
SELREDUCE	Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.
SEMGENERATE	Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.
SEMREDUCE	Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced. In buckling analysis, the differential stiffness matrices are assembled and reduced.
SERESP	Specifies the superelement identification numbers for which the design sensitivity matrices will be generated.
SUPER	Assigns a subcase(s) to a superelement or set of superelements.

Miscellaneous

$	Used to insert comments into the input file. Comment statements may appear anywhere within the input file.
BEGIN BULK	Designates the end of the Case Control Section and/or the beginning of a Bulk Data Section.
INCLUDE	Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.
NSM	Request nonstructural mass distribution selection.
OUTPUT	Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS.
PARAM	Specifies values for parameters.
POST	Controls selection of data to be outut for post-processing.
RIGID	Selects type of rigid element formulations to be used.

$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Format:

$ followed by any characters out to column 80.

Example:

```
$ TEST FIXTURE-THIRD MODE
```

Remarks:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

A2GG Selects a Direct Input Fluid-Structure Coupling Matrix

Selects

Format:

A2GG = name

Example:

A2GG = AGG0

Describer	Meaning
name	Name of a fluid-structure coupling matrix that is input on the DMIG Bulk Data entry.

Remarks:

1. DMIG entries will not be used unless selected by the A2GG command
2. This entry must be above subcase level or in the first subcase.
3. If the A2GG command selects a DMIG entry, then MD Nastran will add the selected fluid-structure coupling matrix to the computed coupling matrix. To replace the computed coupling matrix with the selected A2GG matrix, set PARAM,ASCOUP,NO. The user may still define panels with the panel selection procedure.
4. When filling out the DMIG entries: IFO = 1, NCOL = g-size, GJ-column index corresponds to fluid points, CJ = 0, Gi-row index corresponds to structural points, Ci-corresponds to DOF, Ai-the area values.
5. A2GG is supported in dynamic solutions with fluid-structure coupling.
6. Only one A2GG command should be used. It must appear above any subcase structure.

ACCELERATION Acceleration Output Request

Requests form and type of acceleration vector output.

Format:

$$\text{ACCELERATION}\left[\left(\left[\begin{matrix}\text{SORT1}\\\text{SORT2}\end{matrix}\right],\left[\begin{matrix}\text{PRINT, PUNCH}\\\text{PLOT}\end{matrix}\right],\left[\begin{matrix}\text{REAL or IMAG}\\\text{PHASE}\end{matrix}\right],\left[\begin{matrix}\text{PSDF,ATOC,CRMS}\\\text{or RALL}\end{matrix}\right],\left[\begin{matrix}\text{RPRINT}\\\text{NORPRINT}\end{matrix},\text{RPUNCH}\right],[\text{CID}]\right)\right]=\left\{\begin{matrix}\text{ALL}\\\text{n}\\\text{NONE}\end{matrix}\right\}$$

Examples:

ACCELERATION=5
ACCELERTION(SORT2, PHASE)=ALL
ACCELERTION(SORT1, PRINT, PUNCH, PHASE)=17
ACCELERATION(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
ACCELERATION(PRINT, RALL, NORPRINT)=ALL

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Computes, but does not print or punch, acceleration output.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.

Describer	Meaning
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RALL	Requests all of PSDF, ATOC and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output file, F06 file.
ALL	Accelerations at all points will be output.
n	Set identification of a previously appearing SET command. Only accelerations of points with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	Accelerations at no points will be output.

Remarks:

1. Both PRINT and PUNCH may be requested.
2. Acceleration output is only available for transient and frequency response problems. Acceleration is only available for transient and frequency response problems and when response spectra is requested in eigenvalue analysis.
3. See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2.
4. ACCELERATION=NONE allows overriding an overall output request.

5. The PLOT option is used when curve plots are desired in the magnitude/phase representation and no printer request is present for magnitude/phase representation.
6. Acceleration results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
7. The option of PSDF, ATOC, CRMS and RALL, or any combination of them, can be selected for random analysis. The results can be either printed in .f06 file or punched in the punch file, or output in both files.
8. Note that the CID keyword affects only grid point related output, such as DISPlacement, VELOcity, ACCEleration, OLOAD, SPCForce and MPCForce. In addition, CID keyword needs to appear only once in a grid related output request anywhere in the Case Control Section to turn on the printing algorithm.

ACPOWER Acoustic Power Output Request

Requests output of the power radiated from the wetted surface.

Format:

$$\text{ACPOWER}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right], \left[\begin{matrix}\text{PRINT,PUNCH}\\ \text{PLOT}\end{matrix}\right], [\text{CSV} = \text{unit}]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Describer	Meaning
SORT1	Output will be presented as tabular listing of panels for each excitation frequency
SORT2	Output will be presented as tabular listing of excitation frequencies for each panel (Default)
PRINT	The printer will be the output medium (Default).
PUNCH	The punch file will be the output medium.
PLOT	Results are generated but not output.
CSV	Results will be written to a .csv file.
unit	Unit of the .csv file as used on the ASSIGN statement
ALL	Radiated power will be processed for the wetted surface and all panels.
n	Set identification of a previously defined set of panels. Radiated power will be processed for the wetted surface and all panels in the referenced set.
NONE	Radiated power will not be processed.

ADACT Adaptivity Subcase Selection

Specifies whether or not the subcase is to participate in the adaptivity process.

Format:

$$\text{ADACT} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

Examples:

ADACT=NONE
ADACT=10

Describer	Meaning
ALL	All subcases will participate in the error analysis.
n	The first n modes in a normal modes analysis will participate in the error analysis. (Integer > 0)
NONE	The current subcase will not participate in the error analysis.

Remarks:

1. ADACT is processed only when an adaptive analysis is requested.
2. In a static analysis, ADACT=n is equivalent to ADACT=ALL and ALL means that the results of all subcases will be included in the error analysis. When ADACT=NONE in any subcase, the results of that subcase are excluded from the error analysis and adaptivity.
3. In an eigenvalue analysis, ALL means that the results of all the modes will be included in the error analysis.
4. Only one ADACT command may be specified per SUBCASE.
5. An ADAPT Case Control command must be present in order to have an adaptive analysis.

ADAMSMNF* Control for MD Nastran / MSC.ADAMS Interface

Control for MD Nastran / MSC.ADAMS Interface Modal Neutral File (MNF)

Format:

$$\text{ADAMSMNF}\left[\text{FLEXBODY} = \left\{\begin{matrix}\underline{\text{NO}}\\ \text{YES}\end{matrix}\right\}\right], \left[\text{FLEXONLY} = \left\{\begin{matrix}\underline{\text{YES}}\\ \text{NO}\end{matrix}\right\}\right],$$

$$\left[\text{ADMCHECK} = \left\{\begin{matrix}\underline{\text{NO}}\\ \text{YES}\end{matrix}\right\}\right], \left[\text{ADMOUT} = \left\{\begin{matrix}\underline{\text{NO}}\\ \text{YES}\end{matrix}\right\}\right],$$

$$\left[\text{OUTGSTRS} = \left\{\begin{matrix}\underline{\text{YES}}\\ \text{NO}\end{matrix}\right\}\right], \left[\text{OUTGSTRN} = \left\{\begin{matrix}\underline{\text{YES}}\\ \text{NO}\end{matrix}\right\}\right],$$

$$\left[\text{OUTSTRS} = \left\{\begin{matrix}\underline{\text{NO}}\\ \text{YES}\end{matrix}\right\}\right], \left[\text{OUTSTRN} = \left\{\begin{matrix}\underline{\text{NO}}\\ \text{YES}\end{matrix}\right\}\right]$$

$$\left[\text{V1ORTHO} = \left\{\begin{matrix}\underline{-1.0}\\ \text{value1}\end{matrix}\right\}\right], \left[\text{V2ORTHO} = \left\{\begin{matrix}\underline{1.0\text{e}8}\\ \text{value2}\end{matrix}\right\}\right],$$

$$\left[\text{MINVAR} = \left\{\begin{matrix}\underline{\text{PARTIAL}}\\ \text{CONSTANT}\\ \text{FULL}\\ \text{NONE}\\ \text{RIGID}\end{matrix}\right\}\right], \left[\text{PSETID} = \left\{\begin{matrix}\underline{\text{NONE}}\\ \text{setid}_{\text{plotel}}\\ \text{ALL}\end{matrix}\right\}\right]$$

Example(s):

ADAMSMNF FLEXBODY = *YES*

*MD Nastran / MSC.ADAMS Modal Stress Recovery (MSR) interface is also available. See Remark **19.**

Describer	Meaning
FLEXBODY	Requests that the MD Nastran / MSC.ADAMS interface be executed.
NO	Execute standard MD Nastran.
YES	Execute MD Nastran / MSC.ADAMS interface.
FLEXONLY	Requests standard DMAP solution and data recovery following MD Nastran / MSC.Adams interface execution.
YES	Execute only the MD Nastran / MSC.ADAMS interface.
NO	Execute MD Nastran / MSC.ADAMS interface and standard DMAP solution and data recovery.
ADMCHECK	Requests MD Nastran / MSC.ADAMS diagnostic output.
YES	Print diagnostic output.
NO	Suppress diagnostic output.
ADMOUT	Requests that the MD Nastran / MSC.ADAMS interface outputs MD Nastran OP2 files.
YES	OP2 files are generated.
NO	Requests that OP2 files are not generated.
OUTGSTRS	Controls grid point stress output to OP2 file or MNF or both.
YES	Grid point stress is output to OP2 file or MNF or both.
NO	Grid point stress is not output to OP2 file or MNF.
OUTGSTRN	Controls grid point strain output to OP2 file or MNF or both.
YES	Grid point strain is output to OP2 file or MNF or both.
NO	Grid point strain is not output to OP2 file or MNF.
OUTSTRS	Controls element stress output to OP2 file.
YES	Element stress is output to OP2 file.
NO	Element stress is not output to OP2 file.
OUTSTRN	Controls element strain output to OP2 file.
YES	Element strain is output to OP2 file.
NO	Element strain is not output to OP2 file.
V1ORTHO	Lower frequency bound of the Craig-Bampton modes in cycles / unit time.

Describer	Meaning
value1	Value of lower bound.
V2ORTHO	Higher frequency bound of the Craig-Bampton modes in cycles / unit time.
value2	Value of higher bound.
MINVAR	Requests the type of mass invariants to be computed.
FULL	All nine mass invariants will be calculated.
CONSTANT	Only mass invariants 1, 2, 6, and 7 will be calculated.
PARTIAL	All mass invariants except 5 and 9 will be calculated.
NONE	No mass invariants are computed.
RIGID	No modal information is output to the MNF. Only units, grid point coordinate, element connectivity, interface nodes, and invariants 1, 2, and 7 data are shared in the MNF.
PSETID	Selects a set of elements (including PLOTEL) whose grids are retained in the MNF, and whose connectivity defines face geometry for ADAMS display.
$\text{setid}_{\text{plotel}}$	Specified in the OUTPUT(PLOT) Section of MD Nastran.
ALL	Select all the sets defined in the OUTPUT(PLOT) Section of MD Nastran.

Remarks:

1. This entry represents a collection of PARAM,name,value entries. A license is required for the MD Nastran / MSC.ADAMS interface.
2. ADAMSMNF FLEXBODY = YES is required to execute the MD Nastran / MSC.ADAMS interface, all other ADAMSMNF items are optional. The ADAMSMNF FLEXBODY = YES must occur above subcase level.
3. Mass invariants - The following defines the nine mass invariants:

$$\underset{1 \times 1}{{}^{1}I} = \sum_{p=1}^{N} m_p \qquad \text{Eq. 4-1}$$

$$ {}^{2}I_{3\times 1} = \sum_{p=1}^{N} m_p \boldsymbol{s}_p \qquad \text{Eq. 4-2} $$

$$ {}^{3}I_{j\ 3\times M} = \sum_{p=1}^{N} m_p \Phi_p \quad \text{j} = 1, \ldots, M \qquad \text{Eq. 4-3} $$

$$ {}^{4}I_{3\times M} = \sum_{p=1}^{N} m_p \tilde{\boldsymbol{s}}_p \Phi_p + \mathrm{I}_p \Phi_p^{*} \qquad \text{Eq. 4-4} $$

$$ {}^{5}I_{j\ 3\times M} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \Phi_p \quad \text{j} = 1, \ldots, M \qquad \text{Eq. 4-5} $$

$$ {}^{6}I_{M\times M} = \sum_{p=1}^{N} m_p \Phi_p^{T} \Phi_p + \Phi_p^{*\mathrm{T}} \mathbf{I}_p \Phi_p^{*} \qquad \text{Eq. 4-6} $$

$$ {}^{7}I_{3\times 3} = \sum_{p=1}^{N} m_p \tilde{\boldsymbol{s}}_p^{T} \tilde{\boldsymbol{s}}_p + \mathbf{I}_p \qquad \text{Eq. 4-7} $$

$$ {}^{8}I_{j\ 3\times 3} = \sum_{p=1}^{N} m_p \tilde{\boldsymbol{s}}_p \tilde{\phi}_{pj} \quad \text{j} = 1, \ldots, M \qquad \text{Eq. 4-8} $$

$$ {}^{9}I_{jk\ 3\times 3} = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \tilde{\phi}_{pk} \quad j, k = 1, \ldots, M \qquad \text{Eq. 4-9} $$

where $\boldsymbol{s}_p = [xyz]^T$ are the coordinates of grid point p in basic and

$$ \tilde{\boldsymbol{s}}_p = \begin{bmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{bmatrix} $$

is the skew-symmetric vector cross product operator. Φ_p is the partitioned orthogonal modal matrix that corresponds to the translational degrees-of-freedom of grid *p*, $\mathbf{I}_p$ is the inertia tensor, Φ_p^* is the partitioned orthogonal modal matrix that corresponds to the rotational degrees-of-freedom of grid *p*. $\tilde{\phi}_{pf}$ is the skew-symmetric matrix formed for each grid translational degree-of-freedom for each mode. M is the number of modes and N is the number of grids.

4. The above mass invariant calculation currently depends on a lumped mass formulation. The PARAM,COUPMASS should not be specified when executing the MD Nastran / MSC.ADAMS interface. Since p-elements use a coupled mass formulation they should not be used.
5. If the CONM1 is used, M21, M31, and M32 entries should be left blank.
6. If PARAM,GRDPNT,value specified, mass invariants ${}^1I$, ${}^2I$, and ${}^7I$ will be obtained from a MD Nastran grid point weight generator execution in the basic system.
7. The following DTI,UNITS Bulk Data entry is required for a FLEXBODY=YES run:

   Units:

   Since MSC.ADAMS is not a unitless code (as is MD Nastran), units must be specified. A DTI Bulk Data entry provides ‘UNITS’ (a unique identifier) input as the following example illustrates. Once identified, the units will apply to all superelements in the model. Acceptable character input strings are listed below.

   Format:

   DTI UNITS 1 MASS FORCE LENGTH TIME

   Example:

   DTI UNITS 1 KG N M S

Mass:

kg - kilogram
lbm - pound-mass
slug - slug
gram - gram
ozm - ounce-mass
klbm - kilo pound-mass (1000.lbm)
mgg - megagram
slinch - 12 slugs
ug - microgram
ng - nanogram
uston - US ton

Force:

n - newton
lbf - pounds-force
kgf - kilograms-force
ozf - ounce-force
dyne - dyne
kn - kilonewton
klbf - kilo pound-force (1000.lbf)
mn - millinewton
un - micronewton
nn - nanonewton

Length:

km - kilometer
m - meter
cm - centimeter
mm - millimeter
mi - mile
ft - foot
in - inch
um- micrometer
nm - nanometer
ang - angstrom
yd - yard
mil - milli-inch
uin - micro-inch

Time:

h - hour
min - minute
s - sec
ms - millisecond
us - microsecond
nanosec - nanosecond
d - day

A note of clarification about UNITS and its relation to MD Nastran's WTMASS parameter: WTMASS, though necessary to achieve units consistency in MD Nastran, is ignored in the output for MSC.ADAMS. Units data for MSC.ADAMS is supplied on the UNITS DTI entry. For example, consider a model with mass in grams, force in Newtons, length in meters and time in seconds. A WTMASS parameter equal to 0.001, would ensure that MD Nastran works with a consistent set of units (kg, N and m). The units reported to ADAMS should then be: "DTI, UNITS, 1, GRAM, N, M, S."

8. OUTSTRS or OUTSTRN entries require the use of the standard MD Nastran STRESS= or STRAIN= commands to produce element stress or strain. STRESS(PLOT)= or STRAIN(PLOT)= will suppress stress or strain output to the MD Nastran .f06 file. The OUTSTRS or OUTSTRN entries are required for importing MSC.ADAMS results into MSC.Fatigue. See the MD Nastran / MSC.ADAMS / Durability documentation for more information.
9. OUTGSTRS or OUTGSTRN entries require the use of the standard MD Nastran STRESS= or STRAIN= used in conjunction with GPSTRESS= or GPSTRAIN=commands to produce grid point stress or strain. GPSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain output to the MD Nastran .f06 file.
10. To reduce the FE mesh detail for dynamic simulations, PSETID=set_entry is used to define a set of PLOTELs or other elements used to display the component in ADAMS. If a mass invariant computation is requested, this option can significantly reduce the size of the MNF without compromising accuracy in the ADAMS simulation.

    If PSETID specifies an existing set from the OUTPUT(PLOT) Section of MD Nastran, this single set is used explicitly to define elements to display in ADAMS. Otherwise, the MD Nastran Case Control Section will be searched for a matching set ID. This matching set ID list then represents a list of OUTPUT(PLOT) defined elements sets, the union of which will be used to define a set of PLOTELS or other elements used to display the component in ADAMS. If the user wishes to select all of the sets in the OUTPUT(PLOT) section, then use PSETID=ALL.

    The elements defined may include rigid element IDs. When defining these sets *do not* use EXCLUDE and EXCEPT descriptions.

    If a superelement analysis is being executed, any element defined on the PSETID=set_entry that lies entirely on the superelement boundary (that is to say all of its grids are a-set or exterior to the superelement) must also be specified on a SEELT Bulk Data entry. The SEELT entry would not be required for part superelements, as boundary elements stay with their component.

    ```
    OUTPUT(PLOT)
    SET 7722 = 10001 THRU 10010
    ```
11. The ADMOUT=YES option is intended for users who plan to import ADAMS results into MSC.Fatigue. This option requires the following assignment command:

ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM

in the File Management Section of the MD Nastran file. It causes OP2 files with an .out extension to be generated for input into MSC.Fatigue. FLEXBODY=YES is required with its use. The files output are: DTI-units, SE-number of superelements (9999 if residual), SEIDX-superelement id, ASETX-size of a-set, BGPDTS-grid location table, GEOM2S-element connections, GEOM4S-constraints and sets, MGGEW-physical mass external sort with weight mass removed, VAEXT-a-set partition vector, VGEXT-g-set partition vector, VAPEXT-eigenvalue size partition vector, MAAEW-modal mass, KAAE-modal stiffness, BAAE-modal damping, RAE-modal preload, PAE-modal loads, CMODEXT-component modes, OES1-element stress shapes, OSTR1-element strain shapes, OGS1-grid point stress shapes, OGSTR1-grid point strain shapes, OGSIPL-grid point physical preload stress, OGTRIPL-grid point physical preload strain. The files are output for each superelement and their generation depends on the loading and output requests.

To insure compatibility with the MSC.ADAMS OP2-to-MNF translator, if the ADAMSMNF Case Control command has the keyword ADMOUT=YES, the MD Nastran SYSTEM word OP2NEW is automatically set to OP2NEW=0. This means that any OP2 files generated will have a pre-MSC.Nastran 2004 format.

12. Environment variables controlling MNF generation can be set before submitting the MD Nastran job or by using the MD Nastran keyword 'MNFWRITEOPTIONS' which can be abbreviated to any short unique string such as 'MNFW'. The MD Nastran keyword can be entered on the nastran submittal command line or in a user .nastran rc file.

    For example, interior grids and elements can be removed in the MNF by entering prior to the MD Nastran submittal;
    with Korn shell:
    export MDI_MNFWRITE_OPTIONS=strip_face
    with C shell:
    setenv MDI_MNFWRITE_OPTIONS=strip_face

    Or, at MD Nastran submittal time:
    nastran_submittal_command jid MNFW=strip_face

    Consult the ADAMS/Flex documentation for more information on the use of environment variables during MNF generation.

    The command: nastran_submittal_command help mnfw will generate the description of the symbol keyword.

The command: nastran_submittal_command help all will generate a complete set of MD Nastran submittal keywords.

13. MNF naming convention is as follows: for a single superelement run 'jid.mnf'; for a residual only or multiple superelement run 'jid_seid.mnf', etc., where seid1 and seid2 are the integer numbers of the superelement. The default location of these files is the same directory as the jid.f06 file. See the ASSIGN MNF command to change directory location.

14. When supplying SPOINT/QSET combinations there should be enough to capture correctly the modal shapes. If *n* is the number of modes specified on the EIGR or EIGRL Bulk Data entries and *p* is the number of load cases specified, then the number of SPOINT's(*ns*) should be at least $ns = n + (6 + p)$ assuming that residual flexibility is on. In general, there cannot be too many SPOINTS as excess ones will be simply truncated with no performance penalty.

15. The user can have MD Nastran automatically specify the SPOINT/QSET by including above the Case Control the parameter PARAM,AUTOQSET,YES. In this case no SPOINT/QSET can appear in the Bulk Data. See the PARAM,AUTOQSET description for detailed requirements or limitations.

16. By default, MSC.Nastran 2005 will create a version 6.0 MNF. MSC.ADAMS 2005 is able to read the version 6.0 MNF. Earlier MSC.ADAMS versions are not able to read a 6.0 MNF. MD Nastran can be instructed to write a backward compatible MNF by submitting the MD Nastran job with MNFWRITEOPTIONS=full_str. Alternatively, the user may set the environment variable MDI_MNFWRITE_OPTIONS to 'full_str'. See Remark **12.** for more information on controlling the MNF format.

17. Besides modal stiffness and modal mass matrices, the modal damping matrix may be output to the MNF. The damping allowed is the standard MD Nastran damping matrix consisting of $[B_{gg}]$ viscous damping, $(1/w4)\,[K4_{gg}]$ structural damping, $(g/(w3)+\alpha 2)[K_{gg}]$ structural and Rayleigh damping, $(\alpha 1)[M_{gg}]$ Rayleigh damping. Where g is set by PARAM,G,value, $w3$ is set by PARAM,W3,value, $w4$ is set by PARAM,W4,value, $\alpha 1$ is set by PARAM,ALPHA1,value,0., $\alpha 2$ is set by PARAM,ALPHA2,value,0. Additionally, $[B2H]$ modal damping can be included by use of the Case Control SDAMP=n command. For part superelement or superelement analyses, modal damping for each individual part or superelement can be controlled by PARAM,SESDAMP,YES or for a MD Nastran/MSC.ADAMS interface run PARAM,SESDAMP,FRB (PARAM,SESDAMP,NO is the default).

SESDAMP		
sesdamp=no	modal damping for each superelement using the free boundary modes	SDAMP ⇒ TABDMP1 SDAMP above subcase
sesdamp=yes	modal damping for each superelement using the fixed boundary CMS modes	SDAMP ⇒ TABDMP1 SDAMP in superelement subcase
sesdamp=frb	modal damping for each superelement using the free boundary modes	SDAMP ⇒ TABDMP1 SDAMP in superelement subcase
For PART SUPERELEMENTS each part may have a PARAM,SESDAMP		

The MD Nastran / MSC.ADAMS interface does not allow for adding modal damping to structural damping utilizing PARAM,KDAMP,-1.
Direct input damping may also be included with the Case Control B2GG=n command. For part superelement or superelement analyses, use of this command with the MD Nastran / MSC.ADAMS interface requires fully expanded case control.

18. If preload is present in the model, physical gridpoint stress and strain for the preload may be output to the MNF using standard GPSTRESS= or GPSTRAIN= commands.
If preload is generated in a SOL 106 for a SOL 103 restart and the physical grid point stresses for the preload are desired for the SOL 103 MNF run, then PARAM,FLEXNLS,YES is required above subcase level in the SOL 106 run. For preload generated in a SOL 106 for a SOL 103 restart, the preload subcase must be replicated in the FIRST subcase of the SOL 103 run.

19. After using the MD Nastran / MSC.ADAMS interface to produce a MNF and after preforming an MSC.ADAMS solution it is possible to bring the MSC.ADAMS results into MD Nastran for modal data recovery.
MSC.ADAMS produces OP2 files for input to MD Nastran SOL 111 and SOL 112. The files are binary format with a .mdf extension. The File Management Section requires an assign command for each file:

    ASSIGN INPUTT2='name.mdf' UNIT=ni

    with a DLOAD=ni in the appropriate subcase.

Also, in the Bulk Data Section, the parameter PARAM,ADMPOST=m (m=0, by default no MSR performed) is required. If m=1, rigid body motion is not considered in the structural deformation. If m=2, rigid body motion is considered in the structural deformation.

Full details of the generation of the .mdf files and their use with MD Nastran are to be found in the MD Nastran / MSC.ADAMS Durability documentation.

Caution: In order to obtain consistent results, the MSC.ADAMS results, when brought back into MD Nastran SOL 111 or SOL 112, MUST be restarted off the original MD Nastran database that produced the original MNF which was the basis of the MSC.ADAMS run.

ADAPT Adaptivity Control Selection

Specifies adaptivity control parameters.

Format:

ADAPT=n

Example:

ADAPT=12

Describer	Meaning
n	Set identification for either an ADAPT or PSET Bulk Data entry. (Integer > 0)

Remarks:

1. ADAPT is required only when an analysis with p-elements is requested.
2. A multiple p-level analysis with error analysis is performed whenever the ADAPT command references an ADAPT Bulk Data entry.
3. A single p-level analysis without error analysis is performed whenever the ADAPT command references a PSET Bulk Data entry.
4. Only one ADAPT may appear in the Case Control Section and should appear above all SUBCASE commands.
5. The subcases that will not participate in the error analysis/adaptivity must contain the ADACT=NONE command.

AECONFIG Aeroelastic Configuration Name

Assigns the aerodynamic configuration parameter used to locate the associated datablocks on the aerodynamic and aeroelastic databases.

Format: Assign a DBset member name

AECONFIG =config-name

Example 1: Assign a MASTER file for the aerodynamic and aeroelastic DBsets

AECONFIG =PROTO_A

Describer	Meaning
config-name	The configuration name. This is the Aerodynamic SuperGroup name identified as part of the aeroelastic model. (Character, Default = AEROSG2D)

Remarks:

1. Typically, the aeroelastic configuration name is specified as the Aerodynamic Supergroup as part of the model generation in MSC.FlightLoads.
2. If AECONFIG is not present, aerodynamic and aeroelastic datablocks will be created from the data in the Bulk Data Section and assigned the default value AECONFIG=AEROSG2D.
3. Multiple configuration names are supported.
4. AECONFIG is typically assigned above the subcase level. If it is overridden at the subcase level, it is necessary to attach an existing aerodynamic database.

AERCONFIG Aerodynamic Model to be Used for the Rigid Aerodynamics

Enables the user to select a different mesh for the rigid portion of the aerodynamics than for the elastic portion.

Format:

AERCONFIG=config-name

Examples:

AERC=RAERO

Describer	Meaning
AERC	The configuration name. This is the Aerodynamic Super Group name for the aerodynamic model that is used to create the rigid aerodynamics.

Remarks:

1. If the AERCONFIG command is not present in the subcase, the rigid portion of the aerodynamics is based on the same AECONFIG as the flexible aerodynamics.
2. The rigid aerodynamics must be pre-computed and attached from an assigned database using FMS commands such as:

```
ASSIGN RMASTER = “raero.master”
DBLOCATE WHERE(AECONFIG=’rconfig’) LOGICAL=RMASTER
```

AEROF Aerodynamic Force Output Request

Requests the aerodynamic loads on aerodynamic control points.

Format:

AEROF=n

Examples:

AEROF=ALL
AEROF=5

Describer	Meaning
n	Set identification of a previously appearing SET command. (Integer > 0)
ALL	Forces at all points will be output.

Remarks:

1. This command is supported in SOL 144, SOL 145, SOL 146 (frequency response only) and SOL 200 for ANALYSIS=SAERO or FLUTTER.
2. The SET command references box or body element identification numbers.
3. Output is in the units of force or moment.
4. Only aerodynamic forces on points specified on the SET command will be output.

AESYMXY Aerodynamic Flow Symmetry About The XY Plane

Aerodynamic XY plane of symmetry flag. This is used to indicate whether the aerodynamic model has symmetry with respect to the ground.

Format:

$$\text{AESYMXY} = \left\{ \begin{array}{c} \text{SYMMETRIC} \\ \text{ANTISYMMETRIC} \\ \text{ASYMMETRIC} \end{array} \right\}$$

Example 1:

AESYMXY = ASYMMETRIC

Describer	Meaning
SYMMETRIC	Indicates that the aerodynamic model is moving in a symmetric manner with respect to the XY plane
ANTISYMMETRIC	Indicates that the aerodynamic model is moving in an antisymmetric manner with respect to the XY plane.
ASYMMTRIC	Indicates that the aerodynamic model has no reflection about the XY plane.

Remarks:

1. If AESYMXY is not present in case control, aerodynamic XY symmetry will be determined from the SYMXY field of the AEROS Bulk Data entry for static aeroelastic analysis and from the SYMXY field of the AERO Bulk Data entry for flutter and dynamic aeroelastic analysis.
2. If AESYMXY is present above the subcase level, it is applied to all subcases until overridden.
3. SYMMETRIC implies ground effect while asymmetric implies free air analysis.
4. Multiple aerodynamic symmetries are supported.

AESYMXZ Aerodynamic Flow Symmetry About The XZ Plane

Aerodynamic XZ plane of symmetry flag. This is used to support symmetric models about the centerline.

Format:

$$\text{AESYMXZ} = \left\{ \begin{array}{c} \text{SYMMETRIC} \\ \text{ANTISYMMETRIC} \\ \text{ASYMMETRIC} \end{array} \right\}$$

Example 1:

AESYMXZ = SYMMETRIC

Describer	Meaning
SYMMETRIC	Indicates that a half span aerodynamic model is moving in a symmetric manner with respect to the XZ plane.
ANTISYMMETRIC	Indicates that a half span aerodynamic model is moving in an antisymmetric manner with respect to the XZ plane.
ASYMMETRIC	Indicates that a full aerodynamic model is provided. (Default.)

Remark:

1. If AESYMXZ is not present in case control, aerodynamic XZ symmetry will be determined from the SYMXZ field of the AEROS Bulk Data entry for static aeroelastic analysis and from the SYMXZ field of the AERO Bulk Data entry for flutter and dynamic aeroelastic analysis.
2. If AESYMXZ is present above the subcase level, it is applied to all subcases until overridden.
3. Multiple aerodynamic symmetries are supported.

AEUXREF Define the Reference Aerodynamic Extra Point (Controller) Vector

Reference UXVEC selector for the aeroelastic trim analysis. This is used to indicate an aerodynamic extra point vector about which the stability derivatives are to be computed and printed. The stability derivatives are the change in force due to a unit perturbation of each parameter in the aerodynamic extra point set. Due to the nonlinear nature of the aeroelastic loads, the stability derivatives can be (but are not required to be) a function of the point about which the slope is computed. This input defines which point is to be used in computing the stability derivatives for printing (local slopes will be computed as needed in the trim solver). This selection is typically done within each subcase, but a case control default can be defined by placing an entry above the subcase level.

Format:

$$\text{AEUXREF} = \left\{ \begin{matrix} \text{n} \\ \text{TRIM} \end{matrix} \right\}$$

Examples:

AEUXREF=100
AEUXREF=TRIM

Describer	Meaning
n	The identification number of a UXVEC Bulk Data entry that defines the point about which stability derivatives will be computed in TRIM cases.
TRIM	Indicates that the stability derivatives should be computed about the trimmed state.

Remarks:

1. If, for a particular subcase, AEUXREF is not defined, the “free stream” state will be used (that is, the stability derivatives will be computed about zero values for all parameters). This results in upward compatibility with the linear database paradigm.
2. Only one of TRIM or a UXVEC id may be specified on any given subcase. To see stability derivatives about additional points, you must define additional trim subcases.

ANALYSIS Analysis Discipline Subcase Assignment

Specifies the type of analysis being performed for the current subcase.

Format:

ANALYSIS=type

Examples:

ANALYSIS=STATICS
ANALYSIS=MODES

Describer	Meaning
type	Analysis type. Allowable values and applicable solution sequences (Character):

STATICS	Statics	(SOL 200 only)
MODES	Normal Modes also in SOL 110, 111, 112	
BUCK	Buckling	
DFREQ	Direct Frequency	
MFREQ	Modal Frequency	
MTRAN	Modal Transient	
DCEIG	Direct Complex Eigenvalue Analysis	
MCEIG	Modal Complex Eigenvalue Analysis	
SAERO	Static Aeroelasticity	
DIVERGE	Static Aeroelastic Divergence	
FLUTTER	Flutter	
HEAT	Heat Transfer Analysis	(SOLs 153 and 159 only)
STRUCTURE	Structural Analysis	
NLSTATICS	Nonlinear static analysis	(SOL 400, Remark 5.)
LNSTATICS	Linear static analysis	
NLTRAN	Nonlinar transient analysis	

Remarks:

1. ANALYSIS=STRUC is the default in SOLs 153 and 159.

2. In SOL 200, all subcases, including superelement subcases, must be assigned an ANALYSIS command either in the subcase or above all subcases.
3. ANALYSIS=DIVERG is only available for analysis in SOL 200. Sensitivity and optimization are not supported for this analysis type.
4. In order to obtain normal modes data recovery in SOLs 110, 111, and 112, ANALYSIS = MODES must be specified under one or more separate subcases(s) which contain requests for data recovery intended for normal modes only. For example, in SOL 111:

```
METH=40
SPC=1
SUBCASE 1 $ Normal Modes
  ANALYSIS=MODES
  DISP=ALL
SUBCASE 2 $ Frequency response
  STRESS=ALL
  DLOAD=12
  FREQ=4
```

   All commands which control the boundary conditions (SPC, MPC, and SUPORT) and METHOD selection should be copied inside the ANALYSIS=MODES subcase or specified above the subcase level.
5. For SOL 400, NLSTATICS will perform a nonlinear static analysis; LNSTATICS will perform a linear static analysis; and NLTRAN will perform a nonlinear transient analysis. The default is NLSTATICS. NLSTATICS, LNSTATICS, and NLTRAN can be placed above or below SUBCASE/STEP commands. For example:

```
SUBCASE 1
   STEP 1
       ANALYSIS=LNSTATICS $ Linear statics
       LOAD=1
   STEP 2
       ANALYSIS=NLSTATICS $ Nonlinear statics
       LOAD=2
   STEP 3
       ANALYSIS=NLTRAN $ Nonlinear transient
       DLOAD=3
SUBCASE 2
   ANALYSIS=NLSTATICS $ Nonlinear statics for both steps
   STEP 10
       LOAD 10
   STEP 20
       LOAD=20
SUBCASE 3
   ANALYSIS =NLTRAN $Nonlinear transient for both steps
   STEP 10
       DLOAD=10
   STEP 20
       DLOAD=20
```

APRESSURE Aerodynamic Pressure Output Request

Requests the aerodynamic pressures in static aeroelastic response.

Format:

$$\text{APRES} = \left\{ \begin{array}{c} \text{n} \\ \text{ALL} \end{array} \right\}$$

Examples:

APRES=ALL
APRES=6

Describer	Meaning
n	Set identification number of a previously appearing SET command. Only aerodynamic pressures on the referenced aerodynamic boxes will be output. (Integer > 0)
ALL	Pressures at all points will be output.

AUTOSPC — Constrains Stiffness Singularities via m-sets or s-sets

Requests that stiffness singularities and near singularities be automatically constrained via single or multipoint constraints.

Format:

$$\text{AUTOSPC}\left[\left(\left[\begin{matrix}\text{PRINT}\\ \text{NOPRINT}\end{matrix}\right], \left[\left[\begin{matrix}\text{NOPUNCH}\\ \text{PUNCH}\end{matrix}\right], [\text{SID} = \text{n}]\right], [\text{EPS} = \text{r1}], [\text{EPSSING} = \text{r2}], \left[\begin{matrix}\text{SPC}\\ \text{MPC}\end{matrix}\right], \left[\begin{matrix}\text{ZERO}\\ \text{NOZERO}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right\}$$

Examples:

AUTOSPC=YES
AUTOSPC (PRINT, PUNCH, SID=100, EPS=1.E-6, MPC)=YES

Describer	Meaning
PRINT	Enable the printout of a summary table of singularities. (Default)
NOPRINT	Disable the printout of a summary table of singularities.
NOPUNCH	Disable the creation of SPC or MPC Bulk Data entry in PUNCH file. (Default)
PUNCH	Generate SPC or MPC Bulk Data entry format in PUNCH file.
SID=n	Specify a set identification number for option PUNCH. (Default = 999)
EPS=r1	Identify the singularity with a stiffness ratio smaller than r1 to be automatically constrained with single or multipoint constraint. See Remark 2. (Default=1.E-8)
EPSSING=r2	Identify the potential singularities with stiffness ratios less than r2. See Remark 2. (Default=1.E-8)
SPC	Apply single-point constraints on those degrees of freedom identified as singular. (Default)
MPC	Apply multipoint constraints on those degrees of freedom identified as singular.

Describer	Meaning
ZERO	Request the printout of singularities with zero stiffness ratios in the singularity summary table. (Default)
NONZERO	Disable the printout of those singularities with zero stiffness ratios in the singularity summary table.

Remarks:

1. AUTOSPC specifies the action to take when singularities exist in the stiffness matrix. AUTOSPC=YES means that singularities will be constrained automatically. AUTOSPC=NO means that singularities will not be constrained. If AUTOSPC=NO then the user should take extra caution analyzing the results of the grid point singularity table and the computed epsilons. See "**Constraint and Mechanism Problem Identification in SubDMAP SEKR**" on page 409 of the *MSC.Nastran Reference Guide* for details of singularity and mechanism identification and constraint.
2. Singularity ratios smaller than EPSSING are listed as potentially singular. If AUTOSPC=YES then the identified singularities with a ratio smaller than EPS will be automatically constrained. If EPSSING has the same value as EPS then all singularities are listed. If EPSSING is large than EPS, the printout of singularity ratios equal to exactly zero is suppressed. EPSSING must be greater than or equal to EPS. If not, then the program will set EPSSING equal to EPS.
3. If the PUNCH keyword is specified then automatically generated SPCs or MPCs are placed in SPCi or MPCi Bulk Data entry format on the PUNCH file.
4. Default equals YES for all solutions except for SOLs 106 and 153. The AUTOSPC operation provides the correct action for superelements in all contexts. However, AUTOSPC may over-constrain the residual structure in SOL 129. User PARAMeter, AUTOSPC, not the AUTOSPC command is used for the o-set (omitted set) in the residual structure in SOLs 106 and 153.
5. The MPC option may be somewhat more expensive than the SPC option. It, however, provides a more realistic structural modeling than the SPC. When the MPC option is selected, the multipoint constraint may be applied on some degree of freedom on which the stiffness matrix is identified as singular. If MPC is inapplicable on some degree of freedom, the SPC is used instead.

AUXCASE Auxiliary Model Case Control Delimiter

Delimits Case Control commands for an auxiliary model in SOL 200.

Format:

AUXCASE

Examples:

AUXCAS
AUXC

Remarks:

1. AUXCASE indicates the beginning of Case Control commands for an auxiliary model. AUXCASE commands must follow the primary model Case Control commands.
2. All Case Control commands following this entry are applicable until the next AUXCASE or BEGIN BULK command. Commands from preceding Case Control Sections are ignored.
3. Each auxiliary model Case Control must be delimited with the AUXCASE command.
4. The AUXMODEL command is used to associate the auxiliary model Case Control with a particular auxiliary model.

AUXMODEL Auxiliary Model Identification Number

References an auxiliary model for generation of boundary shapes in shape optimization.

Format:

AUXMODEL=n

Examples:

AUXMODEL=4
AUXM=4

Describer	Meaning
n	Auxiliary model identification number. (Integer>0)

Remarks:

1. AUXMODEL references a particular auxiliary model for analysis and may only be specified in the auxiliary model Case Control Section.
2. See the BEGIN BULK command for the Bulk Data definition of an auxiliary model.

AXISYMMETRIC Conical Shell Boundary Conditions

Selects boundary conditions for an axisymmetric shell problem or specifies the existence of fluid harmonics for hydroelastic problems.

Format:

$$\text{AXISYMMETRIC} = \left\{ \begin{matrix} \text{SINE} \\ \text{COSINE} \\ \text{FLUID} \end{matrix} \right\}$$

Example:

AXISYMMETRIC=COSINE

Describer	Meaning
SINE	Sine boundary conditions will be used.
COSINE	Cosine boundary conditions will be used.
FLUID	Existence of fluid harmonics.

Remarks:

1. This command is required for conical shell problems.
2. If this command is used for hydroelastic problems, at least one harmonic must be specified on the AXIF command.
3. See the "**Surface Elements**" on page 130 of the *MSC.Nastran Reference Guide*, for a discussion of the conical shell problem.
4. See the "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide* for a discussion of the hydroelastic formulation.
5. The sine boundary condition will constrain components 1, 3, and 5 at every ring for the zero harmonic.
6. The cosine boundary condition will constrain components 2, 4, and 6 at every ring for the zero harmonic.
7. SPC and MPC Case Control commands may also be used to specify additional constraints."**Case Control Commands**" on page 193.

B2GG Direct Input Damping Matrix Selection

Selects direct input damping matrix or matrices.

Format:

B2GG=name

Example:

B2GG = BDMIG

B2GG = BDMIG1, BDMIG2, BDMIG3

B2GG = 1.25*BDMIG1, 1.0*BDMIG2, 0.82*BDMIG3

SET 100 = B1, B2
B2GG = 100

Describer	Meaning
name	Name of $[B^2_{gg}]$ matrix that is input on the DMIG Bulk Data entry, or name list with or without factors (see Remark 5.).

Remarks:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the damping matrix before any constraints are applied.
3. The matrix must be symmetric and field 4 on the DMIG,name entry must contain the integer 6.
4. A scale factor may be applied to this input via the PARAM, CB2 entry. See “Parameters” on page 659.
5. The formats of the name list:
   a. Names without factor
      Names separated by comma or blank.
   b. Names with factors
      Each entry in the list consists of a factor followed by a star followed by a name. The entries are separated by comma or blank. The factors are real numbers. Each name must be with a factor including 1.0.

B2PP Direct Input Damping Matrix Selection

Selects direct input damping matrix or matrices.

Format:

B2PP=name

Example:

B2PP = BDMIG

B2PP = BDMIG1, BDMIG2, BDMIG3

B2PP = 5.06*BDMIG1, 1.0*BDMIG2, 0.85*BDMIG3

B2PP = (1.25, 0.5) *BDMIG1, (1.0, 0.0) *BDMIG2, (0.82,-2.2) *BDMIG3

Describer	Meaning
name	Name of $[B_{pp}^{2}]$ matrix that is input on the DMIG or DMIAX Bulk Data entry or name list with or without factors (see Remark 7.) (Character)

Remarks:

1. DMIG entries will not be used unless selected.
2. B2PP is used only in dynamics problems.
3. DMIAX entries will not be used unless selected by the B2PP command.
4. The matrix must be square or symmetric and field 4 on the DMIG,name entry must contain a 1 or 6.
5. It is recommended that PARAM,AUTOSPC,NO be specified. See the "***Constraint and Mechanism Problem Identification in SubDMAP SEKR***" on page 409 of the *MSC.Nastran Reference Guide*.
6. The matrices are additive if multiple matrices are referenced on the B2PP command.
7. The formats of the name list:
   a. Names without factor

      Names separated by comma or blank.

b. Names with factors.

Each entry in the list consists of a factor followed by a star followed by a name. The entries are separated by comma or blank. The factors are either all real numbers, or all complex numbers in the form of two real numbers, separated by a comma, within parenthesis as shown in the above example. The first real number of the pair is for real part, and the second for imaginary part. Either part may be zero or blank, but not both. Mixed real numbers and complex numbers are not allowed. Each name must be with a factor including 1.0 for real and (1.0, 0.0) for complex.

BC Boundary Condition Identification

Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.

Format:

BC=n

Example:

BC=23

Describer	Meaning
n	Identification number. (Integer > 0)

Remarks:

1. In SOLs 103, 105, 145, and 200 BC is required in each subcase if multiple boundary conditions are specified for normal modes, buckling, and flutter analysis.
2. If only one boundary condition is specified, then BC does not have to be specified and n defaults to zero.

BCONTACT (SOL 600/700) Selects 3D Contact Surfaces (SOLs 600 and 700 only)

Selects the contact table and contact parameters for the current subcase. This Case Control command must be entered with one of the options shown below for any 3D contact to occur. This command may be entered above any subcases to apply to all subcases; however, doing so will invoke an option such that rigid contact bodies will be moved until they contact the formable bodies before the nonlinear simulation begins (for MSC.Marc users, this is known as step zero).

If there are multiple subcases, then the BCONTACT Case Control command can be entered before the first subcase to move rigid contact bodies together before the nonlinear solution begins. BCONTACT may be specified within each subcase to define which bodies may make contact during that particular subcase. If no BCONTACT entries are made within a particular subcase and one is entered before any subcase entries, it will be used for that subcase.

Format:

$$\text{BCONTACT}=\left\{\begin{array}{c} n \\ \text{ALL} \\ \text{ALLELE} \\ \text{ALLBODY} \\ \text{BCBOX} \\ \text{BCPROP} \\ \text{BCMATL} \\ \text{NONE} \end{array}\right\}$$

Examples:

BCONTACT = 5
BCONTACT=ALLBODY

Describer	Meaning
n	Identification number of a BCTABLE, BCHANGE, and / or BCMOVE Bulk Data entry.
ALL	All elements in the model can potentially contact with each other (Default). When this option is specified, no 3D contact input is required in the bulk data and if entered will be ignored. Warning - this option may take excessive computer time. This option can only be used if it applies to all subcases.
ALLELE	Same as ALL.

Describer	Meaning
ALLBODY	All bodies defined using all the BCBODY entries can potentially contact each other.This option can only be used if it applies to all subcases.
BCBOX	All elements defined within a box-like region as defined by the Bulk Data entry, BCBOX, can potentially contact each other. See Remark 4.
BCPROP	All elements defined by the Bulk Data entry, BCPROP, can potentially contact each other. See Remark 4.
BCMATL	All elements defined by the Bulk Data entry, BCMATL, can potentially contact each other. See Remark 4.
NONE	All contact definitions (BCTABLE, BCBODY) are ignored.

Remarks:

1. BCONTACT is only recognized in MD Nastran Implicit Nonlinear (SOLs 600 and 700).
2. BCONTACT is required to activate 3D contact in MD Nastran Implicit Nonlinear (SOL 600) and Explicit Nonlinear (SOL 700).
3. Only one form of this entry may be used in any given analysis. Analysis restarts must use the same form as the original run.
4. Bulk Data entries, BCBOX, BCPROP and BCMATL, may be used with BCONTACT=n in which case IDs specified on the BCBODY entry and BCBOX, BCPROP and/or BCMATL entries must match.

BEGIN BULK Case Control and Bulk Data Delimiter

Designates the end of the Case Control Section and / or the beginning of a Bulk Data Section.

Format:

$$\text{BEGIN}\,[\text{BULK}]\begin{bmatrix}\text{AUXMODEL} = \text{auxmid}\\ \text{SUPER} = \text{seid}\end{bmatrix}$$

Examples:

```
BEGIN BULK
BEGIN AUXMODEL=22
```

Describer	Meaning
AUXMODEL	Indicates the beginning of an auxiliary model Bulk Data Section.
auxmid	Auxiliary model identification number. (Integer > 0)
SUPER	Indicates the beginning of partitioned superelement Bulk Data Section.
seid	Superelement identification number. (Integer $\geq$ 0)

Remarks:

1. BEGIN BULK is not required. If not specified, then the program will automatically insert one.
2. For an auxiliary model, AUXMID is referenced by the AUXMODEL Case Control command.
3. Partitioned Bulk Data Sections defined by BEGIN SUPER are used to define only one superelement each. Bulk Data commands which define superelements are ignored in partitioned Bulk Data Sections.

   Superelements specified by a BEGIN SUPER entry can be automatically attached to other superelements based on relative location of grid points. For connection to the downstream superelement, the global coordinate directions of the attachment grid points of the upstream superelement will be internally transformed to the global coordinate directions of the grid points of the downstream superelement. For displacement data recovery, the output will be in the original global coordinate directions.

4. The BEGIN SUPER or BEGIN AUXMODEL Bulk Data must lie between BEGIN BULK and ENDDATA entries.
5. When employing part superelements via the use of the BEGIN BULK SUPER (or BEGIN SUPER) entry, it should be noted that any parameters that are specified in the Main Bulk Data apply only to the residual and not to any of the part superelements. Accordingly, if the user wants certain parameters to apply to all of the superelements, then he must specify them either in the Case Control Section or explicitly in all of the BEGIN BULK SUPER (or BEGIN SUPER) portions of the Bulk Data. A very common example of such a parameter specification is PARAM,POST which is used to request post-processing of results.

BOUTPUT Line Contact or 3D Contact (for SOL 600) Output Requests

Selects contact regions for output.

Format:

$$\text{BOUTPUT}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right],\ \left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Examples:

BOUTPUT=ALL
BOUTPUT=5

Describer	Meaning
SORT1	Output is presented as a tabular listing of slave nodes for each load or time depending on the solution sequence.
SORT2	Output is presented as a tabular listing of load or time for each slave node.
PRINT	The print file is the output media.
PUNCH	The punch file is the output media.
PLOT	Generate output histories for slave nodes, but do not print.
ALL	Histories of all the slave nodes (all nodes for SOL 600) listed in all the BOUTPUT Bulk Data entries are output. If no BOUTPUT Bulk Data entries are specified, histories of all the slave nodes in all the contact regions are output.
n	Set identification of previously appearing SET command. Only contact regions with identification numbers that appear on the SET command are selected for output. If there is a BOUTPUT Bulk Data entry for a contact region selected via the set command, histories for slave nodes listed in the Bulk Data entry are output. If there is no BOUTPUT Bulk Data entry for a contact region selected via the set command, histories for all the slave nodes in that contact region are output.
NONE	Result histories for slave nodes are not calculated or output.

Remarks:

1. BOUTPUT is processed in SOLs 106, 129, 153, 159, and 600 only.
2. SORT1 is the default in SOLs 106 and 153. SORT2 is the default in SOLs 129 and 159.
3. Only SORT1 is available for SOL 600.

CAMPBELL Campbell Diagram Parameters

Specifies Campbell Diagram parameters.

Format:

CAMPBELL= n

Example:

CAMPBELL= 10

Describer	Meaning
n	Identification number of a CAMPBLL Bulk Data entry. (Integer > 0)

CLOAD Static Load Request for Upstream Superelement Loads

Requests a CLOAD Bulk Data entry that defines a list of superelement loads and their scale factors in nonlinear static analysis only.

Format:

CLOAD=n

Example:

CLOAD=15

Describer	Meaning
n	Identification number of a unique CLOAD Bulk Data entry. (Integer>0)

Remarks:

1. This command may only appear in the residual structure subcases (see the Case Control command, "**SUPER**" on page 496) and if used it must be specified in all of them.
2. The CLOAD Bulk Data entry must reference previously processed LSEQ (load sequence) Bulk Data that was requested by LOADSET Case Control commands on the upstream (SUPER ≠ 0) subcases.
3. The resulting load is added to those produced by LOAD and TEMP(LOAD) Case Control commands in the residual structure subcases.

CMETHOD Complex Eigenvalue Extraction Method Selection

Selects complex eigenvalue extraction parameters.

Format:

CMETHOD=n

Example:

CMETHOD=77

Describer	Meaning
n	Set identification of EIGC (and EIGP) Bulk Data entry. (Integer>0)

Remarks:

1. The CMETHOD command must be specified in order to compute complex eigenvalues.
2. See description of the parameter, "**UNSYMF**" on page 837, to perform complex eigenvalue analysis in Solution 106.

CMSENERGY Component Modal Synthesis Energy Output Request

Requests the form and type of CMS energy output.

Format:

$$\text{CMSENERGY}\left(\left[\begin{matrix}\text{PRINT}\\ \text{PUNCH}\\ \text{PLOT}\end{matrix}\right],\left[\begin{matrix}\text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right],\left[\text{ESORT}=\left\{\begin{matrix}\text{MODE}\\ \text{ASCEND}\\ \text{RATIO}\end{matrix}\right\}\right],\right.$$

$$\left[\text{RESPONSE}=\left\{\begin{matrix}\text{BOTH}\\ \text{MODAL}\\ \text{FORCED}\end{matrix}\right\}\right],\left[\text{CMSE}=\left\{\begin{matrix}\text{ALL}\\ \text{NONE}\\ \text{TOTAL}\\ \text{QSET}\end{matrix}\right\}\right],\left[\text{CMKE}=\left\{\begin{matrix}\text{ALL}\\ \text{NONE}\\ \text{TOTAL}\\ \text{QSET}\end{matrix}\right\}\right],$$

$$\left[\text{CMDE}=\left\{\begin{matrix}\text{ALL}\\ \text{NONE}\\ \text{TOTAL}\\ \text{QSET}\end{matrix}\right\}\right],\left[\text{FILTER}=\left\{\begin{matrix}0.001\\ \text{fratio}\end{matrix}\right\}\right],[\text{TOPN}=\text{m}])=\left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Example:

```
CMSENERGY (PHASE,RESPONSE=FORCED,CMSE=TOTAL,CMKE=QSET) = ALL
SET 1001 = 10,40
CMSENERGY (PUNCH,PRINT,RESPONSE=BOTH,CMSE=ALL,FILTER=0.01) = 1001
```

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generate modal fractions for the requested set but no printer output.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output.
PHASE	Requests polar format (magnitude and phase) of complex output.
ESORT	Keyword selecting one of the output sorting options: MODE / RATIO / ASCEND.
MODE	Results are generated in order of increasing CMS natural mode number.
ASCEND	Results are output in order of increasing energy ratio magnitudes.

Describer	Meaning
RATIO	Results are output in order of decreasing energy ratio magnitudes.
RESPONSE	Keyword selecting the types of results to be produced.
MODAL	Specifies that output is to be generated for the free (or real eigenvalue) response solution.
FORCED	Specifies that output is to be generated for the forced response (modal frequency or modal transient) solution.
BOTH	Requests output for both free and forced response solutions.
CMSE	Keyword requesting output of CMS strain energy ratios.
CMKE	Keyword requesting output of CMS kinetic energy ratios.
CMDE	Keyword requesting output of CMS damping energy ratios.
ALL	Requests both TOTAL and QSET output.
TOTAL	Requests CMS energy ratio totals in all component modes of a superelement.
QSET	Requests CMS energy ratios for individual component modes.
NONE	Requests that no CMS energy output be generated.
FILTER	Keyword specifying the value of the printed output data filter.
fratio	Value of output filter ratio. (Default = 0.001)
TOPN	Keyword specifying the number of largest CMS energy ratios to be output.
m	The number of largest CMS energy ratios to be output. (Default is all ratios.)
n	Results for superelement IDs in SET n will be output.
ALL	Results for all recovered superelements will be output.
NONE	No CMS energy ratios will be output.

Remarks:

1. The CMSENERGY command may be requested in the modal solution sequences (SOLs 110, 111, 112, 145, 146, 200) and real eigenvalue analysis (SOLs 103 and 106). It is intended for use when superelements are defined and component modal synthesis techniques are employed. (See the MODALKE and MODALSE commands for other options.)

2. Both PRINT and PUNCH may be requested.
3. ESORT, FILTER and TOPN describers apply only to QSET results output. TOTAL results output is always in increasing order of superelement ID number.
4. QSET CMS energy ratios are output in increasing order of component mode number unless the ESORT keyword specifies a particular sorting order. If a sorting order is specified, the magnitude of the energy ratio is sorted. DESCEND can be used as a synonym for RATIO.
5. The FILTER keyword specifies an absolute value that is used to limit the amount of printed output produced. It is applied to the magnitude of the CMS energy ratio. If the CMS energy ratio magnitude is less than fratio for any natural mode, no output for that natural mode is produced. THRESH can be used as a synonym for FILTER.
6. In order to obtain unforced response (RESPONSE=BOTH or MODAL) output in SOL 111 and SOL 112, a SUBCASE containing the ANALYSIS = MODES option must be present.
7. For modal transient response solution sequences, response quantities are real numbers. There are no imaginary terms. Therefore, polar representations of the data have no meaning.

CSSCHD Aerodynamic Control Surface Schedule

Selects control system schedule information.

Format:

CSSCHD=n

Example:

CSSCHD=10

Describer	Meaning
n	Set identification of a control system schedule that appears on a CSSCHD Bulk Data entry.

Remark:

1. One or more CSSCHD entries can be invoked by this entry.

DATAREC Data Recovery Output for p-Version Elements

Requests form and type of output for p-version elements.

Format:

$$\text{DATAREC}\left[\left(\begin{matrix}\text{SORT1}\\ \text{SORT}\end{matrix}\right)\right] = n$$

Example:

DATAREC=12

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid point results per load case.
SORT2	Output will be presented as a tabular listing of load case per grid point.
n	SID of OUTPUT Bulk Data entry to be used. Only displacements, stresses, and strains of p-version elements with identification numbers that appear on the OUTPUT Bulk Data entry with SID=n will be output. (Integer>0)

Remarks:

1. DATAREC is processed only when an adaptive analysis is requested.
2. Only one command per subcase is allowed.
3. This information is used only for output control and does not in anyway affect the analysis.
4. Displacements, stresses, and strains will be calculated and printed only for p-version elements in OUTPUT entry. Those elements listed that are not p-version elements will be ignored.
5. The coordinates of the view points (points at which the displacements are calculated and printed) can be printed by using the VUGRID command.

DEFAULT (SOL 700) Controls the Default Setting of the Model

Controls the default setting of the model.

Format:

DEFAULT, OPTION

Example:

DEFAULT,DYNA

Description:

OPTION can be DYTRAN (default) or DYNA. The following constants are set depending on the option selected:

Input	DYTRAN	DYNA
PARAM,LSDYNA,ENERGY,HGEN	0	1
PARAM,LSDYNA,ENERGY,SLNTEN	0	1
PARAM,LSDYNA,ENERGY,RYLEN	0	1
PARAM,LSDYNA,RELAX,NRCYCK	0	250
PARAM,LSDYNA, RELAX,DRTOL	0.0	0.001
PARAM,LSDYNA, RELAX,DRFCTR	0.0	0.995
PARAM,LSDYNA, RELAX,DRTERM	0.0	1E20
PARAM,LSDYNA, RELAX,EDTTL	0.0	0.04
PARAM,LSDYNA, OUTPUT,IKEDIT	0	100
PARAM,LSDYNA, OUTPUT,IFLUSH	0	5000
PARAM,LSDYNA, SHELL,WRPANG	0.0	20.
PARAM,LSDYNA, SHELL,ESORT	0	2
PARAM,LSDYNA, SHELL,IRNXX	0	-1
PARAM,LSDYNA, SHELL,THEORY	0	2
PARAM,LSDYNA, SHELL,MITER	0	1
PARAM,LSDYNA, SHELL,BWC	0	2
CONTACT-SOFT	1	0

Input	DYTRAN	DYNA
PARAM,BULKQ	1.0	1.5
PARAM,BULKL	0.0	0.06

DEFORM Element Deformation Static Load

Selects the element deformation set.

Format:

DEFORM=n

Example:

DEFORM=27

Describer	Meaning
n	Set identification number of DEFORM Bulk Data entries. (Integer > 0)

Remarks:

1. DEFORM Bulk Data entries will not be used unless selected by the DEFORM command in the Case Control Section.
2. DEFORM is only applicable in linear statics, inertia relief, differential stiffness, and buckling problems (SOLs 101, 105, 114, and 200), and will produce a fatal message in other solution sequences.
3. The total load applied will be the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM) and constrained displacement loads (SPC, SPCD).
4. Static, thermal, and element deformation loads should have unique identification numbers.
5. In the superelement solution sequences, if this command is used in a cold start, it must also be specified in the restart.

DESGLB Request Design Constraints at the Global Level

Selects the design constraints to be applied at the global level in a design optimization task.

Format:

DESGLB=n

Examples:

DESGLB=10
DESG=25

Describer	Meaning
n	Set identification of a set of DCONSTR or a DCONADD Bulk Data entry identification number. (Integer>0)

Remarks:

1. If used, this command must occur before the first SUBCASE.
2. A DESGLB command is optional and invokes constraints that are to be applied independent of a particular subcase. These constraints could be based on responses that are independent of subcases (e.g., WEIGHT or VOLUME).
3. The DESGLB command can be used to invoke constraints that are not a function of DRESP1 entries; e.g., DRESP2 responses that are not functions of DRESP1 responses are subcase independent.

DESOBJ Design Objective

Selects the DRESP1 or DRESP2 entry to be used as the design objective.

Format:

$$\text{DESOBJ}\left[\left(\begin{matrix}\text{MAX}\\ \text{MIN}\end{matrix}\right)\right] = n$$

Examples:

DESOBJ=10
DESO=25

Describer	Meaning
MIN	Specifies that the objective is to be minimized.
MAX	Specifies that the objective is to be maximized.
n	Set identification of a DRESP1 or DRESP2 Bulk Data entry. (Integer>0)

Remarks:

1. A DESOBJ command is required for a design optimization task and is optional for a sensitivity task. No more than one DESOBJ may appear in Case Control.
2. The referenced DRESPi entry must define a scalar response (for example, WEIGHT or VOLUME).
3. If the DESOBJ refers to a global response, such as weight, it should appear above the first subcase. If the DESOBJ refers to a subcase dependent response, such as an element stress, it should appear in that subcase. If it refers to a subcase dependent response, but is inserted above the first subcase, it will select the response from the first subcase for the objective and ignore the responses in subsequent subcases.

DESSUB Design Constraints Request at the Subcase Level

Selects the design constraints to be used in a design optimization task for the current subcase.

Format:

DESSUB=n

Examples:

DESSUB=10
DESS=25

Describer	Meaning
n	Set identification of a set of DCONSTR and / or a DCONADD Bulk Data entry identification number. (Integer≥0)

Remarks:

1. A DESSUB command is required for every subcase for which constraints are to be applied. An exception to this is ‘global constraints’, which are selected by the DESGLB command.
2. All DCONTR and DCONADD Bulk Data entries with the selected set ID will be used.

DESVAR Design Variable Selection

Selects a set of DESVAR entries for the design set to be used.

Format:

$$\text{DESVAR} = \begin{bmatrix} \text{ALL} \\ n \end{bmatrix}$$

Example:

DESVAR=10

Describer	Meaning
n	Set identification of a previously appearing SET command. (Integer>0). Only DESVAR's with IDs that appear on this SET command will be used in the SOL 200 design task.

Remarks:

1. Only one DESVAR command may appear in the Case Control Section and should appear above all SUBCASE commands.
2. The DESVAR command is optional. If it is absent, all DESVAR Bulk Data entries will be used.

DISPLACEMENT Displacement Output Request

Requests the form and type of displacement or pressure vector output. Note: PRESSURE and VECTOR are equivalent commands.

Format:

$$\text{DISPLACEMENT}\left[\left(\begin{bmatrix}\text{SORT1}\\\text{SORT2}\end{bmatrix},\left[\begin{matrix}\text{PRINT, PUNCH}\\\text{PLOT}\end{matrix}\right],\left[\begin{matrix}\text{REAL or IMAG}\\\text{PHASE}\end{matrix}\right]\left[\begin{matrix}\text{PSDF, ATOC, CRMS}\\\text{RALL}\end{matrix}\right],\right.\right.$$

$$\left.\left.\begin{matrix}\text{RPRINT}\\\text{ORPRINT}\end{matrix},\text{RPUNCH}\right],[\text{CID}],\left[\frac{\text{TM}=f}{\text{T1}=f,\text{T2}=f,\text{T3}=f}\right],\left[\frac{\text{RM}=f}{\text{R1}=f,\text{R2}=f,\text{R3}=f}\right]\right)\right]=\left\{\begin{matrix}\text{ALL}\\\text{n}\\\text{NONE}\end{matrix}\right\}$$

Examples:

DISPLACEMENT=5
DISPLACEMENTS(REAL)=ALL
DISPLACEMENT(SORT2, PUNCH, REAL)=ALL
DISPLACEMENT(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
DISPLACEMENT(PRINT, RALL, NORPRINT)=ALL
DISP (T1=1.-3, T3=1.-2) = ALL
DISP (TM=1.-3, PRINT,PLOT) = ALL
DISP (TM=1.-3,PRINT,PLOT,SORT2) = 20

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of load, frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, displacement data.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
PSDF	Requests the power spectral density function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark **8.**
ATOC	Requests the autocorrelation function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark **8.**
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark **8.**
RALL	Request all of PSDF, ATOC and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control. See Remark **8.**
RPRINT	Writes random analysis results in the print file (Default)
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output file, F06 file.
TM	Translational Magnitude Filter
T1, T2, T3	Translational Component Filters
RM	Rotational Magnitude Filters
R1, R2, R3	Rotational Component Filters
F	Filter value (Real > 0.0)
ALL	Displacements for all points will be output.
n	Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	Displacement for no points will be output.

Remarks:

1. Both PRINT and PUNCH may be requested.
2. The defaults for SORT1 and SORT2 depend on the type of analysis:
   - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis.
   - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.
   - XY plot requests forces SORT2 and overrides SORT1 requests!
3. VECTOR and PRESSURE are alternate forms and are entirely equivalent to DISPLACEMENT.
4. DISPLACEMENT=NONE overrides an overall output request.
5. The PLOT option is used when curve plots are desired in the magnitude/phase representation and no printer output request is present for magnitude/phase representation.
6. The units of translation are the same as the units of length of the model. Rotations are in units of radians.
7. Displacement results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
8. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. The results can be either printed in the .f06 file or punched in the punch file, or output in both files.
9. Note that the CID keyword affects only grid point related output, such as DISPlacement, VELOcity, ACCEleration, OLOAD, SPCForce and MPCForce. In addition, CID keyword needs to appear only once in a grid related output request anywhere in the Case Control Section to turn on the printing algorithm.

10. Displacement components may be selected to control filtering to reduce the amount of output produced. When magnitudes are selected, the component values are ignored. Only a single positive value for f can be supplied and comparisons are performed in the global reference frame. Comparisons are performed after the SET intersection is performed against the domain. Selection of this option does not effect the MAXMIN(GRID) operations. Scalar comparisons are performed using the minimum of all supplied values for the filters. Complex vector magnitudes follow a derivation using a deterministic interpretation for frequency response.
11. When using filters the compound usage of the verbs PRINT, PLOT is allowed. The entries in the printed output are the entries that exceed any threshold, while the remaining entries within the SET are marked as plot to allow for post-processing operations. When SORT2 is selected, then print, plot must be used to allow for table transpose operations to occur. When any entry in the SORT2 format is above the threshold, all values for time or frequency will be printed for the grid.

DIVERG Static Aeroelastic Divergence Request

Selects the divergence parameters in a static aeroelastic divergence problem.

Format:

DIVERG=n

Example:

DIVERG=70

Describer	Meaning
n	Set identification of a DIVERG Bulk Data entry. (Integer>0)

Remark:

1. Static aeroelastic divergence analysis can be performed in SOLs 144 and 200.

DLOAD Dynamic Load Set Selection

Selects a dynamic load or an acoustic source to be applied in a transient or frequency response problem.

Format:

DLOAD=n

Example:

DLOAD=73

Describer	Meaning
n	Set identification of a DLOAD, RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE Bulk Data entry. (Integer>0)

Remarks:

1. RLOAD1 and RLOAD2 may only be selected in a frequency response problem.
2. TLOAD1 and TLOAD2 may be selected in a transient or frequency response problem.
3. Either a RLOADi or TLOADi entry (but not both) must be selected in an aeroelastic response problem. If RLOADi is selected, a frequency response is calculated. If TLOADi is selected, then transient response is computed by Fourier transform. When there are only gust loads (GUST entry), the DLOAD selects a TLOADi or RLOADi entry with zero load along with field 3 of the GUST command.
4. The DLOAD command will be ignored if specified for upstream superelements in dynamic analysis. To apply loads to upstream superelements, please see the LOADSET command.

DRSPAN Response Spanning Set Selection

Selects a set of DRESP1 entries for the current subcase that are to be used in a DRESP2 or DRESP3 response that spans subcases.

Format:

DRSPAN=n

Example:

DRSPAN=10

Describer	Meaning
n	Set identification of a previously appearing SET command. (Integer>0)

Remarks:

1. In SOL 200, DRESP2 or DRESP3 entries can invoke DRESP1 responses that span subcases if these DRESP1 responses have been identified using a DRSPAN command that references a SET request that identifies the DRESP1 entries.
2. The DRSPAN request must be at the subcase level while the SET request can be done above the subcase level.
3. DRESP2 or DRESP3 that SPANS subcases cannot reference another DRESP2 and/or DRESP3.
4. If the DRESP2 or DRESP3 that spans subcases is referenced by the DESOBJ command, the first subcase must contain a DRSPAN command while this command is optional for subsequent subcases.
5. If the DRESP2/3 that spans subcases is referenced by the DESGLB command, the DRESPAN commands can be in any of the subcases and need not be in the first subcase.
6. If it recommended that DOPTPRM parameter P2 be set to output the spanned response value(s) and a check be made to see if the value(s) are as expected.

DSAPRT Design Sensitivity Output Parameters

Specifies design sensitivity output parameters.

Format:

$$\text{DSAPRT}\left[\left[\begin{matrix}\text{FORMATTED}\\ \text{UNFORMATTED}\\ \text{NOPRINT}\end{matrix}\right],\left[\begin{matrix}\text{NOEXPORT}\\ \textit{EXPORT}\end{matrix}\right],\text{START} = \text{i}, \text{BY} = \text{j}, \text{END} = \text{k}\right] = \left[\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right]$$

Examples:

```
DSAPRT(FORMATTED,EXPORT)
DSAPRT(FORMATTED,START=FIRST,BY=3,END=LAST)=101
DSAPRT(UNFORMATTED,START=FIRST)
DSAPRT(UNFORMATTED,EXPORT)
DSAPRT(FORMATTED,END=4)=ALL
DSAPRT(UNFORMATTED,END=SENS)=ALL
DSAPRT(NOPRINT, EXPORT)
```

Describer	Meaning
FORMATTED	Output will be presented with headings and labels.
UNFORMATTED	Output will be printed as a matrix print (see description of the MATPRN module in the *MD Nastran 2006 DMAP Programmer's Guide*).
NOPRINT	No output will be printed.
EXPORT	Output will be exported to an external binary file specified by PARAM,IUNIT.
NOEXPORT	Output will not be exported to an external binary file.
START=i	Specifies the first design cycle for output. (Integer>0 or Character: "FIRST" or "LAST"; Default=1 or "FIRST")
BY=j	Specifies the design cycle interval for output. (Integer≥0; Default=0) See Remark 2.
END=k	Specifies the last design cycle for output. (Integer>0 or Character: "FIRST", "LAST", or "SENS"; Default="LAST")

Describer	Meaning
ALL	All retained design responses (defined on DRESP1, DRESP2 and DRESP3 entries) will be output.
n	Set identification of a previously appearing SET command. Only sensitivities of retained responses with identification numbers that appear on this SET command will be output. (Integer>0)

Remarks:

1. Only one DSAPRT may appear in the Case Control Section and it must occur with or above the first SUBCASE command.
2. Sensitivity data will be output at design cycles i, i+j, i+2j, ..., k. Note that the BY=0 default implies no sensitivity analysis at the intermediate design cycles.
3. END=SENS requests design sensitivity analysis, and no optimization will be performed.
4. If both DSAPRT and PARAM,OPTEXIT, 4, -4, or 7 are specified, then DSAPRT overrides PARAM,OPTEXIT, 4, -4, or 7. PARAM,OPTEXIT values and the equivalent DSAPRT commands are as follows:

OPTEXIT	Equivalent DSAPRT Command
4	DSAPRT(UNFORMATTED, END=SENS)
-4	DSAPRT(NOPRINT, EXPORT, END=SENS)
7	DSAPRT(UNFORMATTED, START=LAST)

5. The n and NONE options are not supported for UNFORMATTED output. Only the UNFORMATTED option is supported for EXPORT.
6. PARAM DSZERO can be used to set a threshold for the absolute value of the formatted sensitivity prints.

DSYM Dihedral Symmetry Option in Cyclic Symmetry

Provides for either one or two planes of overall symmetry in DIH-type cyclic symmetry problems.

Format:

$$\text{DSYM} = \left\{ \begin{array}{c} \text{S} \\ \text{A} \\ \text{SS} \\ \text{SA} \\ \text{AS} \\ \text{AA} \end{array} \right\}$$

Example:

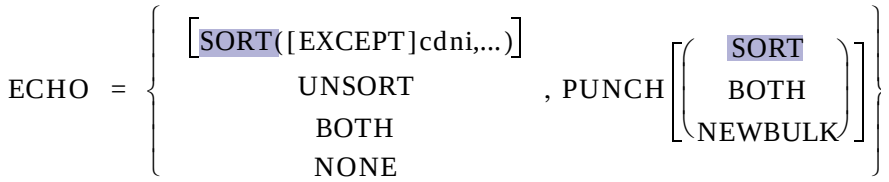
DSYM=AS

Describer	Meaning
S, A	The problem is assumed to be symmetrical (S) or antisymmetrical (A) with respect to the plane containing Side 1 of segment 1.
SS, SA, AS, AA	The problem is assumed to be symmetrical (or antisymmetrical) with respect to the plane containing Side 1 of segment 1 (denoted by the first symbol), and also with respect to a plane perpendicular to Side 1 (denoted by the second symbol).

ECHO Bulk Data Echo Request

Controls echo (i.e., printout) of the Bulk Data.

Format:



Examples:

ECHO=NOSORT
ECHO=BOTH
ECHO=PUNCH, SORT (MAT1, PARAM)
ECHO=SORT (EXCEPT DMI, DMIG)
ECHO=BOTH,PUNCH,FILE

Describer	Meaning
UNSORT	The unsorted Bulk Data will be printed. If SORT is not also specified, the sorted Bulk Data will not be printed.
SORT	The sorted (arranged in alphanumeric order) Bulk Data will be printed.
cdni,...	Bulk Data entry names to be included, or excluded by EXCEPT, in the sorted echo printout. The PUNCH file is not affected by cdni.
EXCEPT	Exclude cdni Bulk Data entries from sorted echo printout. See Remark **6.**
BOTH	Both sorted and unsorted Bulk Data will be printed. This is equivalent to ECHO=SORT, UNSORT.
NONE	Neither sorted nor unsorted Bulk Data will be printed.
PUNCH	The entire Bulk Data will be written to the punch file.
FILE	The entire Bulk Data Echo will be written to the separate file with a default suffix of .BECHO in *.f06 form. User-defined filename must be specified in ASSIGN statement.
NEWBULK	In SOL 200, a complete unsorted Bulk Data file is written to the punch file with updated design model entries.

Remarks:

1. If no ECHO command appears, a sorted Bulk Data will be printed.
2. Comments will appear at the front of the sorted file if ECHO=PUNCH.
3. Portions of the unsorted Bulk Data can be selectively echoed by including the commands ECHOON and ECHOOFF at various places within the Bulk Data. ECHOOFF stops the unsorted echo until an ECHOON command is encountered. Many such pairs of commands may be used. The ECHOON and ECHOOFF command may be used in the Executive and Case Control Sections; however, ECHOOF should not be the first entry and continuation entries are not handled correctly.
4. If the SORT (cdni,...) is specified in a restart in SOLs 101 through 200, then the continuation entries will not be printed.
5. If the SORT (cdni,...) describer is used, then it must appear as the last describer, as in the example above.
6. If EXCEPT is specified then it must be specified before all cdni. All Bulk Data entry types will be listed except those given for cdn1, cdn2, etc. If EXCEPT is not specified, then only those Bulk Data entry types listed under cdn1, cdn2, etc. will be listed.

EDE Element Energy Loss Per Cycle Output Request

Requests the output of the energy loss per cycle in selected elements.

Format:

$$\text{EDE}\left[\left(\left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right]\left[\begin{matrix}\text{AVERAGE}\\ \text{AMPLITUDE}\\ \text{PEAK}\end{matrix}\right][\text{THRESH} = \text{p}]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Examples:

EDE=ALL
EDE(PUNCH, THRESH=.0001)=19

Describer	Meaning
PRINT	Write energies to the print file. (Default)
PUNCH	Write energies to the punch file.
PLOT	Do not write energies to either the punch file or the print file.
AVERAGE	Requests average energy (in frequency response analysis only). (Default)
AMPLITUDE	Requests amplitude of energy (in frequency response analysis only).
PEAK	Requests peak energy (for frequency response analysis only). PEAK is the sum of AVERAGE and AMPLITUDE.
THRESH	Energies for elements having an energy value of less than p% will be suppressed in all output files-print, punch, plot, op2, and xdb. THRESH overrides the value of TINY described in Remark 1. (Default = 0.001)
ALL	Energy for all elements will be computed.
n	Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the EDE command or above all subcases. (Integer > 0)
NONE	Element energy loss will not be output.

Remarks:

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.
4. For frequency response analysis, the energy may be computed in one of three ways as selected by

   AVERAGE:

   $$E_o = \pi\omega(\{u_r\}^T[B_e]\{u_r\} + \{u_i\}^T[B_e]\{u_i\})$$

   AMPLITUDE:

   $$E_a = \pi\omega\sqrt{(\{u_r\}^T[B_e]\{u_r\} - \{u_i\}^T[B_e]\{u_i\})^2 + (2\{u_r\}^T[B_e]\{u_i\})^2}$$

   PEAK:

   $$E_{peak} = E_o + E_a$$

   where:

   E = elemental energy

   $\{u_r\}$ = displacement - real part

   $\{u_i\}$ = displacement - imaginary part

   $[B_e]$ = elemental mass

5. In SOLs 111 and 112, EDE is not available if both PARAM,DDRMM,0 and PARAM,SPARSEDR,NO are specified.
6. Only damping from the viscous dampers (e.g., CVISC, CDAMPi, etc.) are included. Structural damping is not included in the calculation.

EKE Element Kinetic Energy Output Request

Requests the output of the kinetic energy in selected elements.

Format:

$$\text{EKE}\left[\left(\left[\begin{matrix}\text{PRINT}\\ \text{PUNCH}\\ \text{PLOT}\end{matrix}\right]\left[\begin{matrix}\text{AVERAGE}\\ \text{AMPLITUDE}\\ \text{PEAK}\end{matrix}\right][\text{THRESH} = p]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}$$

Examples:

EKE=ALL
EKE(PUNCH, THRESH=.0001)=19

Describer	Meaning
PRINT	Write energies to the print file. (Default)
PUNCH	Write energies to the punch file.
PLOT	Do not write energies to either the punch file or the print file.
AVERAGE	Requests average energy (in frequency response analysis only). (Default)
AMPLITUDE	Requests amplitude of energy (in frequency response analysis only).
PEAK	Requests peak energy (for frequency response analysis only). PEAK is the sum of AVERAGE and AMPLITUDE.
THRESH	Strain energies for elements having a energy value of less than p% will be suppressed in all output files-print, punch, plot, op2, and xdb. THRESH overrides the value of TINY described in Remark 1. (Default = 0.001)
ALL	Strain energy for all elements will be computed.
n	Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the EKE command or above all subcases. (Integer > 0)
NONE	Element kinetic energy will not be output.

Remarks:

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.
4. For frequency response analysis, the energy may be computed in one of three ways as selected by

   AVERAGE:

   $$E_o = \frac{1}{4}(\{v_r\}^T[M_e]\{v_r\} + \{v_i\}^T[M_e]\{v_i\})$$

   AMPLITUDE:

   $$E_a = \frac{1}{4}\sqrt{(\{v_r\}^T[M_e]\{v_r\} - \{v_i\}^T[M_e]\{v_i\})^2 + (2\{v_r\}^T[M_e]\{v_i\})^2}$$

   PEAK:

   $$E_{peak} = E_o + E_a$$

   where:

   E = elemental energy

   $\{v_r\}$ = velocity - real part

   $\{v_i\}$ = velocity - imaginary part

   $[M_e]$ = elemental mass

5. In SOLs 111 and 112, EKE is not available if both PARAM,DDRMM,0 and PARAM,SPARSEDR,NO are specified.

ELSDCON Element Stress Discontinuity Output Request

Requests mesh stress discontinuities based on element stresses.

Format:

$$\text{ELSDCON}\left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Examples:

ELSDCON=ALL
ELSDCON=19

4A CASE

Describer	Meaning
ALL	Stress discontinuity requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) Section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the stress discontinuity output request. (Integer >0)
NONE	No element stress discontinuity output.

Remarks:

1. This output is available in linear static analysis SOLs 101 and 144 only. Output will be presented for each surface or volume as a tabular listing of stress discontinuities for each subcase.
2. Only elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.
3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands. Also, the GPSTRESS command must be present for printed output and the STRFIELD command for postprocessing output using the .xdb file (PARAM,POST,0) for the same surfaces and volumes.

ELSUM Element Summary Output Request

Requests that a summary of element properties grouped by element type and/or element property type are to be printed.

Format:

$$\text{LSUM}([\text{EID, PID, BOTH, EIDSUM, PIDSUM, NSMCONT}]) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

Examples:

ELSUM = 9
ELSUM (PID) = ALL

Describer	Meaning
EID	Element summary output is grouped by element type.
PID	Element summary output is grouped by element property type.
BOTH	Both EID and PID groupings are produced.
EIDSUM	Only a summary of the mass totals for the EID grouping is produced.
PIDSUM	Only a summary of the mass totals for the PID grouping is produced.
NSMCONT	Nonstructural mass contributions from NSM/NSM1/NSML/NSML1 Bulk Data entries are identified.
ALL	Element summary output for all elements
n	Set identification of a previously appearing SET command. Produces output for only those elements whose identification numbers appear in the list of this SET command.
NONE	No element summary output is produced.

Remarks:

1. The ELSUM Case Control command produces a summary of properties for elements. The properties include element-id, material-id, (length or thickness), area, volume, structural mass, non-structural mass, total mass, and the product total mass * WTMASS. Total mass is the sum of structural and non-structural masses.

2. Certain element types produce only partial data. For these element types, no mass data is produced and mass totals will not include any contributions from these element types. Mass data is computed for the following element types: CBAR, CBEAM, CBEND, CHEXA, CMASSi, CONM1, CONM2, CONROD, CPENTA, CQUAD4, CQUAD8, CQUADR, CRAC2D, CRAC3D, CROD, CSHEAR, CTETRA, CTRIA3, CTRIA6, CTRIAR, CTRIAX6 and CTUBE.
3. EIDSUM takes precedence over EID if both are present. Likewise, PIDSUM takes precedence over PID.
4. The ELSUM command is ignored in heat transfer solution sequences.
5. The NSMCONT describer produces various amounts of output depending upon whether the summary option is selected (SUMMARY, PIDSUM or EIDSUM requested). If NSMCONT is combined with PID, a table is produced that identifies the contribution of each NSM type Bulk Data entry to the total element nonstructural mass. If SUMMARY is included with PID and NSMCONT, an additional table is produced that identifies the mass contributions for each property type by property ID.
6. ELSUM output is only available for the PRWT option, not PUNCH or PLOT options used in other commands.

ENDTIME (SOL 700) Specifies Final Analysis Time

Specifies final analysis time for SOL 700.

Format:

ENDTIME = Value

Example:

ENDTIME = 0.01

Describer	Meaning
Value	Time in the applicable units for the model (usually seconds).

ENTHALPY Heat Transfer Enthalpy Output Request

Requests form of enthalpy vector output in transient heat transfer analysis (SOL 159).

Format:

$$\text{ENTHALPY}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right],\ \left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Example:

ENTHALPY=5

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each time.
SORT2	Output will be presented as a tabular listing of time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates but does not print enthalpies.
ALL	Enthalpy for all points will be output.
NONE	Enthalpy for no points will be output.
n	Set identification of previously appearing SET command. Only enthalpies of points with identification numbers that appear on this SET command will be output. (Integer>0)

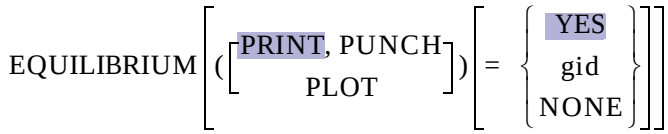
Remark:

1. ENTHALPY=NONE is used to override a previous ENTHALPY=n or ENTHALPY=ALL command.

EQUILIBRIUM Equilibrium Force Output Request

Specifies options for equilibrium force balance output of applied loads, single point constraint forces and forces due to multi-point constraints and rigid elements.

Format:



Examples:

EQUILIBRIUM
EQUILIBRIUM = 501

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, equilibrium force balance data block.
YES	Requests moment summation referenced to origin of basic coordinate system.
gid	Requests moment summation referenced to basic system location specified by the coordinates of grid point gid.
NONE	Equilibrium force balance output will not be generated.

Remarks:

1. The EQUILIBRIUM Case Control command produces a summary of the applied loads, single point forces of constraint (SPC) and multi-point / rigid body element forces of constraint (MPC) as well as a summation of these quantities. In order for the summation to represent all of the forces in the problem, these forces must be available and therefore, the specification of an EQUILIBRIUM command causes the program to automatically compute the SPC and MPC forces. However, if desired, the associated Case Control commands should request output. The single point forces of constraint are

requested by the presence of an SPCFORCE command and the multi-point/RBE constraint forces are requested by an MPCFORCE command. Applied loads are automatically generated by the presence of the LOAD selection command.

2. Results are always output in the basic coordinate system.
3. The EQUILIBRIUM command is applicable to static analysis only and does not produce output if any superelements are present.

ESE Element Strain Energy Output Request

Requests the output of the strain energy in selected elements.

Format:

$$\text{ESE}\left[\left(\left[\begin{matrix}\text{PRINT}\\ \text{PUNCH}\\ \text{PLOT}\end{matrix}\right]\left[\begin{matrix}\text{AVERAGE}\\ \text{AMPLITUDE}\\ \text{PEAK}\end{matrix}\right][\text{THRESH} = p]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ n\end{matrix}\right\}$$

Examples:

ESE=ALL
ESE (PUNCH, THRESH=.0001)=19

Describer	Meaning
PRINT	Write energies to the print file (Default).
PUNCH	Write energies to the punch file.
PLOT	Do not write energies to either the punch file or the print file.
AVERAGE	Requests average energy in frequency response analysis only.
AMPLITUDE	Requests amplitude of energy in frequency response analysis only.
PEAK	Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.
THRESH	Energies for elements having an energy value of less than p% will be suppressed in all output files-print, punch, plot, op2, and xdb. THRESH overrides the value of TINY described in Remark 1. (Default = 0.001)
ALL	Energy for all elements will be computed.
n	Set identification number. energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the ESE command or above all subcases. (Integer >0)
NONE	Element strain energy will not be output.

Remarks:

1. If THRESH = p is not specified then p defaults to the values specified by user parameter TINY.

2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element strain energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.
4. For frequency response analysis, the energy may be computed in one of three ways as selected by

   AVERAGE:

   $$E_o = \frac{1}{4}(\{u_r\}^T[K_e]\{u_r\} + \{u_i\}^T[K_e]\{u_i\})$$

   AMPLITUDE:

   $$E_a = \frac{1}{4}\sqrt{(\{u_r\}^T[K_e]\{u_r\} - \{u_i\}^T[K_e]\{u_i\})^2 + (2\{u_r\}^T[K_e]\{u_i\})^2}$$

   PEAK:

   $$E_{peak} = E_o + E_a$$

   where:

   E = elemental energy

   $\{u_r\}$ = displacement - real part

   $\{u_i\}$ = displacement - imaginary part

   $[K_e]$ = elemental stiffness

5. In SOLs 111 and 112, ESE is not available if both PARAM,DDRMM,0 and PARAM,SPARSEDR,NO are specified.
6. Element data recovery for thermal loads in not currently implemented in dynamics.
7. Element strain energy is available for nonlinear static analysis (SOL 106). All other nonlinear solution sequences do not support element strain energy output.
8. The strain energy for nonlinear elements is calculated by integrating the specific energy rate, the inner product of strain rate and stress, over element volume and time.

$$E = \int_0^t \int_V \dot{\varepsilon}^T \sigma dV d\tau \qquad \text{Eq. 4-10}$$

where:

σ = stress tensor

$\dot{\varepsilon}$ = tensor of the strain rate

V = element volume

t = actual time in the load history

Loads from temperature changes are included in **Eq. 4-10**. If we assume a linear variation of temperatures from subcase to subcase, then the strain energy in **Eq. 4-10**--for the special case of linear material and geometry--becomes

$$E = \frac{1}{2} u^T K_e u - \frac{1}{2} u^T P_{et} \qquad \text{Eq. 4-11}$$

where P_{et} is the element load vector for temperature differences.

For linear elements, the default definition of element strain energy is

$$E = \frac{1}{2} u^T K_e u - u^T P_{et} \qquad \text{Eq. 4-12}$$

where P_{et} is the element load vector for temperature differences and element deformation.

In **Eq. 4-12**, the temperatures are assumed to be constant within a subcase. The default definition of the strain energy for linear elements differs from the definition for nonlinear elements by a factor of 1/2 in the temperature loads. To request the strain energy for linear elements using **Eq. 4-11**, set the parameter XFLAG to 2; the default value for XFLAG is 0, which uses **Eq. 4-12** for the strain energy of linear elements.

EXPORTLD Saves a Load Vector on the Database for Subsequent Reuse

Saves the load vector, for the current subcase, on the database.

Format:

```
EXPORTLD[([LOADID = lid][LOADNAME = Idname])][= {n, ALL}]
```

Examples:

```
EXPO(LOADNAME=LANDGEAR)
EXPORTLD=10
```

Describer	Meaning
LOADID=lid	User assignable value for the LOADID qualifier; used to uniquely identify a saved load vector for later retrieval (using DBLOCATE for example). The default value is the subcase ID.
LOADNAME=Idname	User assignable value (of up to 8 characters) for the LOADNAME qualifier; used to uniquely identify a saved load vector for later retrieval. The default value is blank.
N	Results for grid point components in SET n will be exported.
ALL	Results for ALL grid point components will be exported. Default is ALL.

Remarks:

1. Each load vector is stored individually as one column matrix that is qualified by LOADID and LOADNAME. The combination of both the LOADID and the LOADNAME qualifiers should uniquely identify the load vector to avoid overwriting (or possibly triggering the output twice rule for) an existing load vector.
2. If placed above the SUBCASE entry, then the load vectors for all subcases are saved.
3. The load vectors are qualified by LOADID and LOADNAME for selection using the WHERE clause on those FMS commands that support it.
4. The following table shows the value of the LOADID and LOADNAME qualifiers that are assigned for various EXPORTLD requests.

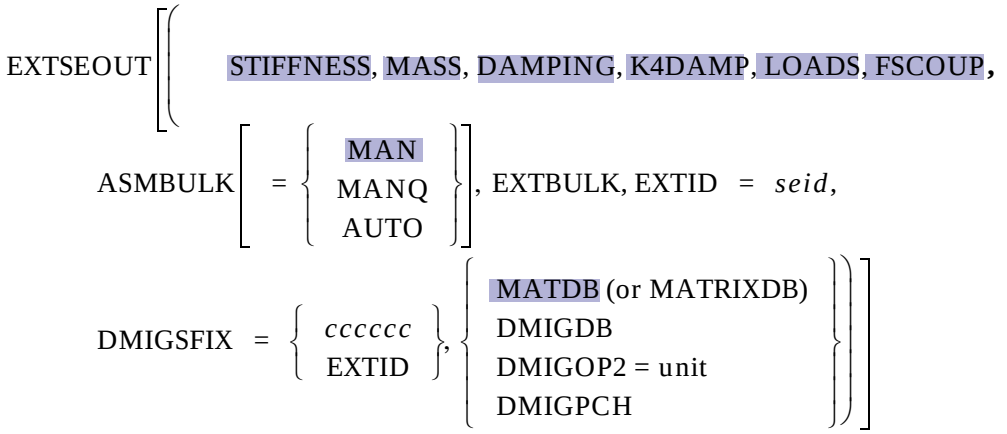
Example	Results
EXPORTLD	LOADID=subcase ID, LOADNAME=' ' - default value applied.
EXPORTLD(LOADID=12)	LOADID=12, LOADNAME=' '
EXPORTLD(LOADID=1,LOADNAME=FORCE12)	LOADID=1, LOADNAME='FORCE12'
EXPORTLD(LOADNAME=ALLCASES)	LOADID=subcase ID, LOADNAME='ALLCASES'

5. The load vector is typically imported into a run using the FMS DBLOCATE command. The imported load is referenced by using its LOADID value on a LOAD Case Control or Bulk Data entry. For example:

```
ASSIGN loads1='run1.MASTER'
DBLOCATE datablk=(EXTLD) WHERE(LOADNAME='ALLCASES'),
CONVERT (LOADID=LOADID+1000) LOGICAL=loads1
...
CEND
LOADS=1001 $ Select external load with LOADID=1001,
imported from previous run.
```

EXTSEOUT External Superelement Creation Specification

Format:



Example(s):

EXTSEOUT(ASMBULK,EXTID=200)
EXTSEOUT(ASMBULK,EXTBULK,EXTID=100)
EXTSEOUT(ASMBULK=AUTO,EXTBULK,EXTID=100)
EXTSEOUT(ASMBULK=MANQ,EXTID=10,DMIGDB)
EXTSEOUT(ASMBULK,EXTID=100,DMIGOP2=26)
EXTSEOUT(ASMBULK,EXTID=100,DMIGPCH)
EXTSEOUT(ASMBULK,EXTID=100,DMIGSFIX=XSE100,DMIGPCH)
EXTSEOUT(ASMBULK,EXTID=200,DMIGSFIX=EXTID,DMIGPCH)

(See also Remarks **17.** and **18.**)

Describer	Meaning
STIFFNESS	Store the boundary stiffness matrix. See Remarks **1.** and **2.**
MASS	Store the boundary mass matrix. See Remark **1.**
DAMPING	Store the boundary viscous damping matrix. See Remarks **1.** and **2.**
K4DAMP	Store the boundary structural damping matrix. See Remark **1.**
LOADS	Store the boundary static loads matrix. See Remark **1.**
FSCOUP	Store the boundary fluid-structure coupling matrix. See Remark **1.**

Describer	Meaning
ASMBULK or ASMBULK = MAN	Generate Bulk Data entries for use in a subsequent superelement assembly process and store them on the assembly punch file (.asm). This data, which is used in the Main Bulk Data portion of a subsequent assembly job, includes an SEBULK entry that specifies MANUAL as the method for searching boundary points and an SECONCT entry that defines connections for boundary grid and scalar points. See Remarks **3.**, **4.** and **15.**
ASMBULK = MANQ	Similar to the ASMBULK = MAN option except that the generated SECONCT entry defines connections not only for boundary grid and scalar points, but also for Q-set points. This allows the user to have control over the Q-set points of the external superelement in the subsequent assembly job. See Remarks **3.**, **5.** and **15.**
ASMBULK = AUTO	Generate Bulk Data for use in a subsequent superelement assembly process and store them on the assembly punch file (.asm). This data, which is to be used in the Main Bulk Data portion of a subsequent assembly job, includes an SEBULK entry that specifies AUTO as the method for searching boundary points and an SECONCT entry that defines connections for boundary scalar points. See Remarks **3.**, **6.** and **15.**
EXTBULK	Generate Bulk Data entries related to the external superelement and store them on the standard punch file (.pch). This data is used in the BEGIN SUPER portion of the Bulk Data of a subsequent assembly job. EXTBULK is *ignored* if DMIGPCH (see description below) is specified. See Remarks **3.**, **7.**, **8.** and **15.**). It should be emphasized here that the EXTBULK keyword is *not required* and is provided solely for the sake of user convenience. In the absence of EXTBULK and the associated output on the standard punch file resulting from it, the subsequent assembly job will retrieve the required data for the external superelement from the medium on which the boundary matrices are stored.

Describer	Meaning
EXTID=*seid*	*seid* (integer > 0) is the superelement ID to be used in the SEBULK and SECONCT (if applicable) Bulk Data entries stored on the assembly punch file (.asm) if ASMBULK is specified and in the BEGIN SUPER Bulk Data entry stored on the standard punch file (.pch) if EXTBULK or DMIGPCH is specified. See Remarks **3.** through **9.**, **13.** and **17.**
DMIGSFIX = *cccccc*	*cccccc* is the suffix (up to six characters) that is to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH option is specified. See Remarks **10.** through **13.** See also Example 3 in Remark **18.**
DMIGSFIX = EXTID	The *seid* defined by the EXTID keyword is the suffix that is to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH option is specified. See Remarks **3.**, **10.** through **13.** See also Example 3 in Remark **18.**
MATDB (or MATRIXDB) (Default)	Store the boundary matrices and other information on the database. See Example 1 in Remark **18.**
DMIGDB	Similar to MATDB (or MATRIXDB) except that the boundary matrices are stored as DMIG Bulk Data entries on the database. See Example 1 in Remark **18.**
DMIGOP2 = *unit*	Store the boundary matrices as DMIG Bulk Data entries on an OP2 file whose Fortran unit number is given by *unit* (integer > 0). See Remark **16.** See also Example 2 in Remark **18.**
DMIGPCH	Store the boundary matrices as DMIG Bulk Data entries on the standard punch file (.pch). See Remarks **3.** and **7.** through **15.** See also Example 3 in Remark **18.**

Remarks:

1. If none of the describers STIFFNESS through FSCOUP is specified, then all matrices are stored.
2. STIFFNESS and DAMPING may be abbreviated to STIF and DAMP, respectively.
3. EXTID with an *seid* value must be specified if one or more of ASMBULK, EXTBULK or DMIGPCH are specified.

If the DMIGSFIX = EXTID form is employed along with the DMIGPCH specification, the value *seid* may not exceed 999999 since this value becomes part of the names given to the DMIG matrices generated on the standard punch file (.pch). See Remark **13.** and Example 3 in Remark **18.**

If PARAM,AUTOQSET,YES is specified to automatically generate the Q-set degrees of freedom (DOFs) (generalized coordinates), the value *seid* may not exceed 999 since this value becomes part of the automatically generated IDs of the SPOINTs representing the Q-set DOFs. See explanation in Item c under Remark **17.**

4. If ASMBULK is specified, the following Bulk Data entries are generated and stored on the assembly punch file (.asm):

   SEBULK *seid* … (specifies MANUAL as the method for searching boundary points)
   SECONCT *seid* …(defines connections for boundary grid and scalar points)
   GRID entries for boundary grid points
   SPOINT entries for boundary scalar points
   CORD2x entries associated with the boundary GRID entries

5. If ASMBULK = MANQ is specified, the following Bulk Data entries are generated and stored on the assembly punch file (.asm):

   SEBULK *seid* ... (specifies MANUAL as the method for searching boundary points)
   SECONCT *seid* ... (defines connections for boundary grid and scalar points as well as for Q-set points)
   GRID entries for boundary grid points
   SPOINT entries for boundary scalar points as well as for Q-set points
   CORD2x entries associated with the boundary GRID entries

6. If ASMBULK = AUTO is specified, the following Bulk Data entries are generated and stored on the assembly punch file (.asm):

   SEBULK *seid* … (specifies AUTO as the method for searching boundary points)
   SECONCT *seid* … (defines connections for boundary scalar points)
   SPOINT entries for boundary scalar points

7. If DMIGPCH is specified, then EXTBULK is ignored even if it is specified.

8. If EXTBULK is specified without DMIGPCH, the following Bulk Data entries are generated and stored on the standard punch file (.pch):

BEGIN BULK *seid*
GRID entries for boundary points
GRID entries for interior points referenced by PLOTEL entries
SPOINT entries for boundary scalar points as well as for Q-set points
CORD2x entries associated with the above GRID entries
EXTRN
ASET
QSET / QSET1
PLOTEL

9. If DMIGPCH is specified, the following Bulk Data entries are generated and stored on the standard punch file (.pch):

   BEGIN SUPER *seid*
   GRID entries for boundary points
   SPOINT entries for boundary scalar points as well as for Q-set points
   CORD2x entries associated with the boundary GRID entries
   ASET / ASET1
   PLOTEL entries referencing the boundary GRID entries
   DMIG entries for the requested boundary matrices

10. The DMIGSFIX keyword is ignored if DMIGPCH is not specified.

11. If DMIGPCH is specified without the DMIGSFIX keyword, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

    KAAX (boundary stiffness matrix)
    MAAX (boundary mass matrix)
    BAAX (boundary viscous damping matrix)
    K4AAX (boundary structural damping matrix)
    PAX (boundary load matrix)
    AAX (boundary fluid-structure coupling matrix)
    See Example 3 in Remark **18.**

12. If the DMIGFIX = *cccccc* form is employed along with the DMIGPCH specification, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

K*cccccc* (boundary stiffness matrix)
M*cccccc* (boundary mass matrix)
B*cccccc* (boundary viscous damping matrix)
K4*cccccc* (boundary structural damping matrix)
P*cccccc* (boundary load matrix)
A*cccccc* (boundary fluid-structure coupling matrix)
See Example 3 in Remark **18.**

13. If the DMIGSFIX = EXTID form is employed along with the DMIGPCH specification, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

    K*seid* (boundary stiffness matrix)
    M*seid* (boundary mass matrix)
    B*seid* (boundary viscous damping matrix)
    K4*seid* (boundary structural damping matrix)
    P*seid* (boundary load matrix)
    A*seid* (boundary fluid-structure coupling matrix)
    where *seid* is the superelement ID specified by the EXTID keyword.
    See Example 3 in Remark **18.**

14. If the DMIGPCH option is specified, the boundary DMIG matrices generated and stored on the standard punch file (.pch) may not be as accurate as the boundary matrices resulting from the other options (MATDB/MATRIXDB or DMIGOP2). Accordingly, this may result in decreased accuracy from the subsequent assembly job utilizing these DMIG matrices.

15. The punch output resulting from EXTSEOUT usage is determined by ASMBULK, EXTBULK and DMIGPCH as follows:

    - No ASMBULK, EXTBULK or DMIGPCH

      No punch output

    - ASMBULK, but no EXTBULK or DMIGPCH

      Punch output is generated and stored on the assembly punch file (.asm) as indicated in Remarks **4.** through **6.**

    - No ASMBULK, but EXTBULK and/or DMIGPCH

      Punch output is generated and stored on the standard punch file (.pch) as indicated in Remarks **8.** or **9.** (as appropriate).

    - ASMBULK and EXTBULK and/or DMIGPCH

Punch output consists of two distinct and separate parts. One part is generated and stored on the assembly punch file (.asm) as indicated in Remarks **4.** through **6.** The other part is generated and stored on the standard punch file (.pch) as indicated in Remarks **8.** or **9.** (as appropriate).

16. If DMIGOP2 = *unit* is specified, an appropriate ASSIGN OUTPUT2 statement must be present in the File Management Section (MFS) for the *unit*. See Example 2 in Remark **18.**

17. The creation of an external superelement using EXTSEOUT involves running a non-superelement MD Nastran job, with the following additional data:

    a. The data for the creation of the external superelement is specified by the EXTSEOUT command.

    b. The boundary points of the external superelement are specified by ASET / ASET1 Bulk Data entries.

    c. If the external superelement creation involves component mode reduction, then Q-set degrees of freedom (DOFs) (generalized coordinates) must be defined. This can be done either by automatically defining them via PARAM,AUTOQSET,YES or by explicitly specifying them via QSET / QSET1 Bulk Data entries. Note that for the latter case of explicit definition, only SPOINTs may be specified as Q-set points; no grid points may be specified.

    If PARAM,AUTOQSET,YES is specified, SPOINTs are automatically generated internally to represent the Q-set degrees of freedom (DOFs). The IDs of these SPOINTs are of the form 9*sssnnnn* where *sss* is the superelement ID *seid* specified by the EXTID keyword and *nnnn* is a mode number. Both *sss* and *nnnn* will have leading zeros inserted in them to ensure that *sss* is a three-digit number and *nnnn* is a four-digit number.

    Thus, for example, the Q-set DOF corresponding to the 8^{th} mode of superelement ID 5 would be represented by an SPOINT with an automatically generated ID of 90050008 while the Q-set DOF corresponding to the 50^{th} mode of superelement ID 25 would be represented by an SPOINT with an automatically generated ID of 90250050.

    d. The fixity of the boundary DOFs for the component mode reduction may be specified using the BSET / BSET1 / BNDFIX / BNDFIX1 and CSET / CSET1 / BNDFREE / BNDFREE1 Bulk Data entries. (The default scenario assumes that all boundary DOFs are fixed for component mode reduction.)

e. The output for the external superelement is generated in the assembly process. This output consists of displacements, velocities, accelerations, SPC forces and element stresses and forces. However, in order for this output to be generated in the assembly process, the output requests must be specified in the external superelement creation run. Normally, the only output requests for the external superelement that are honored in the assembly process are those that are specified in the creation run. There is, however, one important exception to this and that is this: the output for the boundary grid and scalar points as well as for all grid points associated with PLOTEL entries can be obtained in the assembly process *even if there is no output request specified for these points in the creation run.*

18. The following examples illustrate details of job setups for the external superelement creation and the subsequent assembly process for various scenarios. These examples assume that there are three external superelement creation jobs, one each for external SE 10 (extse10.dat), SE 20 (extse20.dat) and SE 30 (extse30.dat), followed by an assembly job.

- Example 1. MATDB / MATRIXDB or DMIGDB Option

  External SE Creation Jobs

  - Case Control Requirement for the MATDB / MATRIXDB option:

    SE 10: EXTSEOUT (ASMBULK, EXTID = 10)
    SE 20: EXTSEOUT (ASMBULK, EXTID = 20)
    SE 30: EXTSEOUT (ASMBULK, EXTID = 30)

    The EXTBULK keyword may be specified, but it is not necessary.

  - Case Control Requirement for the DMIGDB option:

    SE 10: EXTSEOUT (ASMBULK, EXTID = 10, DMIGDB)
    SE 20: EXTSEOUT (ASMBULK, EXTID = 20, DMIGDB)
    SE 30: EXTSEOUT (ASMBULK, EXTID = 30, DMIGDB)

    The EXTBULK keyword may be specified, but it is not necessary.

  For both options, scr = no should be specified on the Nastran job command line to ensure that the databases are saved at the end of the jobs.

  Assembly Job

  - File Management Section (FMS) Requirement:

    (Note: All of the data blocks stored on the databases for the external SEs have the same common name of EXTDB.)

```
ASSIGN dbname10='extse10.MASTER'
ASSIGN dbname20='extse20.MASTER'
ASSIGN dbname30='extse30.MASTER'
DBLOCATE DATABLK=(EXTDB) CONVERT(SEID=10)
   LOGICAL=dbname10
DBLOCATE DATABLK=(EXTDB) CONVERT(SEID=20)
   LOGICAL=dbname20
DBLOCATE DATABLK=(EXTDB) CONVERT(SEID=30)
   LOGICAL=dbname30
```

where dbname10, dbname20 and dbname30 are logical file names.

Bulk Data Requirement:

The following INCLUDEs are ***required***. They may be specified anywhere in the Main Bulk Data.

```
INCLUDE 'extse10.asm'
INCLUDE 'extse20.asm'
INCLUDE 'extse30.asm'
```

The following INCLUDEs are ***optional***. They may be specified if the corresponding .pch files exist. If specified, they must be grouped together and specified at the very end of the Main Bulk Data (just before the ENDDATA entry):

```
INCLUDE 'extse10.pch'
INCLUDE 'extse20.pch'
INCLUDE 'extse30.pch'
```

- Example 2. DMIGOP2 Option

  External SE Creation Jobs

  - Case Control Requirement:

    SE 10: EXTSEOUT (ASMBULK, EXTID = 10, DMIGOP2 = 25)
    SE 20: EXTSEOUT (ASMBULK, EXTID = 20, DMIGOP2 = 26)
    SE 30: EXTSEOUT (ASMBULK, EXTID = 30, DMIGOP2 = 27)

    The EXTBULK keyword may be specified, but it is not necessary.

  - File Management Section (FMS) Requirement:

    ```
    ASSIGN OUTPUT2='extse10_op2', UNIT=25
    ASSIGN OUTPUT2='extse20_op2', UNIT=26
    ASSIGN OUTPUT2='extse30_op2', UNIT=27
    ```

scr = yes may be specified on the Nastran command line since there is no need for the databases to be saved at the end of the jobs.

Assembly Job

- File Management Section (FMS) Requirement:

  ASSIGN INPUTT2='extse10_op2', UNIT=25
  ASSIGN INPUTT2='extse20_op2', UNIT=26
  ASSIGN INPUTT2='extse30_op2', UNIT=27

- Bulk Data Requirement:

  The following INCLUDEs are ***required***. They may be specified anywhere in the Main Bulk Data:

  INCLUDE 'extse10.asm'
  INCLUDE 'extse20.asm'
  INCLUDE 'extse30.asm'

  The following INCLUDEs are ***optional***. They may be specified if the corresponding .pch files exist. If specified, they must be grouped together and specified at the very end of the Main Bulk Data (just before the ENDDATA entry):

  INCLUDE 'extse10.pch'
  INCLUDE 'extse20.pch'
  INCLUDE 'extse30.pch'

- Example 3. DMIGPCH Option

  External SE Creation Jobs

  - Case Control Requirement:

    SE 10: EXTSEOUT (ASMBULK, EXTID = 10, DMIGPCH)
    SE 20: EXTSEOUT (ASMBULK, EXTID = 20, DMIGPCH,
    DMIGSFIX = XSE20)
    SE 30: EXTSEOUT (ASMBULK, EXTID = 30, DMIGPCH,
    DMIGSFIX = EXTID)

    The EXTBULK keyword is not necessary, but will be ignored even if it is specified.

scr = yes may be specified on the Nastran command line since there is no need for the databases to be saved at the end of the jobs.

Assembly Job

- Case Control Requirement:

  K2GG = (KAAX, KXSE20, K30)
  M2GG = (MAAX, MXSE20, M30)
  B2GG = (BAAX, BXSE20, B30)
  K42GG = (K4AAX, K4XSE20, K430)
  P2G = (PAX, PXSE20, P30)
  A2GG = (AAX, AXSE20, A30)

- Bulk Data Requirement:

  The following INCLUDEs are ***required***. They may be specified anywhere in the Main Bulk Data:

  INCLUDE 'extse10.asm'
  INCLUDE 'extse20.asm'
  INCLUDE 'extse30.asm'

  The following INCLUDEs are also ***required***. They must be grouped together and specified at the very end of the Main Bulk Data (just before the ENDDATA entry):

  INCLUDE 'extse10.pch'
  INCLUDE 'extse20.pch'
  INCLUDE 'extse30.pch'

FBODYLD Free Body Load Output Request

Selects a set of submodels for which free body loads are to be produced and stored.

Format:

$$\text{FBODYLD(LID)} = \left\{ \begin{array}{c} \text{ALL} \\ name1,\ name2,\ name3,\ \ldots \end{array} \right\}$$

Examples:

FBODYLD=ALL
FOBDYLD(100)=WINGLD
FBODYLD(200)=WINGLD,TAILLD

Describer	Meaning
LID	Optional User Defined Load ID. If LID is not supplied the subcase ID is used to define this value.
ALL	Loads will be produced for all FBODYLD Bulk Data entries.
name*i*	Name of a FBODYLD Bulk Data entry defines the submodel to be used for the load.

Remarks:

1. It is recommended, but not required that the LID be unique across subcases.
2. A separate load is created for each name*i*.
3. The name list supplies one or more names separated by comma or blank.
4. Each load is stored individually as a one column matrix that is qualified by LID, name*i*, submodel name, load case label and submodel label (where submodel name is the name on the FBODYSB Bulk Data entry, load case label is the label on the FBODYLD Bulk Data entry and submodel is the label on the FBODYSB Bulk Data entry).

FLSFSEL Control for Fluid-Structure Frequency Selection

Control for fluid-structure frequency selection.

Format:

$$\text{FLSFSEL} \quad \left[\text{LFREQFL} = \left\{\frac{0.0}{fl_1}\right\}\right], \left[\text{HFREQFL} = \left\{\frac{1. + 30}{fl_2}\right\}\right],$$

$$\left[\text{LFREQ} = \left\{\frac{0.0}{fs_1}\right\}\right], \left[\text{HFREQ} = \left\{\frac{1. + 30}{fs_2}\right\}\right],$$

$$\left[\text{LMODESFL} = \left\{\frac{0}{mf}\right\}\right], \left[\text{LMODES} = \left\{\frac{0}{ms}\right\}\right],$$

$$\left[\text{FLUIDSE} = \left\{\frac{0}{\text{seidf}}\right\}\right]$$

Example(s):

FLSFSEL HFREQ = 4. HFREQFL = 9

Describer	Meaning
LFREQFL	Requests in Hertz, lower bound frequency for modal fluid calculations.
fl_1	lower freq range for fluid-real number.
HFREQFL	Requests in Hertz, upper bound frequency for modal fluid calculations.
fl_2	upper freq range for fluid-real number.
LFREQ	Requests in Hertz, lower bound frequency for modal structure calculations.
fs_1	lower freq range for structure-real number.
HFREQ	Requests in Hertz, upper bound frequency for modal structure calculations.
fs_2	upper freq range for structure-real number

Describer	Meaning
LMODESFL	lowest modes for fluid portion of model-0 implies LFREQFL-HFREQFL will determine number of modes.
mf	number of lowest modes to use for fluid portion of model.
LMODES	lowest modes for structure portion of model-0 implies LFREQ-HFREQ will determine number of modes.
ms	number of lowest modes to use for structure portion of model.
FLUIDSE	Defines a specified superelement to be used for fluids only.
seidf	Defines a fluid only superelement.

Remarks:

1. This entry represents a collection of PARAM,name,value entries. See "**Parameters**" on page 659 for detailed description of the parameters collected on this entry. The value of any of these parameters may be given as either the character value given in this description or the numeric value given under the parameter description of this guide.
2. If LMODES (or LMODESFL)=0, the retained modes are determined by the parameters LFREQ and HFREQ (or LFREQFL and HFREQFL).

FLSPOUT Control for Fluid-Structure Mode Participation Output

Control for fluid-structure mode participation output.

Format:

$$\text{FLSPOUT}\quad \left[\text{FLUIDMP} = \left\{\begin{matrix}\underline{\text{ALL}}\\ n_{\text{modes}}\\ \text{NONE}\end{matrix}\right\}\right], \left[\text{GRIDFMP} = \left\{setf_{\underline{\text{ALL}}\ \text{participations}}\right\}\right]$$

$$\left[\text{OUTFMP} = \left\{\begin{matrix}\underline{\text{ALL}}\\ p_{\text{highest}}\\ \text{NOPRINT}\end{matrix}\right\}\right], \left[\text{FEPS} = \left\{\frac{1.-11}{epsf}\right\}\right],$$

$$\left[\text{ARF} = \left\{\frac{0.95}{arf\_v}\right\}\right],$$

$$\left[\text{STRUCTMP} = \left\{\begin{matrix}\text{ALL}\\ m_{\text{modes}}\\ \underline{\text{NONE}}\end{matrix}\right\}\right], \left[\text{OUTSMP} = \left\{\begin{matrix}\underline{\text{ALL}}\\ q_{\text{highest}}\\ \text{NOPRINT}\end{matrix}\right\}\right]$$

$$\left[\text{PANELMP} = \left\{\begin{matrix}\underline{\text{ALL}}\\ setp_{\text{participations}}\\ \text{NONE}\end{matrix}\right\}\right], \left[\text{GRIDMP} = \left\{\begin{matrix}\text{ALL}\\ setg_{\text{participations}}\\ \underline{\text{NONE}}\end{matrix}\right\}\right]$$

$$\left[\text{SEPS} = \left\{\frac{1.-11}{epss}\right\}\right], \left[\text{ARS} = \left\{\frac{0.95}{ars\_v}\right\}\right]$$

$$\left[\text{PSORT} = \left(\left\{\begin{matrix}\underline{\text{ABSOLUTE}}\\ \text{REAL}\\ \text{IMAGINARY}\end{matrix}\right\}, \left\{\frac{\underline{\text{DESCENDING}}}{\text{ASCENDING}}\right\}\right)\right], \left[\text{O2E} = \left\{\frac{\text{YES}}{\underline{NO}}\right\}\right]$$

Example(s):

```
SET 23     =  ROOF, DRIVERSD
SET 211    =  1023, 4069, 56790
FLSPOUT    FLUIDMP = 30      STRUCTMP = 40,  OUTSMP = 30,
           PANELMP = 23      GRIDMP = 211
```

Describer	Meaning
FLUIDMP	Requests fluid participation calculation of fluid response on selected fluid points.
ALL	Requests that all the fluid modes extracted be used.
n	Requests that up to the first n fluid modes be used.
NONE	Requests no participation calculation.
GRIDFMP	Requests inclusion or exclusion of specific fluid grids to be used in all the requested types of participation calculations. These are also the fluid grids that can be referred to on plot and op2 tables.
ALL	Requests inclusion in all the requested types of the participation calculations of all fluid points.
setf	Case Control set id listing a selected set of fluid grids to be used in all the requested types of participation calculations.
OUTFMP	Requests what fluid FLUIDMP participation factors be output for print.
ALL	Requests all FLUID FLUIDMP participation factors be output for print.
p	Requests the p highest FLUIDMP participation factors be output.
NOPRINT	Produce tables for plotting but do not print any results.
FEPS	Filter threshold for fluid participation.
epsf	threshold value.
ARF	Acceptance ratio for fluid participation.
arf_v	Fluid participation values < *arf_v** max_value in a column of the output matrix will be set to zero.
STRUCTMP	Requests structural, load, and panel participation calculations on the selected fluid points. FLUIDMP must be specified for this command to become active.
ALL	Requests that all the structural modes extracted be used.
m	Requests that up to the first m structural modes be used.
NONE	Requests no participation calculation.
OUTSMP	Requests what structural STRUCTMP participation factors be output for print.
ALL	Request all STRUCTMP participation factors be output.

Describer	Meaning
q	Requests the q highest STRUCTMP participation factors be output.
NOPRINT	Produce tables for plotting but do not print any results.
PANELMP	Requests inclusion or exclusion of panel participation calculations on the selected fluid points. FLUIDMP and STRUCTMP must both be specified for this command to become active.
ALL	Requests all panels defined be included in the participation calculations on the selected fluid points.
setp	Case Control set id listing selected panels for panel participation calculations on the selected fluid points. the set consists of the character names of the panels (new V2001)
NONE	Requests exclusion from the participation calculations.
GRIDMP	Requests inclusion or exclusion of a structural panel grid participation calculation on the selected fluid points. FLUIDMP and STRUCTMP must both be specified for this command to become active.
ALL	Requests for panels selected that each and every individual panel grid be included as a separate calculation in the participation calculations on selected fluid points.
setg	Case Control set id listing structural panel grids for grid mode participation on the selected fluid points.
NONE	Requests exclusion from the participation calculations.
SEPS	Filter threshold for structure participation.
epss	Threshold value.
ARS	Acceptance ration for structure related fluid participation.
ars_v	Structure fluid participation values < arf_v* max_value in a column of the output matrix will be set to zero.
PSORT	Requests type of sort
O2E	Controls generation of tables of mode participation vs natural frequency for excitation frequencies. These tables are accessible in XYPLOT.

Remarks:

1. This entry represents a collection of PARAM,name,value entries and must appear above subcase level. See "**Parameters**" on page 659 for detailed descriptions of the parameters collected on this entry. The value of any of these parameters may be given as either the character value given in this description or the numeric value given under the parameter description in this guide.
2. If *n*, *m*, *p*, or *q* are greater than the number computed, MD Nastran will invoke the ALL option for the current value.
3. PSORT values must occur in pairs such as (ABSOLUTE,DESCENDING)
4. The underlined item in the { } braces give the value of the keyword if the keyword and its describers are omitted from this entry. E.g., if FLUIDMP is omitted from the FLSPOUT entry then no fluid mode participation will be computed (unless a PARAM,FLUIDMP,value explicitly appears in a SUBCASE or "**Bulk Data Entries**" on page 945).

FLSTCNT Miscellaneous Fluid-Structure Control Parameters

Control for fluid-structure symmetry and force requests.

Format:

$$\text{FLSTCNT} \quad \left[\text{ACSYM} = \left\{\begin{matrix}\underline{\text{YES}}\\ \text{NO}\end{matrix}\right\}\right], \left[\text{ACOUT} = \left\{\begin{matrix}\underline{\text{PEAK}}\\ \text{RMS}\end{matrix}\right\}\right]$$

$$\left[\text{PREFDB} = \left\{\begin{matrix}\underline{1.0}\\ prp\end{matrix}\right\}\right], \left[\text{ASCOUP} = \left\{\begin{matrix}\underline{\text{YES}}\\ \text{NO}\end{matrix}\right\}\right]$$

$$\left[\text{SKINOUT} = \left\{\begin{matrix}\underline{\text{NONE}}\\ \text{PUNCH}\\ \text{PRINT}\\ \text{ALL}\end{matrix}\right\}\right]$$

Example(s):

FLSTCNT ACSYM = *YES* ACOUT = *RMS*

Describer	Meaning
ACSYM	Requests symmetric or nonsymmetric solution for fluid-structure analysis.
YES	Requests symmetrized coupled fluid-structure analysis.
NO	Requests no symmetric coupled fluid-structure analysis.
ACOUT	Requests peak or rms for output to be used with the FORCE request.
PEAK	Requests peak value output to be used with the FORCE request.
RMS	Requests rms value output to be used with the FORCE request.
PREFDB	Specifies the peak reference pressure.
prp	Value for the peak reference pressure.
ASCOUP	Request a coupled or non-coupled fluid-structure analysis.
YES	Request a coupled fluid-structure analysis.
NO	Request a non-coupled fluid-structure analysis.
SKINOUT	Request that sets of grid and element lists be output for both the fluid and structure at the fluid-structure interface.

Describer	Meaning
NONE	Requests no output of sets.
PUNCH	Requests set output to .pch only.
PRINT	Requests set output to .f06 only.
ALL	Requests set output to both .pch and f06.

Remarks:

1. This entry represents a collection of PARAM,name,value entries. See “**Parameters**” on page 659 for detailed descriptions of the parameters collected on this entry. The value of any of these parameters may be given as either the character value given in this description or the numeric value given under the parameter description in this guide.

FLUX Heat Transfer Gradient and Flux Output Request

Requests the form and type of gradient and flux output in heat transfer analysis.

Format:

$$\text{FLUX}\left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Examples:

FLUX=ALL
FLUX(PUNCH,PRINT)=17
FLUX=25

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	The output will be sent to the plot file.
ALL	Flux for all elements will be output.
NONE	Flux for no elements will be output.
n	Set identification of a previously appearing SET command. Only fluxes of elements with identification numbers that appear on this SET command will be output. (Integer>0)

Remarks:

1. FLUX=ALL in SOL 159 may produce excessive output.
2. FLUX=NONE overrides an overall request.

FMETHOD Flutter Analysis Method Parameter Selection

Selects the parameters to be used by the aerodynamic flutter analysis.

Format:

FMETHOD=n

Example:

FMETHOD=72

Describer	Meaning
n	Set identification number of a FLUTTER Bulk Data entry. (Integer>0)

Remarks:

1. An FMETHOD command is required for flutter analysis.
2. A CMETHOD command is also required for the K-method of flutter analysis.
3. If this entry is being used in SOL 200 in conjunction with flutter design conditions, the METHOD selected on the FLUTTER Bulk Data entry must be “PK” or “PKNL”.

FORCE Element Force Output or Particle Velocity Request

Requests the form and type of element force output or particle velocity output in coupled fluid-structural analysis. Note: ELFORCE is an equivalent command.

Format:

$$\text{FORCE}\left[\left(\left[\begin{matrix}\text{SORT1}\\\text{SORT2}\end{matrix}\right],\left[\begin{matrix}\text{PRINT, PUNCH}\\\text{PLOT}\end{matrix}\right],\left[\begin{matrix}\text{REAL or IMAG}\\\text{PHASE}\end{matrix}\right],\left[\begin{matrix}\text{CENTER}\\\text{CORNER or BILIN}\\\text{SGAGE}\\\text{CUBIC}\end{matrix}\right]\right.\right.$$

$$\left.\left.\left[\begin{matrix}\text{PSDF, ATOC, CRMS}\\\text{or RALL}\end{matrix}\right],\left[\begin{matrix}\text{RPRINT}\\\text{NORPRINT}\end{matrix}\text{, RPUNCH}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\\text{n}\\\text{NONE}\end{matrix}\right\}$$

Examples:

```
FORCE=ALL
FORCE(REAL, PUNCH, PRINT)=17
FORCE=25
FORCE(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
FORCE(PRINT, RALL, NORPRINT)=ALL
```

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element type.
PLOT	Generates force output for requested set but no printed output.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RALL	Requests all of PSDF, ATOC and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
CENTER	Output CQUAD4, QUADR, and TRIAR element forces at the center only. The default for QUAD4 is CENTER. The default for QUADR and TRIAR is CORNER.
CORNER or BILIN	Output CQUAD4, QUADR, and TRIAR element forces at the center and grid points using strain gage approach with bilinear extrapolation.
SGAGE	Output CQUAD4 element forces at center and grid points using strain gage approach.
CUBIC	Output CQUAD4 element forces at center and grid points using cubic bending correction.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
ALL	Forces for all elements will be output.
n	Set identification of a previously appearing SET command. Only forces of elements with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	Forces for no elements will be output.

Remarks:

1. ALL should not be used in a transient problem.
2. See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2.
3. ELFORCE is an alternate form and is entirely equivalent to FORCE.
4. FORCE=NONE overrides an overall request.
5. If PARAM,SPARSEDR,NO is specified, then to request force output on damping elements in modal frequency response analysis (e.g., SOL 111), the mode displacement method (PARAM,DDRMM,-1) must be selected.

   Force output on damping elements is not available in transient response analysis.
6. In nonlinear transient analysis, this request is ignored for nonlinear elements.
7. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. Consequently, options specified in subcases other than the first subcase will be ignored.
   - If the STRESS command is specified in the first subcase then the option on the STRESS command is used in all subcases with STRESS, STRAIN, and FORCE commands.
   - If the STRAIN command and no STRESS command is specified in the first subcase, then the option on the STRAIN command is used in all subcases containing STRESS, STRAIN, and FORCE commands.
   - If the FORCE command and no STRESS or STRAIN commands is specified in the first subcase, then the option on the FORCE command is used in all subcases containing STRESS, STRAIN, and FORCE commands.
   - If STRESS, STRAIN, and FORCE commands are not specified in the first subcase, then the CENTER option is used in all subcases containing STRESS, STRAIN, and FORCE commands.
8. The option of PSDF, ATOC, CRMS and RALL, or any combination of them, can be selected for random analysis. The results can be either printed in .f06 file or punched in the punch file, or output in both files.

FREQUENCY Frequency Set Selection

Selects the set of forcing frequencies to be solved in frequency response problems.

Format:

FREQUENCY=n

Example:

FREQUENCY=17

Describer	Meaning
n	Set identification number of FREQ, FREQ1, FREQ2, FREQ3, FREQ4, and FREQ5 Bulk Data entries. (Integer > 0)

Remarks:

1. A frequency set selection is required for a frequency response problem.
2. A frequency set selection is required for transient response by Fourier methods (SOLution 146).
3. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$\left|f_N - f_{N-1}\right| < \text{DFREQ} \cdot \left|f_{MAX} - f_{MIN}\right|$$

where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

GPFORCE Grid Point Force Output Request

Requests grid point force balance at selected grid points.

Format:

$$\text{GPFORCE}\left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\end{matrix}\right\}$$

Examples:

GPFORCE=ALL
GPFORCE=17

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
ALL	Grid point force balance for all grid points will be output.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the grid point force balance output. (Integer >0)

Remarks:

1. The printing of the grid point forces will be suppressed if PARAM,NOGPF,-1 appears in the Bulk Data.
2. The Bulk Data entry PARAM,NOELOF,+1 will cause the output of the grid point forces to be aligned with the edges of the two-dimensional elements. The default value of -1 will suppress this output. See Remark **4.**
3. The Bulk Data entry PARAM,NOELOP,+1 will cause the output of the sum of the forces parallel to the edges of adjacent elements. The default value of -1 will suppress this output. See Remark **4.**
4. The output of grid point forces aligned with the edges of elements is available for the following elements:

   CBAR
   CROD
   CBEAM
   CSHEAR

CONROD
CTRIA3
CQUAD4
CTUBE

The positive direction for grid point forces aligned with the edges of elements is from the reference point to the load point as indicated on the printed output.

5. Grid point force balance is computed from linear and nonlinear elements, the sum of applied and thermal loads, and MPC and SPC forces. Effects not accounted for include those from mass elements in dynamic analysis (inertia loads), general elements, DMIG entries, slideline force contributions and boundary loads from upstream superelements. These effects may lead to an apparent lack of equilibrium at the grid point level. The following table summarizes those effects that are considered and those effects that are ignored in the calculation of grid point forces in the global coordinate system:

Contributions Included	Contributions Ignored
Applied Loads	DMI Forces
SPC Forces	Boundary Loads from Upstream Superelements
Element Elastic Forces	
GENEL Forces	
DMIG Referenced by K2GG Case Control command	
Thermal Loads	
MPC and Rigid Element Forces	

6. Only the element elastic forces are included when the grid point forces are aligned with the edges of elements.
7. In inertia relief analysis, the SPCFORCE and applied load output includes both the effect of inertial loads and applied loads.
8. When pressure loads are applied the GPFDR module uses the discrete load vector and does not include any distributed effects.
9. GPFORCE is not available in SOL 111.

10. Grid point force output is available for nonlinear static analysis (SOL 106). Contribution from slideline elements is ignored. All other nonlinear solution sequences do not support grid point force output. PARAM,NOELOF and PARAM,NOELOP are not supported in nonlinear static analysis; therefore, Remarks 2, 3, 4, and 6 do not apply to SOL 106.

GPKE Grid Point Kinetic Energy Output Request

Requests the output of the kinetic energy at selected grid points in normal modes analysis only.

Format:

$$\text{GPKE}\left[\left[\begin{matrix}\text{PRINT}\\ \text{NOPRINT}\end{matrix}\right], (\text{PUNCH}, \text{THRESH} = \text{e})\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Examples:

GPKE=ALL
GPKE (PRINT, PUNCH)=19

Describer	Meaning
PRINT	The printer will be the output medium.
NOPRINT	Generates, but does not print, grid point kinetic energy output.
PUNCH	The punch file will be the output medium.
e	Minimum energy threshold. Only energies above this value will be printed and/or punch.
ALL	Grid point kinetic energy for all grid points will be output.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in output. (Integer > 0)
NONE	Grid point kinetic energy for no points will be output.

Remarks:

1. Grid point kinetic energy is only available for normal modes analysis.
2. Both PRINT and PUNCH may be requested.
3. GPKE=NONE overrides an overall output request.

4. For models using the lumped mass formulation, the grid point kinetic energy can be used to examine the distribution of kinetic energy among the grid points. It is computed as:

$$E_{k_g} = \Phi_g^{mass} \otimes [M_{gg}\Phi_g^{mass}]$$

where Φ_g^{mass} is the mass-normalized eigenvectors so that the total grid point kinetic energy is scaled to be unity. Note that the operator $\otimes$ indicates term-wise matrix multiplication.

5. The grid point kinetic energy output has limited meaning for a coupled mass formulation. Since this mass formulation produces a coupling of mass across grid points, the sharing of kinetic energy among grid points can occur. In general, this obscures the meaning of the computation as a means of identifying important model parameters to control modal behavior.

GPRSORT Composites Ply Results Sorted Output

Request sorted output of composites ply results (stress, strain and failure indices) by global ply ID for a given element set.

Format:

$$\text{GPRSORT} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \end{matrix} \right\}$$

Examples:

GPRSORT=ALL
GPRSORT=22

Describer	Meaning
ALL	All composite elements referencing a PCOMPG property entry type. See Remarks 1. and 2.
n	Set identification number of a previously appearing SET command.

Remarks:

1. Composite element output will be sorted by Global Ply ID and Element ID. Note this sorted output is only available for composite elements referencing a PCOMPG property entry. Global ply IDs can only be specified on the PCOMPG entry.
2. Composite elements referencing the PCOMP property entry will be excluded from the sorted output.

GPSDCON Grid Point Stress Discontinuity Output Request

Requests mesh stress discontinuities based on grid point stresses.

Format:

$$\text{GPSDCON}\left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\end{matrix}\right\}$$

Examples:

GPSDCON=ALL
GPSDCON=19

Describer	Meaning
ALL	Stress discontinuity requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) Section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the stress discontinuity output request. (Integer>0)
NONE	No grid point stress discontinuity output.

Remarks:

1. This output is available in linear static analysis SOLs 101 and 144 only. Output will be presented for each surface or volume as a tabular listing of stress discontinuities for each subcase.
2. Only elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.
3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands. Also, the GPSTRESS and STRFIELD commands must be present for printed output.

GPSTRAIN Grid Point Strain Output Request for Printing Only

Requests grid point strains for printing only.

Format:

$$\text{GPSTRAIN}\left[\begin{matrix}\text{PRINT}\\ \text{PUNCH}\\ \text{PLOT}\end{matrix}\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Examples:

GPSTRAIN=ALL
GPSTRAIN=19

Describer	Meaning
ALL	Grid point strain requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the grid point strain output request. (Integer>0)
NONE	No grid point strain output.

Remarks:

1. For statics, normal modes, and transient analysis output will be presented for each surface or volume as a tabular listing of grid point strains for each load, eigenvalue, and time step. (See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2.)
2. Only grid points connected to elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.
3. Element strain output (STRAIN) must be requested for elements referenced on SURFACE and VOLUME commands.
4. In nonlinear transient analysis, grid point strains are computed only if parameter LGDISP is -1, which is the default, and for elements with linear material properties only.
5. For the postprocessing of grid point strains using the .xdb file or the computation of mesh strain discontinuities, the STRFIELD command must also be specified.

GPSTRESS Grid Point Stress Output Request for Printing Only

Requests grid point stresses for printing only.

Format:

$$\text{GPSTRESS}\left[\begin{matrix}\text{PRINT}\\ \text{PUNCH}\\ \text{PLOT}\end{matrix}\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Examples:

GPSTRESS=ALL
GPSTRESS=19

Describer	Meaning
ALL	Grid point stress requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) Section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the grid point stress output request. (Integer > 0)
NONE	No grid point stress output.

Remarks:

1. For statics, normal modes, and transient analysis output will be presented for each surface or volume as a tabular listing of grid point stresses for each load, eigenvalue, and time step. (See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2.)
2. Only grid points connected to elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.
3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands.
4. In nonlinear transient analysis, grid point stresses are computed only if parameter LGDISP is -1, which is the default, and for elements with linear material properties only. Grid point stresses are not computed for the hyperelastic elements.

5. For the postprocessing of grid point stresses using the .xdb file or the computation of mesh stress discontinuities, the STRFIELD command must also be specified.
6. Grid point stress is not output for midside nodes.

GROUNDCHECK Rigid Body Motion Grounding Check

Perform grounding check analysis on stiffness matrix to expose unintentional constraints by moving the model rigidly.

Format:

$$\text{GROUNDCHECK}\left[\begin{array}{l}\left(\left[\begin{array}{c}\text{PRINT}\\ \text{NOPRINT}\end{array}\right], \text{PUNCH}, \text{SET} = \left(\left\{\begin{array}{c}\text{G, N, N + AUTOSPC, F, A}\\ \text{ALL}\end{array}\right\}\right)\right.\\ \left.\text{GRID} = \text{gid}, \text{THRESH} = \text{e}, \text{DATAREC} = \left[\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right], (\text{RTHRESH} = \text{r})\right)\end{array}\right] = \left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

Examples:

GROUNDCHECK=YES
GROUNDCHECK(GRID=12,SET=(G,N,A),THRESH=1.E-5,DATAREC=YES)=YES

4A CASE

Describer	Meaning
PRINT	Write output to the print file (Default)
NOPRINT	Do not write output to the print file.
PUNCH	Write output to the punch file.
SET	Selects degree-of-freedom set(s). (Default: SET=G).
gid	Reference grid point for the calculation of the rigid body motion.
e	Maximum strain energy which passes the check. The default value is computed by dividing the largest term in the stiffness matrix by 1.E10.
DATAREC	Requests data recovery of grounding forces (Default: DATAREC=NO)
r	Grounding forces which are larger than r percent of the largest grounding force will be printed if DATAREC=YES. (Default=10%)

Remarks:

1. GROUNDCHECK must be specified above the subcase level.
2. SET=N+AUTOSPC uses the stiffness matrix for the n-set with the rows corresponding to degrees-of-freedom constrained by the PARAM,AUTOSPC operation zeroed out. If AUTOSPC was not performed then this check is redundant with respect to SET=N.

GUST Aerodynamic Gust Load Requests

Selects the gust field in an aeroelastic response problem.

Format:

GUST=n

Example:

GUST=73

Describer	Meaning
n	Set identification of a GUST Bulk Data entry. (Integer>0)

Remark:

1. The choice of transient or frequency response GUST depends upon the type of TLOAD or RLOAD referenced on the selected GUST entry.

HARMONICS Harmonic Analysis or Printout Control

Controls the number of harmonics output in axisymmetric shell or axisymmetric fluid problems; controls the number of harmonics to be used for analysis in cyclic symmetry problems.

Format for Axisymmetric Problems**:**

$$\text{HARMONICS} = \left\{ \begin{matrix} \text{ALL} \\ \text{NONE} \\ \text{h} \end{matrix} \right\}$$

Format for Cyclic Symmetric Problems**:**

$$\text{HARMONICS} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \end{matrix} \right\}$$

Examples:

HARMONICS=ALL
HARMONICS=32

Describer	Meaning
ALL	All harmonics will be output in the case of axisymmetric shell or axisymmetric fluid problems. All harmonics will be used for analysis in cyclic symmetry problems.
NONE	No harmonics will be output. This option is not available for use in cyclic symmetry problems.
h	Available harmonics up to and including harmonic h will be output in the case of axisymmetric shell or axisymmetric fluid problems. (Integer ≥ 0)
n	Harmonics specified in SET n will be used for analysis in cyclic symmetry problems. (Integer > 0)

Remarks:

1. If no HARMONICS command is present in the Case Control Section for axisymmetric shell or fluid problems, printed output is prepared only for the zero harmonic.

2. This command must be present in cyclic symmetry problems with HARMONICS=ALL or n; otherwise, the program will abort without performing any analysis.
3. In cyclic symmetry analysis, n must be defined as a set of integers on a SET command.

HDOT Heat Transfer Rate of Change of Enthalpy Output Request

Requests form of rate of change of enthalpy vector output in transient heat transfer analysis (SOL 159).

Format:

$$\text{HDOT}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right],\ \left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Example:

HDOT=5

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each time.
SORT2	Output will be presented as a tabular listing of time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates but does not print rate of change of enthalpy.
ALL	Rate of change of enthalpy for all points will be output.
NONE	Rate of change of enthalpy for no points will be output.
n	Set identification of previously appearing SET command. Only rates of change of enthalpy for points with identification numbers that appear on this SET command will be output. (Integer>0)

Remark:

1. HDOT=NONE is used to override a previous HDOT=n or HDOT=ALL command.

HOUTPUT Harmonic Output Request in Cyclic Symmetry Problems

Requests harmonic output in cyclic symmetry problems.

Format:

$$\text{HOUTPUT}[(C, S, C^*, S^*)] = \left\{ \begin{array}{c} \text{ALL} \\ K \end{array} \right\}$$

Examples:

HOUTPUT=ALL
HOUTPUT(C,S)=5

Describer	Meaning
C, S, C*, S*	Harmonic coefficients. See Remark 4.
ALL	All harmonics will be output.
k	Set identification number of harmonics for output. (Integer>0)

Remarks:

1. Set k must be defined on a SET command and output will be computed for all available harmonics in SET k.
2. HOUTPUT=ALL requests output for all harmonics specified on the HARMONICS command.
3. Either the HOUTPUT or NOUTPUT command is required to obtain data recovery in cyclic symmetry analysis.
4. C and S correspond to the cosine and sine coefficients when the STYPE field is ROT or AXI on the CYSYM Bulk Data entry.
   - C, S, C*, and S* correspond to the cosine symmetric, sine symmetric, cosine antisymmetric, and sine antisymmetric coefficients, respectively, when the STYPE field is DIH on the CYSYM Bulk Data entry.

HTFLOW Elemental Heat Flow Output Request

Requests heat flow output at selected structural elements.

Format:

$$\text{HTFLOW}\left[\left(\begin{array}{c}\text{PRINT, PUNCH}\\ \text{NOPRINT}\end{array}\right)\right] = \left\{\begin{array}{c}\text{ALL}\\ \text{n}\end{array}\right\}$$

Example:

HTFLOW = ALL
HTFLOW = 15

Describer	Meaning
PRINT	The printer will be the output medium.
NOPRINT	Generate, but do not print out, the output.
PUNCH	The punch file will be the output medium.
ALL	Heat flow for all structural elements will be output.
n	Set identification of previously appearing SET command. Only structural elements with identification numbers that appear on this SET command will be included in the heat flow output. (Integer>0)

Remarks:

1. Elemental heat flow output is available for steady state thermal analysis (SOL 101 and SOL 153) and transient thermal analysis (SOL 159).
2. Heat flow is computed from the applied heat loads and the effect of convection and radiation heat transfer on boundary (CHBDYE, CHBDYG, and CHBDYP) elements.
3. See Remarks 6-8 of the descriptions of CHBDYE Bulk Data for the side conventions of solid elements, shell elements, and line elements.

IC Transient Analysis Initial Condition Set Selection

Selects the initial conditions for transient analysis (SOLs 109, 112, 129 and 159).

Format:

$$IC\begin{bmatrix} PHYSICAL \\ MODAL \\ STATSUB[,DIFFK] \end{bmatrix} = n$$

Examples:

IC = 10
IC(PHYSICAL) = 100
IC(MODAL) = 200
IC(STATSUB) = 1000
IC(STATSUB,DIFFK) = 2000

Describer	Meaning
PHYSICAL	The TIC Bulk Data entries selected by set n define initial conditions for coordinates involving grid, scalar and extra points. (Default).
MODAL	The TIC Bulk Data entries selected by set n define initial conditions for modal coordinates and extra points. See Remark **3.**
STATSUB	Use the solution of the static analysis subcase n as the initial condition. See Remark **4.**
DIFFK	Include the effects of differential stiffness in the solution. See Remarks **4.** and **5.**
n	For the PHYSICAL (the default) and MODAL options, n is the set identification number of TIC Bulk Data entries for structural analysis (SOL 109, 112 and 129) or TEMP and TEMPD entries for heat transfer analysis (SOL 159). For the STATSUB option, n is the ID of a static analysis subcase. (Integer > 0)

Remarks:

1. For structural analysis, TIC entries will not be used (therefore, no initial conditions) unless selected in the Case Control Section.
2. Only the PHYSICAL option (the default) may be specified in heat transfer analysis (SOL 159).

3. IC(MODAL) may be specified only in modal transient analysis (SOL 112).
4. IC(STATSUB) and IC(STATSUB,DIFFK) may not both be specified in the same execution.
5. The DIFFK keyword is meaningful only when used in conjunction with the STATSUB keyword.
6. The following examples illustrate the usage of the various options of the IC Case Control command.

   a. $ SPECIFY INITIAL CONDITIONS FOR PHYSICAL COORDINATES
   $ IN SOL 109 OR SOL 112
   IC(PHYSICAL) = 100
   or
   IC = 100

   b. $ SPECIFY INITIAL CONDITIONS FOR MODAL COORDINATES
   $ IN SOL 112
   IC(MODAL) = 200

   c. $ SPECIFY STATIC SOLUTION AS INITIAL CONDITION
   $ IN SOL 109 OR SOL 112
   $ (DIFFERENTIAL STIFFNESS EFFECT NOT INCLUDED)
   SUBCASE 10 $ STATIC ANALYSIS
   LOAD = 100
   SUBCASE 20 $ TRANSIENT ANALYSIS
   IC(STATSUB) = 10 $ POINTS TO STATIC ANALYSIS SUBCASE ID

   d. $ SPECIFY STATIC SOLUTION AS INITIAL CONDITION
   $ IN SOL 109 OR SOL 112
   $ (DIFFERENTIAL STIFFNESS EFFECT INCLUDED
   SUBCASE 100 $ STATIC ANALYSIS
   LOAD = 1000
   SUBCASE 200 $ TRANSIENT ANALYSIS
   IC(STATSUB,DIFFK) = 100 $ POINTS TO STATIC ANALYSIS SUBCASE ID

INCLUDE Insert External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

Format:

INCLUDE 'filename'

Describer	Meaning
filename	Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks (’).

Example:

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
INCLUDE 'MYCASE.DATA'
BEGIN BULK
ENDDATA
```

Remarks:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example the file:

   /dir123/dir456/dir789/filename.dat

   may be included with the following input:

   ```
   INCLUDE '/dir123
             /dir456
           /dir789/filename.dat'
   ```

3. See the *MSC.Nastran 2005 r2 Installation and Operations Guide* for more examples.

INTENSITY Acoustic Intensity Output Request

Requests output of acoustic intensity on wetted surface.

Format:

$$\text{INTENSITY}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right], \left[\begin{matrix}\text{PRINT,PUNCH}\\ \text{PLOT}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Describer	Meaning
SORT1	Output will be presented as tabular listing of grid points for each excitation frequency (Default)
SORT2	Output will be presented as tabular listing of excitation frequencies for each grid point
PRINT	The printer will be the output medium (Default).
PUNCH	The punch file will be the output medium.
PLOT	Results are generated but not output.
ALL	Intensities will be computed for all grid points of the wetted surface.
n	Set identification of a previously defined set of grid points. Intensities will be computed for the grid points in this set only.
NONE	Acoustic intensities will not be processed.

K2GG Direct Input Stiffness Matrix Selection

Selects direct input stiffness matrix or matrices.

Format:

K2GG=name

Example:

K2GG = KDMIG

K2GG = KDMIG1, KDMIG2, KDMIG3

K2GG = 1.25*KDMIG1, 1.0*KDMIG2, 0.82*KDMIG3

SET 100 = K2, K3, K4
K2GG = 100

Describer	Meaning
name	Name of a $[K^2_{gg}]$ matrix that is input on the DMIG Bulk Data entry, or name list with or without factors (see Remark 6.) (Character)

Remarks:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the stiffness matrix before any constraints are applied.
3. The matrix must be symmetric and field 4 on DMIG,name entry must contain the integer 6.
4. A scale factor may be applied to this input via the PARAM, CK2 entry. See “Parameters” on page 659.
5. The matrices are additive if multiple matrices are referenced on the K2GG command.
6. The formats of the name list:
   a. Names without factor.
      Names separated by comma or blank.
   b. Names with factors.
      Each entry in the list consists of a factor followed by a star followed by a name. The entries are separated by comma or blank. The factors are real numbers. Each name must be with a factor including 1.0.

K2PP Direct Input Stiffness Matrix Selection

Selects direct input stiffness matrix or matrices, which are not included in normal modes.

Format:

K2PP=name

Example:

K2PP = KDMIG

K2PP = KDMIG1, KDMIG2, KDMIG3

K2PP = 5.06*KDMIG1, 1.0*KDMIG2, 0.85*KDMIG3

K2PP = (1.25, 0.5) *KDMIG1, (1.0,0.0) *KDMIG2, (0.82, -2.2) *KDMIG3

Describer	Meaning
name	Name of a $[K^2_{pp}]$ matrix that is input on the DMIG or DMIAX Bulk Data entry, or name list with or without factors (see Remark 6.).

Remarks:

1. DMIG and DMIAX entries will not be used unless selected by the K2PP command.
2. The matrix must be square or symmetric and field 4 on the DMIG,name entry must contain a 1 or 6.
3. It is recommended that PARAM,AUTOSPC,NO be specified. See the "**Constraint and Mechanism Problem Identification in SubDMAP SEKR**" on page 409 of the *MSC.Nastran Reference Guide.*
4. K2PP matrices are used only in dynamic response problems. They are not used in normal modes.
5. The matrices are additive if multiple matrices are referenced on the K2PP command.
6. The formats of the name list:
   a. Names without factor

      Names separated by comma or blank.

b. Names with factors.

Each entry in the list consists of a factor followed by a star followed by a name. The entries are separated by comma or blank. The factors are either all real numbers, or all complex numbers in the form of two real numbers, separated by a comma, within parenthesis as shown in the above example. The first real number of the pair is for real part, and the second for imaginary part. Either part may be zero or blank, but not both. Mixed real numbers and complex numbers are not allowed. Each name must be with a factor including 1.0 for real and (1.0, 0.0) for complex.

K42GG Direct Input Stiffness Element Damping Selection

Selects direct input structural element damping matrix or matrices.

Format:

K42GG=name

Example:

K42GG = KDMIG

K42GG = KDMIG1, KDMIG2, KDMIG3

K42GG = 2.03*KDMIG1, 0.84*KDMIG2

Describer	Meaning
name	Name of a $[K4^2_{gg}]$ matrix that is input on the DMIG Bulk Data entry, or name list with or without factors (see Remark 4.)

Remarks:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the structural element damping matrix before any constraints are applied.
3. The matrix must be symmetric and field 4 on DMIG,name entry must contain the integer 6.
4. The formats of the name list:
   a. Names without factor.
      Names separated by comma or blank.
   b. Names with factors.
      Each entry in the list consists of a factor followed by a star followed by a name. The entries are separated by comma or blank. The factors are real numbers. Each name must be with a factor including 1.0.

LABEL Output Label

Defines a character string that will appear on the third heading line of each page of printer output.

Format:

LABEL=label

Example:

LABEL=DEMONSTRATION PROBLEM

Describer	Meaning
label	Any character string.

Remarks:

1. LABEL appearing at the subcase level will label output for that subcase only.
2. LABEL appearing before all subcases will label any outputs that are not subcase dependent.
3. If no LABEL command is supplied, the label line will be blank.
4. LABEL information is also placed on plotter output as applicable. Only the first 65 characters will appear.

LINE Maximum Lines Per Printed Page

Defines the maximum number of output lines per printed page.

Format:

LINE=n

Example:

LINE=35

Describer	Meaning
n	Maximum number of output lines per page. (Integer>0; Default=50)

Remarks:

1. For 11 inch paper, 50 lines per page is recommended; for 8-1/2 inch paper, 35 lines per page is recommended.
2. The NASTRAN statement keyword NLINES may also be used to set this value. See the "**nastran Command and NASTRAN Statement**" on page 1.

LOAD External Static Load Set Selection

Selects an external static load set.

Format:

LOAD=n

Example:

LOAD=15

Describer	Meaning
n	Set identification of at least one external load Bulk Data entry. The set identification must appear on at least one FORCE, FORCE1, FORCE2, FORCEAX, GRAV, MOMAX, MOMENT, MOMENT1, MOMENT2, LOAD, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX, QVOL, QVECT, QHBDY, QBDY1, QBDY2, QBDY3, PRESAX, RFORCE, SPCD, or SLOAD entry. (Integer>0)

Remarks:

1. A GRAV entry cannot have the same set identification number as any of the other loading entry types. If it is desired to apply a gravity load along with other static loads, a LOAD Bulk Data entry must be used.
2. LOAD is only applicable in linear and nonlinear statics, inertia relief, differential stiffness, buckling, and heat transfer problems.
3. The total load applied will be the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM), and constrained displacement (SPC) loads.
4. Static, thermal, and element deformation loads should have unique set identification numbers.

LOADSET Static Load Set Selection

Selects a sequence of static load sets to be applied to the structural model. The load sets may be referenced by dynamic load commands.

Format:

LOADSET=n

Example:

LOADSET=100

Describer	Meaning
n	Set identification number of at least one LSEQ Bulk Data entry. (Integer>0)

Remarks:

1. When used in superelement analysis, this command must be used for all superelements. The number of static load vectors created for each superelement depends upon the type of analysis. In static analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all LSEQ entries in the Bulk Data; in dynamic analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data.
2. When the LOADSET command is used in superelement statics, the residual structure should have as many loading conditions as the number of unique EXCITEID sets defined on all LSEQ entries. The subcases after the first should contain only SUBTITLE and LABEL information and residual structure output requests. SUBTITLE and LABEL information for all superelements will be obtained from the residual structure.
3. When multiple subcases are specified in the dynamic solution sequences (108, 109, 111, 112, 118, 146), the LOADSET must appear in the first subcase or above all subcases. In SOL 200 with ANALYSIS=DFREQ, MFREQ, or MTRAN, a different LOADSET may be specified in the first subcase pertaining to each ANALYSIS command.
4. In SOL 101, the design sensitivity output will identify all expanded subcases by a sequence of unique integers beginning with n.
5. In the nonlinear static solution sequences (SOLs 106 and 153) the LOADSET must appear above all subcases and only one LOADSET may be specified.

6. Only one LOADSET command is allowed per superelement and must be specified in the superelement's first subcase.
7. It is no longer necessary to employ LOADSET/LSEQ data to specify static loading data for use in dynamic analysis. In the absence of LOADSET request in Case Control, all static loads whose load set IDs match the EXCITEID IDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data are automatically processed.

M2GG Direct Input Mass Matrix Selection

Selects direct input mass matrix or matrices.

Format:

M2GG=name

Example:

M2GG = MDMIG

M2GG = MDMIG1, MDMIG2, MDMIG3

M2GG = 1.25*MDMIG1, 1.0*MDMIG2, 0.82*MDMIG3

SET 200 = M1, M2
M2GG = 200

Describer	Meaning
name	Name of a $[M_{gg}^2]$ matrix that is input on the DMIG Bulk Data entry, or name list with or without factors (see Remark 6.) (Character)

Remarks:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the mass matrix before any constraints are applied.
3. The matrix must be symmetric and field 4 on DMIG, name entry must contain a 6.
4. M2GG input is not affected by PARAM,WTMASS. M2GG input must either be in consistent mass units or PARAM,CM2 may be used.
5. The matrices are additive if multiple matrices are referenced on the M2GG command.
6. The formats of the name list:
   a. Names without factor.
      Names separated by comma or blank.
   b. Names with factors.
      Each entry in the list consists of a factor followed by a star followed by a name. The entries are separated by comma or blank. The factors are real numbers. Each name must be with a factor including 1.0.

M2PP Direct Input Mass Matrix Selection

Selects direct input mass matrix or matrices, which are not included in normal modes.

Format:

M2PP=name

Example:

M2PP = MDMIG

M2PP = MDMIG1, MDMIG2, MDMIG3

M2PP = 5.06*MDMIG1, 1.0*MDMIG2, 0.85*MDMIG3

M2PP = (1.25, 0.5) *MDMIG1, (1.0, 0.0) *MDMIG2, (0.82, -2.2) *MDMIG3

Describer	Meaning
name	Name of a $[M_{pp}^2]$ matrix that is input on the DMIG or DMIAX Bulk Data entry, or name list with or without factors (see Remark 7.). (Character)

Remarks:

1. DMIG and DMIAX entries will not be used unless selected by the M2PP input.
2. M2PP input is not affected by PARAM,WTMASS. M2PP input must be in consistent mass units.
3. The matrix must be square or symmetric and field 4 on the DMIG, name entry must contain a 1 or 6.
4. It is recommended that PARAM,AUTOSPC,NO be specified. See "**Constraint and Mechanism Problem Identification in SubDMAP SEKR**" on page 409 of the *MSC.Nastran Reference Guide*.
5. M2PP matrices are used only in dynamic response problems. They are not used in normal modes.
6. The matrices are additive if multiple matrices are referenced on the M2PP command.
7. The formats of the name list:
   a. Names without factor

      Names separated by comma or blank.

b. Names with factors.

Each entry in the list consists of a factor followed by a star followed by a name. The entries are separated by comma or blank. The factors are either all real numbers, or all complex numbers in the form of two real numbers, separated by a comma, within parenthesis as shown in the above example. The first real number of the pair is for real part, and the second for imaginary part. Either part may be zero or blank, but not both. Mixed real numbers and complex numbers are not allowed. Each name must be with a factor including 1.0 for real and (1.0, 0.0) for complex.

MASTER Redefine the MASTER Subcase

Allows the redefinition of a MASTER subcase.

Format:

SUBCASE n
MASTER

Example:

SUBCASE 10
MASTER

Remarks:

1. All commands in a MASTER subcase apply to the following subcases until a new MASTER subcase is defined.
2. Suppose that superelement 10 has SPC set 10, MPC set 10, and LOAD sets 101 and 102. Suppose also that superelement 20 has SPC set 20, MPC set 20, and LOAD sets 201 and 202.

   The following setup specifies the required subcase structure:

```
        TITLE = MY MODEL
        DISP = ALL
        SEALL = ALL
SUBCASE 101
        MASTER
        SPC = 10
        MPC = 10
        SUPER = 10, 1
        LOAD = 101
        LABEL = SUPER 10
        ESE = ALL
SUBCASE 102
        LOAD = 102
        SUPER = 10, 2
SUBCASE 201
        MASTER
        SPC = 20
        MPC = 20
        SUPER = 20, 1
        LOAD = 201
        LABEL = SUPER 20
SUBCASE 202
        LOAD = 202
        SUPER = 20, 2
```

3. MASTER may also be used to advantage with multiple boundary condition Case Control setups. Suppose that constraint sets 10 and 20 are to be solved with three loading conditions each: 1, 2, and 3 and 4, 5, and 6, respectively. The following Case Control Section may be used.

```
TITLE = MULTIPLE BOUNDARY CONDITIONS
DISP = ALL
SYM 1
MASTER
  SPC = 10
  LOAD = 1
SYM 2
  LOAD = 2
SYM 3
  LOAD = 3
SYM 4
  MASTER
  SPC = 20
  LOAD = 4
SYM 5
  LOAD = 5
SYM 6
  LOAD = 6
SYMCOM 10
  SYMSEQ = 1., 1., 1., -1., -1., -1.
SYMCOM 20
  SYMSEQ = -1., -1., -1., 1., 1., 1.
```

4. The MASTER command must appear immediately after a SUBCASE or SYM command.

MAXLINES Maximum Number of Output Lines

Sets the maximum number of output lines.

Format:

MAXLINES=n

Example:

MAXLINES=150000

Describer	Meaning
n	Maximum number of output lines allowed. (Integer > 0; Default =999999999)

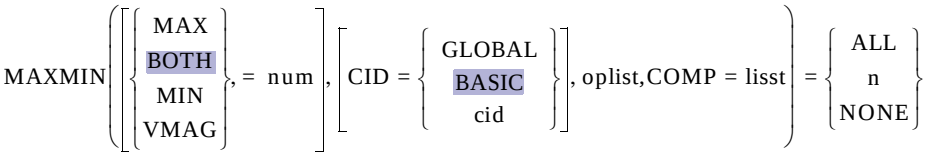
Remarks:

1. Any time MAXLINES is exceeded, the program will terminate.
2. MAXLINES does not override any system parameters such as those on Job Control Language commands.
3. MAXLINES may also be specified on the NASTRAN statement with SYSTEM(14). See the "**nastran Command and NASTRAN Statement**" on page 1.
4. The code actually counts the number of pages and assumes that the number of lines output is the number of lines allowed per page, specified by the "LINES" command, times the number of pages.

MAXMIN MAXMIN Survey Output Request

Specifies options for max/min surveys of certain output data associated with grid points.

Format:



Example:

MAXMIN (BOTH=10,CID=1000,DISP,COMP=T1/T3)=501

Describer	Meaning
MAX	Specifies only maximum values are to be output. See Remark 1.
MIN	Specifies only minimum values are to be output. See Remark 1.
BOTH	Specifies both maximum and minimum values are to be output. See Remark 1.
VMAG	Specifies vector magnitude resultants are to be output. See Remark 2.
num	The maximum number of values that will be output. See Remark 3. (Integer > 0, Default = 5)
CID	Specifies the coordinate system frame of reference in which the max/min values will be output. See Remarks 1. and 3.
GLOBAL	Requests output in the global coordinate system frame of reference.
BASIC	Requests output in the basic coordinate system frame of reference.
cid	Requests output in the local coordinate system defined by cid. (Integer > 0)
oplist	Specifies a list of one or more standard Case Control output requests for which max/min results are to be produced. The list may include any of DISP,SPCDF, OLOAD,MPCF,VELO,ACCE or ALL. See Remark 6. (Character, no default)

Describer	Meaning
list	Specifies a list of grid point degree of freedom (DOF) component directions that will be included in the max/min survey output. The components are separated by slashes and are selected from T1, T2, T3, R1, R2, R3. See Remarks **4.** and **5.** (Character, default=/T1/T2/T3/R1/R2/R3)
ALL	MAXMIN survey results for all points will be output.
NONE	MAXMIN survey results for no points will be output.
n	Set identification of a previously appearing SET command. The max/min results survey will be output only for the points specified SET n. (Integer > 0)

Remarks:

1. The MAXMIN command produces an algebraically ascending sorted list of the output quantities specified for all of the points in the selected set. MAX refers to the largest magnitude positive values, while MIN refers to the largest magnitude negative values. The output format is similar to that of displacement output. All components will be output for a grid point and the order of the grid points will be in sort on the particular component that was surveyed. The output title contains the identification number of the SET of points participating in the max/min output, the coordinate system frame of reference, the number of MAX and MIN values output and the component that was surveyed. When the output being surveyed is in the global output coordinate system reference frame and BASIC or a local output coordinate system is specified as cid, both the sorted system output and the original reference system output are displayed for the grid point if these systems are different.
2. Vector magnitude results are computed for both translations and rotations and displayed under the T1 and R1 column headings. The presence of the COMP keyword is ignored.
3. The default value of 5 generates a minimum of 10 output lines for the BOTH option. There will be 5 maximum values and 5 minimum values produced. In addition, if coordinate system are involved, both surveyed and original data will be output. This could result in as many as 10 more lines of output for each surveyed component.

4. Multiple MAXMIN commands may be specified for a subcase. This permits different output quantities to have different MAXMIN specification within a subcase. For example,

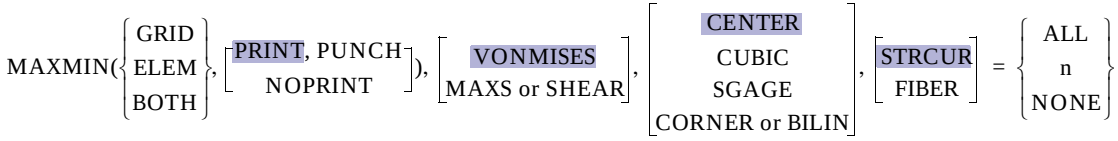
   SET 501=1,3,5,7 THRU 99, 1001, 2001
   MAXMIN (DISP, COMP=T3) = 501
   MAXMIN (SPCF, COMP=T1/R3) = ALL

5. Scalar point output is included only if component T1 is included in the list.
6. MAXMIN output will only be generated for items in the oplist when there is an associated case control command present. For example, a DISP Case Control command must be present in order for the MAXMIN(DISP) = ALL command to produce output. Use of ALL keywords for the oplist requests MAXMIN output for all of the output commands acceptable to MAXMIN that are present in case control.

MAXMIN Requests Output of Maximums and Minimums in Data Recovery

Requests the output of maximums and minimums in data recovery.

Format:



Example:

MAXMIN(GRID) = 12
MAXMIN(ELEM) = ALL
MAXMIN = NONE

4A CASE

Describer	Meaning
GRID	Indicates the request is applied to grid points.
ELEM	Indicates the request is applied to elements.
BOTH	Indicates the request is applied to both elements and grid points. (Default)
PRINT	Compute and write the output to the .f06 file. (Default)
PUNCH	Compute and write output to the punch file.
NOPRINT	Compute but do not write out the results.
ALL	Max/min results will be reported for all elements.
n	Set identification number. The referenced SET command defines a set of elements or grid points to be monitored.
NONE	Max/min results will not be reported.
VONMISES	von Mises stress/strains are output.
MAXS or SHEAR	Maximum shear stress/strains are output.
STRCUR	Strain at the reference plane and curvatures are output for plate elements.
FIBER	Stress/Strain at locations Z1, Z2 are computed for plate elements.
CENTER	Output QUAD4 stress/strains at the center only.

Describer	Meaning
CORNER or BILIN	Output CQUAD4 element stress / strains at the center and grid points. Using strain gage approach with bilinear extrapolation.
SGAGE	Output CQUAD4 element stress / strains at center and grid points using strain gage approach.
CUBIC	Output CQUAD4 element stress / strains at center and grid points using cubic bending correction.

Remarks:

1. MAXMIN is not allowed in REPCASE but is allowed in SUBCOM and SYMCOM.
2. The OTIME command may be used limit the time span of monitoring.
3. No corresponding output request; such as, DISP, STRESS, etc., is required. Also, SET=n may request more elements or grid points for monitoring than is requested by Outputs.

   The output is comprised of two parts: (1) a summary of the maximum / minimum values and the times they occur and (2) the associated output for all components of the element or grid. The first part is always output and the second part is only output if the FULL describer is specified on the MAXMIN(DEF) command. See the next section for a description of the new Case Control commands. Here are sample Case Control commands for the output of maximum von Mises stresses using the BRIEF option:

   ```
   MAXMIN(DEF) STRESS QUAD4 SMAX1 ABSOLUTE(5) BRIEF
   SET 100=4
   MAXMIN(ELEM)=4
   ```

4. See DISPLACEMENT, STRESS or STRAIN Case Control commands for additional keyword implications.

MAXMIN(DEF) Defines Parameters for Monitoring Maximums and Minimums

Defines parameters and output options for the monitoring of maximums and minimums in data recovery. MAXMIN(DEF) must be specified above all subcases. The MAXMIN(ELEM) and/or MAXMIN(GRID) command is then required to printout the max/min results.

Format for Grid Point Output:

```
MAXMIN(DEF){ DISP, VELO, ACCE, MPCF, SPCF, OLOAD}{T1 T2 T3 R1 R2 R3 MAGT MAGR},
        [ABSOLUTE(p) MINALG(q) MAXALG(r)  ][RMS][BRIEF ],
                     ALL(p)               [FULL  ]
        [ CID = { GLOBAL } ]
                { BASIC  }
                { cid    }
```

Format for element output:

```
MAXMIN(DEF){STRESS }{eltype1[, eltype2,...]} { comp1[ / CENTER ] [ , comp2...] },
           {STRAIN }                          {      ALL    }
           {FORCE  }                          {      GROUP  }
                                              {      ENDS   }
        [ABSOLUTE(p) MINALG(q) MAXALG(r)  ][RMS][BRIEF ]
                     ALL(p)               [FULL  ]
```

Examples:

MAXMIN(DEF) stress (cquad4,smax1) MAXA=5
MAXMIN(DEF) disp T1 T2 T3 MAGT RMS FULL

Describer	Meaning
DISP,VELO,etc. STRESS,etc.	Type of result to be monitored.
T1, T2, etc.	Name of the grid point component to be monitored.
MAGT	Specifies the magnitude of the translational components are to be monitored.
MAGR	Specifies the magnitude of the rotational components are to be monitored
eltype1 eltype2 …	Name of the element type(s) to be monitored. At least one element type must be present.

Describer	Meaning
comp1 comp2…	Name of the element component(s) to be monitored; e.g., etmax1 for von Mises stress in the Z1 plane. The component names are defined in the NDDL. Also, the item codes from "**Item Codes**" on page 873 of the *MD Nastran Quick Reference Guide* may be used.
ABSOLUTE(p)	Print out the top p absolute values. (Default for p is 5).
MINALG(q)	Print out the bottom q minimum algebraic values. (Default for q is 5)
MAXALG(r)	Print out the top r maximum algebraic values. (Default for r is 5).
ALL(p)	Print out all options: ABSOLUTE, MINALG, and MAXALG. (Default for p is 5)
BRIEF	Print out only the maxmin results. (Default)
FULL	Print out the maxmin results followed by the standard data recovery format for the elements and grids at the retained set of the maximum or minimum occurrences.
GLOBAL	Selects the global coordinate system (see CD on the GRID entry) for monitoring grid point results.
BASIC	Selects the basic coordinate system for monitoring grid point results. (Default)
cid	Specifies a coordinate system id for a system defined on a CORDij entry for monitoring grid point results.
RMS	Print out the root-mean-square value of each maximum or minimum value requested by ABSOLUTE, MIN, or MAX over all time steps.
CENTER	Component selector when element allows for component name to occur in multiple places. (Default)
ALL	Selects all locations in an element where multiple locations exist.
GROUP	Reduces all occurrences of a component name to a single value before the action is performed.
ENDS	Selects the ends of a BEAM element ignoring intermediate stations.

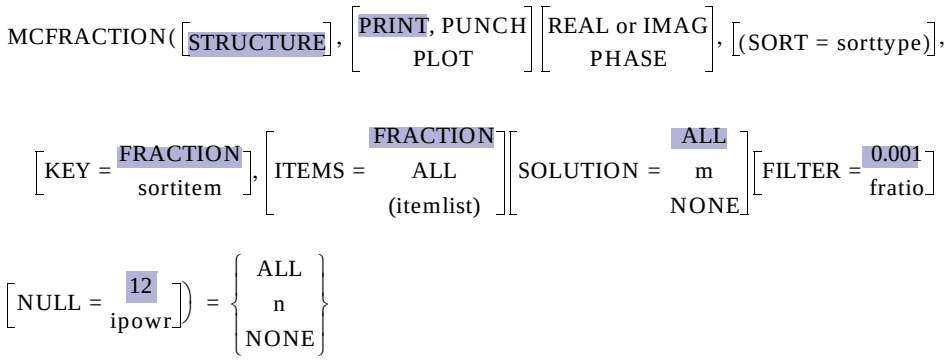
Remarks:

1. MAXMIN(DEF) must be specified above all subcases but is not sufficient to request monitoring of maximums and minimums. The MAXMIN command must also be specified above or inside subcases.
2. MAXMIN(DEF) may be specified more than once.
3. Multiple element types may be grouped together if the same component name is to be monitored across those types by enclosing the element types in parentheses.
4. Multiple component names may be compared collectively to the current maximum (or minimum), but only the maximum (or minimum) component in the group will be reported in the output. This is requested by enclosing the component names in parentheses.
5. Grid point component output is always converted to the basic coordinate system for monitoring when processing "sort1". The global system is the default when processing in "sort2".
6. Results for layers in composite elements or intermediate stations in CBAR and CBEAM elements are not supported.
7. Only real data recovery is currently supported.
8. When no MINA, MAXA or ABSO keyword is supplied, the default values of p, q, and s will be 5. When any keyword is supplied, the other unreferenced keyword values will be set to zero, and no output will be created.
9. The component action keywords of CENTER, ALL, GROUP, and ENDS can only be applied to component names defined in the NDDL and occur at multiple places in element data recovery. They cannot be used with "Item Codes".
10. Specify CQUAD4C and CTRIAC for corner stress of CQUAD4 and CTRIA3 elements.

MCFRACTION Modal Contribution Fractions Output Request

Requests the form and type of modal contribution fraction output.

Format:



Example:

```
SET 1001 = 10.0, 20.0, 100.0
SET 2001 = 716/T3, 809/T3, 412/T1
MCFRACTION (STRUCTURE,PRINT,PUNCH,ITEM=FRACTION,
            SORT=ABSD,KEY=PROJECTION,SOLUTION=1001)=2001

MCFRACTION (ITEMS=(FRACTION,PROJECTION),FILTER=0.01)=2001
```

Describer	Meaning
STRUCTURE	Request pertains to structure points only.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generate modal fractions for the requested set but no printer output.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output.
PHASE	Requests polar format (magnitude and phase) of complex output.
SORT	Keyword selecting one of the sort type options. Default is to produced output in increasing natural mode number order.

Describer	Meaning
sorttype	One of the following modal contribution fraction output sorting options. ABSA - output will be sorted by absolute value in ascending order. ABSD - output will be sorted by absolute value in descending order. ALGA - output will be sorted algebraically in ascending order. ALGD - output will be sorted algebraically in descending order.
KEY	Keyword selecting a sorting operation key quantity.
sortitem	Item from the item list table on which the SORT operation is performed. (Default is FRACTION.)
ITEMS	Keyword specifying data selection options.
itemlist	One (or more) of the following modal contribution fraction output items. If more than one item is selected, the list must be enclosed in parentheses.

Item Identifier	Description
RESPONSE	Each mode's response at each degree-of-freedom selected
PROJECTION	Projection of modal response on solution
FRACTION	Fraction of total displacement per mode (PROJECTION divided by total)
SCALED	Scaled magnitudes (FRACTION divided by largest term in FRACTION)
MODEDISP	Modal displacements (complex solution at each dof by mode number)
MODERESP	Modal response for each mode (polar format wrt total displacement)

SOLUTION	Keyword specifying the output solution time step, forcing frequency or complex eigenvalue selections for which modal fractions will be generated. (Default = ALL)
m	Results for solutions in SET m will be output.
FILTER	Keyword specifying the value of the printed output data filter.
fratio	Value of output filter ratio. (Default = 0.001)

NULL	Keyword specifying the power of ten used to detect a null response quantity.
ipowr	The power of ten used to detect a null response quantity. (Default = 12)
n	Results for grid point components in SET n will be output.
ALL	Results for all solutions and / or grid point components will be output.
NONE	No modal contribution fractions will be output.

Remarks:

1. The MCFRACTION command is useful in modal frequency response (SOL 111), modal transient response (SOL 112) and modal complex eigenvalue analysis (SOL 110) solution sequences only. If superelements are used, its use is restricted to residual structure data recovery operations only.
2. Both PRINT and PUNCH may be requested.
3. Printed output includes results for all of the data items described in the itemlist table.
4. Punched output includes results for only the data items selected by the ITEMS keyword.
5. Modal Contribution Fractions are sorted by increasing order of mode number unless the SORT keywords specifies a particular sorting order. If a sorting order is specified, the KEY keyword selects the particular data item in the printed results tabular output listing that is sorted. When MODEDISP is selected, the magnitude is sorted. When MODERESP is selected, the real portion of the response is sorted.
6. The SOLUTION keyword can be used to select a subset of the solutions available. If SET m is specified, the items in the SET list are forcing frequency values, time step values or complex eigenvalue mode numbers depending upon the solution sequence used.
7. The FILTER keyword specifies a filter ratio value that is used to limit the amount of printed output produced. It applies to the data item selected by the KEY keyword if it is specified. If no KEY keyword is present, the default value od KEY=FRACTION will be used. The maximum value for the selected data item across all natural modes is determined. If the ratio of the data item value to the maximum data item value is less than fratio for any natural mode, no output for that natural mode is produced.

8. If the magnitude of the total response at a selected grid pint component is less than 1.0×10^{-ipowr}, no modal contribution fraction output is generated for that degree-of-freedom. If ipowr is not in the range of 1 to 31, the default value of 12 is used for ipowr producing a null response threshold of 1.0×10^{-12}.
9. For modal transient response solution sequences, response quantities are real numbers. There are no imaginary terms. Therefore, polar representations of the data have no meaning. Furthermore, projections of responses onto the total response are simply the individual modal contribution to the total response at a degree-of-freedom. Thus, the only items available for output are the individual modal response magnitude (PROJECTION), the modal fraction (FRACTION), and the scaled response magnitude (SCALED). Selection of any of the other items from the itemlist table causes selection of the modal response magnitude (PROJECTION) item.

MEFFMASS Modal Effective Mass Output Request

Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis.

Format:

$$\text{MEFFMASS}\left[\left[\begin{matrix}\text{PRINT}\\ \text{NOPRINT}\end{matrix}\right]\left[\begin{matrix}\text{PUNCH}\\ \text{NOPUNCH}\end{matrix}\right],\text{GRID} = \text{gid},\left[\begin{matrix}\text{SUMMARY, PARTFAC,}\\ \text{MEFFM, MEFFW,}\\ \text{FRACSUM, ALL}\end{matrix}\right]\right] = \left\{\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right\}$$

Examples:

MEFFMASS
MEFFMASS(GRID=12, SUMMARY, PARTFAC)

Describer	Meaning
PRINT	Write output to the print file. (Default)
NOPRINT	Do not write output to the print file.
PUNCH	Write output to the punch file.
NOPUNCH	Do not write output to the punch file. (Default)
gid	Reference grid point for the calculation of the Rigid Body Mass Matrix. The default is the origin of the basic coordinate system.
SUMMARY	Requests calculation of the Total Effective Mass Fraction, Modal Effective Mass Matrix, and the Rigid Body Mass Matrix. (Default)
PARTFAC	Requests calculation of Modal Participation Factors.
MEFFM	Requests calculation of the Modal Effective Mass in units of mass.
MEFFW	Requests calculation of the Modal Effective Mass in units of weight.
FRACSUM	Requests calculation of the Modal Effective Mass Fraction.

Remarks:

1. The SUMMARY describer produces three outputs:

   Modal Effective Mass Matrix $[\varepsilon^T][m][\varepsilon]$ where

   ε = Modal Participation Factors: $[m^1][\phi^T][M_{aa}][D_{ar}]$

   m = Generalized mass matrix

ϕ = Eigenvectors

M_{aa} = Mass marix reduced to the a-set (g-set for superelements)

D_{ar} = Rigid body transformation matrix with respect to the a-set

A-set Rigid Body Mass Matrix: $[D_{ar}^T][M_{aa}][D_{ar}]$. For superelement this is computed at the g-set

Total Effective Mass Fraction: i.e., diagonal elements of the Modal Effective Mass Matrix divided by the Rigid Body Mass Matrix.

2. The PARTFAC describer outputs the Modal Participation Factors table: ε
3. The MEFFM describer outputs the Modal Effective Mass table: ε^2, the term-wise square of the Modal Participation Factors table.
4. The MEFFW describer outputs the Modal Effective Weight table; i.e., the Modal Effective mass divided by user PARAMeter WTMASS.
5. The FRACSUM describer outputs the Modal Effective Mass Fraction table; i.e., the Generalized Mass Matrix (diagonal term) multiplied by the Modal Effective Mass and then divided by the Rigid Body Mass Matrix (diagonal term).
6. For superelements the MEFFMASS command uses the residual structure eigenvalues and eigenvectors, by default. If however, PARAM, FIXEDB, -1 is specified then MEFFMASS command uses the component mode eigenvalues and eigenvectors.

METHOD Real Eigenvalue Extraction Method Selection

Selects the real eigenvalue extraction parameters.

Format:

$$\text{METHOD}\left[\begin{array}{c}\text{BOTH}\\ \text{STUCTURE}\\ \text{FLUID}\end{array}\right] = n$$

Examples:

METHOD=33
METHOD(FLUID)=34

Describer	Meaning
BOTH	The referenced EIGR or EIGRL Bulk Data entry will be applied to both the structure and the fluid portion of the model. (Default)
STRUCTURE or FLUID	The referenced EIGR or EIGRL Bulk Data entry is applied to the structural or fluid portion of the model.
n	Set identification number of an EIGR or EIGRL Bulk Data entry for normal modes or modal formulation, or an EIGB or EIGRL entry for buckling. (Integer > 0)

Remarks:

1. An eigenvalue extraction method must be selected when extracting real eigenvalues.
2. If the set identification number selected is present on both EIGRL and EIGR and / or EIGB entries, the EIGRL entry will be used.
3. METHOD(FLUID) and METHOD(STRUCTURE) permits a different request of EIGR or EIGRL for the fluid portion of the model in coupled fluid-structural analysis. See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*.
   - If METHOD(STRUCTURE) or METHOD(FLUID) is also specified then they will override the METHOD(BOTH) selection.
   - The METHOD(FLUID) and METHOD(STRUCTURE) may be specified simultaneously in the same subcase for the residual structure only. Do not specify METHOD(FLUID) in a superelement subcase even if the superelement contains fluid elements.

- The auto-omit feature (see "**Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS**" on page 449 of the *MSC.Nastran Reference Guide*) is not recommended. Therefore, only those methods of eigenvalue extraction that can process a singular mass matrix should be used; e.g., EIGRL entry or MGIV and MHOU on the EIGR entry.

MFLUID Fluid Boundary Element Selection

Selects the MFLUID Bulk Data entries to be used to specify the fluid-structure interface.

Format:

MFLUID=n

Example:

MFLUID=919

Describer	Meaning
n	Set identification number of one or more MFLUID Bulk Data entries. (Integer > 0)

Remark:

1. For a further discussion see "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*.
2. User PARAMeter VMDPT controls how the virtual mass is processed.

MODALKE Modal Kinetic Energy Request

Requests modal kinetic energy calculation and specifies the output form.

Format:

$$\text{MODALKE}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right]\left[\begin{matrix}\text{PRINT}\\ \text{NOPRINT}\end{matrix}\right],\ \text{PUNCH}\right]\left[\begin{matrix}\text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right]\left[\text{ESORT} = \left\{\begin{matrix}\text{MODE}\\ \text{ASCEND}\\ \text{DESCENT}\end{matrix}\right\}\right]$$

$$[\text{THRESH} = e]\left[\left\{\begin{matrix}\text{TIME}\\ \text{FREQ}\end{matrix}\right\} = \left\{\begin{matrix}\text{ALL}\\ r\end{matrix}\right\}\right]\Big)\Big] = \left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}$$

Examples:

MODALKE= ALL
SET 200= 1, 3, 4, 5, 7
MODALKE(ESORT=DESCEND,THRESH=.001)= 200

Describer	Meaning
SORT1	Output will be presented as a tabular listing of modes for each frequency or time step.
SORT2	Output will be presented as a tabular listing of frequencies or time steps for each mode.
PRINT	Write the results to the .f06 file. (Default)
NOPRINT	Do not write the results to the .f06 file.
PUNCH	Write the results to the punch (.f07) file.
ESORT	Present the modal energies sorted by mode number, ascending energy value or descending energy value. (Default is MODE).
THRESH	Write out only those energies greater than e. (Default = 0.001).
MODES	Compute energies for all modes or the set of mode numbers defined in SET n. (Default = ALL).
TIME or FREQ	Compute energies at all time steps or frequencies or the set of frequencies defined by SET r. (Default = ALL).
ALL, n, NONE	Compute modal energies for (1) all modes, (2) the modes defined on SET n, or (3) no modes.

Remarks:

1. Modal kinetic energy calculations will be limited to solutions 112 (modal transient response) and 111 (modal frequency response).
2. The MODES describer selects from the set of the modes selected by the combination of the Case Control command MODESELECT and user PARAMeters LMODES, LFREQ, and HFREQ. If a mode is selected outside this set, a User Warning Message is issued.
3. The TIME (or FREQ) describer selects from the set of the time steps (or forcing frequencies) selected by the OTIME (or OFREQ) Case Control command. If a time or frequency is selected outside this set, a User Warning Message is issued.

Output Format:

The output formats will be as follows:

For SORT1 option:

FREQUENCY RESPONSE OUTPUT (SORT2):

MODE NUMBER = 1

M O D A L K I N E T I C E N E R G Y

FREQUENCY	ACTUAL	NORMALIZED	FRACTIONAL
1.000000E+00	8.147639E-04	1.000000E+00	9.924864E-01
2.000000E+00	4.066131E-03	1.000000E+00	9.936857E-01
3.000000E+00	1.411670E-02	1.000000E+00	9.955439E-01
4.000000E+00	5.744822E-02	1.000000E+00	9.977741E-01
5.000000E+00	7.744190E-01	1.000000E+00	9.996839E-01
6.000000E+00	5.961162E-01	1.000000E+00	9.991935E-01
7.000000E+00	8.745764E-02	1.000000E+00	9.877571E-01
8.000000E+00	3.625540E-02	1.000000E+00	9.107288E-01
9.000000E+00	2.053458E-02	3.622278E-01	2.652422E-01
1.000000E+01	1.350744E-02	7.244376E-01	4.169486E-01

Above form repeated for each time or frequency.

For SORT2 option:

```
TRANSIENT RESPONSE OUTPUT (SORT2):

      MODE NUMBER =         1
                                    M O D A L   K I N E T I C   E N E R G Y

     TIME STEP                      ACTUAL                    NORMALIZED                 FRACTIONAL
   0.000000E+00                    .0                         .0                         .0
   9.999999E-02                   2.997325E-04               1.193021E-01               8.875214E-02
   2.000000E-01                   1.107705E-03               1.306918E-01               1.025178E-01
   3.000000E-01                   2.177984E-03               1.538738E-01               1.193877E-01
   4.000000E-01                   3.184835E-03               1.991635E-01               1.439702E-01
   5.000000E-01                   3.821760E-03               2.977550E-01               1.852529E-01
   6.000000E-01                   3.894964E-03               5.844937E-01               2.506738E-01
   7.000000E-01                   3.382083E-03               8.283827E-01               3.144183E-01
   8.000000E-01                   2.439315E-03               5.028039E-01               2.665924E-01
   9.000000E-01                   1.353515E-03               2.459573E-01               1.108626E-01
   1.000000E+00                   4.552519E-04               4.435274E-02               2.527509E-02
```

The above form is repeated for each mode.

Processing:

Modal kinetic energy will be calculated by the following:

[Actual Kinetic Energy] = $ABS[0.5[diag(\omega_i^2)][M_{hh}][u_h] \cdot [u_h]]$

for frequency response, ω is the excitation frequency and the absolute value of u_h is used.

$0.5[M_{hh}][\dot{u}_h] \cdot [\dot{u}_h]$

for transient response

[Normalized Kinetic Energy] = norm[Actual Kinetic Energy], normalized per column

[Fractional Strain Energy] = [Normalized Kinetic Energy]/[diagonal $[\{1.0\}^T$ [Normalized Kinetic Energy]]], term-by-term division

MODALSE Modal Strain Energy Request

Requests modal strain energy calculation and specifies the output form.

Format:

$$\text{MODALSE}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right]\left[\begin{matrix}\text{PRINT}\\ \text{NOPRINT}\end{matrix}\right],\text{PUNCH}\right]\left[\begin{matrix}\text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right]\left[\text{ESORT}=\left\{\begin{matrix}\text{MODE}\\ \text{ASCEND}\\ \text{DESCENT}\end{matrix}\right\}\right]$$

$$\left[\text{THRESH}=e\right]\left[\left\{\begin{matrix}\text{TIME}\\ \text{FREQ}\end{matrix}\right\}=\left\{\begin{matrix}\text{ALL}\\ r\end{matrix}\right\}\right]\Bigg)=\left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}$$

Examples:

MODALSE= ALL
SET 100= 1, 3, 4, 5, 7
MODALSE(ESORT=ASCEND,THRESH=.0001)= 100

Describer	Meaning
SORT1	Output will be presented as a tabular listing of modes for each frequency or time step.
SORT2	Output will be presented as a tabular listing of frequencies or time steps for each mode.
PRINT	Write the results to the .f06 file. (Default)
NOPRINT	Do not write the results to the .f06 file.
PUNCH	Write the results to the punch (.f07) file.
ESORT	Present the modal energies sorted by mode number, ascending energy value or descending energy value. (Default is MODE).
THRESH	Write out only those energies greater than e. (Default = 0.001).
MODES	Compute energies for all modes or the set of mode numbers defined in SET n. (Default = ALL).
TIME or FREQ	Compute energies at all time steps or frequencies or the set of frequencies defined by SET r. (Default = ALL).
ALL, n, NONE	Compute modal energies for (1) all modes, (2) the modes defined on SET n, or (3) no modes.

Remarks:

1. Modal strain energy calculations will be limited to solutions 112 (modal transient response) and 111 (modal frequency response).
2. The MODES describer selects from the set of the modes selected by the combination of the Case Control command MODESELECT and user PARAMeters LMODES, LFREQ, and HFREQ. If a mode is selected outside this set, a User Warning Message is issued.
3. The TIME (or FREQ) describer selects from the set of the time steps (or forcing frequencies) selected by the OTIME (or OFREQ) Case Control command. If a time or frequency is selected outside this set, a User Warning Message is issued.

Output Format:

The output formats will be as follows:

For SORT1 option:

FREQUENCY RESPONSE OUTPUT (SORT1):

FREQUENCY = 2.000000E+00

M O D A L S T R A I N E N E R G Y

MODE NUMBER	ACTUAL	NORMALIZED	FRACTIONAL
1	3.013590E-02	1.000000E+00	7.759020E-01
2	3.676222E-04	1.219881E-02	9.465083E-03
3	1.649538E-04	5.473664E-03	4.247027E-03
4	8.880605E-06	2.946852E-04	2.286469E-04
5	6.064836E-03	2.012495E-01	1.561499E-01
6	.0	.0	.0
7	8.556397E-05	2.839270E-03	2.202996E-03
8	1.596696E-03	5.298317E-02	4.110975E-02
9	.0	.0	.0
10	3.678207E-05	1.220540E-03	9.470194E-04

The above form is repeated for each time or frequency.

For SORT2 option:

TRANSIENT RESPONSE OUTPUT (SORT1):

TIME STEP = 2.000000E-01

M O D A L S T R A I N E N E R G Y

MODE NUMBER	ACTUAL	NORMALIZED	FRACTIONAL
1	9.122948E-05	5.796646E-02	3.374175E-02
2	1.573832E-03	1.000000E+00	5.820910E-01
3	4.954743E-04	3.148203E-01	1.832541E-01
4	1.456887E-06	9.256940E-04	5.388382E-04
5	7.806947E-06	4.960469E-03	2.887445E-03
6	.0	.0	.0
7	1.082568E-04	6.878546E-02	4.003940E-02
8	1.503170E-05	9.551016E-03	5.559560E-03
9	.0	.0	.0
10	6.410905E-06	4.073436E-03	2.371111E-03

The above form is repeated for each mode.

Processing:

Modal strain energy is calculated by the following:

[Actual Strain Energy]	=	$ABS[0.5[K_{hh}][u_h] \cdot [u_h]]$ for frequency response, use the absolute value of u_h.
[Normalized Strain Energy]	=	norm[Actual Strain Energy], normalized per column
[Fractional Strain Energy]	=	[Normalized Strain Energy]/[diagonal $[\{1.0\}^T$ [Normalized Strain Energy]]], term-by-term division.

MODES Subcase Repeater

Repeats a subcase.

Format:

MODES=n

Example:

MODES=3

Describer	Meaning
n	Number of times the subcase is to be repeated. (Integer>0)

Remarks:

1. This command can be illustrated by an example. Suppose stress output is desired for the first five modes only and displacements for the next two modes and forces for the remaining modes. The following example would accomplish this

```
SUBCASE 1 $ FOR MODES 1 THRU 5
MODES = 5
STRESS = ALL
SUBCASE 6 $ FOR MODES 6 AND 7
DISPLACEMENTS =
MODES = 2
SUBCASE 8 $ FOR MODE 8 AND REMAINING MODES
FORCE = ALL
```

2. This command causes the results for each mode to be considered as a separate, successively numbered subcase, beginning with the subcase number containing the MODES command. In the example above, this means that subcases 1, 2, 3, etc. are assigned to modes 1, 2, 3, etc., respectively.
3. If this command is not used, eigenvalue results are considered to be a part of a single subcase. Therefore, any output requests for the single subcase will apply for all eigenvalues.

4. All eigenvectors with mode numbers greater than the number of subcases defined in Case Control are printed with the descriptors of the last subcase. For example, to suppress all printout for modes beyond the first three, the following Case Control could be used:

```
SUBCASE 1
        MODES = 3
        DISPLACEMENTS = ALL
SUBCASE 4
        DISPLACEMENTS = NONE
BEGIN BULK
```

5. This command may be of no use in non eigenvalue analysis and may cause data recovery processing to be repeated.

MODESELECT Computed Mode Selection

Selects a subset of the computed modes for inclusion or exclusion in modal dynamic analysis.

Format:

(Mode selection based on arbitrary mode numbers)

$$\text{MODESELECT}\left(\left[\begin{matrix}\text{STRUCTURE}\\ \text{FLUID}\end{matrix}\right]\right) = n$$

Alternate Format 1:

(Mode selection based on number of lowest modes)

$$\text{MODESELECT}\left(\left[\begin{matrix}\text{STRUCTURE}\\ \text{FLUID}\end{matrix}\right]\right)\text{LMODES} = lm$$

Alternate Format 2:

(Mode selection based on range of mode numbers)

$$\text{MODESELECT}\left(\left[\begin{matrix}\text{STRUCTURE}\\ \text{FLUID}\end{matrix}\right][\text{LMODENM} = lom][\text{HMODENM} = him]\right)$$

Alternate Format 3:

(Mode selection based on frequency range)

$$\text{MODESELECT}\left(\left[\begin{matrix}\text{STRUCTURE}\\ \text{FLUID}\end{matrix}\right][\text{LFREQ} = lof][\text{HFREQ} = hif]([\text{UNCONSET}] = m)\right)$$

Alternate Format 4:

(Mode selection based on modal effective mass fraction (MEFFMFRA) criteria)

$$\text{MODESELECT}\left(\left[\begin{matrix}\text{STRUCTURE}\\ \text{FLUID}\end{matrix}\right][\text{T1FR}[= t1fr]][\text{T2FR}[= t2fr]][\text{T3FR}[= t3fr]]$$
$$[\text{R1FR}[= r1fr]][\text{R2FR}[= r2fr]][\text{R3FR}[= r3fr]]$$
$$[\text{ALLFR}[= allfr]][\text{UNCONSET} = m]\left[\begin{matrix}\text{SUM}\\ \text{ANYMIN}\\ \text{ALLMIN}\end{matrix}\right]$$

See Remark **14.** for examples illustrating the usage of all of the above formats.

Describer	Meaning
STRUCTURE	The command references computed modes of the structure. (Default)
FLUID	The command references computed modes of the fluid.
$n > 0$	Set identification of a previously appearing SET command. ONLY those modes whose mode numbers are in SET *n* will be included in the analysis. If SET *n* is not defined, then ONLY mode *n* will be included in the analysis. (Integer)
$n < 0$	$\|n\|$ refers to the set identification of a previously appearing SET command. The modes whose mode numbers are in SET $\|n\|$ will be EXCLUDED from the analysis. If SET $\|n\|$ is not defined, then mode $\|n\|$ will be EXCLUDED from the analysis. (Integer)
lm	Number of lowest modes that are to included. (Integer > 0)
lom	Lower limit of the mode number range for selecting the modes. See Remark **5.** (Integer > 0)
him	Upper limit of the mode number range for selecting the modes. See Remark **5.** (Integer > *lom* > 0)
lof	Lower limit of the frequency range for selecting the modes. See Remark **6.** (Real $\geq$ 0.0).
hif	Upper limit of the frequency range for selecting the modes. See Remark **6.** (Real > *lof* $\geq$ 0.0)
UNCONSET	Specifies a single mode or a set of modes for *unconditional* inclusion or exclusion, *regardless of the selection criterion and regardless of the inclusion or exclusion of other modes*.
$m > 0$	Set identification of a previously appearing SET command. Modes whose mode numbers are in SET *m* will be included in the analysis, *regardless of the selection criterion and regardless of the inclusion or exclusion of other modes.* If SET *m* is not defined, then mode *m* will be included in the analysis, *regardless of the selection criterion and regardless of the inclusion or exclusion of other modes.* (Integer)

Describer	Meaning
$m < 0$	$\|m\|$ refers to the set identification of a previously appearing SET command. Modes whose mode numbers are in SET $\|m\|$ will be EXCLUDED from the analysis, *regardless of the selection criterion and regardless of the inclusion or exclusion of other modes*. If SET $\|m\|$ is not defined, then mode *m* will be EXCLUDED from the analysis, *regardless of the selection criterion and regardless of the inclusion or exclusion of other modes*. (Integer)
TiFR / RiFR	Flags explicitly listing the components whose modal effective mass fraction (MEFFMFRA) values are to be considered for mode selection.
tifr / *rifr*	Threshold values for the listed TiFR / RiFR components. See Remark **8.** (0.0 < Real ≤ 1.0)
ALLFR	Flag indicating that the MEFFMFRA values of those components that are not explicitly listed by the TiFR / RiFR flags are also to be considered for mode selection.
allfr	Threshold value for those components that are not explicitly listed by the TiFR / RiFR flags. See Remark **8.** (0.0 < Real ≤ 1.0)
SUM	For *each* specified component, the modes are selected as follows: The modes are first sorted in *descending* order of the corresponding MEFFMFRA values. Then, starting from the first mode in this sorted list, the modes are selected until the *sum* of corresponding MEFFMFRA values equals or just exceeds the threshold value for that component. (Default)
ANYMIN	Any mode whose MEFFMFRA value for *any* specified component equals or exceeds the threshold value for that component will be selected.
ALLMIN	Any mode whose MEFFMFRA values for *all* of the specified components equal or exceed the corresponding threshold values for those components will be selected.

Remarks:

1. This command is meaningful only in modal dynamic analysis (SOL 110, 111, 112, 145, 146 and 200). It is ignored in all other analyses.
2. Only one MODESELECT command is allowed and it should be specified above the subcase level.

3. The various formats of this command may not be combined.
4. It should be noted that the computed modes used for mode selection include the augmented modes (if any) resulting from residual vector calculations.
5. If LMODENM is specified without HMODENM, a default value of 10000000 (ten million) is assumed for HMODENM. If HMODENM is specified without LMODENM, a default value of 1 is assumed for LMODENM.
6. If LFREQ is specified without HFREQ, a default value of 1.0E+30 is assumed for HFREQ. If HFREQ is specified without LFREQ, a default value of 0.0 is assumed for LFREQ.
7. If the format involving the MEFFMFRA criteria is employed, it is *not necessary* for the user to specify a MEFFMASS command in Case Control or, even if such a command is specified, to explicitly request the calculation of the modal effective mass fractions. In the absence of such a command or such a request, the program will automatically perform the necessary calculations internally to ensure that the required modal effective mass fractions are computed.
8. If the T1FR / R1FR / ALLFR keywords are specified without the corresponding *tifr* / *r1fr* / *allfr* threshold values, then a default value of 0.95 (that is, 95%) is assumed for these threshold values if the selection criterion is SUM and a default value of 0.05 (that is, 5%) is assumed if the selection criterion is ANYMIN or ALLMIN.
9. The modal effective mass for a given mode is a measure of how much mass is associated with that mode and indicates the sensitivity of that mode to base excitation. Modal effective mass is meaningful only for fixed base modes. If a structure is not restrained, all the modal effective mass will be associated with its rigid body modes.
10. When the MODESELECT Case Control command is used in conjunction with the parameters LMODES / LMODESFL, LFREQ / LFREQFL and HFREQ / HFREQFL, the hierarchy of their usage is as follows:

    (a) If there is a MODESELECT Case Control command, it takes precedence over the parameters LMODES / LMODESFL, LFREQ / LFREQFL and HFREQ / HFREQFL. (It does not matter whether these parameters are defined directly via PARAM entries or indirectly via the FLSFSEL Case Control command.)

    (b) If there is no MODESELECT Case Control command, then parameter LMODES/LMODESFL takes precedence over the parameters LFREQ/LFREQFL and HFREQ/HFREQFL. In this case, the number of lowest

modes specified by LMODES/LMODESFL will be included in the modal dynamic analysis.

(c) If there is no MODESELECT Case Control command and no LMODES/LMODESFL parameter, then parameters LFREQ/LFREQFL and HFREQ/HFREQFL are honored. In this case, all of the computed modes whose frequencies are in the range specified by LFREQ/LFREQFL and HFREQ/HFREQFL will be included in the modal dynamic analysis.

(d) If there is no MODESELECT Case Control command and no LMODES/LMODESFL, LFREQ/LFREQFL or HFREQ/HFREQFL parameter, then all of the computed modes will be included in the modal dynamic analysis.

11. If a subset of the computed modes is selected for subsequent use in the modal dynamic analysis, the user is informed of this via an user information message. Further, in this case, a new eigenvalue table indicating the actual modes selected for the analysis is output. In addition, if the user has employed a MODESELECT command involving the MEFFMFRA criteria, the modal effective mass fractions for the selected modes are also output.
12. If the mode selection criterion results in no modes being selected for subsequent use in the modal dynamic analysis, the program terminates the job with a fatal message indicating that no modal formulation is possible.
13. If the use of the MODESELECT command results in the selection of all of the computed modes for subsequent use, the user is informed of this via an user information message.
14. The following examples illustrate the usage of the various formats of the MODESELECT command described above.

Examples Illustrating Mode Selection Based on Arbitrary Mode Numbers:

```
$ INCLUDE ONLY STRUCTURE MODES 7, 9 AND 12 IN THE ANALYSIS
SET 100 = 7,9,12
MODESELECT = 100

$ EXCLUDE FLUID MODES 5 AND 6 FROM THE ANALYSIS
SET 200 = 5,6
MODESELECT (FLUID)= -200
```

```
$ EXCLUDE STRUCTURE MODE 5 FROM THE ANALYSIS
MODESELECT = -5  $  (SET 5 NOT DEFINED)
```

Examples Illustrating Mode Selection Based on Number of Lowest Modes:

```
$ INCLUDE THE LOWEST 10 STRUCTURE MODES IN THE ANALYSIS
MODESELECT (LMODES = 10)

$ INCLUDE THE LOWEST 5 FLUID MODES IN THE ANALYSIS
MODESELECT (FLUID LMODES = 5)
```

Examples Illustrating Mode Selection Based on Range of Mode Numbers:

```
$ INCLUDE ONLY STRUCTURE MODES 10 THRU 20 IN THE ANALYSIS
MODESELECT (LMODENM = 10  HMODENM = 20)

$ INCLUDE ALL STRUCTURE MODES HIGHER THAN THE 6th MODE
$ IN THE ANALYSIS
MODESELECT (LMODENM = 7)

$ INCLUDE THE LOWEST 10 FLUID MODES IN THE ANALYSIS
MODESELECT (FLUID  HMODENM = 10)
```

Examples Illustrating Mode Selection Based on Frequency Range:

```
$ INCLUDE ALL STRUCTURE MODES WITH CYCLIC FREQUENCIES
$ IN THE RANGE OF 0.1 HZ. TO 100.0 HZ. IN THE ANALYSIS
MODESELECT (LFREQ = 0.1  HFREQ = 100.0)

$ INCLUDE ALL STRUCTURE MODES WITH CYCLIC FREQUENCIES
$ EQUAL TO OR BELOW 50.0 HZ., BUT INCLUDE THE 10th AND 11th
$ MODES REGARDLESS OF THEIR CYCLIC FREQUENCIES
SET 1000 = 10, 11
MODESELECT (HFREQ = 50.0  UNCONSET = 1000)

$ INCLUDE ALL STRUCTURE MODES WITH CYCLIC FREQUENCIES
$ EQUAL TO OR ABOVE 5.0 HZ., BUT EXCLUDE THE 6 MODE
$ REGARDLESS OF ITS CYCLIC FREQUENCY
MODESELECT (LFREQ = 5.0  UNCONSET = -6)  $  SET 6 NOT DEFINED
```

Examples Illustrating Mode Selection Based on Modal Effective Mass Fraction (MEFFMFRA) Criteria:

```
MODESELECT (T3FR)
```

The default selection criterion of SUM is assumed and a default value of 0.95 (95%) is therefore assumed for the threshold value for component T3.

As many modes with the highest MEFFMFRA(T3) values as possible such that the sum of those values is equal to or just exceeds 0.95 will be selected.

MODESELECT (T1FR = 0.90 T2FR R3FR = 0.85)

The default selection criterion of SUM is assumed and a default value of 0.95 (95%) is therefore assumed for the threshold value for component T2.

As many modes with the highest MEFFMFRA(T1) values as possible such that the sum of those values is equal to or just exceeds 0.90 will be selected.

Similarly, as many modes with the highest MEFFMFRA(T2) values as possible such that the sum of those values is equal to or just exceeds 0.95 will be selected.

Again, as many modes with the highest MEFFMFRA(R3) values as possible such that the sum of those values is equal to or just exceeds 0.85 will be selected.

MODESELECT (T1FR T3FR = 0.10 UNCONSET = -6 ANYMIN)
$ SET 6 NOT DEFINED

Since the selection criterion is specified as ANYMIN, a default value of 0.05 (5%) is assumed for the threshold value for component T1.

All modes, *excluding* mode 6, whose:

MEFFMFRA(T1) values are equal to or greater than 0.05 *OR*
MEFFMFRA(T3) values are equal to or greater than 0.10

will be selected.

SET 1000 = 20, 30
MODESELECT (T2FR = 0.1 R3FR = 0.15 ALLFR UNCONSET = 1000 ALLMIN)

The ALLFR flag indicates that the T1, T3, R1 and R2 components that are not explicitly specified above must also be considered in mode selection. Since the selection criterion is specified as ALLMIN, a default value of 0.05 (5%) is assumed for the threshold value for these components.

All modes whose:

MEFFMFRA(T1) values equal or exceed 0.05 *AND*
MEFFMFRA(T2) values equal or exceed 0.10 *AND*
MEFFMFRA(T3) values equal or exceed 0.05 *AND*
MEFFMFRA(R1) values equal or exceed 0.05 *AND*
MEFFMFRA(R2) values equal or exceed 0.05 *AND*
MEFFMFRA(R3) values equal or exceed 0.15 *AND*

will be selected.

Modes 20 and 30 will be selected regardless of their MEFFMFRA values.

MODTRAK Mode Tracking Request

Selects mode tracking options in design optimization (SOL 200).

Format:

MODTRAK=n

Example:

MODTRAK=100

Describer	Meaning
n	Set identification of a MODTRAK Bulk Data entry. (Integer>0)

Remark:

1. Selection of a MODTRAK Bulk Data entry with the MODTRAK Case Control command activates mode tracking for the current subcase. This request is limited to normal modes subcases (ANALYSIS=MODES) in design optimization (SOL 200).

MONITOR Print Selection for Monitor Data

Specifies options in the print of Monitor Data.

Format:

$$\text{METHOD[REAL or IMAG, PHASE, NODSP1, NOPNT1, NOPNT2, NOPNT3]} = \left\{ \begin{matrix} ALL \\ NONE \end{matrix} \right\}$$

Example:

MONITOR(PHASE,NOPNT1)=ALL
MONITOR(IMAG,NODSP1)=ALL

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
NODSP1	Do not include MONDSP1 results in the MONITOR point prints (default is to provide these prints)
NOPNT1	Do not include MONPNT1 results in the MONITOR point prints (default is to provide these prints).
NOPNT2	Do not include MONPNT2 results in the MONITOR point prints (default is to provide these prints).
NOPNT3	Do not include MONPNT3 results in the MONITOR point prints (default is to provide these prints).
ALL	Perform the print of all monitor point results, except for those deselected.
NONE	Turn off the print of monitor point results

Remarks:

1. The MONITOR command is required in order to obtain MONITOR results in the printed output.
2. The MONITOR command should be above the subcase level or in the first subcase. MONITOR commands in subcases subsequent to the first one are ignored.

MPC Multipoint Constraint Set Selection

Selects a multipoint constraint set.

Format:

MPC=n

Example:

MPC=17

Describer	Meaning
n	Set identification number of a multipoint constraint set. This set identification number must appear on at least one MPC or MPCADD Bulk Data entry. (Integer>0)

Remarks:

1. In cyclic symmetry analysis, this command must appear above the first SUBCASE command.
2. Multiple boundary (MPC sets) conditions are not allowed in superelement analysis. If more than one MPC set is specified per superelement (including the residual), then the second and subsequent sets will be ignored.

MPCFORCES Multipoint Forces of Constraint Output Request

Requests the form and type of multipoint force of constraint vector output.

Format:

$$\text{PCFORCES}\left[\left(\left[\begin{matrix}\text{SORT1}\\\text{SORT2}\end{matrix}\right],\left[\begin{matrix}\text{PRINT, PUNCH}\\\text{PLOT}\end{matrix}\right],\left[\begin{matrix}\text{REAL or IMAG}\\\text{PHASE}\end{matrix}\right],\left[\begin{matrix}\text{PSDF, ATOC, CRMS}\\\text{or RALL}\end{matrix}\right]\right.\right.$$

$$\left.\left.\left[\begin{matrix}\text{RPRINT}\\\text{NORPRINT}\end{matrix}\right],\text{RPUNCH}\right],[\text{CID}])\right] = \left\{\begin{matrix}\text{ALL}\\\text{n}\\\text{NONE}\end{matrix}\right\}$$

Examples:

MPCFORCES=5
MPCFORCES(SORT2, PUNCH, PRINT, IMAG)=ALL
MPCFORCES(PHASE)=NONE
MPCFORCES(SORT2, PRINT,IPSDF, CRMS, RPUNCH)=20
MPCFORCES(PRINT, RALL, NORPRINT)=ALL

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, multipoint constraint forces.
REAL or IMAGE	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.

Describer	Meaning
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RALL	Requests all of PSDF, ATOC and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output file, F06 file.
ALL	Multipoint forces of constraint for all points will be output. See Remarks **3.** and **6.**
NONE	Multipoint forces of constraint for no points will be output.
n	Set identification of a previously appearing SET command. Only multipoint forces constraint for points with identification numbers that appear on this SET command will be output. (Integer > 0)

Remarks:

1. Both PRINT and PUNCH may be requested.
2. See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2. In the SORT1 format only nonzero values will be output.
3. In a statics problem, a request for SORT2 causes loads at all points (zero and nonzero) to be output.
4. MPCFORCES=NONE overrides an overall output request.

5. In SORT1 format, MPCFORCEs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. But if a scalar point is not consecutively numbered, then it will begin a new sextet on a new line of output. If a sextet can be formed and all values are zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
6. MPCFORCE results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
7. MPCFORCE results are not available in SOL 129.
8. In inertia relief analysis, the MPCFORCE output includes both the effects of applied and inertial loads.
9. The option of PSDF, ATOC, CRMS and RALL, or any combination of them, can be selected for random analysis. The results can be either printed in the .f06 file or punched in the punch file, or output in both files.
10. Note that the CID keyword affects only grid point related output, such as DISPlacement, VELOcity, ACCEleration, OLOAD, SPCForce and MPCForce. In addition, CID keyword needs to appear only once in a grid related output request anywhere in the Case Control Section to turn on the printing algorithm.

MPRES Fluid Pressure Output Request

Requests the pressure for selected wetted surface elements when virtual mass (MFLUID) is used.

Format:

$$\text{PRES}\left[\left(\left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right],\left[\begin{matrix}\text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ n\\ \text{NONE}\end{matrix}\right\}$$

Examples:

MPRES=5
MPRES(IMAG)=ALL

4A CASE

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print or punch, data.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Fluid pressures for all elements will be output.
NONE	Fluid pressures for no elements will be output.
n	Set identification number of a previously appearing SET command. Only fluid pressures for elements in this set will be output. (Integer >0)

Remark:

1. If PARAM,SPARSEDR,NO is specified, then PARAM,DDRMM,-1 is also required in the modal solution sequences 111, 112, 146, and 200.

NLIC Nonlinear Initial Condition

Selects a previous exectuved load increment as the initial conditions for a nonlinear transient STEP in SOL 400.

Format:

NLIC [*SUBCASE i* [, *STEP j* [, *LOADFAC f*]]]

Describer	Meaning
i	Specifies the identification number of a previously executed SUBCASE. (Integer, default is the SUBCASE where the current NLIC is located.)
j	Specifies the identification number of a previously executed STEP. (Integer, default is the last STEP)
f	Specified the load factor of a previously executed load increment in linear or nonlinear static analysis. (Real, $1.0 \geq f > 0.0$)

Remarks:

1. The NLIC command can only point to a load increment whose output flag is on - an available restart point in the static analysis. If NLIC is not pointing to an available restart point, a fatal error will be issued and the job will be terminated.
2. NLIC can appear only in the first transient analysis STEP (ANALYSIS=NLTRAN) in a SUBCASE. (Note that the first transient analysis STEP may not be the first STEP of a SUBCASE.) Otherwise, it will be ignored.
3. If NLIC is specified without any of the keywords or NLIC is not present in a transient STEP, the initial condition is taken from the last available restart point in the immediate previous static STEP.
4. In the same STEP, NLIC cannot appear together with the IC case control command. A fatal error message will be issue if NLIC and IC appear in the same STEP. Please note that IC is meaningful only in the first STEP of a SUBCASE and the STEP is a nonlinear transient analysis
5. NLIC can be used in SOL 400 only.

NLLOAD Nonlinear Load Output Request

Requests the form and type of nonlinear load output for transient problems.

Format:

$$\text{NLLOAD}[(\text{PRINT}, \text{PUNCH})] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Example:

NLLOAD=ALL

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
ALL	Nonlinear loads for all solution points will be output.
NONE	Nonlinear loads will not be output.
n	Set identification of a previously appearing SET command. Only nonlinear loads for points with identification numbers that appear on this SET command will be output. (Integer >0)

Remarks:

1. Nonlinear loads are output only in the solution (d or h) set.
2. The output is available in SORT2 format only.
3. Both PRINT and PUNCH may be used.
4. NLLOAD=NONE allows overriding an overall output request.

NLPARM Nonlinear Static Analysis Parameter Selection

Selects the parameters used for nonlinear static analysis.

Format:

NLPARM=n

Example:

NLPARM=10

Describer	Meaning
n	Set identification of NLPARM and NLPCI Bulk Data entries. (Integer>0)

Remarks:

1. NLPARM and NLPCI entries in the Bulk Data will not be used unless selected.
2. NLPARM may appear above or within a subcase.
3. For SOL 600, the only fields used are ID, NINC, DT (creep only), EPSU, EPSP, and EPSW. Use PARAM,MARCOTIM instead of INTOUT. For other fields, advanced convergence controls are available using NLAUTO, NLSTRAT and PARAM,MARCDEF Bulk Data entries.

NLRESTART Nonlinear Restart Request

Request a RESTART execution at a specified point for SOL 400.

Format:

$$\text{NLRESTART}\left[\text{SUBCASE i}\left[,\ \text{STEP j}\left[,\ \begin{matrix}\text{LOADFAC f}\\ \text{TIME t}\end{matrix}\right]\right]\right]$$

Example:

NLTESTART SUBCASE 1, STEP 2, LOADFAC 0.3

Describer	Meaning
i	Specifies the identification number of a previously executed SUBCASE. (Integer, Default is the first SUBCASE)
j	Specifies the identification number of a previously executed STEP. (Integer, Default is the first STEP)
f	Specified the load factor of a previously executed load increment in nonlinear static analysis. (Real, $0.0 \le f \le 1.0$, Default = 0.0)
t	Specified the time of a previously executed time step in nonlinear transient analysis. (Real, $t_0 \le t \le t_n$ where t_0: initial time of STEP j and t_n: last time of STEP j, Default = t_0)

Remarks:

1. The NLRESTART command can be used in SOL 400 (NONLIN) only.
2. The NLRESTART command must appear before any SUBCASE command.
3. To perform a restart, the data base for the original run must be made available by using the ASSIGN FMS command or other equivalent method.
4. The restart run can only be executed at a load increment (or time step) whose output flag is on an available restart point. (See the field INTOUT on NLPARM Bulk Data entry and NO on TSTEPNL). When user specified restart point is not available, the closest previous restart point, that is available will be applied automatically.

5. If only NLRESTART is specified, a restart begins from the last available restart point in the previous run. Otherwise, at lease one set of the SUBCASE i, STEP j or LOADFC f (or TIME t) must be specified.
6. In static analysis, f is reset to 0.0 when f < 0.0 and it makes restart begin from the beginning of STEP j. f is reset to 1.0 when f > 1.0 that makes restart begin from the beginning of the next STEP (after STEP j).
7. In transient analysis, t is reset to t_0 when $t < t_0$ and it makes the restart begin from the beginning of STEP j. t is reset to t_n when $t > t_n$ that makes the restart begin from the beginning of the next STEP (after STEP j).
8. The Case Control command must contain all of the commands used in the original execution up to the point where the restart is requested.
9. All data contained on the database from the restart point will be deleted when the restart begins.

NLSTRESS Nonlinear Element Stress Output Request

Requests the form and type of nonlinear element stress output in SOLs 106 and 400.

Format:

$$\text{NLSTRESS}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right],\ \left[\begin{matrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Examples:

NLSTRESS=5
NLSTRESS (SORT1,PRINT,PUNCH,PHASE)=15
NLSTRESS(PLOT)=ALL

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load.
SORT2	Output will be presented as a tabular listing of load for each element type.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates nonlinear element stresses for requested set but no printer output.
ALL	Stresses for all nonlinear elements will be output. (Default)
n	Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	No nonlinear element stress will be output.

Remarks:

1. Both PRINT and PUNCH may be requested.
2. ALL should not be used in a transient problem due to excessive output.
3. See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2.
4. If the NLSTRESS command is not specified, then NLSTRESS=ALL is the default.

NONLINEAR Nonlinear Dynamic Load Set Selection

Selects nonlinear dynamic load set for transient problems.

Format:

NONLINEAR=n

Example:

NONLINEAR=75

Describer	Meaning
n	Set identification of NOLINi or NLRGAP Bulk Data entry. (Integer>0)

Remark:

1. NOLINi Bulk Data entry will be ignored unless selected in the Case Control Section.

NOUTPUT Normal Output Request in Cyclic Symmetry Problems

Requests physical output in cyclic symmetry problems.

Format:

$$\text{NOUTPUT}\left\{\begin{matrix}\text{k,} & \text{R} \\ & \text{L}\end{matrix}\right\} = \left\{\begin{matrix}\text{ALL} \\ \text{m}\end{matrix}\right\}$$

Examples:

NOUTPUT (R)=ALL
NOUTPUT (2)=5
NOUTPUT (4,L)=10

Describer	Meaning
ALL	Output for all segments is desired.
m	Output for segments specified in SET m is desired. (Integer>0)
k	Used in eigenvalue analysis to request eigenvector and internal force output for harmonics specified in SET k. (Integer>0)
R, L	Output for only the right- or left-half of segments specified as ALL or in SET m. R and L are used in dihedral symmetry only.

Remarks:

1. Sets k and m are defined on SET commands.
2. In cyclic symmetry analysis, this command or the HOUTPUT command is required to obtain data recovery.

NSM Selects Nonstructural Mass Set Entries

Selects Nonstructural Mass (NSM) set for mass generation.

Format:

NSM = n

Example:

NSM = 5

Describer	Meaning
n	Set identification number of a nonstructural mass that appears on a NSM, NSML, NSM1, NSML1, or NSMADD Bulk Data entry. (Integer > 0)

Remark:

1. Different NSM sets may be selected for superelements and residual; but within a superelement or residual it may not change within the subcase structure.

OFREQUENCY Output Frequency Set

Selects a set of frequencies for output requests.

Format:

$$\text{OFREQUENCY} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \end{matrix} \right\}$$

Examples:

OFREQUENCY=ALL
OFREQUENCY=15

Describer	Meaning
ALL	Output for all frequencies will be computed.
n	Set identification of a previously appearing SET command. Output for frequencies closest to those given on this SET command will be output. (Integer>0)

Remarks:

1. In real eigenvalue, buckling, and complex eigenvalue analyses, the OMODES Case Control command allows for an alternate way of selecting the modes to be output based on their mode numbers. In these cases, if both the OMODES and OFREQUENCY requests appear, the OMODES request takes precedence.
2. If this command is not specified in the Case Control Section (or, in the case of real eigenvalue, buckling, and complex eigenvalue analyses, if neither the OMODES nor the OFREQUENCY request is specified), then output will be generated for all frequencies.
3. The number of solutions selected will always be equal to the number of quantities in the selected set. The closest values are used.
4. In flutter analysis (SOL 145), the selected set refers to the imaginary part of the complex eigenvalues. The physical interpretation of this quantity depends on the method of flutter analysis as follows:
    - K- or KE-method: Velocity (input units).
    - PK-method: Frequency.

5. In aeroelastic response analysis (SOL 146) with RLOAD selection, the selected set refers to the frequency (cycles per unit time).
6. In complex eigenvalue analysis (SOLs 107, and 110), the selected set refers to the imaginary part of the complex eigenvalues.
7. If this command is specified in more than one subcase, then it is recommended that the first subcase contain OFREQ=ALL and subsequent subcases contain OFREQ=n. Also, data recovery requests should be specified only in the subsequent subcases. For example:

```
SUBCASE 1
OFREQ = ALL $ 0.0  through 0.5
SUBCASE 2
SET 10 = 0.0 0.1 0.3
OFREQ = 10
DISP = ALL
SUBCASE3
SET 20 = 0.4 0.5
OFREQ = 20
STRESS = ALL
```

OLOAD Applied Load Output Request

Requests the form and type of applied load vector output.

Format:

$$\text{LOAD}\left[\left(\left[\begin{matrix}\text{SORT1}\\\text{SORT2}\end{matrix}\right],\left[\begin{matrix}\text{PRINT, PUNCH}\\\text{PLOT}\end{matrix}\right],\left[\begin{matrix}\text{REAL or IMAG}\\\text{PHASE}\end{matrix}\right],\left[\begin{matrix}\text{PSDF, ATOC, CRMS}\\\text{or RALL}\end{matrix}\right],\right.\right.$$

$$\left.\left.\left[\begin{matrix}\text{RPRINT}\\\text{NORPRINT}\end{matrix}\right],\text{RPUNCH}\right],[\text{CID}])\right] = \left\{\begin{matrix}\text{ALL}\\\text{n}\\\text{NONE}\end{matrix}\right\}$$

Examples:

OLOAD=ALL
OLOAD(SORT1, PHASE)=5
OLOAD(SORT2, PRINT, PSDF, CRMS, RPUNCH=20
OLOAD(PRINT, RALL, NORPRINT)=ALL

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.

Describer	Meaning
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
CRMS	Requests all of PSDF, ATOC and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RALL	Requests all of PSDF, ATOC and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output file, F06 file.
ALL	Applied loads for all points will be output. See Remarks **2.** and **8.**
NONE	Applied load for no points will be output.
n	Set identification of a previously appearing SET command. Only loads on points with identification numbers that appear on this SET command will be output. (Integer > 0)

Remarks:

1. Both PRINT and PUNCH may be requested.
2. See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
3. In a statics problem, a request for SORT2 causes loads at all requested points (zero and nonzero) to be output.
4. OLOAD=NONE overrides an overall output request.

5. In the statics superelement solution sequences, and in the dynamics SOLs 107 through 112, 118, 145, 146, and 200. OLOADs are available for superelements and the residual structure. Only externally applied loads are printed. Loads transmitted from upstream superelements are not printed. Transmitted loads can be obtained with GPFORCE requests.
    - In the nonlinear transient analysis solution sequences SOLs 129 and 159, OLOADs are available only for residual structure points and include loads transmitted by upstream superelements.
6. In nonlinear analysis, OLOAD output will not reflect changes due to follower forces.
7. Loads generated via the SPCD Bulk Data entry do not appear in OLOAD output.
8. In SORT1 format, OLOADs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. But if a scalar point is not consecutively numbered, then it will begin a new sextet on a new line of output. If a sextet can be formed and it is zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
9. OLOAD results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
10. In static inertia relief analysis the OLOAD output includes both the inertia loads and applied loads.
11. The option of PSDF, ATOC, CRMS and RALL, or any combination of them, can be selected for random analysis. The results can be either printed in the .f06 file or punched n the punch file, or output in both files.
12. Note that the CID keyword affects only grid point related output, such as DISPlacement, VELOcity, ACCEleration, OLOAD, SPCForce and MPCForce. In addition, CID keyword needs to appear only once in a grid related output request anywhere in the Case Control Section to turn on the printing algorithm.

OMODES Output Modes Set

Selects a set of modes for output requests.

Format:

$$\text{OMODES} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \end{matrix} \right\}$$

Examples:

OMODES = ALL
OMODES = 20

Describer	Meaning
ALL	Output for all extracted modes will be computed. (Default)
n	Set identification of a previously appearing SET command. Output for those extracted modes appearing on this SET command will be computed.

Remarks:

1. This command is honored only in SOLs 103, 105, 107, 110, 111, 112, and 200. It is ignored in all other analyses.
2. In contrast to the OFREQENCY Case Control request, which affords an alternate way of selecting the modes to be output based on their frequencies, the OMODES command allows mode selection based on integer mode ID. For example:

```
SUBCASE 10
        …
        SET 11 = 1,3,5,7
        OMODES = 11
        DISP = ALL
        …
SUBCASE 20
        …
        SET 21 = 25., 28., 31.
        OFREQ = 21
        DISP = ALL
        …
```

3. If both the OMODES and the OFREQUENCY requests appear, the OMODES request takes precedence.

4. If neither the OMODES nor the OFREQUENCY request is specified, output will be generated for all modes.
5. It is important to note that the OMODES request has no effect whatsoever on the number of modes computed. Instead, it only provides a means for selecting a subset of the computed modes for which output is to be generated.
6. In superelement analysis, the set definition of an OMODES request for an upstream superelement will not be honored unless a similar request appears for its downstream neighbor. The downstream request should either be comprised of the union of all upstream requests, or left blank, as the OMODES default is ALL. Note that the code does not check to see if this condition is satisfied.

OTIME Output Time Set

Selects a set of times for output requests.

Format:

$$\text{OTIME} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

Examples:

OTIME =ALL
OTIME =15

Describer	Meaning
ALL	Output for all times will be computed.
n	Set identification number of a previously appearing SET command. Output for times closest to those given on this SET command will be computed. (Integer>0)

Remarks:

1. If the OTIME command is not supplied in the Case Control Section, then output for all times will be computed.
2. This command is particularly useful for requesting a subset of the output (e.g., stresses at only peak times, etc.).
3. This command can be used in conjunction with the MODACC module to limit the times for which modal acceleration computations are performed.

4. If this command is specified in more than one subcase in the modal solution sequences, then it is recommended that the first subcase contain OTIME=ALL and subsequent subcases contain OTIME=n. Also, data recovery requests should be specified only in the subsequent subcases. For example:

```
SUBCASE 1
        OTIME ALL
SUBCASE 2
        OTIME = 10
        SET10 = . . .
        DISP = ALL
SUBCASE 3
        OTIME = 20
        SET 20 = . . .
        STRESS = ALL
```

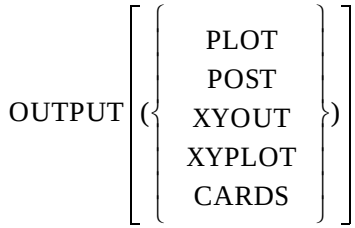

5. The OTIME command is not effective in nonlinear transient analysis (SOL 129). However, the OTIME command can be used in the nonlinear transient thermal analysis (SOL 159) to limit the output to a specified output times.

6. In superelement analysis, the set definition of an OTIME request for an upstream superelement will not be honored unless a similar request appears for its downstream neighbor. The downstream request should either be comprised of the union of all upstream requests, or left blank, as the OTIME default is ALL. Note that the program does not check to see if this condition is satisfied.

OUTPUT Case Control Delimiter

Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS.

Format:



Examples:

```
OUTPUT
OUTPUT(PLOT)
OUTPUT(XYOUT)
```

Describer	Meaning
PLOT	Beginning of the structure plotter request. This command must precede all structure plotter control commands. Plotter commands are described in "**OUTPUT(PLOT) Commands**" on page 539.
POST	Beginning of grid point stress SURFACE and VOLUME commands. This command must precede all SURFACE and VOLUME commands.
XYOUT or XYPLOT	Beginning of curve plotter request. This command must precede all curve plotter control commands. XYPLOT and XYOUT are entirely equivalent. Curve plotter commands are described in "**X-Y PLOT Commands**" on page 580.
CARDS	The OUTPUT(CARDS) packet is used by the MSGSTRESS program. See the *MSGMESH Analyst's Guide* for details. These commands have no format rules. This package must terminate with the command ENDCARDS (starting in column 1).

Remarks:

1. The structure plotter request OUTPUT(PLOT), the curve plotter request OUTPUT(XYOUT or XYPLOT), and the grid point stress requests (OUTPUT(POST)) must follow the standard Case Control commands.

2. If OUTPUT is specified without a describer, then the subsequent commands are standard Case Control commands.
3. Case Control commands specified after OUTPUT(POST) are SURFACE and VOLUME.

OUTRCV P-element Output Option Selection

Selects the output options for the p-elements defined on an OUTRCV Bulk Data entry.

Format:

OUTRCV=n

Examples:

OUTRCV=10
OUTR=25

Describer	Meaning
n	Set identification number of a OUTRCV Bulk Data entry. (Integer>0)

Remark:

1. The OUTRCV command is optional. By default, p-element output uses the defaults specified for CID and VIEW under the OUTRCV Bulk Data entry description.

P2G Direct Input Load Matrix Selection

Selects direct input load matrices.

Format:

P2G=name

Example:

P2G = LDMIG

P2G = LDMIG1, LDMIG2, LDMIG3

SET 100 = LDMIG, L1, L8
P2G = 100

P2G = 1.25*LDMIG1, 1.0*LDMIG2, 0.82*LDMIG3

Describer	Meaning
name	Name of a $[P_g^2]$ matrix to be input on the DMIG Bulk Data entry, or name list with or without factors (see Remark 4.). (Character)

Remarks:

1. Terms are added to the load matrix before any constraints are applied.
2. The matrix must be columnar in form (i.e., Field 4 on DMIG entry - IFO - must contain the integer 9.)
3. A scale factor may be applied to this input using the PARAM,CP2 parameter. See “Parameters” on page 659.
4. The formats of the name list:
   a. Names without factor.
      Names separated by comma or blank.
   b. Names with factors.
      Each entry in the list consists of a factor followed by a star followed by a name. The entries are separated by comma or blank. The factors are real numbers. Each name must be with a factor including 1.0.
5. P2G should be selected above all subcase. The number of columns specified for NCOL on the DMIG Bulk Data entry must equal the number of subcases.

PAGE Page Eject

Causes a page eject in the echo of the Case Control Section.

Format:

PAGE

Example:

PAGE

Remarks:

1. PAGE appears in the printed echo prior to the page eject.
2. PAGE is used to control paging in large Case Control Sections.

PARAM Parameter Specification

Specifies values for parameters. Parameters are described in "**Parameters**" on page 659.

Format:

PARAM,n,V1,V2

Examples:

PARAM,GRDPNT,0
PARAM,K6ROT,1.0

Describer	Meaning
n	Parameter name (one to eight alphanumeric characters, the first of which is alphabetic).
V1, V2	Parameter value based on parameter type, as follows:

Type	V1	V2
Integer	Integer	Blank
Real, single precision	Real	Blank
Character	Character	Blank
Real, double precision	Real, Double Precision	Blank
Complex, single precision	Real or Blank	Real or Blank
Complex, double precision	Real, Double Precision	Real, Double Precision

Remarks:

1. The PARAM command is normally used in the Bulk Data Section and is described in the "**Bulk Data Entries**" on page 945.
2. The parameter values that may be defined in the Case Control Section are described in "**Parameters**" on page 659. Case Control PARAM commands in user-written DMAPs requires the use of the PVT module, described in the *MD Nastran 2006 DMAP Programmer's Guide*.

PARTN Partitioning Vector Specifications

Specifies a list of grid point identification numbers that will be partitioned with the DMAP module MATMOD (Option 17).

Format:

PARTN=n

Example:

PARTN=10

Describer	Meaning
n	Set identification number of a previously appearing SET command. (Integer>0)

Remarks:

1. The PARTN command and the DMAP module MATMOD provide a convenient method for building a partitioning vector for use in the DMAP modules such as PARTN and MERGE.
2. The PARTN command is no longer applicable to coupled fluid-structure analysis. It has been replaced by the FSLPOUT command.

PFGRID Acoustic Grid Participation Factor Output Request

Requests the form and type of acoustic grid participation factor output.

Format:

$$\text{PFGRID}\left[\left(\left[\underline{\text{PRINT}},\ \text{PUNCH}\right],\left[\underline{\text{REAL}}\ \textit{or}\ \text{IMAG}\right],\left[\text{GRIDS}=\left\{\underline{\text{ALL}}\right\}\right],\left[\text{SOLUTION}=\left\{\underline{\text{ALL}}\right\}\right]\right)\right]=\left\{\begin{matrix}\textit{setdof}\\ \text{NONE}\end{matrix}\right\}$$

Example:

```
SET 10 = 11217
SET 20 = 25., 30., 35.
PFGRID(PHASE, SOLUTION=20) = 10
```

Describer	Meaning
PRINT	Output will be written to the .f06 file (Default).
PUNCH	Output will be written to the .pch file.
PLOT	Results are computed but not output.
REAL or IMAG	Real and imaginary part of complex results will be output (Default).
PHASE	Magnitude and phase of complex results will be output.
GRIDS	Keyword selecting the structural grid points to be processed; default is all structural grid points
setg	Identifier of a set containing the identifiers of the structural grid points to be processed
SOLUTION	Keyword selecting a set of excitation frequencies for which the participation factors will be processed; default is all excitation frequencies
setf	Identifier of a set of excitation frequencies
setdof	Identifier of a set of fluid degrees of freedom for which the participation factors are to be processed.

Remarks:

1. All PFMODE(FLUID), PFPANEL and PFGRID case control commands must reference the same set of fluid degrees of freedom.
2. Acoustic grid participation factors are available in a coupled frequency response analysis (SOL 108 and SOL 111).
3. Both PRINT and PUNCH may be requested.
4. The SOLUTION keyword can be used to select a subset of solutions available. If set setf is specified, the items in the set are excitation frequencies.

PFMODE Modal Participation Factor Output Request

Requests the form and type of modal participation factor output.

Format:

$$\text{PFMODE}\left[\left(\left[\frac{\underline{\text{STRUCTURE}}}{\text{FLUID}}\right],\left[\frac{\underline{\text{PRINT}},\text{PUNCH}}{\text{PLOT}}\right],\left[\frac{\underline{\text{REAL}}\ or\ \text{IMAG}}{\text{PHASE}}\right],\right.\right.$$

$$[\text{SORT} = sorttype],[\text{KEY} = sortitem],\left[\text{ITEMS} = \left\{\begin{matrix}\text{ALL}\\(itemlist)\end{matrix}\right\}\right],$$

$$\left[\text{FLUIDMP} = \left\{\begin{matrix}\text{ALL}\\ m_f \\ \underline{\text{NONE}}\end{matrix}\right\}\right],\left[\text{STRUCTMP} = \left\{\begin{matrix}\text{ALL}\\ m_s \\ \underline{\text{NONE}}\end{matrix}\right\}\right],$$

$$\left[\text{PANELMP} = \left\{\begin{matrix}\text{ALL}\\ setp \\ \underline{\text{NONE}}\end{matrix}\right\}\right],\left[\text{SOLUTION} = \left\{\begin{matrix}\underline{\text{ALL}}\\ setf \\ \text{NONE}\end{matrix}\right\}\right],[\text{FILTER} = fratio],$$

$$\left.[\text{NULL} = ipower])\right] = \left\{\begin{matrix}setdof\\ \text{NONE}\end{matrix}\right\}$$

Examples:

```
SET 20 = 11/T3, 33/T3, 55/T3
SET 30 = 420., 640., 660.
PFMODE(STRUCTURE, SOLUTION = 30, FILTER = 0.01,
    SORT = ABSD) = 20
SET 40 = 1222, 1223
SET 50 = 10., 12.
PFMODE(FLUID, STRUCTMP=ALL, PANELMP=ALL,
    SOLUTION=50, SORT=ABSD) = 40
```

Describer	Meaning
STRUCTURE	Requests output of structural participation factors. (Default)
FLUID	Requests output of acoustic participation factors.
PRINT	Output will be written to the .f06 file. (Default)
PUNCH	Output will be written to the .pch file.

Describer	Meaning
PLOT	Results are computed but not output.
REAL or IMAG	Real and imaginary part of complex results will be output (Default).
PHASE	Magnitude and phase of complex results will be output.
SORT	Keyword selecting the sort type. Default is sorting by increasing natural mode number.
sorttype	Sort option: ABSA output will be sorted by absolute value in ascending order ABSD output will be sorted by absolute value in descending order ALGA output will be sorted by algebraic value in ascending order ALGD output will be sorted by algebraic value in descending order
KEY	Keyword selecting the output item to be used for sorting; default is FRACTION.
sortitem	Item from the item list (see below) on which the sort operation is performed.
ITEMS	Keyword specifying data selected for output to the .pch file
itemlist	One or more of the following items: Item Identifier — Description RESPONSE — Modal participation factor PROJECTION — Projected modal participation factor FRACTION — Normalized projected modal participation factor SCALED — Projected modal participation factor divided by largest magnitude of all modal participation factors

Describer	Meaning
	MODEDISP Real and imaginary part of modal participation factors
	MODERESP Magnitude and phase relative to total response of modal participation factors
	If more than one item is selected, the list must be enclosed in parentheses.
FLUIDMP	Keyword to select output of fluid modal participation factors.
mf	Number of fluid modes for which modal participation factors will be computed.
STRUCTMP	Keyword to select output of structural modal participation factors; see Remarks **9.** and **10.**
ms	Number of structural modes for which modal participation factors will be computed.
PANELMP	Keyword to select output of panel modal participation factors
setp	Identifier of a set of panels.
SOLUTION	selects a set of excitation frequencies for which the participation factors will be processed; default is all excitation frequencies.
setf	Identifier of a set of excitation frequencies.
FILTER	Keyword specifying the value of a filter to be applied to the printed output.
fratio	filter value (default is 0.001); see Remark **12.**
NULL	Keyword specifying the power of ten used to detect a null response; see Remark **13.**
ipower	power of ten used to detect a null response (default is 12); see Remark **13.**
setdof	Identifier of a set of fluid degrees of freedom for which the participation factors are to be processed.

Remarks:

1. All PFMODE(FLUID), PFPANEL and PFGRID case control commands must reference the same set of fluid degrees of freedom.

2. Keywords FLUIDMP, STRUCTMP and PANELMP are only valid if FLUID is specified.
3. If STRUCTURE is specified, setdof must reference a set of structural degrees of freedom. If FLUID is specified, setdof must reference a set of acoustic degrees of freedom.
4. Acoustic modal participation factors are available in a coupled modal frequency response analysis (SOL 111) only.
5. Both PRINT and PUNCH may be requested.
6. Printed output includes results for all the data items described in the itemlist table.
7. Punched output includes results for only the data items selected by the ITEMS keyword.
8. Modal participation factors are sorted by increasing order of mode number unless the SORT keyword specifies a different sorting order. If a sorting order is specified, the KEY keyword selects the item that is used for sorting. When MODEDISP is selected, sorting is based on the magnitude. When MODERESP is selected, sorting is based on the real part.
9. If STRUCTURE is specified, STRUCTMP requests output of structural modal participation factors. The default is ALL.
10. If FLUID is specified, STRUCTMP requests output of acoustic structural modal participation factors. The default is NONE.
11. The SOLUTION keyword can be used to select a subset of solutions available. If set setf is specified, the items in the set are excitation frequencies.
12. The filter is applied to the real part of the normalized projected participation factors. Only participation factors that pass the filter are output.
13. If the magnitude of the total response at a selected response degree of freedom is less than 10^{-ipowr}, then no modal participation factors are processed. If ipower is not in the range of 1 to 31, the default of 12 is used.
14. Acoustic panel modal participation factors are normalized using the panel response instead of the total response.
15. If present, output of acoustic structural modal participation factors includes the load participation factor. The load participation factor has a mode number of 0 and a resonance frequency of 0.

PFPANEL Acoustic Panel Participation Factor Output Request

Requests the form and type of acoustic panel participation factor output.

Format:

$$\text{PFPANEL}\left[(\left[\begin{matrix}\underline{\text{PRINT}},\ \text{PUNCH}\\ \text{PLOT}\end{matrix}\right],\ \left[\begin{matrix}\underline{\text{REAL}}\ or\ \text{IMAG}\\ \text{PHASE}\end{matrix}\right],\ \left[\text{PANEL}\ =\ \left\{\begin{matrix}\underline{\text{ALL}}\\ setp\end{matrix}\right\}\right]\right.$$

$$,\ [\text{SORT}\ =\ sorttype],\ [\text{KEY}\ =\ sortitem],\ \left[\text{ITEMS}\ =\ \left\{\begin{matrix}\text{ALL}\\ (itemlist)\end{matrix}\right\}\right],$$

$$\left.\left[\text{SOLUTION}\ =\ \left\{\begin{matrix}\underline{\text{ALL}}\\ setf\\ \text{NONE}\end{matrix}\right\}\right],\ [\text{FILTER}\ =\ fratio],\ [\text{NULL}\ =\ ipower])\right]$$

$$=\ \left\{\begin{matrix}setdof\\ \text{NONE}\end{matrix}\right\}$$

Example:

```
SET 10 = 10., 12.
SET 20 = 1222, 1223
PFPANEL(SOLUTION=10, FILTER=0.01, SORT=ABSD) = 20
```

Describer	Meaning
PRINT	Output will be written to the .f06 file. (Default)
PUNCH	Output will be written to the .pch file.
PLOT	Results are computed but not output.
REAL or IMAG	Real and imaginary part of complex results will be output. (Default)
PHASE	Magnitude and phase of complex results will be output.
PANEL	Keyword to select the panels to be processed; default is all panels.
setp	Identifier of a set of panels.
SORT	Keyword selecting the sort type. Default is alphabetic sorting by panel name.

Describer	Meaning
sorttype	Sort option: ABSA output will be sorted by absolute value in ascending order ABSD output will be sorted by absolute value in descending order ALGA output will be sorted by algebraic value in ascending order ALGD output will be sorted by algebraic value in descending order
KEY	Keyword selecting the output item to be used for sorting; default is FRACTION.
sortitem	Item from the item list (see below) on which the sort operation is performed.
ITEMS	Keyword specifying data selected for output to the .pch file.
itemlist	One or more of the following items: Item Identifier — Description RESPONSE — Modal participation factor PROJECTION — Projected modal participation factor FRACTION — Normalized projected modal participation factor SCALED — Projected modal participation factor divided by largest magnitude of all modal participation factors MODEDISP — Real and imaginary part of modal participation factors MODERESP — Magnitude and phase relative to total response of modal participation factors If more than one item is selected, the list must be enclosed in parentheses.
SOLUTION	selects a set of excitation frequencies for which the participation factors will be processed; default is all excitation frequencies.

Describer	Meaning
setf	Identifier of a set of excitation frequencies.
FILTER	Keyword specifying the value of a filter to be applied to the printed output.
fratio	filter value (Default is 0.001); see Remark **8.**
NULL	Keyword specifying the power of ten used to detect a null response; see Remark **9.**
ipower	power of ten used to detect a null response (Default is 12); see Remark **9.**
setdof	Identifier of a set of fluid degrees of freedom for which the participation factors are to be processed.

Remarks:

1. All PFMODE(FLUID), PFPANEL and PFGRID Case Control commands must reference the same set of fluid degrees of freedom.
2. Acoustic panel participation factors are available in a coupled frequency response analysis (SOL 108 and SOL 111).
3. Both PRINT and PUNCH may be requested.
4. Printed output includes results for all the data items described in the itemlist table.
5. Punched output includes results for only the data items selected by the ITEMS keyword.
6. Panel participation factors are alphabetically sorted by panel names unless the SORT keyword specifies a different sorting order. If a sorting order is specified, the KEY keyword selects the item that is used for sorting. When MODEDISP is selected, sorting is based on the magnitude. When MODERESP is selected, sorting is based on the real part.
7. The SOLUTION keyword can be used to select a subset of solutions available. If set setf is specified, the items in the set are excitation frequencies.
8. The filter is applied to the real part of the normalized projected participation factors. Only participation factors that pass the filter are output.
9. If the magnitude of the total response at a selected response degree of freedom is less than 10^{-ipowr}, then no modal participation factors are processed. If ipower is not in the range of 1 to 31, the Default of 12 is used.

10. If present, output includes the load participation factor. The panel name of the load participation factors is –LOAD-.

PLOTID Plotter Identification

Defines a character string that will appear on the first frame of any plotter output.

Format:

PLOTID=title

Example:

PLOTID=BLDG. 125 BOX 91

Describer	Meaning
title	Any character string.

Remarks:

1. PLOTID must appear before the OUTPUT(PLOT) or OUTPUT(XYOUT) commands.
2. The presence of PLOTID causes a special header frame to be plotted with the supplied identification plotted several times. The header frame allows plotter output to be identified easily.
3. If no PLOTID command appears, no ID frame will be plotted.
4. The PLOTID header frame will not be generated for the table plotters.

POST Post-Processor Data Specifications

Controls selection of data to be output for post-processing functions via the OUTPUT2 module interface for selected commercial post-processor products.

Format:

$$\text{POST} \left\{ \begin{matrix} \text{TOFILE} \\ \text{TOCASE} \end{matrix} \right\} \left\{ \begin{matrix} \text{furn} \\ \text{filename} \end{matrix} \right\} [\text{ppname}][\text{oplist}]$$

Examples:

POST PATRAN TOFILE 51 NOSTRESS
POST TOFILE SUBCASE8
POST TOCASE SUFNAME1

Describer	Meaning
TOFILE	Keyword to specify the destiny of output files. (No default if it appears above all subcases.)
TOCASE	Keyword to specify the destiny of subcase results to user-defined output files. (No default if it appears above all subcases.)
furn	Fortran file unit reference number where data will be written. (Integer > 0)
filename	Suffix filename (see Remark 8.) (Char8)
ppname	Name of the target post-processor program. (Default = PATRAN)
oplist	Names of output items to be processed.

Remarks:

1. The POST Case Control command controls the placement of output data on external fortran files for use by commercial post-processors. Use of the POST command generates the proper value for the POST DMAP parameter associated with the particular post-processor. All of the other parameter controls related to the POST DMAP parameter remain in effect and are described in “Parameters” on page 659. The products supported are identified in the following table. PATRAN is the default post-processor name used for ppname. DBC output (POST=0) cannot be controlled by the POST command.

ppname	Product	PARAM,POST,Value
PATRAN	MSC.Patran V3	-1
SDRC	SDRC IDEA-S	-2
NF	MSC / LMS NF	-4
FEMTOOLS	DDS / FemTools	-5
UNIGRAHICS	EDS / Unigraphics	-6

2. The TOFILE describer is followed by the specification of either a FORTRAN unit reference number or a file name associated with the external file that receives the output data. If a FORTRAN unit number is used, the file must be associated with it via the ASSIGN File Management Statement. If POST appears above all subcases, TOFILE must be used to specify either a FORTRAN unit reference number or a file name. The default value of TOFILE, which appears under a subcase, will inherit from the value given in the POST above all subcases. If the unit reference number is associated with a form=formatted file, changes in unit numbers across subcases are not allowed.
3. The data that can be controlled for each post-processor product is limited and is identified under the description of the POST and related DMAP parameters in "**Parameters**" on page 659. The keywords that can be used for the oplist options are shown in the following table. If an output item supported by a particular post-processor is described in "**Parameters**" on page 659 but is not listed here, then the POST command cannot be used to control its output to the external file.

Output Item	oplist Keyword	Case Command
Displacements	[NO]DISPLACE	DISP
Forces of Single Point Constraint	[NO]SPCFORCE	SPCFORCE
Element Forces	[NO]FORCES	ELFO / FORCE
Element Stresses	[NO]STRESS	ELST / STRESS
Element Strain Energy	[NO]ESE	ESE
Grid Point Force Balance	[NO]GPFORCE	GPFORCE
Stress at Grid Points	[NO]GPSIGMA	STRESS
Strain / Curvature at Grid Points	[NO]GPEPSILON	STRAIN

Output Item	oplist Keyword	Case Command
Composite Element Failure Indices	[NO]PLYFAILURE	STRESS
Element Kinetic Energy	[NO]EKE	EKE
Element Energy Loss	[NO]EDE	EDE
Multi-point Constraint Forces	[NO]MPCFORCE	MPCFORCE
Composite Lamina Stresses	[NO]PLYSIGMA	STRESS
Composite Lamina Strains	[NO]PLYEPSILON	STRAIN
Element Strains	[NO]STRAIN	STRAIN
Grid Point Stresses	[NO]GPSTRESS	GPSTRESS
Grid Point Strains	[NO]GPSTRAIN	GPSTRAIN
Applied Loads	[NO]LOAD	OLOAD
No items to be output	NONE	----------
Structure Mode Participation Factors	[NO]SMPF	PFMODE

4. Output data items must have been generated via the appropriate case control command in order for the data to be available for post-processing options. For example, the specification of SPCF in the oplist of the POST command will not produce forces of single point constraint on the POST output file unless there is a SPCF Case Control command present. Refer to the tables under the POST parameter description in "**Parameters**" on page 659 for a list of the output items supported by each post-processor.
5. Any data generated by a case control output request is automatically included in the oplist of the POST command. If output data is not wanted for a particular case, then the characters "NO" should be the first two characters of the keyword in the oplist. For example, NODISP specifies that displacements are not to be posted to the output file even though they have been requested via the DISP Case Control command. Alternatively, the related POST parameters may be used. For example, to avoid outputting any displacements whatsoever to the .op2 file, use a "PARAM, OUG, NO" Bulk Data entry.
6. Certain data (e.g. geometry) is always generated and is not dependent upon the presence of a case control command in the input data. The POST command affects the placement of this data on the external file only insofar as the selection of the post-processor defines the value of the POST DMAP

parameter value. The actions described in "**Parameters**" on page 659 under the POST parameter description will prevail for the particular value of POST associated with the selected post-processor. The primary purpose of the POST command is to give the user more control over subcase-dependent output data being stored on the external OUTPUT2 file.

7. If a POST command is present within any subcase, a POST command must also be present above the subcase level. The placement of the POST command above the subcase level causes a cumulative effect on POST commands in subsequent subcases. Any options specified above the subcase level propagate down into the POST command within a subsequent subcase. Thus, if a POST command specifies NODISP (no displacement output wanted) above the subcase level, then a POST command with the DISP option would be required within a subcase to generate any output to the OUTPUT2 file for displacements. This also implies that changing the OUTPUT2 file unit reference number with the TOFILE option in a subcase causes **all** output quantities currently scheduled for output to be switched to the new unit number, not just those in the oplist for the current POST command.
8. When the name of an output file is specified by keyword TOFILE, the ASSIGN statement in the File Management Section (FMS) can be used to specify the full path of its root name. the logical-keyword for the root name is OUTPUT2F. The default root name is the MD Nastran job name. FORTRAN unit reference number 19 has been reserved by MD Nastran for OUTPUT2F, although the user can assign other FORTRAN unit number to it. The full file name is in the form of <root name>.<suffix filename>.
9. When the name of an output file is specified by keyword TOCASE, the ASSIGN statement in the File Management Section (FMS) can be used to specify the full path of its root name. the logical-keyword for the root name is OPCASE. The default root name is the MD Nastran job name. FORTRAN unit reference number 22 has been reserved by MD Nastran for OPCASE. Although the user can assign other .FORTRAN unit numbers to it. The full file name is in the form of <root name>.<suffix filename>. Also ppname and oplist are not required. If ppname and oplist are specified, they will be ignored. Suffix filename must be specified with keyword TOCASE.
10. POST commands using TOCASE for Structure Mode Participation Factor output (SMPF) are not supported and will be ignored.

PRESSURE Pressure Output Request

Requests form and type pressure output. Analogous to the DISPLACEMENT Case Control command.

See the description of the Case Control command, “**DISPLACEMENT**” on page 268.

RANDOM Random Analysis Set Selection

Selects the RANDPS and RANDT1 Bulk Data entries to be used in random analysis.

Format:

RANDOM=n

Example:

RANDOM=177

Describer	Meaning
n	Set identification number of RANDPS and RANDT1 Bulk Data entries to be used in random analysis. (Integer>0)

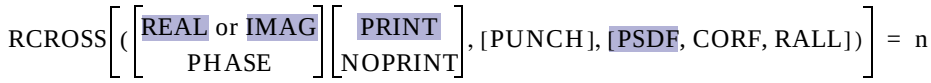
Remarks:

1. RANDOM must select RANDPS Bulk Data entries to perform random analysis.
2. RANDOM must appear in the first subcase of the current loop. RANDPS Bulk Data entries may not reference subcases in a different loop. Loops are defined by a change in the FREQUENCY command or changes in the K2PP, M2PP, or B2PP commands.
3. If RANDPS entries are used in a superelement analysis, the RANDOM command may be specified above the subcase level if a condensed subcase structure (SUPER=ALL) is used. If a condensed subcase structure is not used, then a unique RANDOM selection of a unique RANDPS entry must be specified within each of the desired superelement subcases.
4. P-elements are not supported in random analysis.

RCROSS Cross-power Spectral Density and Cross-correlation Function Output Request

Requests computation and output of cross-power spectral density and cross-correlation functions in random analysis.

Format:



Example:

RCROSS(PHASE, PSDF, CROF) = 10
RCROSS(RALL, NOPRINT, PUNCH) = 20
RCROSS = 30

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output (for cross-power spectral density function). Use of either REAL or IMAG yields the same output. (Default)
PHASE	Requests polar format (magnitude and phase) of complex output (for cross-power spectral density function). Phase output is in degree.
PRINT	Write output to print file. (Default)
NOPRINT	Do not write output to print file.
PUNCH	Write output to punch file.
PSDF	Requests the cross-power spectral density function be calculated and output for random analysis post-processing. (Default)
CORF	Requests the cross-correlation function be calculated and output for random analysis post-processing.
RALL	Requests both the cross-power spectral density function and cross-correlation function be calculated and output for random analysis post-processing.
n	Identification number of Bulk Data entry, RCROSS, to be used in random analysis. (Integer > 0)

Remarks:

1. Case Control RCROSS must be used along with Case Control, RANDOM. See Remarks under RANDOM.
2. Response quantities, such as DISPLACEMENT, STRESS and FORCE, must be requested by corresponding Case Control commands in order to compute cross-power spectral density and cross-correlation functions between the two response quantities specified by the Bulk Data entry, RCROSS. It is recommended that those requests be put above the subcase level to avoid the unfortunate situation that some response quantities are missing when it comes to the random analysis post-processing.
3. The response quantities must belong to the same superelement. The cross-power spectral density and cross-correlation functions between the two responses, which belong to the different superelements, are not supported.

REPCASE Repeat Output Subcase Delimiter

Delimits and identifies a repeated output subcase.

Format:

REPCASE=n

Example:

REPCASE=137

Describer	Meaning
n	Subcase identification number. (Integer > 1)

Remarks:

1. n must be strictly increasing (i.e., must be greater than all previous subcase identification numbers).
2. REPCASE defines a subcase that is used to make additional output requests for the previous real subcase. This command is required because multiple output requests for the same item are not permitted in the same subcase.
3. REPCASE may only be used in statics and normal modes problems. However, in normal modes, only output for the first mode is obtained. This feature is not supported in SOLs 106 or 153.
4. One or more repeated subcases (REPCASEs) must immediately follow the subcase (SUBCASE) to which they refer.
5. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the REPCASE with a TEMP(LOAD) Case Control command or the element deformation state with a DEFORM command.

RESVEC Controls Residual Vectors

Specifies options for and calculation of residual vectors.

Format:

$$\text{RESVEC}\left[\left(\left[\begin{matrix}\text{INRLOD}\\ \text{NOINRL}\end{matrix}\right],\left[\begin{matrix}\text{APPLOD}\\ \text{NOAPPL}\end{matrix}\right],\left[\begin{matrix}\text{ADJLOD}\\ \text{NOADJLOD}\end{matrix}\right]\left[\begin{matrix}\text{RVDOF}\\ \text{NORVDO}\end{matrix}\right],\left[\begin{matrix}\text{DAMPLOD}\\ \text{NODAMP}\end{matrix}\right],\left[\begin{matrix}\text{DYNRSP}\\ \text{NODYNRSP}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{SYSTEM / NOSYSTEM}\\ \text{COMPONENT / NOCOMPONENT}\\ \text{BOTH or YES}\\ \text{NO}\end{matrix}\right\}$$

Examples:

RESVEC=SYSTEM
RESVEC(NOINRL)=COMPONENT
RESVEC=NO

Describer	Meaning
INRLOD / NOINRL	Controls calculation of residual vectors based on inertia relief. (Default = INRLOD)
APPLOD / NOAPPL	Controls calculation of residual vectors based on applied loads. (Default = APPLOD)
ADJLOD / NOADJLOD	Controls calculation of residual vectors based on adjoint load vectors (SOL 200 only; Default = ADJLOD)
RVDOF / NORVDOF	Controls calculation of residual vectors based on RVDOFi entries (Default = RVDOF)
DAMPLOD / NODAMP	Controls calculation of residual vectors based on viscous damping. (Default = DAMPLOD)
DYNRSP / NODYNRSP	Controls whether the residual vectors will be allowed to respond dynamically in the modal transient or frequency response solution. See Remark 5. (Default = DYNRSP)
SYSTEM / NOSYSTEM	Controls calculation of residual vectors for system (a-set) modes. For NOSYSTEM, describers inside the parentheses are ignored. See Remark 2. for default.

4A CASE

Describer	Meaning
COMPONENT / NOCOMPONENT	Controls calculation of residual vectors for component (superelement or o-set) modes. For NOCOMPONENT, describers inside the parentheses are ignored. See Remark 2. for default.
BOTH or YES	Requests calculation of residual vectors for both system modes and component modes. See Remark 2. for default.
NO	Turns off calculation of residual vectors for both system and component modes and describers inside the parentheses are ignored. See Remark 2. for default.

Remarks:

1. RESVEC=SYSTEM / NOSYSTEM and RESVEC=COMPONENT / NOCOMPONENT may be specified in the same subcase.
2. RESVEC=BOTH is the default in all solution sequences except SOLs 103, 106, (with PARAM,NMLOOP) and 115 wherein RESVEC=COMPONENT is the default.
3. If the RESVEC command is specified then the user PARAMeters RESVEC and RESVINER are ignored.
4. The lower frequency cut-off on the EIGR or EIGRL should be left blank or set to a value below the minimum frequency. Residual vectors may not be calculated if all modes below the maximum frequency cut-off are not determined. If low-frequency modes are to be excluded from the analysis, use the MODESELECT Case Control command or PARAM,LFREQ.
5. Caution needs to be exercised when allowing the residual vectors to respond dynamically in a modal solution. The best approach is to always include enough normal modes to capture the dynamics of the problem, and rely on the residual vectors to help account for the influence of the truncated modes on the quasi-static portion of the response. This is not the default setting for this capability. When choosing to allow the residual vectors to respond dynamically, it is important to be aware of the frequency content of the excitation, as it will have the ability to excite these augmentation modes. If this is undesirable, then the forcing function should be filtered in advance to remove any undesired frequency content, or specify the NODYNRSP keyword.

RIGID Rigid Element Method

Selects processing rigid elements method for RBAR, RBAR1, RJOINT, RROD, RTRPLT, RTRPLT1, RBE1, RBE2, and RBE3.

Format:

$$\text{RIGID} = \left\{ \begin{matrix} \text{LINEAR} \\ \text{LAGRAN} \\ \text{LGELIM} \end{matrix} \right\}$$

Example:

RIGID=LAGR

4A CASE

Describer	Meaning
LINEAR	Selects the linear elimination method.
LAGR	Selects the Lagrange multiplier method.
LGELIM	Selects the Lagrange multiplier method with elimination.

Remarks:

1. The RIGID command must be above the SUBCASE level.
2. The RIGID command can be used in SOL 101, 103, 105, and 400 only. For all other solution sequences, RIGID command is ignored and RIGID=LINEAR is used.
3. If the RIGID command is not specified. RIGID=LINEAR is used for all solution sequences except SOL 400. For SOL 400, the default is RIGID=LAGRAN.
4. RIGID=LGELIM is not available for SOL 400.
5. LINEAR processing will not compute the thermal loads. Also, for SOLs 103, 105, LINEAR processing will not compute the differential stiffness. In order to compute the thermal load or the differential stiffness, LAGRAN or LGELIM methods must be used.
6. For SOL 400, the LINEAR rigid elements are valid for small rotation only. The LAGRAN method is valid for both small rotation or large rotation (parameter LGDISP=1).

7. For the LINEAR method, the dependent degrees-of-freedom are eliminated and placed in the mp-set. For the LAGRAN method, both independent and dependent degrees-of-freedoms are placed in l-set. Lagrange multiplier degrees-of-freedoms are created internally for the dependent degrees-of-freedom and placed in l-set. For the LGELIM method, first the LAGRAN rigid elements are created. Then both the Lagrange degrees-of-freedom and the dependent degrees-of-freedom are eliminated, and the dependent degrees-of-freedom are placed in the mr-set. Both mp-set and mr-set are subset of m-set. Please see "**Degree-of-Freedom Set Definitions**" on page 940.
8. Between LAGRAN and LGELIM, LAGRAN is the preferred method. LGELIM is a backup method if difficulty is encountered using the LAGRAN method.
9. The parameters LMFACT and PENFN can be used as scale factor and penalty function for the LAGRAN method of processing.

RGYRO Activates Gyroscopic Effects and Selects RGYRO or UNBALNC Entries

The RGYRO command will activate the rotodynamics capability. The RGYRO command will select the RGYRO Bulk Data entry for use in complex modes, frequency response, and static analysis. For transient response, the RGYRO command will select the UNBALNC Bulk Data entry. If the UNBALNC entry is not required, setting RGYRO to YES will include the gyroscopic effects in the transient response calculation. Setting RGYRO to NO will deactivate gyroscopic effects in all solutions.

Format:

For complex modes, frequency response, and static analysis:

RGYRO = n or YES / NO

Examples:

RGYRO = 100

For Transient Response:

RGYRO = YES
or
RGYRO = 200

SACCELERATION Solution Set Acceleration Output Request

Requests the form and type of solution set acceleration output.

Format:

$$\text{SACCELERATION}\left[\left(\left[\begin{matrix}\text{SORT1}\\\text{SORT2}\end{matrix}\right],\text{PRINT},\text{PUNCH},\left[\begin{matrix}\text{REAL or IMAG}\\\text{PHASE}\end{matrix}\right]\right)\right]=\left\{\begin{matrix}\text{ALL}\\\text{n}\\\text{NONE}\end{matrix}\right\}$$

Examples:

SACCELERATION=ALL
SACCELERATION(PUNCH,IMAG)=142

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Acceleration for all solution set points (modes) will be output.
NONE	Acceleration for no solution set points (modes) will be output.
n	Set identification number of a previously appearing SET command. Only accelerations of points with identification numbers that appear on this SET command will be output. (Integer>0)

Remarks:

1. Acceleration output is only available for transient and frequency response problems.

2. The defaults for SORT1 and SORT2 depend on the type of analysis and is discussed in Remark 2 under the "**DISPLACEMENT**" on page 268 command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL then the remaining commands will also be SORT1.
3. SACCELERATION=NONE allows an overall output request to be overridden.

SDAMPING Structural Damping Selection

Requests modal damping as a function of natural frequency in modal solutions or viscoelastic materials as a function of frequency in direct frequency response analysis.

Format:

$$\text{SDAMPING}\left[\begin{pmatrix}\text{STRUCTURE}\\ \text{FLUID}\end{pmatrix}\right]=\text{n}$$

Example:

SDAMPING=77

Describer	Meaning
STRUCTURE or FLUID	Modal damping is requested for the structural or fluid portion of the model.
n	Set identification number of a TABDMP1 or TABLEDi Bulk Data entry. (Integer>0)

Remarks:

1. In the modal solutions (e.g., SOLs 110, 111, 112, 145, 146, and 200), SDAMPING must reference a TABDMP1 entry.
2. In direct frequency response (e.g., SOL 108), SDAMPING must reference a TABLEDi entry which defines viscoelastic (frequency-dependent) material properties. See "**Viscoelastic Materials in Frequency Response Analysis**" on page 888 of the *MSC.Nastran Reference Manual*.
3. SDAMPING may be requested for superelements as long as PARAM,SESDAMP,YES is specified.

SDISPLACEMENT Solution Set Displacement Output Request

Requests the form and type of solution set displacement output.

Format:

$$\text{SDISPLACEMENT}\left[\left(\begin{bmatrix}\text{SORT1}\\\text{SORT2}\end{bmatrix},\text{PRINT},\text{PUNCH},\begin{bmatrix}\text{REAL or IMAG}\\\text{PHASE}\end{bmatrix}\right)\right]=\left\{\begin{matrix}\text{ALL}\\\text{n}\\\text{NONE}\end{matrix}\right\}$$

Examples:

SDISPLACEMENT=ALL
SDISPLACEMENT(SORT2,PUNCH,PHASE)=NONE

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Displacements for all solution set points (modes) will be output.
NONE	Displacements for no solution set points (modes) will be output.
n	Set identification number of a previously appearing SET command. Only displacements on points with identification numbers that appear on this SET command will be output. (Integer > 0)

Remarks:

1. The defaults for SORT1 and SORT2 depend on the type of analysis and is discussed in Remark 2 under the "**DISPLACEMENT**" on page 268 command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL then the remaining commands will also be SORT1.
2. SDISPLACEMENT=NONE allows an overall output request to be overridden.
3. The SDISPLACEMENT command is required to output normalized complex eigenvectors.

SEALL Superelement Generation and Assembly

Specifies the superelement identification numbers of Phase 1 processing in which all matrices and loads are generated and assembled. Controls execution of the solution sequence.

Format:

$$\text{SEALL} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{i} \end{matrix} \right\}$$

Examples:

SEALL=ALL
SEALL=7

Describer	Meaning
ALL	Generate and assemble all superelements.
n	Set identification number of a previously appearing SET command. Only superelements with identification numbers that appear on this SET command will be generated and assembled. (Integer>0)
i	Identification number of a single superelement that will be generated and assembled. (Integer>0)

Remarks:

1. If this command is present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure and can only appear as a member of a SET.
3. For a further discussion of this command, see "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. This command combines, in one command, the functions of the SEMG, SELG, SEKR, SELR, and SEMR commands.
6. This command does not control superelement data recovery (Phase 3). See the "**SEDR**" on page 456 Case Control command description.

7. SEALL=ALL is the default but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SEDR Superelement Data Recovery

Specifies the superelement identification numbers for which data recovery will be performed.

Format:

$$\text{SEDR} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{i} \end{matrix} \right\}$$

Examples:

SEDR=ALL
SEDR=7

Describer	Meaning
ALL	Performs data recovery for all superelements.
n	Set identification number of a previously appearing SET command. Data recovery will be performed for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which data recovery will be performed. (Integer>0)

Remarks:

1. If this command is present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure and can only appear as a member of a SET.
3. For a further discussion of this command, see "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If this command is not present, data recovery is performed for all superelements for which there are output requests (i.e., the default for this command is SEDR=ALL).

SEDV Superelement Design Variable Processing

Specifies the superelement identification numbers for which the design variables will be processed.

Format:

$$\text{SEDV} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{i} \end{matrix} \right\}$$

Examples:

SEDV=ALL
SEDV=18

Describer	Meaning
ALL	Requests design variable processing for all superelements. This is the default value if SEDV is missing in the file.
n	Set identification number of a previously appearing SET command. Design variable processing will be performed for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which design variable processing will be performed. (Integer>0)

Remarks:

1. If this command is present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure and can only appear as a member of a SET.
3. For a further discussion of superelement sensitivity analysis, see the *MSC.Nastran Design Sensitivity and Optimization User's Guide.*
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If both the SEDV and SERESP commands are not present, then the design variable processing and design sensitivity matrix generation will be performed for all superelements.

SEEXCLUDE Superelement Matrix and Load Assembly Exclusion

Specifies the superelement identification numbers for which all matrices and loads will not be assembled into the downstream superelement.

Format:

$$\text{SEEXCLUDE} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{i} \end{matrix} \right\}$$

Examples:

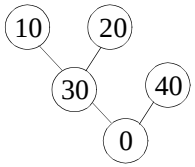
SEEXCLUDE=ALL
SEEXCLUDE=18

Describer	Meaning
ALL	All upstream superelements will be excluded.
n	Set identification number of a previously appearing SET command. Only those superelements with identification numbers that appear on this SET command will be excluded. (Integer>0)
i	Identification number of a single superelement for which matrices will be excluded. (Integer>0)

Remarks:

1. If this command is present, then it must be located before the first SUBCASE command.
2. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
3. This command is not meaningful when applied to the residual structure.
4. For a further discussion of this command, see the *MSC.Nastran Handbook for Superelement Analysis.*

5. If the SEEXCLUDE command is specified in a restart of SOLs 101 through 200, then PARAM,SERST,MANUAL must be specified. Also the SEKR command must be specified for the superelement immediately downstream from the excluded superelement. For example, if superelement 10 is excluded in the superelement tree:



then you must specify the following commands in the Case Control Section:

```
SEKR = 30
PARAM,SERST,MANUAL
```

SEFINAL Final Superelement for Assembly

Specifies the superelement identification number for the final superelement to be assembled.

Format:

$$\text{SEFINAL} = \left\{ \begin{matrix} n \\ i \end{matrix} \right\}$$

Example:

SEFINAL=14

Describer	Meaning
n	Set identification of a previously appearing SET command. Each superelement identification number appearing on the SET command must belong to a disjoint model. (Integer>0)
i	Identification number of the final superelement to be assembled. (Integer>0)

Remarks:

1. If this command is not present, the program selects the order of the superelements for assembly operations.
2. If this command is present, then it must be located before the first SUBCASE command.
3. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
4. This command can be used on restarts to ensure that minor modeling changes do not also change the processing order. For this usage, inspect the SEMAP table to determine which superelements were final superelements on the prior run.
5. See the *MSC.Nastran Handbook for Superelement Analysis* for a further discussion of this command.

SEKREDUCE Superelement Stiffness Matrix Assembly and Reduction

Specifies the superelement identification numbers for which stiffness matrices are assembled and reduced.

Format:

$$\text{SEKREDUCE} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{i} \end{matrix} \right\}$$

Examples:

SEKREDUCE=ALL
SEKREDUCE=9

Describer	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which the stiffness matrix will be assembled and reduced. (Integer>0)

Remarks:

1. If this command is present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure and can only appear as a member of a SET.
3. For a further discussion of this command, see "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.
4. SEKREDUCE is an alternate form and is entirely equivalent to the obsolete command SEMASSEMBLE.
5. SEALL=ALL is the default but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SELGENERATE Superelement Load Generation

Specifies the superelement identification numbers for which static loads will be generated.

Format:

$$\text{SELGENERATE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

Examples:

SELGENERATE=ALL
SELGENERATE=18

Describer	Meaning
ALL	Generates static loads for all superelements.
n	Set identification number of a previously appearing SET command. Static load matrices will only be generated for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which load matrices will be generated. (Integer>0)

Remarks:

1. If this command is present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure and can only appear as a member of a SET.
3. For a further discussion of this command, see "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. SEALL=ALL is the default but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SELREDUCE Superelement Load Assembly and Reduction

Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.

Format:

$$\text{SELREDUCE} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{i} \end{matrix} \right\}$$

Examples:

SELREDUCE=ALL
SELREDUCE=9

Describer	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will be assembled only for superelements with identification numbers that appear on this SET command. (Integer > 0)
i	Identification number of a single superelement for which the load matrices will be assembled and reduced. (Integer > 0)

Remarks:

1. If this command is present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure and can only be appear as a member of a SET.
3. For a further discussion of this command, see "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.
4. This command is used on restarts to selectively assemble and reduce load matrices.
5. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
6. In superelement static analysis, SELREDUCE is equivalent to SELASSEMBLE.

7. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.
8. SEALL=ALL is the default but can be overridden by specifying and of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SEMGENERATE Superelement Matrix Generation

Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.

Format:

$$\text{SEMGENERATE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

Examples:

SEMGENERATE=ALL
SEMGENERATE=7

Describer	Meaning
ALL	Generates structural matrices for all superelements.
n	Set identification number of a previously appearing SET command. Structural matrices will only be generated for superelements with identification numbers that appear on this SET command. (Integer > 0)
i	Identification number of a single superelement for which structural matrices will be generated. (Integer > 0)

Remarks:

1. If this command is present, then is must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure and can only appear as a member of a SET.
3. For a further discussion of this command, see "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. SEALL=ALL is the default in the Structured SOLs 101 through 200. This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SEMREDUCE Superelement Mass and Damping Assembly and Reduction

Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced. In buckling analysis, the differential stiffness matrices will be assembled and reduced.

Format:

$$\text{SEMREDUCE} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{i} \end{matrix} \right\}$$

Examples:

SEMREDUCE=ALL
SEMREDUCE=9

Describer	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command. (Integer > 0)
i	Identification number of a single superelement for which the load matrices or the mass and damping matrices will be assembled and reduced. (Integer > 0)

Remarks:

1. If this command is present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure and can only appear as a member of a set.
3. This command is used on restart to selectively assemble and reduce mass and damping matrices. For a further discussion of this command, see "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.

6. This command has no function in static analysis.
7. SEALL=ALL is the default but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SERESP Superelement Response Sensitivity

Specifies the superelement identification numbers for which the design sensitivity matrices will be generated.

Format:

$$\text{SERESP} = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{i} \end{matrix} \right\}$$

Examples:

SERESP=ALL
SERESP=18

Describer	Meaning
ALL	Requests design sensitivity matrix generation for all superelements. This is the default value if SERESP is missing.
n	Set identification number of a previously appearing SET command. Design sensitivity matrices will be generated for superelements with identification numbers that appear on this SET command. (Integer>0)
i	Identification number of a single superelement for which the design sensitivity matrix will be generated.

Remarks:

1. If this command is present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure and can only appear as a member of a SET.
3. For a further discussion of this command, see the *MSC.Nastran Reference Manual*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If both the SEDV and SERESP commands are not present, then the design variable processing and design sensitivity matrix generation will be performed for all superelements.

SET Set Definition, General Form

Sets are used to define the following lists:

1. Identification numbers (point, element, or superelement) for processing and output requests.
2. Frequencies for which output will be printed in frequency response problems or times for transient response, using the OFREQ and OTIME commands, respectively.
3. Surface or volume identification numbers to be used in GPSTRESS or STRFIELD commands.
4. DRESP1 entries that are used in the spanning of subcases.
5. Grid point number and component type code to be used by the MCFRACTION command.

Formats:

SET n = {i_1[,i_2, i_3, THRU i_4, EXCEPT i_5, i_6, i_7, i_8, THRU i_9]}
SET n = {r_1[,r_2, r_3, r_4]}
SET = ALL
SET n = {i_1 / c_1[,i_2 / c_2, i_3 / c_3, i_4 / c_4]}
SET n = {l_1, [l_2, l_3]}

Examples:

```
SET 77=5
SET 88=5, 6, 7, 8, 9, 10 THRU 55 EXCEPT 15, 16, 77, 78, 79, 100 THRU 300
SET 99=1 THRU 100000

SET 101=1.0, 2.0, 3.0
SET 105=1.009, 10.2, 13.4, 14.0, 15.0
SET 1001=101/T1, 501/T3, 991/R3
SET 2001=M1,M2
```

Describer	Meaning
n	Set identification number. Any set may be redefined by reassigning its identification number. SETs specified under a SUBCASE command are recognized for that SUBCASE only. (Integer >0)
i_1, c_1	Grid point identification numbers and component codes. The c values must be of T1, T2, T3, R1, or R3.

Describer	Meaning
l_1, l_2 etc.	Identification names of literals used for matrix or group selection.
i_1, i_2, etc.	Identification numbers. If no such identification number exists, the request is ignored. (Integer$\geq$0)
i_3THRUi_4	Identification numbers ($i_4 > i_3$). (Integer>0)
EXCEPT	Set identification numbers following EXCEPT will be deleted from output list as long as they are in the range of the set defined by the immediately preceding THRU. An EXCEPT list may not include a THRU list or ALL.
r_1, r_2, etc.	Frequencies or times for output. The nearest solution frequency or time will be output. EXCEPT and THRU cannot be used. If an OFREQ or OTIME command references the set then the values must be listed in ascending sequences, $r_1 < r_2 < r_3 < r_4$ ...etc., otherwise some output may be missing. If an OFREQ or OTIME command is not present, all frequencies or times will be output. (Real>0.0)
ALL	All members of the set will be processed.

Remarks:

1. A SET command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation. Place a number after the THRU.
2. Set identification numbers following EXCEPT within the range of the THRU must be in ascending order.
3. In SET 88 above, the numbers 77, 78, etc., are included in the set because they are outside the prior THRU range.
4. SET commands using the grid point / component code format cannot contain THRU. SETs using this format should be selected only by the MCFRACTION Case Control command.
5. SET commands using literals apply only to direct matrix input such as K2PP etc. or FLSPOUT panel grouping.

SETP Process Set Definition

Process sets are used to define lists of SET identifications to be processed individually for data recovery:

Formats:

ETP n = $\{i_1[i_2, i_3 \text{THRU } i_4 \text{EXCEPT } i_5, i_6, i_7, i_8 \text{THRU } i_9]\}$

Examples:

SET 77=5
SET 88=5, 6, 7, 8, 9, 10 THRU 55

Describer	Meaning
n	SETP identification number. Any SETP may be redefined by reassigning its identification number. SETPs specified under a SUBCASE command are recognized for that SUBCASE only. (Integer > 0) SET identification numbers. If no such identification number exists, the request is ignored. (Integer > 0)
EXCEPT	Set identification numbers following EXCEPT will be deleted from output list as long as they are in the range of the set defined by the immediately preceding THRU. An EXCEPT list may not include a THRU list or ALL.

Remarks:

1. A SETP command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation. Place a number after the THRU.
2. Set identification numbers following EXCEPT within the range of the THRU must be in ascending order.

In SET 88 above, the numbers 77, 78, etc., are included in the set because they are outside the prior THRU range.

SET Set Definition OUTPUT(PLOT)

Defines a set of element or grid point numbers to be plotted.

Format:

```
SET n = [ ALL [ ELEMENTS ] [ EXCEPT { type1 type2 ... typej } ] ],
        [ GRID POINTS      ]        { k1    k2    ... kj THRU kk BY incj }

[ [ INCLUDE ] [ ELEMENTS ] { type1 type2 ... typej } ],
  [ EXCLUDE ] [ GRID POINTS } { k1    k2    ... kj THRU kk BY incj }

[ EXCEPT { type1 typem ... typen } ]
         { k1    km    ... kn THRU ko BY incn }
```

Examples:

1. SET 1 consists of elements 1, 5, 10, 11, 13, 14, 15, 20, 22, 24, and 26.

   SET 1=INCLUDE 1, 5, 10 THRU 15 EXCEPT 12, INCLUDE 20 THRU 26 BY 2
2. SET 2 consists of all CTRIA3 and CQUAD4 elements except element 21.

   SET 2=QUAD4 TRIA3 EXCEPT 21
3. SET 10 includes all CTRIAR elements plus elements 70 through 80.

   SET 10 TRIAR INCLUDE ELEMENTS 70 THRU 80
4. SET 15 includes all elements from 15 to 20 and 26 to 100.

   SET 15=15 THRU 100 EXCEPT 21 THRU 25
5. SET 2 includes all elements except CTETRA elements.

   SET 2=ALL EXCEPT TETRA

Describer	Meaning
n	Sets identification number. (0<Integer<999999)
ALL	Selects all elements or grid points. See typei.
ELEMENTS	Specifies that all identification numbers refer to elements.
GRID POINTS	Specifies that all identification numbers refer to grid points.
INCLUDE	Includes specified element or grid point identification numbers or elements in the set.
EXCLUDE	Excludes specified element or grid point identification numbers or element types in the set.

Describer	Meaning
EXCEPT	Modifies a prior ALL, INCLUDE, or EXCLUDE specification.
typei	Element types. The allowed element types are (Character):

typei	Element Entry Name
TRIA3	CTRIA3
TRIA6	CTRIA6
TRIAR	CTRIAR
QUAD4	CQUAD4
QUAD8	CQUAD8
QUADR	CQUADR
HEXA	CHEXA
PENTA	CPENTA
TETRA	CTETRA

THRU	Specifies a range of identification numbers.
BY	Specifies an increment for a THRU specification.
inci	Increment for THRU range. (Integer>0)

Remarks:

1. This form of the SET command must and can only be specified after an OUTPUT(PLOT) delimiter.
2. The INCLUDE, EXCLUDE, and EXCEPT specifications may be specified more than once in the same set. See previous examples.
3. Commas or spaces may be used as separators.
4. Not all of the identification numbers in a THRU range have to correspond to elements or grid points. For example, elements 2, 4, 7, and 9 may be selected with 2 THRU 9, even if elements 3, 5, 6, and 8 do not exist. This is called an open set. It should be noted that large open sets can cause higher computational costs.

SETS DEFINITION Case Control Processing Delimiter

Delimites the various type of commands under grid point stress and/or p-version element set definitions. This command is synonymous with OUTPUT(POST).

Format:

SETS DEFINITION

Example:

SETS DEFINITION

Remark:

1. SETS DEFINITION is synonymous with OUTPUT(POST). Either SETS DEFINTIION or OUTPUT(POST) may be specified but not both.

SKIP Case Control Processing Delimiter

Activates or deactivates the execution of subsequent commands in the Case Control (including plot commands).

Format:

$$\text{SKIP}\left\{\begin{array}{c}\text{ON}\\ \text{OFF}\end{array}\right\}$$

Example:

SKIPOFF

Remarks:

1. SKIPON and SKIPOFF commands may appear as many times as needed in the Case Control.
2. Commands that are skipped will be printed.
3. SKIPON ignores subsequent commands until either a SKIPOFF or BEGIN BULK command is encountered. This allows the user to omit requests without deleting them from his data. In the following example, plot commands will be skipped.

```
TITLE=EXAMPLE
SPC=5
LOAD=6
SKIPON$SKIP PLOT REQUEST
OUTPUT (PLOT)
SET 1 INCLUDE ALL
FIND
PLOT
BEGIN BULK
```

SMETHOD Iterative Solver Method Selection

Selects iterative solver method and parameters.

Format:

$$\text{SMETHOD}\left\{\begin{array}{c}\text{ELEMENT}\\ n\\ \text{MATRIX}\end{array}\right\}$$

Example:

SMETHOD = ELEMENT $ selects element-based iterative solver defaults.

SMETHOD = MATRIX $ selects matrix based iterative solver defaults.

SMETHOD = 1000 $ specifies ID of ITER Bulk Data entry to select iterative.

4A CASE

Describer	Meaning
ELEMENT	Selects the element-based iterative solver with default control values.
MATRIX	Selects the matrix-based iterative solver with default control values.
n	Sets identification of an ITER Bulk Data entry. (Integer > 0)

Remarks:

1. The matrix-based iterative solver is available in SOLs 101, 106, 108 and 111 and allows use of all features.
2. The element-based iterative solver is only available in SOL 101. It is intended primarily for very large solid element models and does not handle p-elements. See the ITER Bulk Data entry for a list of restrictions.

SPC Single-Point Constraint Set Selection

Selects a single-point constraint set to be applied.

Format:

SPC=n

Example:

SPC=10

Describer	Meaning
n	Set identification number of a single-point constraint that appears on a SPC, SPC1, or SPCADD Bulk Data entry. (Integer > 0)

Remarks:

1. In cyclic symmetry analysis, this command must appear above the first SUBCASE command.
2. Multiple boundary conditions are only supported in SOLs 101, 103, 105, 145 and 200. Multiple boundary conditions are not allowed for upstream superelements. The BC command must be specified to define multiple boundary conditions for the residual structure in SOLs 103, 105, 145 and 200.

SPCFORCES Single-Point Forces of Constraint Output Request

Requests the form and type of single-point force of constraint vector output.

Format:

$$\text{PFORCES}\left[\left(\left[\begin{matrix}\text{SORT1}\\\text{SORT2}\end{matrix}\right],\left[\begin{matrix}\text{PRINT, PUNCH}\\\text{PLOT}\end{matrix}\right],\left[\begin{matrix}\text{REAL or NOZPRINT}\\\text{PHASE}\end{matrix}\right]\left[\begin{matrix}\text{PSDF, ATOC, CRMS}\\\text{or RALL}\end{matrix}\right],\right.\right.$$

$$\left.\left.\left[\begin{matrix}\text{RPRINT}\\\text{NORPRINT}\end{matrix}\right],\text{RPUNCH}\right],[\text{CID}])\right] = \left\{\begin{matrix}\text{ALL}\\\text{n}\\\text{NONE}\end{matrix}\right\}$$

Examples:

SPCFORCES = 5
SPCFORCES(SORT2, PUNCH, PRINT, IMAG) = ALL
SPCFORCES(PHASE) = NONE
SPCFORCES(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
SPCFORCES(PRINT, RALL, NORPRINT)=ALL

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, single-point forces of constraint.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
NOZPRINT	Print only non-zero SPCForces appearing in SORT2 output. This keyword does not affect SORT1 output.

Describer	Meaning
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RALL	Requests all of PSDF, ATOC and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RPRINT	Writes random analysis results in the print file. (Default)
NORPRINT	Disables the writing of random analysis results in the print file.
PRUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output file, F06 file.
ALL	Single-point forces of constraint for all points will be output. See Remarks **2.** and **5.**
NONE	Single-point forces of constraint for no points will be output.
n	Set identification of a previously appearing SET command. Only single-point forces constraint for points with identification numbers that appear on this SET command will be output. (Integer>0)

Remarks:

1. Both PRINT and PUNCH may be requested.
2. See Remark 2 under **“DISPLACEMENT”** on page 268 for a discussion of SORT1 and SORT2. In the SORT1 format only nonzero values will be output.
3. In a statics problem, a request for SORT2 causes loads at all points (zero and nonzero) to be output.

4. SPCFORCES=NONE overrides an overall output request.
5. In SORT1 format, SPCFORCESs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. But if a scalar point is not consecutively numbered, then it will begin a new sextet on a new line of output. If a sextet can be formed and all values are zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
6. SPCFORCES results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
7. In SOLs 129 and 159, SPCFORCES results do not include the effects of mass and damping elements.
8. In all solution sequences except SOLs 129 and 159, SPCFORCES results do include the effects of mass and damping, except damping selected by SDAMPING command. PARAM,DYNSPCF,OLD may be specified to obtain SPCFORCES results, which do not include mass and damping effects.
9. In inertia relief analysis, the SPCFORCES output is interpreted differently for SOLs 1, 101 and 200:
    - In SOL 1, the SPCFORCE output reflects the effects due to the applied loads only and not the inertial loads.
    - In SOLs 101 and 200, the SPCFORCE output includes both the effects due to inertial loads and applied loads.
10. The option of PSDF, ATOC, CRMS and RALL, or any combination of them, can be selected for random analysis. The results can be either printed in the .f06 file or punched in the punch file, or output in both files.
11. Note that the CID keyword affects only grid point related output, such as DISPlacement, VELOcity, ACCEleration, OLOAD, SPCForce and MPCForce. In addition, CID keyword needs to appear only once in a grid related output request anywhere in the Case Control Section to turn on the printing algorithm.

STATSUB Static Solution Selection for Differential Stiffness

Selects the static solution to use in forming the differential stiffness for buckling analysis, normal modes, complex eigenvalue, frequency response and transient response analysis.

Format:

$$\text{STATSUB}\left(\begin{bmatrix}\text{BUCKLING}\\ \text{PRELOAD}\end{bmatrix}\right) = n$$

Examples:

STATSUB=23
STAT=4
STATSUB(PREL)=7

Describer	Meaning
BUCKLING	Subcase id number corresponding to static subcase of buckling or varying load. (Default in buckling analysis.)
PRELOAD	Subcase id number corresponding to static subcase of preload or constant load. (Default in dynamic analysis.)
n	Subcase identification number of a prior SUBCASE specified for static analysis (Integer>0)

Remarks:

1. STATSUB may be used in SOLs 103, 105, 107 through 112, and SOL 200 (ANALYSIS = BUCKLING only).
2. STATSUB must be specified in the same subcase that contains the METHOD selection for buckling or normal modes, CMETHOD for complex eigenvalue analysis, TSTEP for transient response and FREQ for frequency response.
3. In SOL 105, if it is intended that results from the first static subcase are used to compute the differential stiffness, then the STATSUB command is not required. In other words, the default for STATSUB is the first static subcase identification. In SOLs 103 and 107 through 112, 115, and 116, STATSUB must reference a separate static subcase.

4. In dynamic analysis only one STATSUB command may be specified in each dynamic subcase. In buckling analysis with a pre-load, both STATSUB (BUCKLING) and STATSUB(PRELOAD) must be specified in each buckling subcase. STATSUB(PRELOAD) is not supported in SOL 200.
5. In dynamic analysis, any subcase that does not contain a CMETHOD command in SOLs 107 and 110, FREQUENCY command in SOLs 108 and 111, and a TSTEP command in SOLs 109 and 112 will be treated as a static subcase.

STEP Step Delimiter

Delimits and identifies a nonlinear analysis step for SOL 400.

Format:

STEP=n

Examples:

STEP=10

Describer	Meaning
n	Step identification number. (Integer > 0)

Remarks:

1. The STEP command can only be used in nonlinear solution sequence SOL 400 (NONLIN).
2. The STEP command is to be used below the SUBCASE Case Control command. If no SUBCASE specified, MD Nastran creates a default SUBCASE 1.
3. The STEP identification number, n, in a SUBCASE must be in increasing order and less than 9999999.
4. The following example illustrates a typical usage of SUBCASE and STEP:

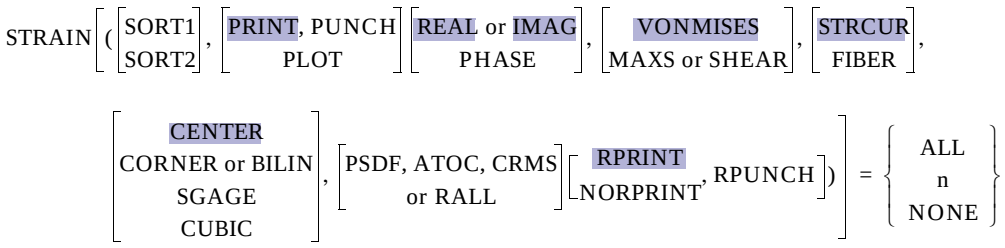
```
SUBCASE 1
   STEP 1
      LOAD = 1
   STEP 2
      LOAD = 2
SUBCASE 2
   STEP 10
      LOAD = 10
   STEP 20
      LOAD = 20
```

5. The solutions of all SUBCASEs are independent of each other. However, the solution of any STEP is a continuation of the solution of the previous STEP.

STRAIN Element Strain Output Request

Requests the form and type of strain output.

Format:



Examples:

STRAIN=5
STRAIN(CORNER)=ALL
STRAIN(PRINT,PHASE)=15
STRAIN(PLOT)=ALL
STRAIN(PRINT, PSDF, CRMS, RPUNCH)=20
STRAIN(PRINT, RALL,NORPRINT)=ALL

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates strain for the requested set but no printer output.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RALL	Requests all of PSDF, ATOC and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
VONMISES	von Mises strains are output.
MAXS or SHEAR	Maximum shear strains are output.
STRCUR	Strain at the reference plane and curvatures are output for plate elements.
FIBER	Strain at locations Z1, Z2 are computed for plate elements.
CENTER	Output QUAD4 strains at the center only.
CORNER or BILIN	Output CQUAD4 element strains at the center and grid points. Using strain gage approach with bilinear extrapolation.
SGAGE	Output CQUAD4 element strains at center and grid points using strain gage approach.
CUBIC	Output CQUAD4 element strains at center and grid points using cubic bending correction.
RPRINT	Writes random analysis results in the print file. (Default)
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
ALL	Strain for all elements will be output.

Describer	Meaning
n	Set identification of a previously appearing SET command. Only strain for elements with identification numbers that appear on this SET command will be output. (Integer>0)
NONE	No element strain will be output.

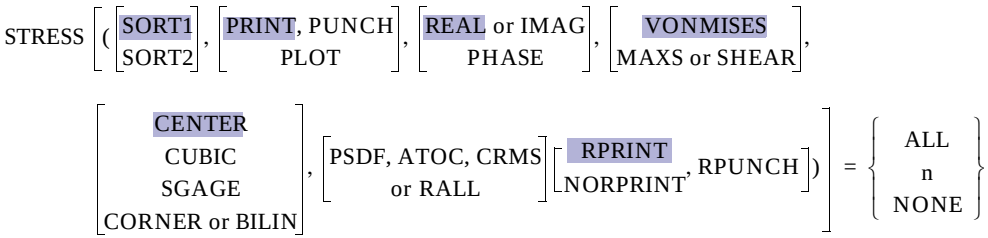
Remarks:

1. In SOLs 106 and 129, the STRAIN request pertains only to linear elements and only if the parameter LGDISP is -1, which is the default. Nonlinear strains for nonlinear elements are requested by the STRESS command and appear in the nonlinear stress output.
2. Both PRINT and PUNCH may be requested.
3. STRAIN=NONE overrides an overall output request.
4. The PLOT option is used when strains are requested for postprocessing but no printer output is desired.
5. Definitions of stress, strain, curvature, and output locations are given in the "**Structural Elements**" on page 47 of the *MSC.Nastran Reference Guide*.
6. If the STRCUR option is selected, the values of Z1 will be set to 0.0. and Z2 will be set to -1.0 on the output.
7. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.
8. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also Remark 8 under the "**FORCE**" on page 318 command for further discussion.
9. See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2.
10. The option of PSDF, ATOC, CRMS and RALL, or any combination of them, can be selected for random analysis. The results can be either printed in the .f06 file or punched in the punch file, or output in both files.

STRESS Element Stress Output Request

Requests the form and type of element stress output. Note: ELSTRESS is an equivalent command.

Format:



Examples:

STRESS=5
STRESS(CORNER)=ALL
STRESS (SORT1,PRINT,PUNCH,PHASE)=15
STRESS(PLOT)=ALL
STRESS(PRINT, PSDF, CRMS, RPUNCH)=20
STRESS(PRINT, RALL, NORPRINT)=ALL

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element type.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates stresses for requested set but no printer output.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
VONMISES	Requests von Mises stress.
MAXS or SHEAR	Requests maximum shear in the plane for shell elements and octahedral stress for solid elements.
CENTER	Requests CQUAD4, QUADR and TRIAR element stresses at the center only. The default for QUAD4 is CENTER. The default for QUADR and TRIAR is CORNER.
CUBIC	Requests CQUAD4 element stresses at the center and grid points using strain gage approach with cubic bending correction.
SGAGE	Requests CQUAD4 element stresses at center and grid points using strain gage approach.
CORNER or BILIN	Requests CQUAD4, QUADR. and TRIAR element stresses at center and grid points using bilinear extrapolation.
RPRINT	Writes random analysis results in the print file. (Default)
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
ALL	Stresses for all elements will be output.

Describer	Meaning
n	Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output. (Integer > 0)
NONE	No element stress will be output.

Remarks:

1. Both PRINT and PUNCH may be requested.
2. ALL should not be used in a transient problem due to excessive output.
3. See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2.
4. ELSTRESS is an alternate form and is entirely equivalent to STRESS.
5. STRESS=NONE overrides an overall output request.
6. The PLOT option is used when contour plots of stresses are requested but no printer output of stresses is desired. However in nonlinear analysis, the nonlinear stresses will still be printed unless NLSTRESS(PLOT) is specified.
7. The VONMISES option is ignored for ply stresses.
8. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.
9. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also Remark 8 under the "**FORCE**" on page 318 command for further discussion.
10. For composite ply output, the grid point option for CQUAD4 elements will be reset to the default option (CENTER).
11. MAXS for shell elements is not an equivalent stress.
12. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. The results can be either printed in the .f06 file or punched in the punch file, or output in both files.

STRFIELD Grid Point Stress Output Request

Requests the computation of grid point stresses for graphical postprocessing and mesh stress discontinuities.

Format:

$$\text{STRFIELD} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

Examples:

STRFIELD=ALL
STRFIELD=21

Describer	Meaning
ALL	Grid point stress requests for all surfaces and volumes defined in the OUTPUT(POST) section will be saved for postprocessing.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command and in the OUTPUT(POST) section will be included in the grid point stress output request for postprocessing. (Integer>0)

Remarks:

1. The STRFIELD command is required for the graphical display of grid point stresses in postprocessors that use the .xdb file (PARAM,POST,0), or when the GPSDCON or ELSDCON commands are specified, and does not provide printed output. The GPSTRESS command can be used to obtain printed output.
2. Only grid points connected to elements used to define the surface or volume are output. See the SURFACE and VOLUME commands.
3. Element stress output (STRESS) must be requested for elements referenced on requested SURFACE and VOLUME commands.
4. In nonlinear static and transient analysis, grid point stresses are computed only if parameter LGDISP is -1, which is the default. Also, in nonlinear transient analysis, grid point stresses are computed for elements with linear material properties only.

SUBCASE Subcase Delimiter

Delimits and identifies a subcase.

Format:

SUBCASE=n

Example:

SUBCASE=101

Describer	Meaning
n	Subcase identification number. (Integer>0)

Remarks:

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. Plot requests and RANDPS requests refer to n.
3. See the MODES command for use of this command in normal modes analysis.
4. If a comment follows n, then the first few characters of the comment will appear in the subcase label in the upper right-hand corner of the output.

SUBCOM Combination Subcase Delimiter

Delimits and identifies a combination subcase.

Format:

SUBCOM = n

Example:

SUBCOM = 125

Describer	Meaning
n	Subcase identification number. (Integer>2)

Remarks:

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. A SUBSEQ command must follow this command.
3. SUBCOM may only be used in statics problems.
4. Output requests above the subcase level will be used.
5. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SUBCOM with a TEMP(LOAD) command or the element deformations with a DEFORM command.
6. SUBCOMs may be specified in superelement analysis with the following recommendations:
   - For each superelement, specify its SUBCASEs consecutively, directly followed by its SUBCOM(s).
   - Specify a SUPER command with a new load sequence number under each SUBCOM command.

   The following example is given for a model with one superelement and one load combination:

```
SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SUBCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SUBSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SUBCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SUBSEQ=1.,1.
```

SUBSEQ Subcase Sequence Coefficients

Gives the coefficients for forming a linear combination of the previous subcases.

Format:

SUBSEQ=R1 [, R2, R3, ..., Rn]

Example:

SUBSEQ=1.0, -1 .0, 0.0, 2.0

Describer	Meaning
Ri	Coefficients of the previously occurring subcases. See Remark 4. (Real)

Remarks:

1. The SUBSEQ command can only appear after a SUBCOM command.
2. This command may only be used in statics problems.
3. This command list is limited to a maximum of 200 numbers.
4. R1 to Rn refer to the immediately preceding subcases. In other words Rn is applied to the most recently appearing subcase and R(n - 1) is applied to the second most recently appearing subcase, and so on. The comments ($) describe the following example:

```
DISPL = ALL
SUBCASE 1
SUBCASE 2
SUBCOM 3
SUBSEQ = 1.0, -1.0 $ SUBCASE 1 - SUBCASE 2
SUBCASE 11
SUBCASE 12
SUBCOM 13
SUBSEQ = 0.0, 0.0, 1.0, -1 .0 $ SUBCASE 11 - SUBCASE 12
or
SUBSEQ = 1.0, - 1.0 $ EQUIVALENT TO PRECEDING COMMAND.  USE ONLY ONE.
```

SUBTITLE Output Subtitle

Defines a subtitle that will appear on the second heading line of each page of printer output.

Format:

SUBTITLE=subtitle

Example:

SUBTITLE=PROBLEM NO. 5-1A

Describer	Meaning
subtitle	Any character string.

Remarks:

1. SUBTITLE appearing under a SUBCASE command will appear in the output for that subcase only.
2. SUBTITLE appearing before all SUBCASE commands will appear in the output for all subcases except those in Remark **1.**
3. If no SUBTITLE command is present, the subtitle line will be blank.
4. The subtitle also appears on plotter output.

SUPER Superelement Subcase Assignment

Assigns a subcase(s) to a superelement or set of superelements.

Format:

$$\text{SUPER} = \left\{ \begin{array}{c} \text{ALL} \\ \left\{ \begin{array}{c} n \\ i \end{array} \right\} [\, , l] \end{array} \right\}$$

Examples:

SUPER=17, 3
SUPER=15
SUPER=ALL

Describer	Meaning
i	Superelement identification number. (Integer > 0)
ALL	The subcase is assigned to all superelements and all loading conditions. (Default)
n	Set identification number of a previously appearing SET command. The subcase is assigned to all superelements with identification numbers that appear on this SET command. (Integer > 0)
l	Load sequence number. (Integer > 0; Default=1)

Remarks:

1. All subcases with requests for specific superelement(s) must contain the SUPER command. If no SUPER command is specified in the Case Control Section then all subcases will be assigned to all superelements. In other words, SUPER=ALL is the default.
2. All subcases associated with superelements must precede those for the residual structure except when SUPER=ALL or SUPER=n and the selected set includes the residual structure.

3. The load sequence number is only used in static analysis and frequency response analysis when there are multiple loading conditions. Also, the residual structure must have a subcase specified for each unique load condition. This is required because the number of residual structure subcases is used to determine the number of load conditions for all superelements.
4. The load sequence number is associated with the order of the subcases for the residual structure; i.e., the third loading condition is associated with the third subcase for the residual structure.
5. Subcases are required for superelements when there is a load, constraint, or output request.
6. If a set is referenced by n, then the SET identification number must be unique with respect to any superelement identification numbers. In addition, the same sets must be used for all loading conditions.
7. If the ALL option is used, it must be used for all loading conditions.

SUPORT1 Fictitious Support Set Selection

Selects the fictitious support set (SUPORT1 or SUPORT6 entries only) to be applied to the model.

Format:

SUPORT1=n

Examples:

SUPORT1=15
SUPO=4

Describer	Meaning
n	Set identification of fictitious support set defined on the SUPORT1 or SUPORT6 Bulk Data entries. (Integer > 0)

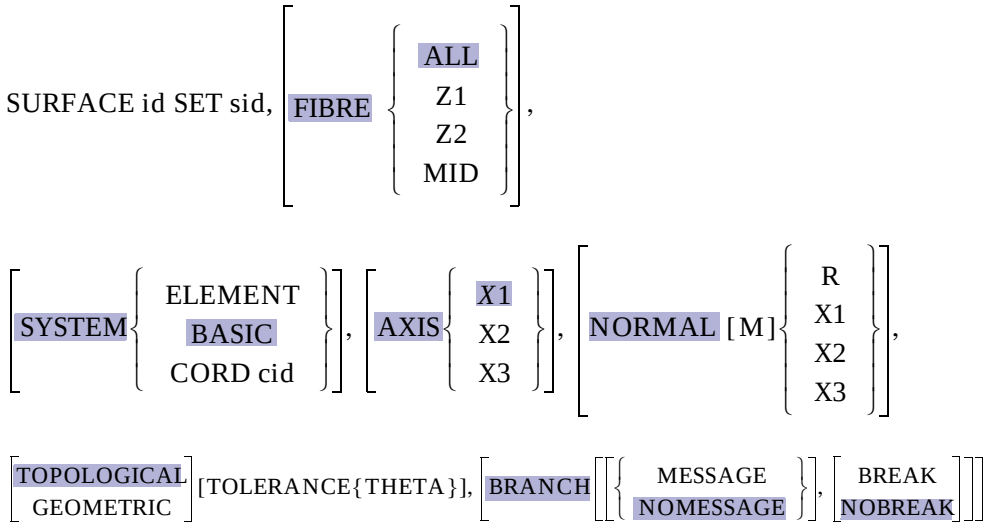
Remarks:

1. SUPORT1 or SUPORT6 entries will not be used unless selected in Case Control by the SUPORT1 command.
2. SUPORT entries will be applied in all subcases.
3. For SOL 600, Case Control command, SUPORT1 must reference a SUPORT6 Bulk Data entry with ID = N.

SURFACE Surface Definition

Defines a surface for the calculation of grid point stresses, strains, or mesh stress discontinuities.

Format:



Example:

SURFACE 10 SET 9 NORMAL X3

Describer	Meaning
id	Surface identification number (required).
SET	References a SET command that defines the elements in the surface (required). Either form of the SET command may be used.
sid	Set identification number.
FIBRE	Specifies the fiber location at which stresses will be calculated.
ALL	Requests output at all fibre locations; i.e., z=Z1, Z2, and MID.
Z1	Requests output z=Z1 only.
Z2	Requests output z=Z2 only.
MID	Requests output z=(Z1+Z2)/2 only.
SYSTEM	Specifies the coordinate system to be used to define the output coordinate system.

Describer	Meaning
ELEMENT	Specifies the element coordinate system for output.
CORD cid	Specifies the coordinate system defined on a CORDij Bulk Data entry for output.
BASIC	Specifies the basic coordinate system for output.
AXIS	Specifies the axis of the coordinate system to be used as the x output axis and the local x-axis when geometric interpolation is used.
X1, X2, X3	Specifies the direction of the axis or the normal. X, Y, and Z may be substituted for X1, X2, and X3, respectively.
NORMAL	Specifies the reference direction for positive fiber and shear stress output, but has no effect when ELEMENT is specified.
M	Specifies the reverse of the direction given by R, X1, X2, or X3.
R	Specifies the radius vector from the origin of reference coordinate system to the grid point.
TOPOLOGICAL GEOMETRIC	Specifies the method to calculate the average grid point stress or strain. The default is TOPOLOGICAL.
theta	Specifies the interelement slope difference tolerance (in degrees) to detect stress discontinuity between elements (not used with TOPOLOGICAL). (Real; Default = 0.0)
BRANCH	Selects whether multiple element intersections (BREAK / NOBREAK) are to be treated as discontinuities and if warning messages (MESSAGE / NOMESSAGE) are to be issued.
BREAK NOBREAK	Multiple element intersections are (or are not) to be treated as stress discontinuities.
MESSAGE NOMESSAGE	A warning message will (or will not) be issued when multiple element intersections are encountered.

Remarks:

1. SURFACE commands must be specified after OUTPUT(POST).
2. The surface identification number must be referenced on a SET command appearing after OUTPUT(POST). The SET identification number may then be referenced on a GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON commands. The seid on the surface entry must reference a SET defined after OUTPUT(POST).

3. The surface normal is also used in the definition of the local reference surface for geometric interpolation. Two options are available. In the first option, the radius vector (R) from the origin of the reference coordinate system to the grid point is used. In the second option, one axis (X1, X2, or X3) of the coordinate system is used. The direction can be reversed using the modification parameter, M. The positive side of an element is defined as a side from which the NORMAL direction emerges rather than the side determined by the connection specified on the element connection entries.
4. When the parameter ELEMENT is present, the element stresses or strains are used unmodified (defaults to output stresses in output system). The CORD keyword references a CORDij Bulk Data entry with coordinate system identification number cid.
5. When theta=0, no testing is made. When theta is negative, grid point stresses will be calculated for each element connected to an exception point; otherwise, the best estimation of the grid point stress will be output.
6. BREAK is the default if theta is nonzero.
7. For all elements defined in SET 9 of the previous example:
    - All fiber locations are output.
    - The basic output system is used.
    - The x-axis is x-axis of the basic system.
    - The surface normal direction point is z-axis of the basic system.
    - The topological interpolation method is used.
    - No tolerance test is made.
    - No branch test is made.

    The example illustrates a good choice for regular two-dimensional problems in the x-y plane.

SVECTOR Solution Set Eigenvector Output Request

Requests the form and type of solution set eigenvector output.

Format:

$$\text{SVECTOR}[(\text{PRINT}, \text{PUNCH})] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

SVECTOR=ALL
SVECTOR(PUNCH)=NONE

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
ALL	Displacements for all points (modes) will be output.
NONE	Displacements for no points (modes) will be output.
n	Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output. (Integer>0)

Remarks:

1. Both PRINT and PUNCH may be requested.
2. SVECTOR=NONE overrides an overall output request.
3. Output will be presented as a tabular listing of grid points for each eigenvector.

SVELOCITY Solution Set Velocity Output Request

Requests the form and type of solution set velocity output.

Format:

$$\text{SVELOCITY}\left[\left(\left[\begin{matrix}\text{SORT1}\\ \text{SORT2}\end{matrix}\right], \text{PRINT}, \text{PUNCH}, \left[\begin{matrix}\text{REAL or IMAG}\\ \text{PHASE}\end{matrix}\right]\right)\right] = \left\{\begin{matrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{matrix}\right\}$$

Examples:

SVELOCITY=5
SVELOCITY(SORT2,PUNCH,PRINT,PHASE)=ALL

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Velocity for all solution points (modes) will be output.
NONE	Velocity for no solution points (modes) will be output.
n	Set identification of a previously appearing SET command. Only velocities of points with identification numbers that appear on this SET command will be output. (Integer > 0)

Remarks:

1. Both PRINT and PUNCH may be requested.
2. Velocity output is only available for transient and frequency response problems.

3. The defaults for SORT1 and SORT2 depend on the type of analysis and is discussed in Remark 2 under the "**DISPLACEMENT**" on page 268 command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL then the remaining commands will also be SORT1.
4. SVELOCITY=NONE overrides an overall output request.

SYM Symmetry Subcase Delimiter

Delimits and identifies a symmetry subcase.

Format:

SYM=n

Example:

SYM=123

Describer	Meaning
n	Subcase identification number. (Integer > 0)

Remarks:

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. Plot commands should refer to n.
3. Overall output commands will not propagate into a SYM subcase (i.e., any output desired must be requested within the subcase).
4. SYM may only be used in statics or inertia relief problems.

SYMCOM Symmetry Combination Subcase Delimiter

Delimits and identifies a symmetry combination subcase.

Format:

SYMCOM=n

Example:

SYMCOM=123

Describer	Meaning
n	Subcase identification number. (Integer > 2)

Remarks:

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. SYMCOM may only be used in statics problems.
3. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SYMCOM by use of a TEMP(LOAD) command, or the element deformations by a DEFORM command.
4. An alternate command is the SUBCOM command.
5. SYMCOMs may be specified in superelement analysis with the following recommendations:
   - For each superelement, specify its SUBCASEs consecutively, directly followed by its SYMCOM(s).
   - Specify a SUPER command with a new load sequence number under each SYMCOM command.

   The following example is given for a model with one superelement and one load combination:

```
SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SYMCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SYMSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SYMCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SYMSEQ=1.,1.
```

SYMSEQ Symmetry Sequence Coefficients

Gives the coefficients for combining the symmetry subcases into the total structure.

Format:

SYMSEQ=R1 [,R2,R3,..., Rn]

Example:

SYMSEQ=1.0, -2.0, 3.0, 4.0

Describer	Meaning
Ri	Coefficients of the previously occurring n SYM subcases. (Real)

Remarks:

1. SYMSEQ may only appear after a SYMCOM command.
2. The default value for the coefficients is 1.0 if no SYMSEQ command appears.
3. SYMSEQ may only be used in statics or inertia relief.
4. Ri is limited to a maximum of 200 numbers.

TEMPERATURE Temperature Set Selection

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

Format:

$$\text{TEMPERATURE}\left[\begin{array}{c}\text{INITIAL}\\ \text{MATERIAL}\\ \text{LOAD}\\ \text{BOTH}\end{array}\right] = n$$

Examples:

TEMPERATURE(LOAD)=15
TEMPERATURE(MATERIAL)=7
TEMPERATURE=7

Describer	Meaning
MATERIAL	The selected temperature set will be used to determine temperature-dependent material properties indicated on the MATTi Bulk Data entries. See Remarks **6.**, **7.**, and **8.**
LOAD	The selected temperature set will be used to determine an equivalent static load and to update material properties in a nonlinear analysis. See Remarks **2.**, **5.**, **6.**, and **7.**
BOTH	Both MATERIAL and LOAD will use the same temperature set.
n	Set identification number of TEMP, TEMPD, TEMPP1, TEMPRB, TEMPF, or TEMPAX Bulk Data entries. (Integer > 0)
INITIAL	The selected temperature table will be used to determine initial temperature distribution in nonlinear static analysis. See Remarks **4.**, **6.**, **7.**, **8.**, **9.**, and **12.**

Remarks:

1. In linear analysis, only one temperature-dependent material request should be made in any problem and should be specified above the subcase level. If multiple requests are made, then only the last request will be processed. See also Remarks **6.** and **7.**

2. The total load applied will be the sum of external (LOAD command), thermal (TEMP(LOAD) command), element deformation (DEFORM command) and constrained displacement (SPC command) loads.
3. Static, thermal, and element deformation loads should have unique set identification numbers.
4. INITIAL is used in steady state heat transfer analysis for conduction material properties and provides starting values for iteration.
5. In superelement data recovery restarts, TEMPERATURE(LOAD) requests must be respecified in the Case Control Section.
6. In linear static analysis, temperature strains are calculated with:

$$\varepsilon_T = A(T_o) \cdot (T - T_o)$$

where $A(T_o)$ is the thermal expansion coefficient defined on the MATi Bulk Data entries, T is the load temperature defined with TEMPERATURE(LOAD) and T_o is the initial temperature defined below. The following rules apply for TEMPERATURE(INITIAL), TEMPERATURE(MATERIAL), and TREF on the MATi entries:

- If TEMPERATURE(INITIAL) and TREF are specified, then the TEMPERATURE(INITIAL) set will be used as the initial temperature to calculate both the loads and the material properties.
- If TEMPERATURE(MATERIAL) and TREF are specified, then TREF will be used as the initial temperature in calculating the load and the TEMPERATURE(MATERIAL) set will be used for the calculation of material properties.
- If neither TEMPERATURE(INITIAL), TEMPERATURE(MATERIAL), nor TEMPERATURE(BOTH) is present, TREF will be used to calculate both the load and the material properties and be obtained from the MATi entry. The MATTi is not used in this case.

7. In nonlinear static analysis, temperature strains are calculated with

$$\varepsilon_T = A(T) \cdot (T - \text{TREF}) - A(T_o) \cdot (T_o - \text{TREF})$$

where $A(T)$ is the thermal expansion coefficient defined on the MATi Bulk Data entries. T is the load temperature defined with TEMPERATURE(LOAD) and T_o is the initial temperature defined with TEMPERATURE(INITIAL). The following rules apply

- The specification of TEMPERATURE(INITIAL) is required above the subcase level. The specification of TEMPERATURE(MATERIAL) or TEMPERATURE(BOTH) will cause a fatal error.
- If a subcase does not contain a TEMPERATURE(LOAD) request, then the thermal load set will default to the TEMPERATURE(INITIAL) set.
- TEMPERATURE(LOAD) will also cause the update of temperature-dependent material properties due to the temperatures selected in the thermal load set. Temperature-dependent material properties are specified with MATi, MATTi, MATS1, and/or TABLEST Bulk Data entries.
- If TREF and TEMPERATURE(INITIAL) are specified, then the TEMPERATURE(INITIAL) set will be used as the initial temperature to calculate both the loads and the material properties. Both are used in the definition of thermal strain.

For SOL 600, TREF and TEMP(INIT) must be consistent (the same values) or unexpected results may occur.

8. TEMPERATURE(MATERIAL) and TEMPERATURE(INITIAL) cannot be specified simultaneously in the same run.
9. TEMP(INIT) is not used with TEMPAX.
10. Temperature loads cause incorrect stresses in dynamic analysis.
11. In linear analysis, TEMPERATURE(MATERIAL) is not supported for hyperelastic elements (MATHP). TEMP(INIT) must be placed above the subcase level and TEMP(LOAD) placed within the subcase.
12. For layered composites, neither the TREF specified or the material entries or TEMP(INIT) or TEMP(MATE) are used to determine ply reference temperature. The TREF on the PCOMP or PCOMPG entries is used for all plies of the element. This is true for both linear and nonlinear analysis.

TFL Transfer Function Set Selection

Selects the transfer function set(s) to be added to the direct input matrices.

Format:

TFL=n

Example:

TFL=77

TFL = 1, 25, 77

Describer	Meaning
n	Set identification of a TF Bulk Data entry. (Integer > 0)

Remarks:

1. Transfer functions will not be used unless selected in the Case Control Section.
2. Transfer functions are supported in dynamics problems only.
3. Transfer functions are described in the *MSC.Nastran Dynamics Users Guide.*
4. It is recommended that PARAM,AUTOSPC,NO be specified when using transfer functions. See "**Constraint and Mechanism Problem Identification in SubDMAP SEKR**" on page 409 of the *MSC.Nastran Reference Guide*.
5. The transfer functions are additive if multiple TF values are referenced on the TFL command.

THERMAL Temperature Output Request

Requests the form and type of temperature output.

Format:

$$\text{THERMAL}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix},\ \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

Examples:

THERMAL=5
THER(PRINT,PUNCH)=ALL

Describer	Meaning
SORT1	Output is presented as a tabular listing of point temperatures for each load or time step.
SORT2	Output is presented as a tabular listing of loads or time steps for each.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Compute temperatures but do not print.
ALL	Temperatures for all points will be output.
NONE	Temperatures for no points will be output.
n	Set identification of a previously appearing SET command. Only temperatures of points with identification numbers that appear on this SET command will be output. (Integer > 0)

Remarks:

1. The THERMAL output request is designed for use with the heat transfer option. The printed output will have temperature headings. The PUNCH option produces TEMP Bulk Data entries, and the SID on the entries will be the subcase number (=1 if no SUBCASES are specified).
2. SORT1 is the default in steady state heat transfer analysis. SORT2 is the default in transient heat transfer analysis.

3. In a transient heat transfer analysis, the SID on the punched TEMP Bulk Data entries, equal the time step number.

TITLE Output Title

Defines a character string that will appear on the first heading line of each page of MD Nastran printer output.

Format:

TITLE=title

Example:

TITLE=RIGHT WING, LOAD CASE 3.

Describer	Meaning
title	Any character string.

Remarks:

1. If this command appears under a SUBCASE command, then the title appears in the output for that subcase only.
2. If this command appears before all SUBCASE commands, then the title is used in all subcases without a TITLE command.
3. If no TITLE command is present, then the title line will contain data and page numbers only.
4. The title also appears on plotter output.

TRIM Aerodynamic Trim Variable Constraint Selection

Selects trim variable constraints in static aeroelastic response.

Format:

TRIM=n

Example:

TRIM=1

Describer	Meaning
n	Set identification number of a TRIM Bulk Data entry. (Integer>0)

Remark:

1. Aerodynamic extra points (trim variables) that are not constrained by a TRIM Bulk Data entry will be free during the static aeroelastic response solution.

TSTEP Transient Time Step Set Selection

Selects integration and output time steps for linear or nonlinear transient analysis.

Format:

TSTEP=n

Example:

TSTEP=731

Describer	Meaning
n	Set identification number of a TSTEP or TSTEPNL Bulk Data entry. (Integer>0)

Remarks:

1. A TSTEP entry must be selected to execute a linear transient analysis (SOLs 109 or 112) and TSTEPNL for a nonlinear transient analysis (SOLs 129 and 159).
2. A TSTEPNL entry must be selected in each subcase to execute a nonlinear transient problem.
3. For the application of time-dependent loads in modal frequency response analysis (SOLs 111 and 146), or TSTEP entry must be selected by the TSTEP command. The time-dependent loads will be recomputed in frequency domain by a Fourier Transform.
4. In one subcase or STEP for SOL 400, users should only specify TSTEP or TSTEPNL, but not both at the same time.

TSTEPNL Transient Time Step Set Selection for Nonlinear Analysis

See the description of the “TSTEP” on page 517.

TSTRU Temperature Set ID for a Structures Run

Defines a temperature set ID for a structures run based on a heat transfer subcase.

Format:

TSTRU=n

Example:

TSTRU=999

Describer	Meaning
n	Set identification for use on TEMP(LOAD)=n or TEMP(INIT)=n

Remarks:

1. TSTRU should be placed in heat transfer subcase.
2. If TSTRU does not explicitly appear in the heat transfer subcase, it is defaulted to TSTRU=Heat Transfer Subcase ID.
3. In a structures run, a temperature set generated from a heat transfer run will override an existing temperature set with identical set ID defined with TEMP, TEMPD, TEMPF, TEMPP1, TEMPRB or any combination.
4. TSTRU may be placed in the first subcase of a PARAM,HEATSTAT,YES run.
5. TSTRUs may be placed in each subcase of an APPHEAT run. The associated structures run then requires the following:

```
ASSIGN heat_run='heat transfer job name.MASTER'
DBLOC DATABLK=(UG,EST,BGPDTS,CASECCR/CASEHEAT) LOGICAL=heat_run
```

6. Heat transfer runs and structure runs must have the same mesh. P-elements should also have the same geometry description. P-order between runs can be different.
7. For nonlinear heat transfer SOL 106 or SOL 153 the INOUT field on the NLPARM Bulk Data entry must be blank or NO if the results of the run are to be transferred to a linear structures run.

   PARAM,NLHTLS,-1

   should be placed in the nonlinear heat run. This will place UG heat transfer on the database.

VECTOR Displacement Output Request

Requests the form and type of displacement vector output.

See the description of the “**DISPLACEMENT**” on page 268.

VELOCITY Velocity Output Request

Requests the form and type of velocity vector output.

Format:

$$\text{ELOCITY}\left[\left(\left[\begin{matrix}\text{SORT1}\\\text{SORT2}\end{matrix}\right],\left[\begin{matrix}\text{PRINT, PUNCH}\\\text{PLOT}\end{matrix}\right],\left[\begin{matrix}\text{REAL or IMAG}\\\text{PHASE}\end{matrix}\right]\left[\begin{matrix}\text{PSDF, ATOC, CRMS}\\\text{or RALL}\end{matrix}\right],\right.\right.$$

$$\left.\left.\left[\begin{matrix}\text{RPRINT}\\\text{NORPRINT}\end{matrix}\right],\text{RPUNCH}\right],[\text{CID}]\right) = \left\{\begin{matrix}\text{ALL}\\\text{n}\\\text{NONE}\end{matrix}\right\}$$

Examples:

VELOCITY=5
VELOCITY(SORT2,PHASE,PUNCH)=ALL
VELOCITY(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
VELOCITY(PRINT, RALL, NORPRINT)=ALL

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generates, but does not print, velocities.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.

Describer	Meaning
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
CRMS	Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the Case Control.
RPRINT	Writes random analysis results in the print file. (Default)
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output file, F06 file.
ALL	Velocity for all solution points will be output.
NONE	Velocity for no solution points will be output.
n	Set identification of a previously appearing SET command. Only velocities of points with identification numbers that appear on this SET command will be output. (Integer>0)

Remarks:

1. Both PRINT and PUNCH may be requested.
2. Velocity output is only available for transient and frequency response problems.
3. See Remark 2 under "**DISPLACEMENT**" on page 268 for a discussion of SORT1 and SORT2.
4. VELOCITY=NONE overrides an overall output request.
5. The PLOT option is used when curve plots are desired in the magnitude / phase representation and no printer request is present for magnitude / phase representation.

6. Velocity results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
7. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. The results can be either printed in the .f06 file or punched in the punch file, or output in both files.
8. Note that the CID keyword affects only grid point related output, such as DISPlacement, VELOcity, ACCEleration, OLOAD, SPCForce and MPCForce. In addition, CID keyword needs to appear only once in a grid related output request anywhere in the Case Control Section to turn on the printing algorithm.

VOLUME Volume Definition

Defines a volume for the calculation of grid point stresses, strains, or mesh stress discontinuities.

Format:

VOLUME id SET sid, [PRINCIPAL, DIRECT STRESS], [SYSTEM{ELEMENT / CORD cid / BASIC}]

Example:

VOLUME 21 SET 2

Describer	Meaning
id	Volume identification number.
sid	Set identification number of a SET command that defines the elements in the volume. Either form of the SET command may be used. The default is all elements.
PRINCIPAL	Requests principal stresses or strains, direction cosines, mean pressure, and von Mises equivalent stresses or strains to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both.
DIRECT	Requests direct stress or strains, mean pressure stress and von Mises equivalent stress to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both.
SYSTEM	Used to specify the reference coordinate system to be used to define the output stress orientation coordinate system.
ELEMENT	Specifies the element coordinate system.
CORD cid	Specifies the coordinate system specified on a CORDij entry.
BASIC	Specifies the basic coordinate system.

Remarks:

1. VOLUME commands must be specified after OUTPUT(POST).

2. The volume identification number must be referenced on a SET command appearing after OUTPUT(POST). The SET identification number may then be referenced on GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON commands.
3. If ELEMENT is specified, element stresses or strains are not transformed.
4. In the example above, for all elements in SET 2:
    - Both PRINCIPAL and DIRECT stress are output.
    - The BASIC output system is used.

VUGRID View Geometry Output for p-Version Analysis

Requests output of view grid and view element entries used in p-version element data recovery.

Format:

$$\text{VUGRID}\left[\left(\begin{array}{c}\text{PRINT, PUNCH}\\ \text{PLOT}\end{array}\right)\right]=\left\{\begin{array}{c}\text{ALL}\\ \text{n}\end{array}\right\}$$

Example:

VUGRID(PRINT)=n

Describer	Meaning
ALL	All view element and grid entries will be output.
n	Set identification of a previously appearing SET command. Only those p-version elements with identification numbers that appear on this SET command will be output. (Integer>0)
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
PLOT	Generate entries but do not print or punch.

Remarks:

1. VUGRID is processed only when an analysis with p-version elements is requested.
2. Only one VUGRID command per analysis is allowed.
3. The VUGRID command is used only for output control and does not in anyway affect the p-version analysis.
4. See parameters VUHEXA, VUTETRA, and VUPENTA in “**Parameters**” on page 659, for renaming element entries.
5. See parameters VUELJUMP and VUGJUMP in “**Parameters**” on page 659, for numbering of view grid and view element entries.

WEIGHTCHECK Rigid Body Mass Reduction Check

At each stage of the mass matrix reduction, compute rigid body mass and compare with the rigid body mass t the g-set.

Format:

$$\text{WEIGHTCHECK}\left[\begin{array}{l}\left(\left[\begin{array}{c}\text{PRINT}\\ \text{NOPRINT}\end{array}\right],\text{SET}=\left(\left\{\begin{array}{c}\text{G, N, N + AUTOSPC, F, A,V}\\ \text{ALL}\end{array}\right\}\right)\right.\\ \left.\text{GRID}=\text{gid},\text{CGI}=\left[\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right],\left[\begin{array}{c}\text{WEIGHT}\\ \text{MASS}\end{array}\right]\right)\end{array}\right]=\left\{\begin{array}{c}\text{YES}\\ \text{NO}\end{array}\right\}$$

Examples:

WEIGHTCHECK=YES
WEIGHTCHECK(GRID=12,SET=(G,N,A),MASS)=YES

Describer	Meaning
PRINT	Write output to the print file (Default).
NOPRINT	Do not write output to the print file.
SET	Selects degree-of-freedom set(s). (Default SET=G)
gid	Reference grid point for the calculation of the rigid body motion. The default is the origin of the basic coordinate system.
CGI	For SET ≠ G , CGI = YES requests output of center of gravity and mass moments of inertia. (Default: CGI = NO).
WEIGHT / MASS	Selects output in units of weight or mass. (Default = WEIGHT)

Remarks:

1. WEIGHTCHECK must be specified above the subcase level.
2. For SET=N, N+AUTOSPC, F, or A, the WEIGHTCHECK command also outputs a percentage loss or gain in the reduced rigid body mass matrix (e.g., MAA) as compared to the g-set rigid body mass matrix (e.g., MGG). G must also be requested to obtain this comparison; e.g., WEIGHTCHECK(SET=(G,A))=YES.
3. SET=N+AUTOSPC uses the mass matrix for the n-set with the rows corresponding to degrees-of-freedom constrained by the PARAM, AUTOSPC operation zeroed out. If AUTOSPC was not performed then this check is redundant with respect to SET=N.

4. WEIGHTCHECK is available in all SOLs. However, in SOLs 101, 105, 114, and 116, because no mass reduction is performed, only WEIGHTCHECK(SET=G) is available.

4.3 Case Control Applicability Tables

The following tables describe the applicability of Case Control commands to Solution Sequences:

Table 4-1 and **Table 4-2**	SOLs (101 through 200) -- Subcase Definition, Superelement Control, and Auxiliary Model Control
Table 4-3 and **Table 4-4**	SOLs (101 through 200) -- Data Selection
Table 4-5 and **Table 4-6**	SOLs (101 through 200) -- Output Selection

Table 4-1 Case Control Commands in SOLs 101 Through 112 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
ADACT	X	X								
ANALYSIS										
AUXCAS										
AUXMODEL										
BEGIN BULK	X	X	X	X	X	X	X	X	X	X
MASTER	X	X	X	X	X	X	X	X	X	X
MODES		X								
OUTPUT(blank)	X	X	X	X	X	X	X	X	X	X
OUTPUT(PLOT)	X	X	X	X	X	X	X	X	X	X
OUTPUT (POST) or SETS DEFINITION	X	X	X	X	X	X	X	X	X	X
OUTPUT(XYPLOT)	X	X	X	X	X	X	X	X	X	X
REPCASE	X	X								
SEALL	X	X	X	X	X	X	X	X	X	X
SEDR	X	X	X	X	X	X	X	X	X	X
SEDV										
SEEXCLUD	X	X	X	X	X	X	X	X	X	X
SEFINAL	X	X	X	X	X	X	X	X	X	X
SEKR	X	X	X	X	X	X	X	X	X	X
SELG	X		X	X		X	X		X	X
SELR	X		X	X		X	X		X	X
SEMR	X	X	X	X	X	X	X	X	X	X
SEMR		X	X		X	X	X	X	X	X
SERE										
SUBCASE	X	X	X	X	X	X	X	X	X	X
SUBCOM	X									
SUBSEQ	X									
SUPER	X	X	X	X	X	X	X	X	X	X
SYM	X									
SYMCOM	X									
SYMSEQ	X									

Table 4-2 Case Control Commands in SOLs 114 Through 200 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number										
	114	115	116	118	129	144	145	146	153	159	200
ADACT											
ANALYSIS									X		X
AUXCAS											X
AUXMODEL											X
BEGIN BULK	X	X	X	X	X	X	X	X	X	X	X
MASTER	X	X	X	X	X	X	X	X	X	X	X
MODES		X	X								X
OUTPUT(blank)	X	X	X	X	X	X	X	X	X	X	X
OUTPUT(PLOT)	X	X	X	X	X	X	X	X	X	X	X
OUTPUT (POST) or SETS DEFINITION	X	X	X	X	X	X	X	X	X	X	X
OUTPUT(XYPLOT)	X	X	X	X	X	X	X	X	X	X	X
REPCASE											X
SEALL	X	X	X	X	X	X	X	X	X	X	X
SEDR	X	X	X	X	X	X	X	X	X	X	X
SEDV											X
SEEXCLUD	X	X	X	X	X	X	X	X	X	X	X
SEFINAL	X	X	X	X	X	X	X	X	X	X	X
SEKR	X	X	X	X	X	X	X	X	X	X	X
SELG	X		X	X	X	X		X	X	X	X
SELR	X		X	X	X	X		X	X	X	X
SEMR	X	X	X	X	X	X	X	X	X	X	X
SEMR		X	X	X	X	X	X	X		X	X
SERE											X
SUBCASE	X	X	X	X	X	X	X	X	X	X	X
SUBCOM											X
SUBSEQ											X
SUPER	X	X	X	X	X	X	X	X	X	X	X
SYM											X
SYMCOM											X
SYMSEQ											X

Table 4-3 Case Control Commands in SOLs 101 Through 112 -- Data Selection

Command Name	Structured Solution Number									
	101	103	105	106	107	108	109	110	111	112
ADAPT	X	X								
AUTOSPC	X	X	X	X	X	X	X	X	X	X
AXISYMME	X	X	X		X	X	X	X	X	X
B2GG	X	X	X	X	X	X	X	X	X	X
B2PP					X	X	X	X	X	X
BC		X								
CLOAD				X						
CMETHOD					X			X		
DEFORM	X		X							
DESGLB										
DESOBJ										
DESSUB										
DLOAD					X	X	X	X	X	X
DSYM										
FMETHOD										
FREQUENC						X			X	
GUST										
HARMONIC	X	X	X		X	X	X	X	X	X
IC							X			
K2GG	X	X	X	X	X	X	X	X	X	X
K2PP					X	X	X	X	X	X
LOAD	X		X	X		X	X		X	X
LOADSET	X		X	X		X	X		X	X
M2GG	X	X	X	X	X	X	X	X	X	X
M2PP					X	X	X	X	X	X
METHOD		X		X	X	X	X	X	X	X
MFLUID		X			X	X	X	X	X	X
MODTRAK										
MPC	X	X	X	X	X	X	X	X	X	X
NLPARM				X						
NONLINEA							X			X
OMODES		X	X					X*	X*	X*
P2G	X		X	X		X	X		X	X
RANDOM						X			X	
RESVEC		X		X	X	X	X	X	X	X

Table 4-3 Case Control Commands in SOLs 101 Through 112 -- Data Selection (continued)

Command Name	Structured Solution Number									
	101	103	105	106	107	108	109	110	111	112
SDAMPING								X	X	X
SDENSITY	X	X	X							
SMETHOD	X					X				
SPC	X	X	X	X	X	X	X	X	X	X
STATSUB*		X	X		X	X	X	X	X	X
SUPORT1	X	X	X	X	X	X	X	X	X	X
TEMPER(INIT)				X						
TEMPER(LOAD)	X		X	X		X	X		X	X
TEMPER(MATE)	X	X	X	X	X	X	X	X	X	X
TFL					X	X	X	X	X	X
TRIM										
TSTEP							X			X
WEIGHTCHECK	X	X	X	X	X	X	X	X	X	X

*If STATSUB is specified, then the Case Control commands that select static loads become applicable to the solution sequence supporting STATSUB.

Table 4-4 Case Control Commands in SOLs 114 Through 200 -- Data Selection

Command Name	Solution Number										
	114	115	116	118	129	144	145	146	153	159	200
ADAPT											
AUTOSPC	X	X	X	X	X	X	X	X	X	X	X
AXISYMME											X
B2GG	X	X	X	X	X	X	X	X	X	X	X
B2PP				X	X		X	X		X	X
BC											
CLOAD									X		
CMETHOD							X				
DEFORM	X		X			X					X
DESGLB											X
DESOBJ											X
DESSUB											X
DLOAD				X	X		X	X		X	X
DSYM	X	X	X	X							
FMETHOD							X				X

Table 4-4 Case Control Commands in SOLs 114 Through 200 -- Data Selection (continued)

Command Name	Solution Number										
	114	115	116	118	129	144	145	146	153	159	200
FREQUENC				X				X			X
GUST								X	X		
HARMONIC	X	X	X	X							X
IC										X	
K2GG	X	X	X	X	X	X	X	X	X	X	X
K2PP				X	X		X	X		X	X
LOAD	X		X	X	X	X		X	X		X
LOADSET	X		X	X	X	X		X	X	X	X
M2GG	X	X	X	X	X	X	X	X	X	X	X
M2PP				X	X		X	X		X	X
METHOD		X	X	X	X	X	X	X	X	X	X
MFLUID		X	X	X	X	X	X	X			X
MODTRAK											
MPC	X	X	X	X	X	X	X	X	X	X	X
NLPARM									X		
NONLINEA					X					X	
OMODES		X									X
P2G	X		X	X	X	X		X	X		X
RANDOM				X				X			
RESVEC		X		X	X	X	X	X	X	X	X
SDAMPING							X	X			X
SMETHOD											
SPC	X	X	X	X	X	X	X	X	X	X	X
STATSUB				X							X
SUPORT1	X	X	X	X	X	X	X	X	X	X	X
TEMPER(INIT)											
TEMPER(LOAD)	X			X		X		X			X
TEMPER(MATE)	X	X	X	X	X	X	X	X	X	X	X
TFL				X	X		X	X		X	X
TRIM						X					X
TSTEP					X			X		X	X
WEIGHTCHECK	X	X	X	X	X	X	X	X			X

Table 4-5 Case Control Commands in SOLs 101 Through 112 -- Output Selection

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
ACCELERA		X				X	X		X	X
AEROF										
APRESS										
BOUTPUT				X						
CMSENERGY		X		X				X	X	X
DATAREC	X	X				X	X		X	X
DISPLACE	X	X	X	X	X	X	X	X	X	X
DSAPRT										
ECHO	X	X	X	X	X	X	X	X	X	X
ECHOOFF	X	X	X	X	X	X	X	X	X	X
ECHOON	X	X	X	X	X	X	X	X	X	X
EDE		X			X	X	X	X*	X	X
EKE		X			X	X	X	X*	X	X
ELSDCON	X		X							
ELSUM	X	X	X	X	X	X	X	X	X	X
ENTHALPY										
ESE	X	X	X	X	X	X	X	X*	X	X
FLUX	X									
FORCE	X	X	X	X	X	X	X	X	X	X
GPFORCE	X	X	X	X			X^+			X^+
GPKE		X						X*	X*	X*
GPSDCON	X		X							
GPSTRAIN	X	X		X			X			X
GPSTRESS	X	X		X			X			X
GROUNDCHECK	X	X	X	X	X	X	X	X	X	X
HARMONIC	X	X	X		X	X	X	X	X	X
HDOT										
HOUTPUT										
LABEL	X	X	X	X	X	X	X	X	X	X
LINE	X	X	X	X	X	X	X	X	X	X
MAXLINES	X	X	X				X			X
MAXMIN	X	X	X	X	X	X	X	X	X	X
MPCFORCE	X	X	X		X	X	X	X	X	X

Table 4-5 Case Control Commands in SOLs 101 Through 112 -- Output Selection (continued)

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
MPRESSURE		X			X	X	X	X	X	X
NLLOAD							X			X
NOUTPUT										
OFREQUEN						X			X	
OLOAD	X		X	X		X	X		X	X
OTIME							X			X
OUTRCV	X	X				X	X			X
PAGE	X	X	X	X	X	X	X	X	X	X
PARTN	X	X	X	X	X	X	X	X	X	X
PLOTID	X	X	X	X	X	X	X	X	X	X
POST	X									
PRESSURE		X			X	X	X			
SACCELER						X	X		X	X
SDISPLAC		X	X		X	X	X		X	X
SET	X	X	X	X	X	X	X	X	X	X
SKIP	X	X	X	X	X	X	X	X	X	X
SPCFORCE	X	X	X	X	X	X	X	X	X	X
STRAIN	X	X	X	X	X	X	X	X	X	X
STRESS	X	X	X	X	X	X	X	X	X	X
STRFIELD	X	X	X				X			X
SUBTITLE	X	X	X	X	X	X	X	X	X	X
SURFACE	X	X		X			X			X
SVECTOR		X	X					X	X	X
SVELOCITY						X	X		X	X
THERMAL	X									
TITLE	X	X	X	X	X	X	X	X	X	X
VECTOR	X	X	X	X	X	X	X	X	X	X
VELOCITY						X	X		X	X
VOLUME	X	X		X			X			X
VUGRID	X	X				X	X		X	X

*For modal part of solution.

+Forces limited to stiffness contributions.

Table 4-6 Case Control Commands in SOLs 114 Through 200 -- Output Selection

Command Name	Solution Number										
	114	115	116	118	129	144	145	146	153	159	200
ACCELERA				X	X			X			X
AEROF						X	X	X			X
APRESS						X					X
BOUTPUT					X				X	X	
CMSENERGY							X	X			X
DATAREC											X
DISPLACE	X	X	X	X	X	X	X	X	X	X	X
DSAPRT											X
ECHO	X	X	X	X	X	X	X	X	X	X	X
ECHOFF	X	X	X	X	X	X	X	X	X	X	X
ECHOON	X	X	X	X	X	X	X	X	X	X	X
EDE		X		X							X
EKE		X		X							X
ELSDCON						X					X
ELSUM	X	X	X	X	X	X	X	X			X
ENTHALPY										X	
ESE	X	X	X	X		X					X
FLUX	X								X	X	X
FORCE	X	X	X	X	X	X	X	X	X	X	X
GPFORCE	X	X	X	X							X
GPKE		X									X
GPSDCON	X					X			X		X
GPSTRAIN	X	X	X		X			X			X
GPSTRESS	X	X	X		X	X					X
GROUNDCHECK	X	X	X	X	X	X	X	X			X
HARMONY	X	X	X	X							X
HDOT										X	
HOUTPUT	X	X	X	X							
LABEL	X	X	X	X	X	X	X	X	X	X	X
LINE	X	X	X	X	X	X	X	X	X	X	X
MAXLINES	X	X	X	X	X	X	X	X	X	X	X
MAXMIN											
MPCFORCE	X	X	X	X		X	X	X			X
MPRESSURE		X		X		X	X	X			X
NLLOAD										X	
NOUTPUT	X	X	X	X							

Table 4-6 Case Control Commands in SOLs 114 Through 200 -- Output Selection (continued)

Command Name	Solution Number										
	114	115	116	118	129	144	145	146	153	159	200
OFREQUEN				X				X			X
OLOAD	X		X	X	X	X		X	X	X	X
OTIME	X		X	X	X	X		X	X	X	X
OUTRCV											
PAGE	X	X	X	X	X	X	X	X	X	X	X
PARTN	X	X	X	X	X	X	X	X	X	X	X
PLOTID	X	X	X	X	X	X	X	X	X	X	X
PRESSURE											X
SACCELER				X	X			X			X
SDISPLAC		X		X	X		X	X		X	X
SET	X	X	X	X	X	X	X	X	X	X	X
SKIP	X	X	X	X	X	X	X	X	X	X	X
SPCFORCE	X	X	X	X	X	X	X	X	X	X	X
STRAIN	X	X	X	X	X	X	X	X	X	X	X
STRESS	X	X	X	X	X	X	X	X	X	X	X
STRFIELD	X	X	X			X					X
SUBTITLE	X	X	X	X	X	X	X	X	X	X	X
SURFACE	X	X	X								X
SVECTOR		X	X	X			X		X		X
SVELOCITY				X	X			X			X
THERMAL	X								X	X	X
TITLE	X	X	X	X	X	X	X	X	X	X	X
VECTOR	X	X	X	X	X	X	X	X	X	X	X
VELOCITY				X	X			X			X
VOLUME	X	X	X								X
VUGRID											

4.4 OUTPUT(PLOT) Commands

The PLOT command requests the generation of undeformed, deformed, or contour plots. All other commands specify how the model will be plotted, type of projection, view angles, scales, etc. All commands have default actions if not specified by the user. The FIND command may be used to calculate an optimal SCALE, ORIGIN, and / or VANTAGE POINT to allow the construction of a plot in a user-specified region of the paper or film. All the commands used in the generation of the various plots will be printed out as part of the output, whether they are directly specified, defaulted or established using the FIND command. Initialization of commands to default values occurs only once. Subsequently, these values remain until altered by direct command input. The only exceptions are the view angles, scale factors, vantage points, and the origins. Whenever the plotter or the method of projection is changed, the view angles are reset to the default values, unless they are respecified by the user. In addition, the scale factors, vantage points, and the origin must be respecified by the user.

The commands are listed here in a logical sequence; however, they need not be so specified. Any order may be used, but if a command is specified more than once, the value or choice stated last will be used.

PLOTTER	Selects format of plot file for interpretation by plotter post-processor.
ORTHOGRAPHIC	Selects orthographic projection.
PERSPECTIVE	Selects perspective projection.
STEREOSCOPIC	Selects stereoscopic projection.
AXES	Assigns axes of the basic coordinate system to the observer's coordinate system.
VIEW	Defines the angular relationship between observer's coordinate system (r, s, and t axes specified on the AXES command) and basic coordinate system.
MAXIMUM DEFORMATION	Defines the magnification of the maximum displacement.
SCALE	Defines reduction, as a scale factor, of model's dimensions so that model fits on a plot frame.
DISTORTION	Specifies the distortion scale factors of the axes in the basic coordinate system.

CSCALE	Defines scale factor for characters in the plot frame.
ORIGIN	Defines the origin of the plot frame with respect to the origin of the (r, s, t) coordinate system defined on the AXES command.
VANTAGE POINT	Defines the location of the observer with respect to the model in the (r, s, t) coordinate system defined on the AXES command for perspective and stereoscopic projections only.
PROJECTION PLANE	Defines the separation, along the r-axis, between the observer and the projection plane if not already specified on the VANTAGE POINT command. Used by stereoscopic projections only.
OCULAR SEPARATION	Defines the separation of the left and right eye vantage points along the s-axis for stereoscopic projections.
CAMERA	Specifies microfilm plotter options.
PAPER SIZE	Defines the size and type of the paper.
PEN	Generates a message on the printed output to inform the plotter operator as to what size and color pen point to mount in the various pen holders.
PTITLE	Defines a character string that will appear at the top of the plot frame on the line below the sequence number.
SET	Defines a set of elements and / or grid points to be plotted.
FIND	Requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN i, and / or VANTAGE POINT commands.
CONTOUR	Specifies contour plot options for stress, displacement, or temperature.
PLOT	Generates an undeformed plot or a deformed plot per subcase, mode number, frequency, or time step. A contour plot may also be requested with an undeformed or deformed plot.

AXES Orientation of Observer's Coordinate System

Assigns axes of the basic coordinate system to the observer's coordinate system.

Format:

AXES r s t [SYMMETRIC / ANTISYMMETRIC]

Example 1: View toward negative x-axis of model.

AXES MX, Y, MZ

Example 2: Mirror image of model.

AXES Y X Z

Describers	Meaning
r,s,t	Assigns axes of basic coordinate system to axes of observer's coordinate system: (Default = X, Y, Z)
	X X-direction of basic coordinate system. (Default for r)
	Y Y-direction of basic coordinate system. (Default for s)
	Z Z-direction of basic coordinate system. (Default for t)
	MX Negative X-direction of basic coordinate system
	MY Negative Y-direction of basic coordinate system
	MZ Negative Z-direction of basic coordinate system
SYMMETRIC	Specifies a symmetric orientation of the view. See Remark 5. (Default)
ANTISYMMETRIC	Specifies an antisymmetric orientation of the view. See Remark 5.

Remarks:

1. If no AXES command is specified then AXES X, Y, Z is the default.
2. The direction of view is in the negative r-direction; i.e., the projection plane is parallel to the s-t plane.
3. The VIEW command depends on the AXES command specification and defines the angular relationship between observer's coordinate system and the basic coordinate system.

4. The AXES command can be used to preposition the object in 90° increments in such a manner that only rotations less than 90° are required by the VIEW command to obtain the desired orientation. MX, MY, MZ can be used to define left-handed coordinate systems. Note that the default system is right-handed.
5. An undeformed or deformed plot of the symmetric portion of an object can be obtained by reversing the sign of the axis that is normal to the plane of symmetry. In the case of multiple planes of symmetry, the signs of all associated planes should be reversed. The ANTISYMMETRIC option should be specified when a symmetric model is loaded in an unsymmetric manner. This will cause the deformations to be plotted antisymmetrically with respect to the specified plane or planes. Since the AXES command applies to all parts (SETs) of a single frame, symmetric and antisymmetric combinations cannot be made with this command (see the symmetry option on the "**PLOT**" on page 558 command in this section).
6. To avoid a mirror image, ensure that the r, s, and t axes obey the right-hand rule.

CAMERA Microfilm Plotter Options

Specifies microfilm plotter options.

Format:

CAMERA [PAPER / FILM / BOTH] BLANK FRAME n

Example:

CAMERA FILM

Describers	Meaning
FILM	Requests 35mm or 16mm film and positive or negative images.
PAPER	Requests positive prints.
BOTH	Requests positive prints and 35mm or 16mm film.

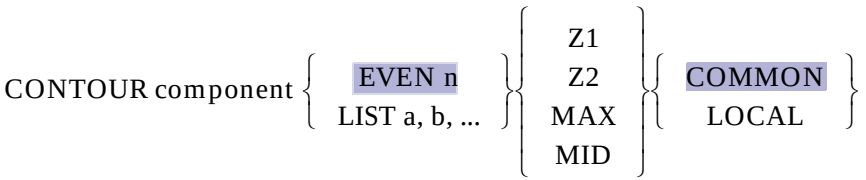
Remarks:

1. If the CAMERA command is not specified then CAMERA PAPER BLANK FRAMES 0 is assumed.
2. If FILM or BOTH is specified then these options must be communicated to the plotter operator through normal means of communications at the installation.
3. If FILM or BOTH are specified and if n is greater than 0 then n blank frames will be inserted between plots. The plotter must be operated in the manual mode in order to have blank frames inserted between positive prints. If blank frames are desired only on film, and not on paper, the plotter must be operated in the automatic mode.

CONTOUR Contour Plot Options

Specifies contour plot options for stress, displacement, or temperature.

Format:



Example:

```
CONTOUR  MAGNIT  LIST  2., 4., 6., 8., 10.
```

Describers	Meaning
component	Name of stress, displacement or temperature component. (Character, Default="MAJPRIN")
	MAJPRIN — Major principal stress. Not available for nonlinear elements. (Default)
	MINPRIN — Minor principal stress. Not available for nonlinear elements.
	EQUIVAL — von Mises stress. When STRESS(MAXS) is requested in Case Control, von Mises stress is used for plotting. For nonlinear analysis, Mohr-Coulomb or Drucker-Prager stress may also be plotted in conjunction with the MATS1 command.
	XNORMAL — X component of normal stress.
	YNORMAL — Y component of normal stress.
	ZNORMAL — Z component of normal stress.
	XYSHEAR — XY component of shear stress.
	XZSHEAR — XZ component of shear stress.
	YZSHEAR — YZ component of shear stress.
	XDISP — T1 component of displacement in global coordinate system.
	YDISP — T2 component of displacement in global coordinate system.

Describers	Meaning	
	ZDISP	T3 component of displacement in global coordinate system.
	MAGNIT	Magnitude of displacement or temperature.
EVEN n	Number of contours. (50≥Integer>0, Default is EVEN 10)	
LIST a, b, ...	List of stresses, displacements or temperatures which define the contours. (Real)	
Z1	Stresses at Z1 from neutral plane. (Default).	
Z2	Stresses at Z2 from neutral plane.	
MAX	Maximum of stress at Z1 and Z2.	
MID	Average of stress (membrane stress) at Z1 and Z2.	
COMMON	Plot stress contours in basic coordinate system (Default).	
LOCAL	Plot stress contours in element coordinate system. This is the coordinate system used in printed output.	

Remarks:

1. The CONTOUR command should be specified immediately before its associated PLOT command.
2. A STRESS request must appear in the Case Control Section for all elements included in a CONTOUR request. If printed output is not desired then STRESS(PLOT)=sid should be specified.
3. In linear analysis, stress contour plots are available for the following elements: CTRIA3, CQUAD4, CSHEAR, and CTRlAX6. In nonlinear analysis, stress contour plots are available for CQUAD4 and CTRlA3 elements. The Bulk Data element connection entries for all elements must list the grid points in either clockwise or counterclockwise order. Mixing the order will result in meaningless or confusing plots.
4. When selecting contour options, note:
    - MAJPRIN, MINPRIN, EQUIVAL are the same in COMMON and LOCAL.
    - ZNORMAL, XZSHEAR, YZSHEAR, if selected in LOCAL, will be changed to COMMON.
    - CSHEAR elements only have the MAXSHEAR value.

5. The CTRIAX6 element stress contour plots are different in that they must be selected as COMMON. Also, the following equivalences apply:

   XNORMAL is radial
   YNORMAL is azimuthal
   ZNORMAL is axial
   XYSHEAR is shear
   XZSHEAR is maximum principal
   YZSHEAR is von Mises
   EQUIVAL is octahedra

CSCALE Character Scale Factor

Defines scale factor for characters in the plot frame.

Format:

CSCALE cs

Example:

CSCA 2.0

Describer	Meaning
cs	Scale factor applied to characters in the plot frame. (Default = .5)

Remarks:

1. CSCALE is used to control the spacing of characters when plots are made with the NASTRAN plotter and they are postprocessed with the MSC / NASPLOT routine. For example, if the SCALE FACTOR on the NASPLOT data command is 2.0, a value for cs of 0.5 will result in characters of default size (.07 inches) at the regular spacing. A value of 1.8 produces good spacing when using the post-processing plotter programs NASTPLT, TEKPLT, and NEUPS. On the other hand, if the user wishes to double the size of both the plot and the characters, the SCALE FACTOR and the CSCALE FACTOR on the NASPLOT data command should both be set equal to 2.0.
2. The CSCALE command must immediately precede the PLOTTER command. If a second CSCALE command is specified then a second PLOTTER command must also be specified.

DISTORTION Distortion Scale Factors

Specifies the distortion scale factors of the axes in the basic coordinate system.

Format:

DISTORTION dx dy dz

Example:

DIST 0.5 1.0 1.0

Describers	Meaning
dx	Distortion scale factor of the basic coordinate system's x-axes. (Default=1.0)
dy	Distortion scale factor of the basic coordinate system's y-axes. (Default=1.0)
dz	Distortion scale factor of the basic coordinate system's z-axes. (Default=1.0)

Remarks:

1. If no DISTORTION command is specified then no distortion is applied.
2. If DISTORTION is specified then all three values for dx, dy, and dx must be specified even though one or two will use the default.
3. The distortion factors are applied prior to any other scaling commands: SCALE, MAXIMUM DEFORMATION, CSCALE, etc.

FIND Automatic Calculation of Scale, Origin, and Vantage Point

Requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN i, and / or VANTAGE POINT commands.

Format:

FIND [SCALE ORIGIN oid VANTAGE POINT SET setid ,
REGION { le be re te }]

Example:

FIND SCALE ORIGIN 5 SET 2

Describers	Meaning
oid	Origin identification number. (Integer>0)
setid	Set identification number. (Integer>0)
le	Fractional distance of left edge of plot region from the lower left corner of the image area. (Real, Default=0.0)
be	Fractional distance of bottom edge of plot region from the lower left corner of the image area. (Real, Default=0.0)
re	Fractional distance of right edge of plot region from the lower left corner of the image area. (Real, Default=1.0)
te	Fractional distance of top edge of plot region from the lower left corner of the image area. (Real, Defaul=1.0)

Remarks:

1. The FIND command is recommended over the specification of SCALE, ORIGIN, and VANTAGE POINT commands and should be specified prior to its associated PLOT or CONTOUR command.
2. The FIND command requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN, and / or VANTAGE POINT commands and based on the specification of the
   - PLOTTER command
   - PROJECTION PLANE command
   - SET and REGION specifications on the FIND command
   - VIEW and / or AXES commands,
   - MAXIMUM DEFORMATION command and

- PAPER SIZE command. All of these commands must precede the associated FIND command.

3. The FIND command can be used to compute any or all of SCALE, ORIGIN, or VANTAGE POINT as long as they have not been previously specified.
4. If SET is not specified then the first defined SET will be used.
5. If no options are specified on the FIND command, a SCALE and VANTAGE POINT are selected and an ORIGIN is located, using the first defined SET, so that the plotter object is located within the image area.
6. The plot region is defined as some fraction of the image area (image area = 0., 0., 1., 1. and first quadrant = .5, .5, 1., 1.). The image area is located inside the margins on the paper.

MAXIMUM DEFORM

Length of Maximum Displacement

Defines the magnification of the maximum displacement. All other displacements are scaled accordingly.

Format:

MAXIMUM DEFORMATION d

Example:

Magnify the displacements such that the maximum displacement is equal to two units of length of the model.

MAXI DEFO 2.

Describer	Meaning
d	Specifies the length, in units of the model and not of the plot frame, to which the maximum displacement is scaled. (Default=5% of the largest dimension of the model represented by the elements in the SET specification on the PLOT command.)

Remarks:

1. If no MAXIMUM DEFORMATION command is specified then the default shown above is assumed.
2. If you wish the FIND command to utilize the d value, a MAXIMUM DEFORMATION command should precede the FIND command.
3. If you wish the plot deformation scaling to be different then the FIND scaling a different MAXIMUM DEFORMATION command can appear first before the PLOT command.
4. For nonlinear plotting MAXIMUM DEFORMATION d and the MAXIMUM DEFORMATION field on the PLOT command should have the same value.

OCULAR SEPARATION Separation of the Vantage Points

Defines the separation of the left and right eye vantage points along the s-axis for stereoscopic projections.

Format:

OCULAR SEPARATION os

Example:

OCULAR SEPARATION 2.0

Describer	Meaning
os	Separation, in inches, of the two vantage points along the s-axis. (Real, Default=2.756 inches)

Remark:

1. The default value is the separation used in the standard stereoscopic cameras and viewers (70mm). The default value is recommended.

ORIGIN Origin of Plot Frame

Defines the origin of the plot frame with respect to the origin of the (r, s, t) coordinate system defined on the AXES command.

Format:

ORIGIN oid u v

Example:

ORIG 3 -1. -2.

Describers	Meaning
oid	Origin identification number which may be specifed after the ORIGIN describer on the PLOT command. (Integer>0)
u	Horizontal displacement of plot frame origin from the rst origin. (Real, Default=0.0)
v	Vertical displacement of paper origin from the rst origin. (Real, Default=0.0)

Remarks:

1. In the transformation performed for any of the three projections, the origins of both the basic coordinate system and the observer's coordinate system are coincident. The ORIGIN command may be used to locate the plot frame origin (lower left hand corner) from the rst origin. The units are inches and are not subject to the scaling of the plotted object.
2. The ORIGIN command is not recommended for general use. See the "**FIND**" on page 549 command to have the origin optimally located so as to place the plotted object in the center of the plot frame.
3. Ten origins may be specified at one time. However, any one can be redefined at any time. An eleventh origin is also provided if more than ten origins are erroneously defined; i.e., only the last of these surplus origins will be retained.
4. If a projection; e.g., ORTHOGRAPHIC, STEREOSCOPIC, or PERSPECTIVE, is changed in the plot packet, or the PLOTTER command is changed, then all previously defined origins are deleted.

ORTHOGRAPHIC, etc. Type of Projection

Selects type of projection.

Format:

ORTHOGRAPHIC
PERSPECTIVE
STEROSCOPIC

Describers	Meaning
ORTHOGRAPHIC	Selects orthographic projection. (Default)
PERSPECTIVE	Selects perspective projection.
STEREOSCOPIC	Selects stereoscopic projection.

Remark:

1. If none of the above projections are specified then ORTHOGRAPHIC is used.

PAPER SIZE Paper Dimensions

Defines the size and type of the paper.

Format:

PAPER SIZE h X or BY v [TYPE ptype]

Example:

PAPER SIZE 10. BY 10.

Describers	Meaning
h	Horizontal size of paper in inches. (Real, Default=20.0)
v	Vertical size of paper in inches. (Real, Default=20.0)
ptype	Paper type. (Character, Default="VELLUM")

Remarks:

1. The default paper size for the PLOTTER NAST is 20 by 20 inches which is converted to a 7 by 7 inch plot frame by the NASPLOT postprocessor.
2. PAPER SIZE can be specified along with the NASPLOT postprocessor to create rectangular plots. For example, the command will result in a 14 by 7 inch plot frame if the default value of 1.0 is used for the SCALE FACTOR on the NASPLOT command. The SCALE FACTOR on the NASPLOT data command can be used to make larger plots having the shape defined with PAPER SIZE.
3. PAPER SIZE also affects the raster count for the NASTRAN plotter. The default raster count is 1000 for a paper size of 20 by 20. Doubling the paper size to 40 by 40 will double the raster count to 2000.

PEN Pen Color and Size Assignments

Generates a message on the printed output which may be used to inform the plotter operator as to what size and color pen point to mount in the various pen holders.

Format:

PEN pn [COLOR cname]

Example:

PEN 2 COLOR RED

Describers	Meaning
pn	Pen identification number. (Integer>0)
COLOR	Flag indicating the next word is a color name.
cname	Pen color. (Character)

Remarks:

1. The actual number of pens available will depend on the plotter hardware configuration at each installation.
2. The PEN command does not control the pen used in generating the plot. See the PEN describer on the "**PLOT**" on page 558 command.
3. The PEN command is optional and is not appropriate for microfilm plotters.

PERSPECTIVE Selects Perspective Projection

See the description of the **“ORTHOGRAPHIC, etc.**” on page 554.

PLOT Undeformed or Deformed Plot Request

Generates an undeformed plot of the model or a deformed plot for a subcase, mode number or time step.

Format:

PLOT [analysis][dtype][CONTOUR][i1, i2 THRU i3, i4, etc.] [{ RANGE f1, f2 / TIME t1, t2 }] ,

[{ PHASE LAG ϕ / MAGNITUDE }] [MAXIMUM DEFORMATION d] ,

[SET sid1] [ORIGIN oid1] [{ SYMMETRY / ANTISYMMETRY } w] [{ PEN / DENSITY } p] [SYMBOLS m[,n]] ,

[LABEL label] [[SHAPE / OUTLINE]] [VECTOR v] , [PRINT] ,

[SHRINK t, o] [NORMALS] ,

[SET sid2] [ORIGIN oid2] etc.

Describers	Meaning	
analysis	Type of analysis results. (Character, default results in an undeformed plot or undeformed underlay for contour plots)	
	STATIC	Plot static deformations.
	MODAL	Plot mode shapes or eigenvectors.
	CMODAL	Plot complex mode shapes or eigenvectors.
	TRANSIENT	Plot transient solutions.
	FREQUENCY	Plot frequency solutions.
	SENOMOVE	Plot undeformed superelements in their original position; i.e., ignore SELOC Bulk Data entry.
dtype	Specifies plot quantity. (Character, Default = “DEFORMATION”)	
	DEFORMATION	Plot displacements or temperatures in the Z direction. (Default)

Describers	Meaning
	VELOCITY Plot velocities.
	ACCELERATION Plot accelerations.
CONTOUR	Request for contour plot.
i1, i2,...	Subcase identification numbers. See SHAPE and VECTOR for use of “0" (underlay) command. See Remark 3. (Integer≥0, Default is all subcases)
RANGE f1 f2	Specifies range of natural frequencies, eigenvalues, excitation
TIME t1,t2	frequencies, time steps, or load factors. Used to minimize the amount of plotted data. See Remark 4. (Real)
PHASE LAG ϕ	Specified phase lag, in degrees, for plotting complex quantities. See Remark 5. (Real, Default=0.0)
MAGNITUDE	Requests magnitude of complex quantities.
MAXIMUM DEFORMATION d	Specifies the magnification of the maximum displacement. See Remark 6. (Real)
SET sid	Set identification number which defines the set of elements or grid points to be plotted. (Default is first SET command)
ORIGIN oid	Origin identification number. (Default is first origin defined by the ORIGIN or FIND command)
SYMMETRY w ANTISYMMETRY w	Request plot of the symmetric portion of the symmetrically or antisymmetrically loaded model. This symmetric portion will be located in the space adjacent to the region originally defined by ORIGIN oid, and will appear as a reflection of the antisymmetrically deformed model about the plane whose normal is oriented parallel to the coordinate direction w. See Remark 7. (Default is no action)
PEN p	Specifies pen number that is used to generate the plot. (Integer>0)
DENSITY d	Specifies line density scale factor for film plotters which is d times heavier than a line density of 1. (Integer>0)

Describers	Meaning
SYMBOLS m[,n]	All the grid points associated with the specified set will have symbol m overprinted with symbol n printed at its location. If n is not specified, only symbol m will be printed. See Remark **8.**

m or n	Symbol
0	none
1	X
2	*
3	+
4	-
5	.
6	×
7	[ ]
8	< >
9	/ \

LABEL label	Specifies labeling options at elements and grid points:	
	GRID	All the grid points included in the specified set have their identification number printed to the right of the undeformed or deformed location (undeformed location in the case of superimposed plots). (Default)
	ELEMENTS	All the elements included in the specified set are identified by the element identification number and type at the center of each element (undeformed location in the case of superimposed plots). See Remarks **11.** and **12.**
	BOTH	Both GRID and ELEMENT options.

Describers	Meaning	
	GSPC	Label those degrees-of-freedom that are constrained to zero through permanent single point constraints on GRID and GRDSET Bulk Data entries, or are constrained through SPC and SPC1 Bulk Data entries. The label consists of the grid point ID number and the constrained degrees-of-freedom.
	EPID	Label elements with their respective property identification (PID) numbers. The label consists of the standard element labels and element PID.
SHAPE	All the elements included in the specified set are shown by connecting the associated grid points in a predetermined manner. See Remark **9.**	
OUTLINE	Only the outline of all the elements in the specified set are shown. Elements not supported by contour plots are ignored. Outlines are always drawn with PEN 1. See Remark **9.**	
VECTOR v	A line will be plotted at the grid points of the set representing by length and direction the deformation of the point. See Remark **10.** Possible values of v are:	
	X, Y, or Z	Requesting individual components.
	XY, XZ, or YZ	Requesting two specified components.
	XYZ	Requesting all three components.
	RXY, RXZ, or RYZ	Requesting vector sum of two components.
	R	Requesting total vector deformation.
	N	Used with any of the above combinations to request no underlay shape be drawn.
PRINT	List of the average stresses at the interior grid points in the set will be printed for contour stress plots.	

Describers	Meaning
SHRINK t,o	t is the ratio of the reduction to the original dimensions of all two-dimensional elements except the CQUAD8 and CTRIA6 ($0.0 \leq t \leq 1.0$) (Real, Default=0.1 which results in a 10% reduction. o is the ratio of the reduction in length to the original length for one-dimensional elements. There is no default value for o. t must be specified to shrink one-dimensional elements.
NORMALS	Plot vector normal to CHBDYP and CHBDYG elements.

***Examples:**

See after Remarks Section.

Remarks:

1. If PLOT is specified with no describers then a picture of the undeformed model will be prepared using the first defined set and the first defined origin.
2. Describers analysis through PHASE LAG must be specified in the order shown above.
3. The following should be noted when using subcase numbers for plotting eigenvectors.
    - If subcase numbers are specified then the convention for displacement vectors is that the list of subcases must refer to subcase IDs whenever the number of modes recovered is equal to or less than the number of subcases defined. If the number of modes recovered is more than the subcases defined, the plot request for those modes associated with the subcases must refer to subcase IDs. After the mode associated with the last defined subcase, higher modes will be identified by incrementing the last defined subcase ID by one for each such higher mode.
    - For the display of element quantities in contour plots, the automatic incrementing beyond the last defined subcase does not occur. All subcase numbers to be plotted must be defined. A MODES command in the Case Control Section may be used for this purpose.

- In problems using cyclic symmetry solution sequences, the plot requests for segments of the model must refer to the coded subcase identification numbers (see "**Theory**" on page 811 of the *MSC.Nastran Reference Manual*). All eigenvectors recovered for the segment will be plotted. The RANGE option can be used to select a subset of all eigenvectors for plotting without use of coded subcase IDs.
- RANGE does not require the use of subcase numbers.

4. RANGE specifies the range of values using requested subcases for which plots will be prepared. If only one number is input, it is used as the lower bound and the upper bound is set to the highest value computed. Unless otherwise noted, the default range is all values computed.
    - In real eigenvalue analysis, the values are natural frequencies, in units of cycles per unit time.
    - In buckling analysis, the values are eigenvalues.
    - In frequency response, the values are excitation frequencies in units of cycles per unit time.
    - In transient response, the values are in units of time.
    - In static nonlinear analysis (SOLs 106 and 153), the values are load factors. The default range is the highest load factor of each subcase.
    - In transient nonlinear analysis (SOLs 129 and 159), the values are in units of time. The default range is the last time step for each subcase.
5. PHASE LAG ϕ is used in the equation:

$$u_R \cos\phi - u_I \sin\phi$$

    where u_R an u_I are the real and imaginary parts of the response quantity, respectively. The printed output for magnitude and phase uses the convention of a phase lead.
6. MAX DEFO is not recommended for general use. Each subcase is separately scaled according to its own maximum if this item is absent. If d is omitted, the set will be scaled to the maximum within the set being plotted.
7. w specifies the basic coordinates X, Y or Z or any combination thereof. This option allows the plotting of symmetric and / or antisymmetric combinations, provided that an origin is selected for the portion of the model defined in the Bulk Data Section that allows sufficient room for the complete plot. This does not permit the combination of symmetric and antisymmetric subcases, as each plot must represent a single subcase. In the case of a double

reflection, the figure will appear as one reflected about the plane whose normal is parallel to the first of the coordinates w, followed by a reflection about the plane whose normal is oriented parallel to the second of the coordinates w. This capability is primarily used in the plotting of models that are loaded in a symmetric or an antisymmetric manner. The plane of symmetry must be one of the basic coordinate planes.

8. Grid points excluded from the set will not have a symbol. Grid points in an undeformed underlay will be identified with symbol 2.
9. In order to get a deformed shape, either SHAPE or OUTLINE must be present in the PLOT command. Both deformed and undeformed shapes or outlines may be specified. All the deformed shapes relating to the subcases listed may be underlaid on each of their plots by including “0" with the subcase string on the PLOT command. The undeformed plot will be drawn using PEN 1 or DENSITY 1 and symbol 2 (if SYMBOLS is specified).
10. All plots requesting the VECTOR option will have an underlay generated of the undeformed shape using the same sets, PEN 1 or DENSITY 1, and symbol 2 (if SYMBOLS is specified). If SHAPE and VECTOR are specified, the underlay will depend on whether “0" is used along with the subcases with DEFORMATION. It will be the deformed shape when not used and will be both deformed and undeformed shapes when it is used. The part of the vector at the grid point will be the tail when the underlay is undeformed and the head when it is deformed. If the v=“N” then no shape will be drawn but other options such as SYMBOLS will still be valid.
11. Element type labels are: (Plot labels QH and TH indicate hyperelastic elements)

Element Type	Plot Label
CAERO1	AE
CAXIF2	A2
CAXIF3	A3
CAXIF4	A4
CBAR	BR
CBEAM	BM
CBEND	BD
CONEAX	CN

Element Type	Plot Label
PLOTEL	PL
CQUAD	QH
CQUAD4	Q4 or QH
CQUAD8	Q8 or QH
CQUADR	QR
CQUADX	QH
CROD	RD
CSHEAR	SH

Element Type	Plot Label
CONROD	CR
CDUMI	Di
CFLUID2	F2
CFLUID3	F3
CFLUID4	F4
CHBDYG	HB
CHBDYP	HB
CHEXA	HA
CPENTA	HA

Element Type	Plot Label
CSLOT3	S3
CSLOT4	S4
CTETRA	TE
CTRIAX6	D1
CTRIA3	T3 or TH
CTRIA6	T6 or TH
CTRIAR	TR
CTRIAX	TH
CTUBE	TU
CVISC	VS

12. The heat transfer boundary condition elements CHBDYG and CHBDYP can be plotted for undeformed plots. There are several types of CHBDYi elements, as follows:

Type	No. of Primary Grid Points	Normals Available
POINT	1	yes
LINE	2	yes
AREA3	3	yes
AREA4	4	yes
REV	2	no
ELCYL	2	no
TUBE	2	yes
FTUBE	2	yes
AREA6	6	yes
AREA6	8	yes

The secondary grid points are used for ambient conditions and are ignored by the plotter. Type POINT must have a nonzero associated area (see AF on the associated PHBDY entry) and a defined normal direction (see V1, V2, V3 on the CHBDYP entry) to be plotted. It is plotted as a hexagon with approximately the correct area. Type LINE must have a nonzero width (see AF on the associated PHBDY entry) and a normal defined in order to plot.

13. To assign PLOT command to superelements it requires an SEUPPLOT or a SEPLOT command.

Examples:

1. Undeformed SHAPE using first defined SET, first defined ORIGIN and PEN 1 (or DENSITY 1).

   PLOT

2. Undeformed SHAPE using SET 3, ORIGIN 4, PEN 2 (or DENSITY 2) with each grid point of the set having a + placed at its location, and its identification number printed adjacent to it.

   PLOT SET 3 ORIGIN 4 PEN 2 SHAPE SYMBOLS 3 LABEL

3. Modal deformations as defined in subcase 5 using first defined SET, first defined ORIGIN, and PEN 1 (or DENSITY 1).

   PLOT MODAL DEFORMATION 5 SHAPE

4. STATIC deformations as defined in subcases 3, 4, 5, and 8 deformed SHAPE; drawn with PEN 4, using first defined SET and ORIGIN, underlaid with undeformed SHAPE drawn with PEN 1. This command will cause four plots to be generated.

   PLOT STATIC DEFORMATION 0, 3 THRU 5, 8 PEN 4, SHAPE

5. Deformations as defined in· subcases 1, 2, 3, 4, and 5 undeformed underlay with PEN 1, consisting of SET 2 at ORIGIN 3, SET 2 at ORIGIN 4 (with a < placed at each grid point location), and SET 35 at ORIGIN 4. Deformed data as follows: SHAPE using SET 2 at ORIGIN 3 (PEN 3) and SET 35 at ORIGIN 4 (PEN 4); 3 VECTORS (X, Y and Z) drawn at each grid point of SET 2 at ORIGIN 4 (PEN 4) (less any excluded grid points), with o placed at the end of each vector.

   ```
   PLOT STATIC DEFORMATION 0 THRU 5,
      SET 2 ORIGIN 3 PEN 3 SHAPE,
      SET 2 ORIGIN 4 PEN 4 VECTORS XYZ SYMBOLS 0,
      SET 35 SHAPE
   ```

6. Static deformations as defined in subcases 3 and 4, both halves of a problem solved by symmetry using the X-Y principal plane as the plane of symmetry. SET 1 at ORIGIN 2 and SET 2 at ORIGIN 3, with the deformed shape plotted using DENSITY 3 and the undeformed model plotted using DENSITY 1. The deformations of the "opposite" half will be plotted to correspond to symmetric loading. This command will cause two plots to be generated.

   ```
   PLOT STATIC DEFORMATIONS 0, 3, 4,
      SET 1 ORIGIN 2 DENSITY 3 SHAPE,
      SET 1 SYMMETRY Z SHAPE,
      SET 2 ORIGIN 3 SHAPE,
      SET 2 SYMMETRY Z SHAPE
   ```

7. Transient deformations as defined in subcase 1 for time = 0.1 to time = 0.2, using SET 1 at ORIGIN 1. The undeformed SHAPE using PEN or DENSITY 1 with an * at each grid point location will be drawn as an underlay for the resultant deformation vectors using PEN or DENSITY 2 with an < typed at the end of each vector drawn. In addition, a plotted value of 2.0 will be used for the single maximum deformation occurring on any of the plots produced. All other deformations on all other plots will be scaled relative to this single maximum deformation. This command will cause a plot to be generated for each output time step which lies between 0.1 and 0.2.

   ```
   PLOT TRANSIENT DEFORMATION, TIME 0.1, 0.2,
      MAXIMUM DEFORMATION 2.0, SET 1, ORIGIN 1, PEN 2, SYMBOLS 2,
      VECTOR R
   ```

8. Contour plot of x-component of normal stress for elements in SET 2 in basic coordinate system at a distance Z1 from neutral plane with 10 contour lines, an outline of elements in SET 2, and using ORIGIN 4.

   ```
   CONTOUR XNORMAL
   PLOT CONTOUR, SET 2, ORIGIN 4, OUTLINE
   ```

9. Contour plot of magnitude of displacements at grid points associated with elements in SET 5 with 5 contours having values of 2., 4., 6., 8., 10., and an outline of the elements in SET 5 using ORIGIN 4.

   ```
   CONTOUR MAGNIT, LIST 2., 4., 6., 8., 10.
   PLOT CONTOUR, SET 5, OUTLINE
   ```

10. Plot the imaginary part of the complex eigenvector in SET 1.

    ```
    PLOT CMODAL DEFORMATION PHASE LAG 90.  SET 1 VECTOR R
    ```

PLOTTER Plot File Format

Selects format of plot file for interpetation by plotter post-processor.

Format:

PLOTTER { NAST / *SC* }

Example:

PLOTTER NAST

Describers	Meaning
NAST	Specifies format suitable for Postscript plotters. (Default)
SC	Specifies Stromberg-Carlson microfilm plotter format.

Remark:

1. If no PLOTTER command is specified then PLOTTER NAST is the default.

PROJECTION Separation Between Projection Plane and Observer

Defines the separation along the r-axis and between the observer and the projection plane if not already specified on the VANTAGE POINT command. Used by stereoscopic projections only.

Format:

PROJECTION PLANE SEPARATION do

Example:

PROJ PLAN SEPA 1.5

Describer	Meaning
do	Separation of the observer and the projection plane on the r-axis in model units. The VANTAGE POINT command may also specify the separation. (Real, Default=2.0)

Remarks:

1. The PROJECTION PLANE SEPARATION command is not recommended. The FIND command is recommended because it automatically calculates the optimum separation.
2. A theoretical description of projection plane separation is contained in "**Plotting**" on page 527 of the *MSC.Nastran Reference Guide*.

PTITLE Plot Frame Title

Defines a character string that will appear at the top of the plot frame on the line below the sequence number.

Format:

PTITLE ptitle

Example:

PTITLE RIGHT WING -- LOAD CASE 3

Describer	Meaning
ptitle	Any character string. (Character, Default=blank)

Remarks:

1. PTITLE may not be continued to the next command line.
2. Up to four lines of title information will be printed in the lower left-hand corner of each plot. The text for the top three lines is taken from the TITLE, SUBTITLE, and LABEL commands in the Case Control Section. (See the "**Case Control Commands**" on page 181 for a description of the TITLE, SUBTITLE, and LABEL commands). The text for the bottom line may be of two forms depending on the type plot requested. One form contains the word UNDEFORMED SHAPE. The other form contains the type of plot (statics, modal, etc.) subcase number, load set or mode number, frequency or eigenvalue or time, and (for complex quantities) the phase lag or magnitude. The sequence number for each plot is printed in the upper corners of each frame. The sequence number is determined by the relative position of each PLOT execution command in the plot package. The information on the PTITLE command will be printed on the line below the sequence number. The date and (for deformed plots) the maximum deformation are also printed at the top of each frame.

SCALE Scale Factor

Defines reduction, as a scale factor, of model's dimensions so that model fits on a plot frame.

Format:

SCALE a [b]

Example:

SCALE 0.5

Describers	Meaning
a	Scale factor. (Default=1.0)
b	Ratio of model size / real object size for stereoscopic projection only.

Remarks:

1. The SCALE command is not recommended. The FIND command is recommended because it automatically calculates the optimum scale factor.
2. For orthographic or perspective projections, a is the ratio of the plotted object in inches to the real object in the units of model; i.e., one inch of paper equals one unit of model.
3. For stereoscopic projections, the stereoscopic effect is enhanced by first reducing the real object to a smaller model according to b, and then applying a. The ratio of plotted / real object is then the product of a and b.
4. If the NASTRAN general purpose plotter is used in combination with the PLOTPS postprocessing routine, a scale factor may computed as follows:

$$a = p \cdot \frac{20}{7} \cdot K$$

where

p = ratio of plot size to object size. For instance, if your model is 100 inches long and your plot size is 7 inches:

$$p = \frac{7}{100} = .007$$

$\frac{20}{7}$ = ratio of default PAPER SIZE to default PLOTPS frame size.

K = SCALE value on PLOTPS command (Default = 1.0). (See "**Using the Utility Programs**" on page 209 of the *MD Nastran 2006 Installation and Operations Guide*

SEPLOT Superelement Plot Delimiter

Assigns the subsequent PLOT or XYPLOT commands to one or more superelements.

Format:

SEPLOT seid1 [seid2 ...]

Examples:

SEPLOT 5

SEPLOT 0 3 7 200

Describer	Meaning
seidi	Superelement identification number. (Integer≥0)

Remarks:

1. See also related command SEUPPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEPLOT (or SEUPPLOT) commands will apply in all SEPLOT (or SEUPPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEPLOT command with each PLOT. For the special case where the PLOTs or XYPLOTs refer to the same superelements and use the same FIND, a single SEPLOT followed by a single FIND may be placed above all commands.

SET Set Definition Under OUTPUT(PLOT)

Defines a set of elements or grid point numbers to be plotted.

Remark:

1. The SET command specifies sets of elements or grid points, corresponding to portions of the model, which may be referenced by PLOT and FIND commands. The SET command is required. Each set of elements defines by implication a set of grid points connected by those elements. The set may be modified by deleting some of its grid points. The elements are used for creating the plot itself and element labeling while the grid points are used for labeling, symbol printing, and drawing deformation vectors.

Element Type	Name on SET Command
CAXIF2	AXIF2
CAXIF3	AXIF3
CAXIF4	AXIF4
CBAR	BAR
CBEAM	BEAM
CBEND	BEND
CONEAX	CONE
CONROD	CONROD
CDUMi	DUMi
CFLUID2	FLUID2
CFLUID3	FLUID3
CFLUID4	FLUID4
CHBDYG	HBDY
CHBDYP	HBDY
CHEXA	HEXA
CPENTA	PENTA
PLOTEL	PLOTEL

Element Type	Name on SET Command
CQUAD	QUAD
CQUAD4	QUAD4
CQUAD8	QUAD8
CQUADR	QUADR
CQUADX	QUADX
CROD	ROD
CSHEAR	SHEAR
CSLOT3	SLOT3
CSLOT4	SLOT4
CTETRA	TETRA
CTRIAX6	TRIAX6
CTRIA3	TRIA3
CTRIA6	TRIA6
CTRIAR	TRIAR
CTRIAX	TRIAX
CTUBE	TUBE
CVISC	VISC

SEUPPLOT Superelement Plot Delimiter

Assigns the subsequent PLOT or XYPLOT commands to a superelement and all of its upstream superelements.

Format:

SEUPPLOT seid

Example:

SEUPPLOT 7

Describer	Meaning
seid	Superelement identification number. (Integer ≥ 0)

Remarks:

1. See also related command SEPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEUPPLOT (or SEPLOT) commands will apply in all SEUPPLOT (or SEPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEUPPLOT command with each PLOT. For the special case where the PLOTs or XYPLOTs refer to the same superelements and use the same FIND, a single SEUPPLOT followed by a single FIND may be placed above all the commands.

STEREOSCOPIC Selects Stereoscopic Projection

See the description of the “**ORTHOGRAPHIC, etc.**” on page 554.

VANTAGE POINT Location of the Observer

Defines the location of the observer with respect to the model in the (r, s, t) coordinate system defined on the AXES command for perspective and stereoscopic projections only.

Format:

VANTAGE POINT ro so to do sor

Example:

VANT 100.

Describers	Meaning
ro	Location of the observer on the r-axis in model units. (Real)
so	Location of the observer and left eye of the observer on the s-axis, in model units, for perspective and stereoscopic projections, respectively. (Real)
to	Location of the observer on the t-axis in model units. (Real)
do	Separation of the observer and the projection plane on the r-axis in model units. The PROJECTION PLANE SEPARATION command may also specify the separation. (Real)
sor	Location of the of the observer's right eye for stereoscopic projections in model units. (Real)

Remarks:

1. VANTAGE POINT or the FIND command must be specified if the PERSPECTIVE or STEREOSCOPIC command is also specified.
2. The VANTAGE POINT command is not recommended. The FIND command is recommended because it automatically calculates the optimum vantage point.
3. A theoretical description of the vantage point is contained in "**Plotting**" on page 527 of the *MSC.Nastran Reference Guide*.

VIEW Angular Relationship of Observer's Coordinate System

Defines the angular relationship between observer's coordinate system (r, s, and t axes specified on the AXES command) and basic coordinate system.

Format:

VIEW gamma beta alpha

Describers	Meaning
gamma	Angle of rotation, in degrees, of t axis specified on AXES command. (Default=34.27)
beta	Angle of rotation, in degrees, of s axis specified on AXES command. (Default=23.17 if ORTHOGRAPHIC or STEREOSCOPIC command is specified and 0.0 if PERSPECTIVE command is specified)
alpha	Angle of rotation, in degrees, of r axis specified on AXES command. (Default=0.0)

Example 1:

View the model from the r-axis.

VIEW 0. 0. 0.

Example 2:

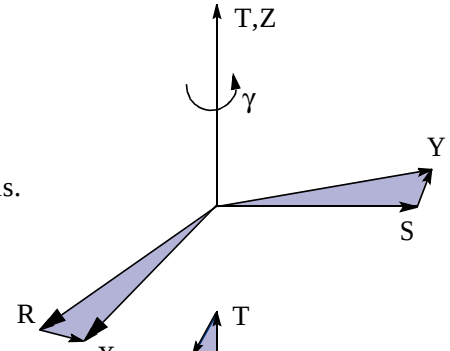
View the model midway between the r- and s-axes.

VIEW 45. 0. 0.

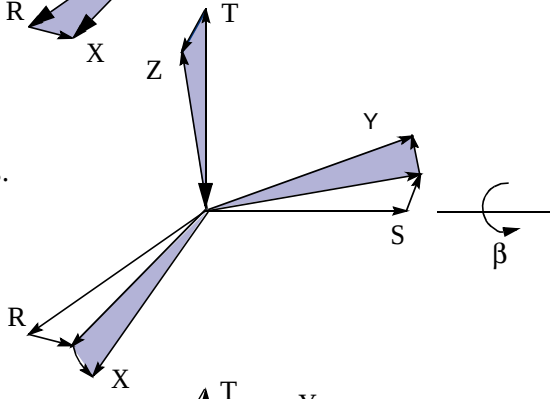
Remarks:

1. If no VIEW command is specifed then VIEW 34.27 23.17 0.0 is assumed for orthographic and steroscopic projections; and VIEW 34.27 0.0 0.0 for perspective projections. The default values produce a plot in which unit vectors on the axes of the basic coordinate system have equal lengths.
2. The angles are rotated in sequence: gamma rotates the t-axes, followed by beta which rotates the s-axes, followed by alpha which rotates the r-axes.

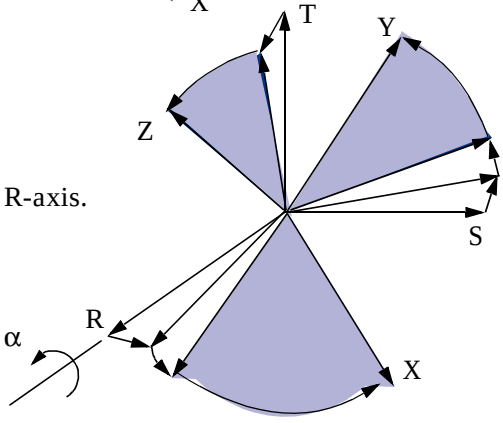
(a) g-rotation about T-axis.



(b) b-rotation about S-axis.



(c) a-rotation about R-axis.



3. The VIEW command specifies the position of the model with respect to the s-t plane. gamma and beta represent the angles of turn and tilt. alpha is normally not used since it does not affect the orientation of the s-t plane, but only it's orientation on the plot frame.

4.5 X-Y PLOT Commands

The X-Y output request packet of the Case Control Section includes all commands between either OUTPUT(XYPLOT) or OUTPUT(XYOUT) and either BEGIN BULK or or OUTPUT(PLOT). The remainder of this section describes the X-Y output commands.

A single set of plotted X-Y pairs is known as a curve. Curves are the entities that the user requests to be plotted. The surface (paper, microfilm frame, etc.) on which one or more curves is plotted is known as a fme. Curves may be plotted on a whole frame, an upper-half frame, or a lower-half frame. Grid lines, tic marks, axes, axis labeling and other graphic control options may be chosen by the user. The program will select defaults for parameters not selected by the user.

Only two commands are required for an X-Y output request.

1. Only one of OUTPUT(XYPLOT) or OUTPUT(XYOUT) at the beginning of the X-Y output command packet.
2. At least one of the commands XYPLOT, XYPEAK, XYPRINT, XYPUNCH, XYPAPLOT

The commands OUTPUT(XYPLOT) and OUTPUT(XYOUT) are equivalent. If the X-Y output is to be printed and / or punched, a PLOTTER command is not required.

If only the required commands are used, the graphic control options will all assume default values. Curves using all default parameters have the following general characteristics.

1. Tic marks are drawn on all edges of the frame. Five spaces are provided on each edge of the frame.
2. All tic marks are labeled with their values.
3. Linear scales are used.
4. Scales are selected such that all points fall within the frame.
5. The plotter points are connected with straight lines.
6. The plotted points are not identified with symbols.

The above characteristics may be modified by inserting any of the parameter definition commands in the next section, ahead of the XY____ command(s). The use of a parameter definition command sets the value of that parameter for all following command operation commands unless the CLEAR command is inserted. If grid lines are requested, they will be drawn at the locations of all tic marks that result from

defaults or user request. The locations of tic marks (or grid lines) for logarithmic scales cannot be selected by the user. Values for logarithmic spacing are selected by the program. The values for the number of tic marks (or grid lines) per cycle depend on the number of logarithmic cycles required for the range of the plotted values.

The definition and rules for the X-Y output commands follow. The form of X-Y output commands differ in many instances from that of similar commands used in the OUTPUT(PLOT) section.

X-Y Output Command Summary

Commands Applied To All Curves

PLOTTER	Selects format of plot file for interpetation by plotter post-processor.
CAMERA	Selects plotter media.
PENSIZE	Selects pen number.
DENSITY	Selects the line density for microfilm plotters only.
XPAPER	Defines the size of the paper in x-direction.
YPAPER	Defines the size of the paper in y-direction.
XMIN	Specifies the minimum value on the x-axis.
XMAX	Specifies the maximum value on the x-axis.
XLOG	Selects logarithmic or linear x-axis.
YAXIS	Controls the plotting of the y-axis on all curves.
XINTERCEPT	Specifies the location of the x-axis on the y-axis.
UPPER TICS	Specifies how to draw tic marks on upper edge.
LOWER TICS	Specifies how to draw tic marks on lower edge.
CURVELINE	Selects lines and / or symbols to be drawn through the x-y points.
XDIVISIONS	Specifies spacing of tic marks on the x-axis for all curves.
XVALUE SKIP	Specifies how often to print the x-values alongside the x-axis tic marks.
CLEAR	Resets X-Y Plot commands to their default value.
XTITLE	Defines a character string that will appear along the x-axis.
TCURVE	Defines a character string that will appear at the top of the plot frame.

Commands Applied To All Curves

LONG	Controls amount of curve's summary printout.
CSCALE	Defines scale factor for characters in the plot frame.

Commands Applied to Whole Frame Curves Only

YMIN	Specifies the minimum value on the y-axis.
YMAX	Specifies the maximum value on the y-axis.
XAXIS	Controls the plotting of the x-axis.
YINTERCEPT	Specifies the location of the y-axis on the x-axis.
YLOG	Selects logarithmic or linear y-axis.
LEFT TICS	Specifies how to draw tic marks on left edge.
RIGHT TICS	Specifies how to draw tic marks on right edge of the frame.
ALLEDGE TICS	Specifies how to draw tic marks on all edges of the frame.
YDIVISIONS	Specifies spacing of tic marks on the y-axis.
YVALUE PRINT	Specifies how often to print the y-values alongside the y-axis tic marks applies.
XGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at at the x-axis tic marks.
YGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at at the y-axis tic marks.
YTITLE	Defines a character string that will appear along the y-axis

Commands Applied to Upper Half Frame Curves Only

YTMIN	Specifies the minimum value on the y-axis.
YTMAX	Specifies the maximum value on the y-axis.
YTAXIS	Controls the plotting of the y-axis.
YTINTERCEPT	Specifies the location of the y-axis on the x-axis.
YTLOG	Selects logarithmic or linear y-axis.
TLEFT TICS	Specifies how to draw tic marks on the left edge.
TRIGHT TICS	Specifies how to draw tic marks on all edges.
TALL EDGE TIC	Specifies how to draw tic marks on all edges.

Commands Applied to Upper Half Frame Curves Only

YTDIVISIONS	Specifies spacing of tic marks on the y-axis.
YTVALUE PRINT	Specifies how often to print the y-values alongside the y-axis tic marks.
XTGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YTGRID LINES	Controls the drawing of the grid lines parallel to the x-axisat the y-axis tic marks.
YTTITLE	Defines a character string that will appear along the y-axis.

Commands Applied to Lower Half Frame Curves Only

YBMIN	Specifies the minimum value on the y-axis.
YBMAX	Specifies the maximum value on the y-axis.
XBAXIS	Controls the plotting of the x-axis.
YBINTERCEPT	Specifies the location of the y-axis on the x-axis.
YBLOG	Selects logarithmic or linear y-axis.
BLEFT TICS	Specifies how to draw tic marks on left edge.
BRIGHT TICS	Specifies how to draw tic marks on right edge.
BALL EDGE TIC	Specifies how to draw tic marks on all edges.
YBDIVISIONS	Specifies spacing of tic marks on the y-axis.
YBVALUE PRINT	Specifies how often to print the y-values alongside the y-axis tic marks.
XBGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YBGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.
YBTITLE	fines a character string that will appear along the y-axis.

X-Y Plot Generation Commands

XYPAPLOT	Generate X-Y plots for a printer
XYPEAK	Print only the summary for all curves
XYPLOT	Generate X-Y plots for a plotter

X-Y Plot Generation Commands

XYPRINT	Generate table of X-Y pairs for a printer
XYPUNCH	Generate table of X-Y pairs for the PUNCH file

ALLEDGE TICS — Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the frame.

Format:

ALLEDGE TICS tic

Example:

ALLEDGE -1

Describer	Meaning	
tic	Specifies how to draw tic marks. (Integer, Default=0)	
	-1	Draw tic marks only.
	0	Do not draw tic marks or associated values. (Default)
	1	Draw tic marks and associated values.

Remarks:

1. ALLEDGE TICS applies to whole frame curves only.
2. To determine if on any given edge (a) tic marks will be drawn without values, (b) no tic marks or values will be drawn, or (c) tic marks with values will be drawn, the following sum must be computed by the user. Add the tic values of the edge in question to its associated ALLEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS tic values. If the resulting value is less than zero, tic marks will be drawn without values. If the resulting value is zero, no tic marks or values will be drawn. If the resulting value is greater than zero, tic marks with values will be drawn. The user should be careful in his use of the ALLEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS commands. For example, the use of only the ALLEDGE TICS = -1 command will result in no tic marks or values being drawn since the default values for individual edges is +1. Tic values input may only be -1, 0, or 1.

BALL EDGE TICS — Controls Drawing of Tic Marks on Lower Half

Specifies how to draw tic marks on lower half of frame.

Format:

BALL EDGE TICS tic

Example:

BALL EDGE TICS -1

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default=0)
	-1 Draw tic marks only.
	0 Do not draw tic marks or associated values. (Default)
	1 Draw tic marks and associated values.

Remarks:

1. BALL EDGE TICS applies to lower frame curves only.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.

BLEFT TICS — Controls Drawing of Tic Marks on Left Edge

Specifies how to draw tic marks on left edge of lower half of frame.

Format:

BLEFT TICS tic

Example:

BLEFT TICS -1

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default=1)
	-1 Draw tic marks only.
	0 Do not draw tic marks or associated values.
	1 Draw tic marks and associated values. (Default)

Remarks:

1. BLEFT TICS applies to lower frame curves only.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.
3. See related command "**BRIGHT TICS**" on page 588.

BRIGHT TICS — Controls Drawaing of Tic Marks on Right Edge

Specifies how to draw tic marks on right edge of lower half of frame.

Format:

BRIGHT TICS tic

Example:

BRIGHT TICS -1

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default=1)
	-1 Draw tic marks only.
	0 Do not draw tic marks or associated values.
	1 Draw tic marks and associated values. (Default)

Remarks:

1. BRIGHT TICS applies to lower frame curves only.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.

CAMERA Plotter Media Selection

Selects plotter media.

Format:

CAMERA ctype

Example:

CAMERA 1

Describer	Meaning
ctype	Camera type. (Integer 1, 2, or 3; Default=2)
	1 Film
	2 Paper (Default)
	3 Both

Remark:

1. If the CAMERA command is not specified then CAMERA 2 is assumed.

CLEAR Resets X-Y Plot Commands

Resets X-Y Plot commands to their default value.

Format

CLEAR

Remark:

1. All commands except XTITLE, YTITLE, YTTITLE, YBTITLE, and TCURVE will revert to their default values.

CSCALE Character Scale Factor

Defines scale factor for characters in the plot frame. See the command, “**CSCALE**” on page 591 in the OUTPUT(PLOT) Section.

CURVELINESYMBOL Curve, Line and Symbol Selection

Selects lines and / or symbols to be drawn through the x-y points.

Format:

CURVELINESYMBOL symtype

Example:

CURV 4

Describer	Meaning
symtype	Specifies the symbol drawn at the x-y points. If symtype is 0 then only lines will be drawn through the points with no symbol. If symtype is less than zero then only the symbol and not the lines will be drawn. If symtype is greater than zero then both the symbol and the lines will be drawn. (-9≤Integer≤9, Default=0)

symtype	Symbol
0	none
1	X
2	*
3	+
4	-
5	.
6	×
7	[ ]
8	< >
9	/ \

Remark:

1. If more than one curve is plotted per frame then the symbol number is incremented by 1 for each curve.

DENSITY Microfilm Plotter Line Density

Selects the line density for microfilm plotters only.

Format

DENSITY d

Example

DENS 3

Describer	Meaning
d	Specifies line density scale factor for microfilm plotters. A line density of d is d times heavier than a line density of 1. (Integer≥0, Default=1)

LEFT TICS Controls Drawing of Tic Marks on Left Edge

Specifies how to draw tic marks on left edge of whole frame curves.

Format:

LEFT TICS tic

Example:

LEFT -1

Describer	Meaning
tic	Specifies how to draw tic marks. (Integer, Default=1)
	-1 Draw tic marks only.
	0 Do not draw tic marks or associated values.
	1 Draw tic marks and associated values. (Default)

Remarks:

1. LEFT TICS applies to whole frame curves only.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.
3. See related command "**RIGHT TICS**" on page 599.

LONG Summary Print Control

Controls amount of curve's summary printout.

Format:

LONG{ YES / NO }

Describers	Meaning
YES	One page for each curve's summary. (Default)
NO	Condensed curve summary.

Remark:

1. If LONG is not specified then LONG=NO is assumed.

LOWER TICS Controls Drawing of Tic Marks on Lower Edge

Specifies how to draw tic marks on lower edge.

Format:

LOWER TICS tic

Example:

LOWER -1

Describers	Meaning
tic	Specifies how to draw tic marks. (Integer, Default=1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

Remarks:

1. LOWER TICS applies to all curves.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.

PENSIZE Pen Selection

Selects pen number.

Format:

PENSIZE p

Example:

PENS 3

Describer	Meaning
p	Specifies pen number that is used to generate the plot. (Integer>0, Default=1)

PLOTTER X-Y Plot File Format

See the command, “**PLOTTER**” on page 598 in the OUTPUT(PLOT) Section.

RIGHT TICS — Controls Drawing of Tic Marks on Right Edge

Specifies how to draw tic marks on right edge of the frame.

Format:

RIGHT TICS tic

Example:

RIGHT -1

Describers	Meaning
tic	Specifies how to draw tic marks. (Integer, Default=1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

Remarks:

1. RIGHT TICS applies to whole frame curves only.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.
3. See related command "**LEFT TICS**" on page 594.

SEPLOT Superelement Plot Delimiter

Assigns the subsequent PLOT or XYPLOT commands to one or more superelements.

Format:

SEPLOT seid1 [seid2 ...]

Examples:

SEPLOT 5

SEPLOT 0 3 7 200

Describer	Meaning
seidi	Superelement identification number. (Integer≥0)

Remarks:

1. See also related command SEUPPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEPLOT (or SEUPPLOT) commands will apply in all SEPLOT (or SEUPPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEPLOT command with each PLOT. For the special case where the PLOTs or XYPLOTs refer to the same superelements and use the same FIND, a single SEPLOT followed by a single FIND may be placed above all commands.

SEUPPLOT Superelement Plot Delimiter

Assigns the subsequent PLOT or XYPLOT commands to a superelement and all of its upstream superelements.

Format:

SEUPPLOT seid

Example:

SEUPPLOT 7

Describer	Meaning
seid	Superelement identification number. (Integer ≥ 0)

Remarks:

1. See also related command SEPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEUPPLOT (or SEPLOT) commands will apply in all SEUPPLOT (or SEPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEUPPLOT command with each PLOT. For the special case where the PLOTs or XYPLOTs refer to the same superelements and use the same FIND, a single SEUPPLOT followed by a single FIND may be placed above all the commands.

TALL EDGE TICS Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the upper half of the frame.

Format:

TALL EDGE TICS tic

Example:

TALL -1

Describers	Meaning
tic	Specifies how to draw tic marks. (Integer, Default = 0)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values. (Default)
1	Draw tic marks and associated values.

Remarks:

1. TALL EDGE TICS applies to upper half frame curves only.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.

TCURVE Curve Title

Defines a character string that will appear at the top of the plot frame.

Format:

TCURVE ctitle

Example:

TCUR RIGHT WING -- LOAD CASE 3

Describer	Meaning
ctitle	Any character string. (Character, Default=blank)

Remark:

1. TCURVE may not be continued to the next command line.

TLEFT TICS Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on the left edge of the upper half of the frame.

Format:

TLEFT TICS tic

Example:

TLEFT -1

Describers	Meaning
tic	Specifies how to draw tic marks. (Integer, Default=1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

Remarks:

1. TLEFT TICS applies to upper half frame curves only.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.
3. See related command "**TRIGHT TICS**" on page 605.

TRIGHT TICS Controls Drawing of Tic Marks on the Right Edge

Specifies how to draw tic marks on all edges of the upper half of the frame.

Format:

TRIGHT TICS tic

Example:

TRIGHT -1

Describers	Meaning
tic	Specifies how to draw tic marks. (Integer, Default=1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

Remarks:

1. TRIGHT TICS applies to upper half frame curves only.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.
3. See related command "**TLEFT TICS**" on page 604.

UPPER TICS Controls Drawing Of Tic Marks On Upper Edge

Specifies how to draw tic marks on upper edge.

Format:

UPPER TICS tic

Example:

UPPER -1

Describers	Meaning
tic	Specifies how to draw tic marks. (Integer, Default=1)
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values. (Default)

Remarks:

1. UPPER TICS applies to all curves.
2. See Remark 2 under "**ALLEDGE TICS**" on page 585.
3. See related command "**LOWER TICS**" on page 596.

XAXIS X-Axis Plot Control

Controls the plotting of the x-axis on whole frame curves only.

Format:

XAXIS { YES / NO }

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis. (Default)

Remarks:

1. XAXIS applies to whole frame curves only.
2. See related command "**YAXIS**" on page 631.

XBAXIS X-Axis Plot Control

Controls the plotting of the x-axis on lower half frame curves only.

Format:

XBAXIS { YES / NO }

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis. (Default)

Remark:

1. XBAXIS applies to lower half frame curves only.

XBGRID LINES Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on lower half frame curves only.

Format:

XBGRID LINES { YES / NO }

Describers	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines. (Default)

Remarks:

1. XBGRID applies to lower half frame curves only.
2. See related command "**YBGRID LINES**" on page 634.

XDIVISIONS Tic Spacing on Y-Axis

Specifies spacing of tic marks on the x-axis for all curves.

Format:

XDIVISIONS xd

Example:

XDIV 10

Describer	Meaning
xd	Number of spaces between tic marks on x-axis. (Integer>0, Default=5)

Remarks:

1. XDIVISIONS applies to all curves and to the commands: UPPER TICS, LOWER TICS, and YINTERCEPT.
2. XDIVISIONS is ignored for logarithmic x-axes.

XGRID LINES Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on whole frame curves only.

Format:

XGRID LINES { YES / NO }

Describers	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines. (Default)

Remarks:

1. XGRID applies to whole frame curves only.
2. See related command "**YGRID LINES**" on page 642.

XINTERCEPT Location of X-Axis on Y-Axis

Specifies the location of the x-axis on the y-axis.

Format:

XINTERCEPT xi

Example:

XINT 50.

Describer	Meaning
xi	Location of x-axis on the y-axis. (Real, Default=0.0)

XLOG Logarithmic or Linear X-Axis

Selects logarithmic or linear x-axis.

Format:

XLOG { YES / NO }

Describers	Meaning
YES	Plot a logarithmic x-axis.
NO	Plot a linear x-axis. (Default)

Remarks:

1. XLOG applies to all curves.
2. The default value for tic division interval depends on the number of log cycles. The default values for tic divisions are given below but will range over whole cycles:

Number of Cycles	Intermediate Values
1, 2	2., 3., 4., 5., 6., 7., 8., 9.
3	2., 3., 5., 7., 9.,
4	2., 4., 6., 8.,
5	2., 5., 8.
6, 7	3., 6.
8, 9, 10	3.

XMAX Maximum X-Axis Value

Specifies the maximum value on the x-axis.

Format:

XMAX xmax

Example:

XMAX 100.

Describer	Meaning
xmax	Maximum value on the x-axis. (Real)

Remarks:

1. If XMAX is not specified then the maximum value is set to the highest value of x.
2. See related commands "**XMIN**" on page 615, "**YMIN**" on page 645, and "**YMAX**" on page 644.

XMIN Minimum X-Axis Value

Specifies the minimum value on the x-axis.

Format:

XMIN xmin

Example:

XMIN 100.

Describer	Meaning
xmin	Minimum value on the x-axis. (Real)

Remarks:

1. XMIN applies to all curves.
2. If XMIN is not specified then the minimum value is set to the lowest value of x.
3. See related commands "**XMAX**" on page 614, "**YMIN**" on page 645, and "**YMAX**" on page 644.

XPAPER Paper Size in X-Direction

Defines the size of the paper in x-direction.

Format:

XPAPER xsize

Example:

XPAP 10.

Describer	Meaning
xsize	Size of paper in x-direction and in inches. (Real, Default=20.0)

Remarks:

1. The default paper size is 20 by 20 inches.
2. See related command "**YPAPER**" on page 646.

XTAXIS X-Axis Plot Control

XYAXIS applies to upper half frame curves only.

Format:

XTAXIS { YES / NO }

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis. (Default)

XTGRID LINES

Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on upper half frame curves only.

Format:

XTGRID LINE { YES / NO }

Describers	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines. (Default)

Remark:

1. XTGRID applies to upper half frame curves only.

XTITLE X-Axis Title

Defines a character string that will appear along the x-axis.

Format:

XTITLE xtit

Example:

XTIT RIGHT WING CASE 3 - TIME

Describer	Meaning
xtit	Any character string. (Character, Default=blank)

Remarks:

1. XTITLE may not be continued to the next command line.
2. XTITLE applies to all curves.

XYPAPLOT Generate X-Y Plots for a Printer

Generate X-Y plots for a printer.

"XYPLOT" on page 622 for Format, Describers, and additional Remarks.

Remarks:

1. The x-axis moves vertically along the page and the y-axis moves horizontally along the page.
2. An asterisk (*) identifies the points associated with the first curve of a frame, then for successive curves on a frame the points are designated by symbols O, A, B, C, D, E, F, G and H.

XYPEAK Print Curve Summary

Print only the summary for all curves. The summary output is titled:

"X Y - O U T P U T S U M M A R Y"

and is also printed under XYPLOT, XYPUNCH, XYPRINT, and XYPAPLOT. This output contains the maximum and minimum values of y for the range of x.

"XYPLOT" on page 622 for Format, Describers, and additional Remarks.

XYPLOT Generate X-Y Plots for a Plotter

Generate X-Y Plots for a Plotter.

Format:

```
XYPLOT      yvtype  ptype  [ i1, i2, i3,... ] /
            id11 (itemu11 [, iteml11] ) , id12 (itemu12 [, iteml12] ) , ... /
            id21 (itemu21 [, iteml21] ) , id22 (itemu22 [, iteml22] ) , ... / ....
```

Example(s):

BEGIN BULK or OUTPUT(PLOT) command is shown as a reminder to the user to place his X-Y output request packet properly in this Case Control Section, i.e., at the end of the Case Control Section or just ahead of any structure plot requests.

Example 1:

```
OUTPUT (XYPLOT)
CSCALE = 1.8
XYPLOT SDISP / 16(T1)
BEGIN BULK
```

This sequence causes a single whole frame to be plotted for the T1 displacement component of solution set point 16 using the default parameter values. If 16(T1) is not in the solution set, a warning message will be printed and no plot will be made. The plot will be generated for the NASTRAN plotter on file PLT which must be available.

Example 2:

```
OUTPUT (XYOUT)
CSCALE = 1.8
XYPLOT, XYPRINT VELO RESPONSE 1,5  / 3(R1,), 5(,R1)
OUTPUT (PLOT)
```

This sequence causes two frame plots (each consisting of an upper half frame and a lower half frame) to be plotted, one for subcase 1 and one for subcase 5, using the default parameter values. The velocity of the first rotational component of grid point 3 will be plotted on the upper half frame and that of grid point 5 will be plotted on the lower half frame. Tabular printer output will also be generated for both curves.

Example 3:

```
OUTPUT (XYPLOT)
CSCALE = 1.8
YDIVISIONS = 20
XDIVISIONS = 10
SGRID LINES = YES
YGRID LINES = YES
XYPLOT DISP 2,5 / 10(T1),10(T3)
BEGIN BULK
```

This sequence causes two whole frame plots to be generated, one for subcase 2 and one for subcase 5. Each plot contains the T1 and T3 displacement component for grid point 10. The default parameters will be modified to include grid lines in both the x-direction and y-direction with 10 spaces in the x-direction and 20 spaces in the y-direction. The plot will be generated for the NASTRAN plotter on file PLT.

Example 4


```
OUTPUT (XYPLOT)
CSCALE = 1.8
XAXIS = YES
YAXIS = YES
XPAPER = 40.
YPAPER = 20.
XYPLOT STRESS 3 / 15(2) / 21(7)
OUTPUT (PLOT)
```

This sequence causes two whole frame plots to be generated using the results from subcase 3. The first plot is the response of the axial stress for ROD element number 15. The second plot is the response of the major principal stress at Z1 for TRIA3 element number 21. The default parameters will be modified to include the x-axis and y-axis drawn through the origin. Each plot will be initially scaled to fit on 40 x 20-inch paper. The plots will be generated for the NASPLT postprocessor and NASTRAN file PLT2 which must be defined. NASPLT will redefine the plot to 14 x 7-inch paper (with default options).

Example 5

```
OUTPUT (XYPLOT)
CURVELINESYMBOL = -1
XYPLOT XYPAPLOT VG / 1(G,F) 2(G,F) 3(G,F) 4(G,F)
OUTPUT (PLOT)
```

This sequence is an example of plotting in a Flutter Analysis for which a split frame plot will be made; the upper half is V-g and the lower half is V-f. Data from the first four loops will be plotted. Distinct symbols are used for data from each loop and no lines are drawn between points (since the flutter analyst must sometimes exercise judgement about which points should be connected). The plots will also be printed in the normal output. These plots will not have all the features of the external plots, but can be very useful in getting a quick picture of the curves.

Example 5

```
XTITLE=EXCITATION FREQUENCY FROM 2.5 TO 250 HERTZ SC 200
YTITLE=FLUID MODE PF AT FLUID POINT 204 FOR NATURAL MODE 2
XYPLOT,XYPEAK FMPF(2) MODE 200 / 204
```

Example 6

```
YTITLE=EXCITATION FREQUENCY FROM 2.5 TO 250 HERTZ
YTITLE=PSD MPF FOR FLUID GRID 204 FOR NATURAL MODE 2
XYPLOT,XYPEAK FMPF(2) PSDF  / 204
```

Describers	Meaning	
yvtype	Type of y-value to be plotted: (Character)	
	ACCE	Acceleration in the physical set
	BOUT	Slideline contact output
	DISP	Displacement in the physical set
	ELFORCE	Element force
	ENTHALPY	Enthalpy in the physical set
	FLUX	Element Heat flux
	FMPF (mode_id or frequency_id)	Fluid mode participation factors. See Remark 8.
	GMPF (mode_id or frequency_id, panel_name, panel_grid_id)	Panel grid mode participation factors. See Remark 8.
	HDOT	Rate of change of enthalpy in the physical set
	MPCF	Multiple-point force of constraint

Describers	Meaning	
	LMPF	Load mode participation factors. See Remark 8.
	NONLINEAR	Nonlinear applied load
	OLOAD	Applied Load
	PMPF (mode_id or frequency_id, panel_name)	Panel mode participation factors. See Remark 8.
	PRESSURE	Pressure of fluid-structure body
	SACCE	Acceleration in the solution set
	SDISP	Displacement in the solution set
	SMPF (mode_id or frequency_id)	Structural mode participation factors. See Remark 8.
	SPCF	Single-point force of constraint
	STEMP	Temperature in the solution set
	STRAIN	Element Strain
	STRESS	Element stress
	SVELO	Velocity in the solution set
	TEMP	Temperature in the physical set
	VECTOR	Displacement in the physical set
	VELO	Velocity in the physical set
	VG	Flutter analysis
ptype	Plot type defining the meaning of i1, i2, ... etc., idi, itemui and itemli. (Character, Default="RESPONSE")	
	AUTO	Autocorrelation function on whole frame curves only.
	FREQ	frequency - for given excitation frequency plot mode partcipation vs natural frequency - Oxx2E tables - point plot only. See Remark 8.
	MODE	mode - for given fluid mode plot mode participation vs excitation freq - Oxx2M tables.

Describers	Meaning	
	PSDF	Power spectral density function on whole frame curves only.
	RESPONSE	Time or frequency in SORT2 format or grid identification numbers in SORT1 format. (Default)
	SPECTRAL	Response spectrum on whole frame curves only.
i1, i2,...	Subcase identification numbers for ptype=RESPONSE. The list must be specified in ascending order. For ptype=SPECTRAL, the subcase refers to the RECNO in the DTI,SPSEL Bulk Data entry. The list is ignored for ptype=AUTO and PSDF. (Integer≥0, Default is all subcases)	
idij	Element, grid, scalar, or extra point identification number for y-value for frame i. For yvtype=VG, idij refers to the loop count of a flutter analysis. (Integer>0)	
itemuij,	Item code for y-value. itemuij is for for upper half or whole itemlij curves on frame i and itemlij is for lower half curves only on frame i. If itemlij is not specified then whole frame curves will be plotted with itemuij. itemlij is ignored for ptype="AUTO", "PSDF", and "SPECTRAL". (Character or Integer>0)	
	For elements the code represents a component of the element stress, strain, or force and are described in **Table 6-1** and **Table 6-2** of the Guide. For ptype="AUTO"and "PSDF", the complex stress or strain item codes need to be used. As the output quantities are real, you can use either the real or the imaginary item code. Both will give the same result.	
	For grid points and pty="RESPONSE", the code is one of the mnemonics T1, T2, T3, R1, R2, R3, T1RM, T2RM, T3RM, R1RM, R2RM, R3RM, T1IP, T2IP, T3IP, R1IP, R2IP, or R3IP, where Ti stands for the i-th translational component, Ri stands for the i-th rotational component, RM means real or magnitude, and IP means imaginary or phase. For scalar or extra points or heat transfer analysis, use T1, T1RM, or T1IP.	
	For grid points and ptype="AUTO" or "PSDF", the code is one of the mnemonics T1, T2, T3, R1, R2, R3. For scalar or extra points use T1.	
	For yvtype=VG, itemui and/or or itemli can be "F" for frequency or "G" for damping.	

Remarks:

1. Multiple XYPLOT, XYPUNCH, XYPRINT, XYPEAK and/or XYPAPLOT commands may be specified in the OUTPUT(XYPLOT) section.
2. Solution set requests are more efficient, because the time-consuming recovery of the dependent displacements can be avoided.
3. The item codes also appear in printed summaries as “CURVE ID” for grid points as well as element data.
4. The information after each slash (/) specifies the curve(s) that are to be plotted on the same frame. The describer idij identifies the grid point j or element j associated with the frame number i. All plot requests on one command are sorted on idij to improve the efficiency of the plotting process. Symbols are assigned in order by idij.
5. If any of the item codes, itemlij or itemuij, are not specified; e.g., (8,) or (,5), the corresponding half frame curve is not plotted. If both the comma (,) and itemlij not specified; e.g., (8), then whole frame curves will be plotted. Also, for any single frame, the specifications of “(itemuij,itemlij)” must be consistently half frame (upper and/or lower) or whole frame. For example on half frame curves, if iteml11 and the comma is not specified then either iteml12 or itemu12 must not be specified and on whole frame curves, the commas, iteml11, and iteml12 must not be specified. In other words, the curves on each plot frame must be all whole or half (upper and/or lower).
6. The XYPLOT command may be continued on the next line as long as “XYPLOT yvtype ptype [i1, i2, i3,...] /” is specified on the first line.
7. Specifying a nonexistent grid point may cause the program to exit in the XYTRAN module and missing plots to occur.
8. mode_id is used for natural frequency selection of Oxx2m participation versus excitation frequency output.
   frequency_id is used for excitation frequency selection of Oxx2E participation versus natural frequency output. frequency_id is an integer value; e.g., (2) would represent the second frequency calculated.

XYPRINT Generate Table of X-Y Pairs for a Printer

Generate tabular printer output of the X-Y pairs.

See “XYPLOT” on page 622 for Format, Describers, and additional Remarks.

XYPUNCH Generate Table of X-Y Pairs for the PUNCH File

Generate tabular punch output of the X-Y pairs. Same as XYPRINT except the output is written to the PUNCH file.

See “XYPLOT” on page 622 for Format, Describers, and additional Remarks.

XVALUE PRINT SKIP

Print Values on X-Axis Tic Marks

Specifies how often to print the x-values alongside the x-axis tic marks.

Format:

XVALUE PRINT SKIP xvps

Example:

XVAL 5

Describer	Meaning
xvps	Number of tic marks to be skipped between labeled tic marks with their corresponding values. (Integer≥0)

Remark:

1. XVALUE applies to all curves.

YAXIS Y-Axis Plot Control

Controls the plotting of the y-axis on all curves.

Format:

YAXIS { YES / NO }

Describers	Meaning
YES	Plot the y-axis.
NO	Do not plot the y-axis. (Default)

YBDIVISIONS Tic Spacing on Y-Axis

Specifies spacing of tic marks on the y-axis for lower half frame curves only.

Format:

YBDIVISIONS ybd

Example:

YBDI 10

Describer	Meaning
ybd	Number of spaces between tic marks on y-axis. (Integer>0, Default=5)

Remarks:

1. YBDIVISIONS applies to lower half frame curves only.
2. YBDIVISIONS is ignored for logarithmic y-axes.

YBINTERCEPT Location of Y-Axis on X-Axis

Specifies the location of the y-axis on the x-axis for lower half frame curves only.

Format:

YBINTERCEPT ybi

Example:

YBINT 50

Describer	Meaning
ybi	Location of y-axis on the x-axis. (Real, Default=0.0)

Remark:

1. YBINTERCEPT applies to lower half frame curves only.

YBGRID LINES

Plot Y-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on lower half frame curves only.

Format:

YBGRID LINES { YES / NO }

Describers	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines. (Default)

Remark:

1. YBGRID applies to lower half frame curves only.

YBLOG Logarithmic or Linear Y-Axis

Selects logarithmic or linear y-axis for lower half frame curves only.

Format:

YBLOG { YES / NO }

Describers	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis. (Default)

Remarks:

1. YBLOG applies to lower half frame curves only.
2. See Remark 2 under "**XLOG**" on page 613.

YBMAX Maximum Y-Axis Vaue

Specifies the maximum value on the y-axis for lower half frame curves only.

Format:

YBMAX ymax

Example:

YBMAX 100

Describer	Meaning
ymax	Maximum value on the y-axis. (Real)

Remarks:

1. YBMAX applies to lower half frame curves only.
2. If YBMAX is not specified then the maximum value is set to the highest value of y.
3. See related command "**YBMIN**" on page 637.

YBMIN Minimum Y-Axis Value

Specifies the minimum value on the y-axis for lower half frame curves only.

Format:

YBMIN ymin

Example:

YBMIN 100

Describer	Meaning
ymin	Minimum value on the y-axis. (Real)

Remarks:

1. YBMIN applies to lower half frame curves only.
2. If YBMIN is not specified then the minimum value is set to the lowest value of y.
3. See related command "**YBMAX**" on page 636.

YBTITLE Y-Axis Title

Defines a character string that will appear along the y-axis for lower half frame curves only.

Format:

YBTITLE ytit

Example:

YBTIT RIGHT WING LOADS - CASE 3

Describer	Meaning
ytit	Any character string. (Character, Default=blank)

Remarks:

1. YBTITLE may not be continued to the next command line.
2. YBTITLE applies to lower half frame curves only.

YBVALUE PRINT SKIP

Print Values on Y-Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on lower half frame curves only.

Format:

YBVALUE PRINT SKIP yvps

Example:

YBVAL 5

Describer	Meaning
yvps	Number of tic marks to be skipped between labeled tic marks with their cooresponding values. (Integer≥0)

Remark:

1. YBVALUE applies to lower half frame curves only.

YDIVISIONS Tic Spacing on Y-Axis

Specifies spacing of tic marks on the y-axis for whole frame curves only.

Format:

YDIVISIONS yd

Example:

YDIV 10

Describer	Meaning
yd	Number of spaces between tic marks on y-axis. (Integer>0, Default=5)

Remarks:

1. YDIVISIONS applies to whole frame curves only and to the commands: LEFT TICS, RIGHT TICS, and XINTERCEPT.
2. YDIVISIONS is ignored for logarithmic y-axes.

YINTERCEPT Location of Y-Axis on X-Axis

Specifies the location of the y-axis on the x-axis for whole frame curves only.

Format:

YINTERCEPT yi

Example:

YINT 50

Describer	Meaning
yi	Location of y-axis on the x-axis. (Real, Default=0.0)

Remark:

1. YINTERCEPT applies to lower half frame curves only.

YGRID LINES

Plot Y-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on whole frame curves only.

Format:

YGRID LINES { YES / NO }

Describers	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines. (Default)

Remark:

1. YGRID applies to whole frame curves only.

YLOG Logarithmic or Linear Y-Axis

Selects logarithmic or linear y-axis for whole frame curves only.

Format:

YLOG { YES / NO }

Describers	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis. (Default)

Remarks:

1. YLOG applies to whole frame curves only.
2. See Remark 2 under "**XLOG**" on page 613.

YMAX Maximum Y-Axis Value

Specifies the maximum value on the y-axis.

Format:

YMAX ymax

Example:

YMAX 100

Describer	Meaning
ymax	Maximum value on the y-axis. (Real)

Remarks:

1. If YMAX is not specified then the maximum value is set to the highest value of y.
2. See related command "**YMIN**" on page 645.

YMIN Minimum Y-Axis Value

Specifies the minimum value on the y-axis.

Format:

YMIN ymin

Example:

YMIN 100

Describer	Meaning
ymin	Minimum value on the y-axis. (Real)

Remarks:

1. YMIN applies to all curves.
2. If YMIN is not specified then the minimum value is set to the lowest value of y.
3. See related command "YMAX" on page 644.

YPAPER Paper Size in Y-Direction

Defines the size of the paper in y-direction.

Format:

YPAPER ysize

Example:

YPAP 10

Describer	Meaning
ysize	Size of paper in y-direction and in inches. (Real, Default=20.0)

Remarks:

1. The default paper size is 20 by 20 inches.
2. See related command "**XPAPER**" on page 616

YTAXIS Y-Axis Plot Control

Controls the plotting of the y-axis on upper half frame curves only.

Format:

YTAXIS { YES / NO }

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis. (Default)

Remark:

1. YTAXIS applies to upper half frame curves only.

YTDIVISIONS The Spacing on Y-Axis

Specifies spacing of tic marks on the y-axis for upper half frame curves only.

Format:

YTDIVISIONS ytd

Example:

YTDI 10

Describer	Meaning
ytd	Number of spaces between tic marks on y-axis. (Integer>0, Default=5)

Remarks:

1. YTDIVISIONS applies to upper half frame curves only.
2. YTDIVISIONS is ignored for logarithmic y-axes.

YTGRID LINES Plot Y-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on upper half frame curves only.

Format:

TYGRID LINES { YES / NO }

Describers	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines. (Default)

Remark:

1. YTGRID applies to upper half frame curves only.

YTINTERCEPT Location of Y-Axis on X-Axis

Specifies the location of the y-axis on the x-axis for upper half frame curves only.

Format:

YTINTERCEPT yti

Example:

YTINT 50

Describer	Meaning
yti	Location of y-axis on the x-axis. (Real, Default=0.0)

Remark:

1. YTINTERCEPT applies to upper half frame curves only.

YTITLE Y-Axis Title

Defines a character string that will appear along the y-axis for whole frame curves only.

Format:

YTITLE ytit

Example:

YTIT RIGHT WING LOADS - CASE 3

Describer	Meaning
ytit	Any character string. (Character, Default=blank)

Remarks:

1. YTITLE may not be continued to the next command line.
2. YTITLE applies to whole frame curves only.

YTLOG Logarithmic or Linear Y-Axis

Selects logarithmic or linear y-axis for upper half frame curves only.

Format:

YT(LOG){ YES / NO }

Describers	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis. (Default)

Remarks:

1. YTLOG applies to upper half frame curves only.
2. See Remark 2 under "**XLOG**" on page 613.

YTMAX Maximum Y-Axis Value

Specifies the maximum value on the y-axis for upper half frame curves only.

Format:

YTMAX ymax

Example:

YTMAX 100

Describer	Meaning
ymax	Maximum value on the y-axis. (Real)

Remarks:

1. YTMAX applies to upper half frame curves only.
2. If YTMAX is not specified then the maximum value is set to the highest value of y.
3. See related command "YTMIN" on page 654.

YTMIN Minimum Y-Axis Value

Specifies the minimum value on the y-axis for upper half frame curves only.

Format:

YTMIN ymin

Example:

YTMIN 100

Describer	Meaning
ymin	Minimum value on the y-axis. (Real)

Remarks:

1. YTMIN applies to upper half frame curves only.
2. If YTMIN is not specified then the minimum value is set to the lowest value of y.
3. See related command "**YTMAX**" on page 653.

YTTITLE Y-Axis Title

Defines a character string that will appear along the y-axis for upper half frame curves only.

Format:

YTTITLE ytit

Example:

YTTIT RIGHT WING LOADS - CASE 3

Describer	Meaning
ytit	Any character string. (Character, Default=blank)

Remarks:

1. YTTITLE may not be continued to the next command line.
2. YTTITLE applies to upper half frame curves only.

YTVALUE PRINT SKIP

Print Values on Y-Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on upper half frame curves only.

Format:

YTVALUE PRINT SKIP yvps

Example:

YTVAL 5

Describer	Meaning
yvps	Number of tic marks to be skipped between labeled tic marks with their cooresponding values. (Integer≥0)

Remark:

1. YTVALUE applies to upper half frame curves only.

YVALUE PRINT SKIP

Print Values on Y-Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on whole frame curves only.

Format:

YVALUE PRINT SKIP yvps

Example:

YVAL 5

Describer	Meaning
yvps	Number of tic marks to be skipped between labeled tic marks with their cooresponding values. (Integer≥0)

Remark:

1. YVALUE applies to whole frame curves only.

CHAPTER 5

Parameters

- Parameter Descriptions
- Parameter Applicability Tables

5.1 Parameter Descriptions

Parameters are used extensively in the solution sequences for input of scalar values and for requesting special features. Parameters values are specified on PARAM Bulk Data entries or PARAM Case Control commands. The PARAM Bulk Data entry is described in the "**Bulk Data Entries**" on page 945. The PARAM Case Control command is described in the "" on page 193. A complete alphabetical list of PARAMeter names and their functions is given in this section.

ACOUT Default = PEAK

ACOUT specifies the type of output to be used with the FORCE Case Control command in coupled fluid-structural analysis (see "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*. ACOUT=RMS requests root-mean-square output.

To obtain sound pressure level in units of dB and dBA given by the FORCE command, a peak reference pressure must be specified with PARAM, PREFDB. The dB level is defined as:

$$\text{dB} = 20 \cdot \log\left(\frac{P}{\text{PREFDB}}\right)$$

See also the Case Control command "**FLSTCNT**" on page 314.

ACSYM Default = YES

By default, the dynamic equations for coupled fluid-structure analysis in frequency response are symmetrized for efficiency. PARAM,ACSYM,NO requests the pre-Version 69 formulation which involves no symmetrization and will require more CPU time. See "**Formulation of Dynamic Equations in SubDMAP GMA**" on page 429 of the *MSC.Nastran Reference Guide*. See also the Case Control command "**FLSTCNT**" on page 314.

If the iterative solver is selected (see the ITER=YES keyword on the NASTRAN statement) then the external work diagnostic will be different between ACSYM=YES and ACSYM=NO.

ADJMETH Default = 0

This parameter selects the processing method used in a triple matrix product in module DSADJ. The default is usually preferred, but ADJMETH=1 can be used when disk space is critical. ADJMETH=2 only holds the active solution vectors.

ADPCON Default = 1.0

Initial penalty values used in contact analysis are calculated automatically by the program and are given by $k \cdot SFAC \cdot |ADPCON|$ where k is a number selected for each slave node based on the diagonal stiffness matrix coefficients that are in the contact region, and SFAC is the value specified by the user in the SFAC field of the BCONP Bulk Data entry. The ADPCON value applies to all the contact regions in the model. During the analysis, if convergence problems are encountered, penalty values are automatically reduced. Still there may be some problems where convergence can not be achieved with default values. In such cases, analysis may be restarted with a lower value of ADPCON.

In some cases the default penalty values may be low. In such situations analysis may be restarted with a higher value of ADPCON.

Generally, penalty values are recalculated every time there is a change in stiffness. However, if ADPCON is negative, penalty values are calculated only at the beginning of each subcase, and penalty values are not adjusted during analysis. This is useful if the contact between two elastic bodies is being analyzed.

ADSTAT Default = YES (SOL 109 and 112 only)

Used in transient analysis data recovery with differential stiffness (see the Case Control command, "**STATSUB**" on page 481) to request whether the static solution (displacements, SPC forces, and MPC forces) is to be included (YES) or excluded (NO) in the transient results.

AESDISC Default = 1.E-8

Tolerance for discarding generalized coordinates in the RITZ method (see AESMETH) which are not linearly independent.

AESMAXIT Default = 15

Maximum number of iterations for the ITER method (see AESMETH).

AESMETH Default = SELECT

Solution method for static aeroelastic analysis.

SELECT	selects the DIRECT method on models with less than 5000 DOF in the solution set; otherwise selects AUTO.
AUTO	selects the reduced basis method for an approximate solution, which is used as starting vectors for an ITER solution.
DIRECT	selects the direct solution.
RITZ	selects the reduced basis approximate solution.
ITER	selects the iterative solution.

AESRNDM Default = 2

Number of random vectors to use as generalized functions in the RITZ method (see AESMETH).

AESTOL Default = 1.E-10

Convergence criteria for the iterative solver.

ALPHA1 Default = 0.0, 0.0

ALPHA2

In frequency and transient response analysis, if PARAM,ALPHA1 and/or ALPHA2 are not equal to zero, then Rayleigh damping is added to the viscous damping. ALPHA1 is the scale factor applied to the mass matrix and ALPHA2 to the structural stiffness matrix. The scale factors are applied to the d-set or h-set matrices. See "**Formulation of Dynamic Equations in SubDMAP GMA**" on page 429 of the *MSC.Nastran Reference Guide*.

$$[B'] = [B] + \text{ALPHA1} \cdot [M] + \text{ALPHA2} \cdot [K]$$

If ξ_i is the damping ratio for the i-th mode ω_i (cycles/unit time), then ALPHA1 (α_1) and ALPHA2 (α_2) may be computed as

$$\begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = \frac{2\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \begin{bmatrix} \omega_j & -\omega_i \\ \frac{-1}{\omega_j} & \frac{1}{\omega_i} \end{bmatrix} \begin{Bmatrix} \xi_i \\ \xi_j \end{Bmatrix}$$

5 PARAM

and the damping ratio for any other ξ_l mode becomes

$$\xi_l = \frac{\omega_i \omega_j}{\omega_j^2 - \omega_i^2}\left[\left(\frac{\omega_j}{\omega_l} - \frac{\omega_l}{\omega_j}\right)\xi_i - \left(\frac{\omega_i}{\omega_l} - \frac{\omega_l}{\omega_i}\right)\xi_j\right]$$

Note: The use of Rayleigh damping with non-zero values of ALPHA1 is not appropriate for enforced motion problems involving large mass since the resulting damping matrix will essentially violate the assumption of large mass in the problem and thus give wrong answers.

ALPHA1FL Default = 0.0, 0.0

ALPHA2FL In frequency and transient response analysis, if PARAM,ALPHA1FL and / or ALPHA2FL are not equal to zero, then Rayleigh damping is added to the viscous damping. ALPHA1FL is the scale factor applied to the mass matrix and ALPHA2FL to the fluid stiffness matrix. The scale factors are applied to the d-set or h-set matrices. See "**Formulation of Dynamic Equations in SubDMAP GMA**" on page 429 of the *MSC.Nastran Reference Guide*.

$$[B'] = [B] + \text{ALPHA1FL} \cdot [M] + \text{ALPHA2FL} \cdot [K]$$

If ξ_i is the damping ratio for the i-th mode ω_i (cycles / unit time), then ALPHA1FL (α_1) and ALPHA2FL (α_2) may be computed as

$$\begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = \frac{2\omega_i \omega_j}{\omega_j^2 - \omega_i^2}\begin{bmatrix} \omega_j & -\omega_i \\ \frac{-1}{\omega_j} & \frac{1}{\omega_i} \end{bmatrix}\begin{Bmatrix} \xi_i \\ \xi_j \end{Bmatrix}$$

and the damping ratio for any other ξ_l mode becomes

$$\xi_l = \frac{\omega_i \omega_j}{\omega_j^2 - \omega_i^2}\left[\left(\frac{\omega_j}{\omega_l} - \frac{\omega_l}{\omega_j}\right)\xi_i - \left(\frac{\omega_i}{\omega_l} - \frac{\omega_l}{\omega_i}\right)\xi_j\right]$$

Note: The use of Rayleigh damping with non-zero values of ALPHA1FL is not appropriate for enforced motion problems involving large mass since the resulting damping matrix will essentially violate the assumption of large mass in the problem and thus give wrong answers.

ALTRED Default = NO

ALTRED=YES requests the alternate stiffness and load reduction technique for superelement analysis in SOLs 101 and 114. This technique is described in "**Static Solutions in SubDMAP SEKRRS**" on page 417, "**Static and Dynamic Load Generation**" on page 421, and "**Data Recovery Operations in SubDMAP SEDISP**" on page 433 of the *MSC.Nastran Reference Guide.*

ALTSHAPE Default = 0

ALTSHAPE selects the set of displacement shape functions to be used in p-version analysis. PARAM,ALTSHAPE,0 selects the MacNeal set. PARAM,ALTSHAPE,1 selects the Full Product Space set. For ALTSHAPE=1, IN=1 and ISOP=1 must be specified on the PSOLID entry.

ARBMFEM Default = YES

ARBMFEM controls the generation of '.bdf' file which contains the Finite Element Model of the arbitrary beam cross section. This parameter is functional for PBRSECT and PBMSECT only. To turn off the capability, set value of ARBMFEM to 'NO'.

ARBMPS Default = YES

ARBMPS controls the generation of outline plot for arbitrary beam cross section in PostScript format. This parameter is functional for PBRSECT and PBMSECT only. To turn off capability, set value of ARBMPS to 'NO'.

ARBMSS Default = NO

ARBMSS controls the stress recovery for the whole arbitrary beam cross section and the companion 'screened' stresses. The stress recovery for the whole cross section is available in 'OP2' format and suitable for post-processing. The 'screened' stresses for BAR and BEAM elements is available in print file (f06) and can be utilized for design optimization via RTYPE=ABSTRESS on DRESP1. This parameter is functional for PBRSECT and PBMSECT only. To turn on the capability, set value of ARBMSS to 'YES'.

Note: The recovery of 'screened' stresses will be turned on automatically if RTYPE=ABSTRESS is in use on DRESP1.

ARF Default = 0.95

See FLUIDMP.

ARS Default = 0.95

See FLUIDMP.

ASCOUP Default = YES

In coupled fluid-structure analysis, by default, the coupling between the fluid and structure is computed. This interaction will be ignored if PARAM,ASCOUP,NO is specified. See also the Case Control command "**FLSTCNT**" on page 314 for alternative selections.

ASING Default = 0

ASING specifies the action to take when singularities (null rows and columns) exist in the dynamic matrices (or $[K_{ll}]$ in statics). If ASING=-1, then a User Fatal Message will result.

If ASING=0 (the default), singularities are removed by appropriate techniques depending on the type of solution being performed. The procedures used are described in "**Data Recovery Operations in SubDMAP SEDRCVR**" on page 439 and "**Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS**" on page 449 of the *MSC.Nastran Reference Guide*.

AUNITS Default = 1.0

AUNITS is used in SOL 144 to convert accelerations specified in units of gravity on the TRIM Bulk Data entry to units of distance per time squared. Accelerations are divided by this parameter. To convert accelerations input in g's into physical, consistent units, set AUNITS to 1 / g.

AUTOADJ Default = YES

In SOL 200, a value of YES will automatically select the direct or the adjoint method for sensitivity analysis based on the performance criteria. AUTOADJ=NO requests that the adjoint method not be selected for sensitivity analysis and the direct is enforced. The default should be preferred in all cases. However, the presence of this parameter allows investigation of the alternative of using direct methods in the place of adjoint methods in the sensitivity calculations.

AUTOMSET Character, Default = NO MD Nastran Implicit Nonlinear (SOL 600) only.

For use in SOL 600, AUTOMSET=Yes maps to MSC.Marc's new parameter AUTOMSET. By default or if PARAM,AUTOSET,NO is entered, MSC.Marc's AUTOMSET will be omitted.

NO MSC.Marc's AUTOMSET parameter is omitted.

YES MSC.Marc's AUTOMSET parameter is activated.

Note: See Volume C of MSC.Marc for further details.

AUTOQSET Default = NO

AUTOQSET=YES requests the automatic calculation of component modes without the need to define a q-set (generalized coordinates).

1. The calculation of component modes is attempted on all superelements including the residual structure.
2. All component modes are treated like SPOINTs which means that all component modes are "brought down" to and assigned to the q-set in the residual structure. In other words, component modes may not be assigned interior to a superelement.
3. Selected component modes may not be removed (constrained).
4. Since the generalized coordinates are automatically defined, the following entries may not be specified: QSETi, SEQSETi, SENQSET, or PARAM,NQSET.
5. This feature is not supported with:
   a. Multiple boundary conditions
   b. Design optimization (SOL 200)
   c. Aerodynamic analyses (SOLs 144, 145, 146)
   d. Cyclic symmetry analyses (SOLs 114, 115, 116, 118)

AUTOSPC This parameter and the related parameters EPPRT, EPZERO, PRGPST, and SPCGEN are replaced by the AUTOSPC Case Control command.

AUTOSPCR Default = NO (SOLs 106 and 153 only)

In nonlinear static analysis only, AUTOSPCR specifies the action to take when singularities exist in linear stiffness matrix of the residual structure after multipoint constraints are processed. AUTOSPCR=YES means that singularities will be constrained and AUTOSPCR=NO means they will not be constrained. It is recommended that all degrees-of-freedom attached to nonlinear elements be specified on ASETi entries. Parameters EPPRT, EPZERO, PRGPST, and SPCGEN may be used with AUTOSPCR.

AUTOSPRT Default = YES

By default, free-free models will be automatically constrained for calculation of residual vectors (RESVEC Case Control command) as long as F1<=0.0 on the EIGR (or EIGRL) Bulk Data entry. The auto-SUPORT method may be deactivated by specifying a SUPORTi entry, PARAM,AUTOSPRT,NO, or F1>0.0.

FZERO is the maximum frequency assumed for a rigid body mode. FZERO is used by the auto-SUPORT method to extract the rigid body frequencies. The default is 1.0E-4.

BAILOUT Default = 0

See MAXRATIO.

BEAMBEA, Real, Default = 1000.0

Value Equivalent radius to be used for beam-beam contact problems. For tubes or round bars, use the outer radius. If the radii are different enter the largest outer radius. For beams, enter an equivalent radius calculated as follows:

I=0.5*(Ix+Iy)
R=sqrt(A / pi2+2*I / A)

where A, Ix, Iy are the cross-section properties and pi2 = Π2.

BEIGRED Default = NO (if DOMAINSOLVER ACMS is not specified)
Default = YES (if DOMAINSOLVER ACMS is specified)

PARAM,BEIGRED,YES requests a more efficient method be to reduce the viscous damping matrices coming from upstream superelements. The CPU and disk space savings may be significant if there are a small number of viscous damping elements in tip superelements within a multilevel superelement tree.

BIGER, BIGER1, BIGER2 Default = 0.0
See S1.

BUCKLE Default = -1

BUCKLE=1 requests a nonlinear buckling analysis in a restart run of SOLs 106 or 153. See the *MSC.Nastran Handbook for Nonlinear Analysis*. (Not supported for SOL 600.)

BUCKLE=2 requests buckling in a SOL 106 cold start run. (Must be in Bulk Data for SOL 600.)

CB1 Default = (1.0, 0.0)

CB2 CB1 and CB2 specify factors for the total damping matrix. The total damping matrix is:

$$[B_{jj}] = \text{CB1} \cdot [B^{x}_{jj}] + \text{CB2} \cdot [B^{2}_{jj}]$$

where

$$[B^{2}_{jj}]$$

is selected via the Case Control command B2GG and

$$[B^{x}_{jj}]$$

comes from CDAMPi or CVlSC element Bulk Data entries. These parameters are effective only if B2GG is selected in the Case Control Section.

CDIF Default = YES for shape optimization with or without property optimization

Default = NO for property optimization only

CDIF may be used to override the default finite difference scheme used in the calculation of pseudo loads in SOL 200. PARAM,CDIF,YES forces the selection of the central difference scheme used in the semianalytic approach regardless of the type of optimization requested. PARAM,CDIF,NO forces the selection of the forward difference scheme.

CDITER Default = 0

CDPCH Default = NO

CDPRT Default = YES

If CDITER>0, perform constrained displacement iterations in SOL 101. The value is the maximum number of iterations. If CDPRT=YES, print those negative displacements and tension forces which do not satisfy constraints. If CDPCH=YES, punch DMIG CDSHUT entries for final state; by default all gaps are closed. These can be used for initial conditions for restart. Potential contact points must be specified on the SUPORTi entries. The SUPORTi points must be in the residual structure. Optional DMIG entries to define the initial shut vector may be specified. Degrees-of-freedom that are specified on the SUPORT entry and have a value of 1.0 defined on the DMIG,CDSHUT entry will be considered closed initially.

If the DMIG,CDSHUT entry is not supplied, then all degrees-of-freedom specified on the SUPORT entries will be considered shut initially. A fatal message will be issued if this parameter is used and PARAM,INREL is specified.

CFDIAGP Default = NO

If YES, randomly deleted CFAST elements will be printed. (See CFRANDEL)

CFRANDEL Default = 0.

Represents a percent, expressed as a decimal fraction, of the number of CFAST elements to be randomly deleted.

CHECKOUT Default = NO

CHECKOUT=YES requests a model checkout in SOLs 101 through 200. See "**Geometry Processing in SubDMAP PHASE0**" on page 400 of the *MSC.Nastran Reference Guide*. The run will terminate prior to phase 1 of superelement analysis. The PARAM,POST options are also available with PARAM,CHECKOUT,YES. The following options and their user parameters are also available with PARAM,CHECKOUT,YES:

1. PARAM,PRTGPL,YES

   Prints a list of external grid and scalar point numbers in internal sort. It also lists external grid and scalar point numbers along with the corresponding sequence numbers in internal sort. The sequence numbers are defined as (1000*external number) and will reflect any user-requested resequencing.

2. PARAM,PRTEQXIN,YES

   Prints a list of external and internal grid and scalar numbers in external sort. It also lists external grid and scalar numbers with the corresponding coded SIL number in external sort. The coded SIL numbers are defined as:

$$10 \cdot \text{SIL} + \begin{cases} 1 \text{ for grid point} \\ 2 \text{ for scalar point} \end{cases}$$

   The SIL numbers correspond to degrees-of-freedom, i.e., one SIL number for scalar point and six SIL numbers for a grid point.

3. PARAM,PRTGPDT,YES

   Prints, for each grid and scalar point, the following information in internal sort:

   - Coordinate system ID in which grid point geometry is defined (ID=-1 for scalar points).
   - Spatial location of grid points in the “CP” coordinate system. For scalar points, all entries are zero.
   - Coordinate system ID for grid point displacements, forces, and constraints (ID=0 for scalar points).
   - Permanent single-point constraints defined on GRID Bulk Data entries. A zero is entered for scalar points.

4. PARAM,PRTCSTM,YES

   Prints for each coordinate system type the transformation matrix from the global to the basic coordinate system, and the origin of the indicated coordinate system in the basic coordinate system. Coordinate system types are: 1 = rectangular; 2 = cylindrical; 3 = spherical.

5. PARAM,PRTBGPDT,YES

   Prints all grid and scalar points listed in internal sort with their x, y, and z coordinates in the basic coordinate system. In addition, the coordinate system ID for grid point displacements, forces, and constraints is indicated for each grid point (ID=-1 for scalar points). The x, y, and z coordinates of scalar points are zero.

6. PARAM,PRTGPTT,YES

   Prints, for each temperature load set, information on element and grid point temperatures.

7. PARAM,PRTMGG,YES

   Prints the g-size mass matrix labeled by grid point / degree-of-freedom.

8. PARAM,PRTPG,YES

   Prints the g-size load vectors labeled by grid point / degree-of-freedom.

9. The summation of forces and moments of applied loads in the basic coordinate system is automatically output for each loading condition requested in the Case Control Section. Related parameters are GPECT, PROUT, and EST.

CK1 Default = (1.0, 0.0)

CK2 CK1 and CK2 specify factors for the total stiffness matrix. The total stiffness matrix (exclusive of GENEL entries) is

$$[K^{x}_{jj}] = \text{CK1} \cdot [K^{z}_{jj}] + \text{CK2} \cdot [K^{2}_{jj}]$$

where

$$[K^{2}_{jj}]$$

is selected via the Case Control command K2GG and

$$[K^{z}_{jj}]$$

is generated from structural element (e.g., CBAR) entries in the Bulk Data. These are effective only if K2GG is selected in Case Control. A related parameter is CK3.

Note: Stresses and element forces are not factored by CK1, and must be adjusted manually.

CK3 Default = (1.0, 0.0)

CK3 specifies a factor for the stiffness derived from GENEL Bulk Data entries. The total stiffness matrix is

$$[K_{jj}] = [K_{jj}^{x}] + \text{CK3} \cdot [K_{jj}^{y}]$$

where

$$[K_{jj}^{y}]$$

comes from the GENEL Bulk Data entries and

$$[K_{jj}^{x}]$$

is derived using PARAMs CK1 and CK2. CK3 is effective only if GENEL entries are defined. Related parameters include CK1 and CK2.

CLOSE Default = (1.0, 0.0)

See SCRSPEC.

CM1 Default = (1.0, 0.0)

CM2 CM1 and CM2 specify factors for the total mass matrix. The total mass matrix is

$$[M_{jj}] = \text{CM1} \cdot [M_{jj}^{x}] + \text{CM2} \cdot [M_{jj}^{2}]$$

where

$$[M_{jj}^{2}]$$

is selected via the Case Control command M2GG and

$$[M_{jj}^{x}]$$

is derived from the mass element entries in the Bulk Data Section. These are effective only if M2GG is selected in the Case Control Section.

COMPMATT Default = NO

In nonlinear statics (SOL 106), composite materials compute temperature-dependent properties for the plies only at the reference temperature given on the PCOMP Bulk Data entry. The ply properties are smeared and used for all load steps, regardless of whether the temperature is changing through application of thermal loads.

If the parameter is set to YES, the temperature-dependent properties for the plies are updated and smeared at the current temperature for each load step.

If the parameter is set to NONSMEAR, the temperature-dependent properties for the plies are updated at the current temperature for each load step. This option, only available for the CQUADR and CTRIAR elements, is an alternative to the smeared approach.

This parameter only applies to SOL 106, and only applies to composite CQUAD4, CTRIA3, CQUADR and CTRIAR and elements.

CONFAC Default = 1.E-5

In superelement analysis, CONFAC specifies the tolerance factor used in checking the congruence of the location and displacement coordinate systems of the boundary points between image superelements and their primaries (see the Bulk Data entry, "**CSUPER**" on page 1327). Specification of this parameter is recommended instead of DIAG 37 (DIAG 37 ignores User Fatal Messages 4277 and 4278).

COUPMASS Default = -1

COUPMASS>0 requests the generation of coupled rather than lumped mass matrices for elements with coupled mass capability, as listed in **Table 3-1** in the *MSC.Nastran Reference Guide*. This option applies to both structural and nonstructural mass for the following elements: CBAR, CBEAM, CONROD, CQUAD4, CHEXA, CPENTA, CQUAD8, CROD, CTETRA, CTRIA3, CTRlA6, CTRIAX6, CTUBE. A negative value (the default) causes the generation of lumped mass matrices (which may include torsion inertia for beam elements, and some coupling if there are beam offsets) for all of the above elements.

If SYSTEM(414) is greater than zero, then a negative value causes the generation of lumped mass matrices (translational components only) for all of the above elements.

P-elements are always generated with coupled mass and are not affected by COUPMASS.

CP1 Default = (1.0, 0.0)

CP2 The load vectors are generated from the equation

$$\{P_j\} = \text{CP1} \cdot \left\{P_j^x\right\} + \text{CP2} \cdot \left\{P_j^2\right\}$$

where

$$\left\{P_j^2\right\}$$

is selected via the Case Control command P2G, and

$$\left\{P_j^x\right\}$$

comes from Bulk Data static load entries.

CURV Default = -1

PARAM,CURV,1 requests that the CTRIA3 and CQUAD4 element stress and / or strain output be computed in a material coordinate system (normal output is in the element or basic coordinate system) and / or to interpolate it to grid points. (CQUAD4 element corner stress output is not supported.)

The integer parameter OG controls the calculation of stress and / or strain data at grid points. If OG is set to -1, the calculation for stresses and / or strain data at grid points is not performed. The default value of zero provides the calculation of these quantities at those grid points to which the selected elements connect.

User parameters S1G, S1M, S1AG, and S1AM, set to 1, request the printout of stresses at grid points, stresses in the material coordinate system, strains at grid points and strains in the material coordinate system, respectively.

The integer parameter OUTOPT may be set in accordance with the below options to select print, punch, and / or plotter output for stress and / or strain data that are computed in user-defined material coordinate systems.

OUTOPT Value	Description
0	Default-standard MD Nastran device codes are used.
1	Print only
2	Plot only
4	Punch only

The above values may be combined additively to select two or more forms of output. For example, OUTOPT=6 requests both plot and punch output. Related parameters include BIGER, CURVPLOT, DOPT, NUMOUT, NINTPTS, S1G, S1M.

For stress and / or strain / curvature output in a user-defined material coordinate system MCSID must be defined on MAT1 and MAT2 Bulk Data entries. The values of MCSID reference CORDiR, CORDiC, and CORDiS Bulk Data entries. A value of zero for MCSID does not imply the basic coordinate and will eliminate all elements which reference the MATi from the subject calculations.

1. If these data are requested at the element centers, the program will compute the unit vector $\bar{i}_m$ along the T1 or x-axis of the material coordinate system, and compare

   $$\left|\bar{n} \cdot \bar{i}_m\right|$$

   for each element that references the material coordinate system, where n is the normal to the surface of the element. If

   $$\left|\bar{n} \cdot \bar{i}_m\right|^2 \geq .4$$

   the projection of the y-axis on the surface of element is taken as the reference axis. Otherwise, the projection of the x-axis on the surface of the element is taken as the reference axis. The angle between the x-axis of the element coordinate system and the projection of the selected reference axis of the material coordinate system is used to transform the stress and / or strain data into the material coordinate system at the element centers.

2. If, on the other hand, the user requests these data at the grid points to which the elements connect the program will interpolate the results from (a) to the grid points to which the elements connect. The parameter NlNTPTS=N, the stress and / or strain data at the N closest element centers to the grid point in question will be used in the interpolation. The program may include more that N points in the interpolation if the distance of other element centers is not more than 10% greater than the closest N element centers.

The following specifies the output headings for stresses and / or strains in the material coordinate system.

Element stresses (PARAM,S1M,1)

1. Available in CQUAD4 and CTRlA3 elements

2. Page headings:

   STRESSES IN QUADRILATERAL ELEMENTS (CQUAD4)

   STRESSES IN TRIANGULAR ELEMENTS (CTRlA3)

3. Under the column FlBER DISTANCE:

   Z1 is replaced by MCSID.

   Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

Grid point stresses (PARAM,S1G,1 and PARAM,OG,1)

1. Available for CQUAD4 and CTRIA3 elements
2. Page heading:

   STRESSES AT GRID POINTS

3. Under the column are:

$$\left\{\begin{matrix} \text{MAT1} - \text{COORD1} - \text{ID} \\ \text{PROJ-CODE} \end{matrix}\right\}$$

   Z1 is replaced by MCSlD:

   Z2=A+10*N where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

Element strains (PARAM,S1AM,1)

1. Available for CQUAD4 and CTRlA3 elements
2. Page headings:

   STRAINS IN QUADRlLATERAL ELEMENTS (CQUAD4)

   STRAINS IN TRIANGULAR ELEMENTS (CTRIA3)

3. Under the column FIBER DISTANCE:

   Z1 is replaced by MCSID.

Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

Grid point strains (PARAM,S1AG,1 and PARAM,OG,1)

1. Available for CQUAD4 and CTRIA3 elements.
2. Page heading:

   Z2=A+10*N where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

   STRAINS AND CURVATURES AT GRID POINTS
3. Under the column are:

$$\left\{\begin{array}{c}\text{MAT1} - \text{COORD1} - \text{ID} \\ \text{PROJ-CODE}\end{array}\right\}$$

   Z1 is replaced by MCSID.

CURVPLOT Default = -1

PARAM,CURVPLOT,1 requests that x-y (or curve) plots whose abscissas are a sequence of grid points and whose ordinates may be displacements, loads, SPC forces, or grid point stresses. To obtain stress plots, set the CURV parameter to +1. The default for DOPT is the length between grid points.

Specify the XYOUTPUT Case Control command in the usual manner, replacing the point ID with the SID of SET1 Bulk Data entries.

The SET1 Bulk Data entries must contain unique SIDs for each set of grid points to be plotted.

User requests for xy-plots of output quantities appear in the Case Control Section in the standard form. For example,

```
.
.
.
OUTPUT(XYOUT)
.
.
XYPLOT DISP 1/4(T3)
.
.
XYPLOT SPCF 2/5(T1)
.
.
BEGIN BULK
```

The first XYPLOT command will produce an xy-plot from the displacement output of subcase 1. The abscissa of the curve will reflect the grid point IDs listed on the SET1 entry with an SID of 4, and the ordinate will reflect the T3 component of displacement at these grid points. The second XYPLOT command will produce an xy-plot whose ordinates are the T1 components of the forces of constraint in subcase 2 at the grid points listed on the SET1 entry with an SlD of 5.

The user has some degree of control over the scaling of the abscissas on these xy-plots. This control is exercised through the parameter DOPT on a PARAM Bulk Data entry. The legal values of this parameter provide the following scaling options for the abscissas.

Value of DOPT	Scaling for Abscissa
0 (default)	$\|g_j - g_i\|$
1	$\|x_j - x_i\|$
2	$\|y_j - y_i\|$
3	$\|z_j - z_i\|$
4	1.

Thus, the default value of DOPT will place the first grid point listed on the referenced SET1 entry at the origin, and subsequent grid points will be located along the abscissa at intervals proportional to the distance between that grid point and its predecessor. Values of DOPT equal to 1, 2, or 3 will scale the abscissa so that the interval between adjacent grid points is proportional to the difference in the X, the Y, and the Z components of the subject grid points respectively. DOPT=4 will space the grid points equally along the abscissa.

CWDIAGP Default = NO

For CWELD element: prints elements randomly deleted if set to yes.

CWLDIGNR Integer, Default = 0, SOL 700 only
Option to completely ignore CWELD entries.

0 Process (or fatal out CWELD entries if not supported yet) action depends on version.

1 Skip all CWELD entries

Note: The following entries will also be ignored if CWLDIGNR=1
CSHEAR
CGAP
PBEND
CBUSH
PBUSH
PGAP

CWRANDEL Default = 0.0

For CWELD element: if not zero, then it specifies as a decimal percent for the number of fasteners to randomly delete.

DBALL Default = DBALL

By default, all data to be stored on the database for restart purposes will be located on the DBALL database set DBset). These parameters permit the storage of some data blocks on DBsets other than DBALL, which are defined by the user and specified on the INIT File Management statement. Any or all of these parameters may be set to SCRATCH in order to reduce overall disk space usage; e.g., PARAM,DBUP,SCRATCH or PARAM,DBALL,SCRATCH. However, automatic restarts will be less efficient because data normally assigned to a permanent DBset will have to be recomputed.

A unique value for each superelement may be specified in the Case Control Section for the parameters DBALL, DBDN, DBRCV, and DBUP. Certain DBsets may be taken offline depending on which phase (see "**Summary of Solution Sequence Operations**" on page 441 of the *MSC.Nastran Reference Guide*) of superelement analysis is being performed. PARAM,DBALL specifies the default value for parameters DBDN, DBUP, and DBRCV.

The DBDN DBset contains data blocks necessary for "downstream" processing. For example, the stiffness, mass, damping, and static loads matrices that have been reduced to the boundary of the superelement are stored in this DBset.

The DBRCV DBset contains data blocks that must be online during the first pass through data recovery (Phase 3). These data blocks are used to recover the total displacement vector u_g of the superelement. This operation is performed by the SSG3 and SDR1 modules. On subsequent data recovery restarts, this DBset may be taken offline. Its default is determined from the value of DBUP.

The DBUP DBset contains data blocks necessary for "upstream" processing. For example, the geometry and property tables along with the stiffness, mass, damping, and static loads matrices related to the interior grid points of the superelement are stored in this DBset. These matrices and tables must be online during the reduction (Phase 1) and data recovery (Phase 3) of the superelement.

The IFP DBset contains data blocks that are required for all phases of the analysis. These data blocks are related to the entire model; examples are Bulk Data, superelement map, IFP module outputs, and resequenced grid points. This DBset must be online for all runs.

DBCCONV Default = XL

See POST=0.

DBCDIAG Default = 0

See POST=0.

DBCOVWRT Default = YES

See POST.

DBDICT Default = -1

Controls the printout of the database directory at the beginning and end of the run. See DBDICT FMS statement description in Section 2. If DBDlCT=0, then the database directory will be printed at the start of the run. If DBDICT=1, then the directory will be printed at the end of the run. If DBDlCT≥2, then it will be printed at the beginning and end of the run. If DBDICT=-1 (the default), the directory is not printed at the beginning or end of the run.

If multiple versions and/or projects exist on the database, then the parameters DBDRPRJ and DBDRVER allow the user to select the desired project and version, respectively. The appropriate values may be found in the Project/Version Table that is printed upon restart. If DBDRVER=0 (or DBDRPRJ=0), then the current version (or project) is selected. If DBDRPRJ=-1 (or DBDRVER=-1), then all projects (or versions) are selected.

DBDN Default = value of PARAM,DBALL.

See DBALL.

DBDRPRJ Default = 0

Specifies the desired project-identification number. See DBDICT.

DBDRVER Default = 0

Specifies the desired version-identification number. See DBDICT.

DBEXT Default = DBALL

Specifies the DBset location to store the external superelement information. External superelement information is generated by the EXTSEOUT Case Control command and the user PARAMeter EXTOUT.

DBRCV Default = value of PARAM,DBUP.

See DBALL.

DBUP Default = value of PARAM,DBALL.

See DBALL.

DDRMM Default = 0

DDRMM is only recognized if PARAM,SPARSEDR,NO is specified.

By default, the matrix method of data recovery is used in the modal transient and frequency response solutions. DDRMM=-1 will force calculation of complete g-set solution vectors by the mode displacement method.

DESPCH Default = 0

For sizing and shape optimization, DESPCH specifies in SOL 200 when the optimized (updated) bulk data entries are written to the PUNCH file. Currently, all the property entries, material entries, and connectivity entries that can be designed and DESVAR, DRESP1, and GRID entries can be written. Notice that the DRESP1 entries will be written if only when a mode tracking is performed and the DRESP1 responses have type FREQ or EIGN.

For topology optimization, DESPCH specifies when the topology optimized element density vaues are written to the topology element density history file jobname.des. This file can be directly read in MSC.PATRAN to displace and animate the topology optimization results.

DESPCH < 0 Never

DESPCH = 0 at the last design cycle only (default)

DESPCH > 0 at every design cycle that is a multiple of DESPCH and the last design cycle. For example, if n=2 and the maximum number of design cycles is 5 (DESMAX=5 on the DOPTPRM entry), then, DESVAR and GRID entries at design cycle 2, 4, and 5 are written in the punch file.

DESPCH1 Default = 0

DESPCH1 specifies in SOL 200 the amount of data to be written to the .pch and .des file. A positive DESPCH1 value request large field formats while a negative value requests small field formats. For a shape optimization job, if DESPCH1<>0, the updated GRID entries of the whole model will be written in the .pch file.

Descriptions of various DESPCH1 values are given below:

DESPCH1 = 0, write no data.

DESPCH1 = $\pm$1, write the property entries that are designed.

DESPCH1 = $\pm$2, write all the property entries of a given type when one or more property of that type is designed.

DESPCH1 = $\pm$4, write DESVAR and DRESP1 entries.

DESPCH1 = $\pm$n, write combine quantities by summing the DESPCH1 values. For example, n=1+4=5 requests writing all the designed property entries, DESVAR and DRESP1 entries to the .pch file for normal modes analysis.

DESPCH1 $\geq$ 0, write all topology designed and non-designed element density values to the topology element

DESPCH1 < 0, write all topological designed element density values to the topology element density history file jobname.des.

DFREQ Default = 10^{-5}

DFREQ specifies the threshold for the elimination of duplicate frequencies on all FREQi Bulk Data entries. Two frequencies, f_1 and f_2, are considered duplicated if

$$|f_1 - f_2| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

DOPT Default = 0

See CURVPLOT.

DPEPS Default = 1.0E-4

In SOL 200, if any difference between the property value on the property entries and the value calculated from the design variable values on the DESVAR entry (through DVCRELi, DVMRELi, DVPRELi relations) is greater than DPEPS, the design model values override the analysis values. If all the differences are less than DPEPS, analysis results from a previous run are accepted in a subsequent sensitivity / optimization task, thereby avoiding a reanalysis. The PTOL parameter on the DOPTPRM entry is a related parameter that checks the maximum difference.

DPHFLG Default = 0

In eigenvector sensitivity analysis DPHFLG=1 selects the subspace iteration method, which can accommodate repeated roots. Unless repeated eigenvalues are anticipated, the default value is recommended.

DSNOKD Default = 0.0

DSNOKD specifies a scale factor to the differential stiffness matrix in buckling design sensitivity analysis. If DSNOKD > 0.0, the effect of the differential stiffness matrix is included in buckling the design sensitivity analysis.

If PARAM,DSNOKD > 0 is specified in SOL 105, under the original Design Sensitivity Analysis (DSA), the differential stiffness sensitivity calculation is performed under the assumption that all the displacements are enforced; i.e., the change in the stiffness matrix due to the changes in the displacements are not computed. Therefore, PARAM,DSNOKD,0.0 is recommended in SOL 105. If PARAM,DSNOKD,1.0 is specified in SOL 200, the differential stiffness sensitivity calculation is performed more accurately; i.e., the change in the stiffness matrix due to the changes in the displacements are computed. However, the calculation is more expensive than with PARAM,DSNOKD,0.0.

Non-zero values of PARAM,DSNOKD cannot be used in SOL 200 with multiple buckling design subcases less each subcase contains the same STATSUB command.

DSZERO Default = 0.0

DSZERO specifies the minimum absolute value for the printout of design sensitivities.

DYBEAMIP Integer, Default = 0, SOL 700 d3plot output only

Number of beam integration points for output. This option does not apply to beams that use a resultant formulation

DYBLDTIM Real, No Default, SOL 700 only

The value entered for DYBLDTIM is the number of seconds over which a static load is built up. This parameter is only used in conjunction with PARAM,DYSTATIC,2.

DYBULKL Real $\geq$ 0.0, Default set internally by Dytran LS-DYNA), SOL 700 only

Value of the linear coefficient in the bulk viscosity equation.

DYBULKLQ1 Real $\geq$ 0.0, Default set internally by Dytran LS-DYNA), SOL 700 only

Quadratic bulk viscosity coefficient (Dytran LS-DYNA Q1 value).

DYCMPFLG Integer, Default = 0, SOL 700 d3plot output only

Orthotropic and anisotropic material stress and strain output in local material coordinate system for solids, shells and thick shells.

EQ.0 global,

EQ.1 local.

DYCONECDT* Integer $\geq$ 0, Default = 0, SOL 700 only

Time step size override for eroding contact:

= 0 contact time size may control Dt.

= 1 contact is not considered in Dt determination

*Large field parameter only.

DYCONENMASS* Integer $\geq$ 0, Default = 0, SOL 700 only

Treatment of the mass of eroded grids in contact. This option effects the adaptive contact algorithm (ADAPT=YES on the BCBODY option). Generally, the removal of eroded grids makes the calculation more stable; however, in problems where erosion is important the reduction of mass will lead to incorrect results.

= 0 eroding grids are removed from the calculation

= 1 eroding grids of solid elements are retained and continue to be active in contact

= 2 the eroding grids of solid and shell elements are retained and continue to be active in contact

*Large field parameter only.

DYCONIGNORE* Integer $\geq$ 0, Default = 0, SOL 700 only

Ignore initial penetrations during contact. 'Initial' in this context refers to the first time step that a penetration is encountered. This option can also be specified for each BCBODY individually. The value defined here will be the default.

= 0 move nodes to eliminate initial penetrations in the model definition

= 1 allow initial penetrations to exist by tracking the initial penetrations

= 2 allow initial penetrations to exist by tracking the initial penetrations.

However, penetration warning messages are printed with the original coordinates and the recommended coordinates of each slave node given.

*Large field parameter only.

DYCONPENOPT* Integer $\geq$ 0, Default = 1, SOL 700 only

Penalty stiffness value option. This only applies to contacts with SOFT=0 on the BCBODY entry.

= 0 the default is set to 1

= 1 minimum of master segment and slave node

= 2 use master segment stiffness

= 3 use slave node value

= 4 use slave node value, area or mass weighted

= 5 same as 4, but inversely proportional to the shell thickness. This may require special scaling and is not generally recommended.

Options 4 and 5 are recommended for metal forming calculations.

*Large field parameter only.

DYCONRWPNAL* Real ≥ 0.0, Default = 0.0, SOL 700 only

Scale factor for rigid wall penalties for treating rigid bodies interacting with fixed rigid walls.

= 0.0 rigid bodies interacting with rigid walls are not considered.

> 0.0 rigid bodies interact with fixed rigid walls. A value of 1.0 is recommended. Seven variables are stored for each slave node. This can increase memory requirements significantly if all nodes are slaved to the rigid walls.

*Large field parameter only.

DYCONSKIPRWG* Integer ≥ 0, Default = 0, SOL 700 only

Flag not to display stationary rigid wall

= 0 generate three extra nodes and one shell element to visualize a stationary rigid wall

= 1 do not generate nodes and shell element

*Large field parameter only.

DYCONSLSFAC* Real ≥ 0.0, Default = 0.1, SOL 700 only

Default scale factor for contact forces

*Large field parameter only.

DYCONTHKCHG* Integer ≥ 0, Default = 0, SOL 700 only

Shell thickness changes considered in single surface contact

= 0 no consideration (default)

= 1 shell thickness changes are included.

*Large field parameter only.

DYCOWPRD Real, no Default, SOL 700 only

If this parameter is entered, the real value is the value of D in the Cowper Symonds strain rate equation (see MSC.Dytran Reference Manual Remark 7 for YLDVM).

DYCOWPRP Real, no Default, SOL 700 only

If this parameter is entered, the real value is the value of P in the Cowper Symonds strain rate equation (see the MSC.Dytran Reference Manual Remark 7 for YLDVM).

DYDCOMP Integer, Default = 1, SOL 700 d3plot output only

Data compression to eliminate rigid body data:

EQ.1 off (default), no rigid body data compression,

EQ.2 on, rigid body data compression active,

EQ.3 off, no rigid body data compression, but nodal velocities and accelerations are eliminated from the database.

EQ.4 on, rigid body data compression active and nodal velocities and accelerations are eliminated from the database

DYDTOUT,DT Real, No Default, SOL 700 only

This parameter is used to determine the increment for d3plot outputs. It is used in conjunction with DISP=ALL, VELOCITY=ALL, ACCELERATION=ALL, STRESS=ALL, etc. For example, if DISP=ALL is entered and PARAM,DYDTOUT,DT is entered, the following Dytran LS-DYNA Case Control will be generated.

```
TYPE(D3PLOT) = D3PLOT
TIMES(D3PLOT) = 0,THRU,END,BY,DT
```

DYDYLOAD Integer, Default = 1, SOL 700 only

Determines if MD Nastran dynamic loads are passed directly to Dytran LS-DYNA or translated by the internal translator in MD Nastran

0 Dynamic loads are translated by the internal translator in MD Nastran using the same methods employed for version up to and included 2005 r2.

1 Dynamic loads are passed directly to Dytran LS-DYNA. For this option to work property, if loads other than what is listed below are used, the job may fail or obtain the wrong results. This option requires version 2005 r3 or greater of MD Nastran and Dytran LS-DYNA.

Case Control
- DLOAD
- LOADSET
- LOAD (dynamic relaxation only)

Bulk Data
- LSEQ
- DLOAD
- DELAY
- LOAD (dynamic relaxation only)
- DAREA
- TLOAD1
- TABLED1, TABLED2, TABLED3
- FORCE
- FORCE2
- MOMENT
- MOMENT2
- GRAV
- PLOAD
- PLOAD4
- RFORCE (CID, METHOD, RACC, MB fields are not available)
- SPCD

Note: 1. This parameter may be placed in rc files.

DYELAS1C, Integer, No Default, SOL 700 only.

ITABLE If this parameter is entered, and ITABLE is positive, all CELAS1 entries will e converted to CELAS1D. ITABLE will be used directly to specify TABLED1 associated with each of these CELAS1 springs.

If ITABLE is negative, all CELAS1 entries will be converted to CELAS1D. New TABLED1 entries will be created for each PELAS with an ID the same as PELAS entry. The Y (force) values of the new TABLED1 entries are calculated by multiplying the spring stiffness of each original PELAS times the Y values of corresponding points n the TABLES1 curve with ID of ITABLE (which should be considered to by a “unit value” curve). The unit curve is limited to 20 points.

TABLED1 with an ID of ITABLE must exist if this parameter is entered.

DYELAS1F, Integer, No Default, SOL 700 only

IFOLLOW This parameter is only active if PARAM,DYELAS1C is also entered. IFOLLOW should be set to 1 if the spring is a follower spring. If the spring is not a follower spring, set IFOLLOW=0 or leave the parameter out.

DYELAS1R, Integer, No Default, SOL 700 only

ICID This parameter is only active if PARAM,DYELAS1C is also entered. ICID is the reference coordinate system the spring degrees-of-freedom are defined in. If the system is the basic coordinate system, leave this parameter out or set ICID=0.

DYELPLET, Real, No Default, SOL 700 only

RETAN This parameter is only active if PARAM,DYELPLSY is also entered. In addition, RETAN=ETAN / E (the ratio of tangent modulus to Young’s modulus, E) to be used for all MAT1 conversions. Retan must be greater than zero.

DYELPLFL, Integer, No Default, SOL 700 only

FAIL This parameter is only active if PARAM,DYELPLSY is also entered. FAIL is the plastic failure strain which must be ≥ 0.0

DYELPLSY, Real, No Default, SOL 700 only

RSIGY If this parameter is entered, all elastic MAT1 materials will be converted to elastic-plastic MATD024 materials. In addition, RSIGY=SIGY / E (the ratio of yield stress to Young’s modulus, E) to be used for all MAT1 conversions. RSIGY must be greater than zero.

DYENDTIM Integer, Default = 1, SOL 700 only

If DYENDTIM = -1, MD Nastran TSTEP and TSTEPNL entries will be translated directly to Dytran LS-DYNA. This option requires that fields 7 and beyond as well as the continuation entries be eliminated.

If DYENDTIM=0 or 1 or if parameter is omitted, MD Nastran TSTEP and TSTEPNL entries will be translated to the MSC.Dytran Case Control ENDTIME command with the ENDTIME where the ENDTIME value is calculated using the number of time steps times the delta time from the TSTEP or TSTEPNL entry (only the first such specification will be translated)

DYENERGYHGEN* Integer > 0, Default = 1, SOL 700 only

Hourglass energy calculation option. This option requires significant additional storage and increases cost by ten percent.

= 1 hourglass energy is not computed (default)

= 2 hourglass energy is computed and included in the energy balance.

The hourglass energies are reported in the ASCII output files GLSTAT and MATSUM.

*Large field parameter only.

DYENGFLG Integer, Default = 1, SOL 700 d3plot output only

Flag for including shell internal energy density and thickness in the Dytran LS-DYNA database:

EQ.1 include (default),

EQ.2 exclude.

DYHRGIHQ Integer > 0, Default = 1, SOL 700 only

Default hourglass viscosity type

= 1 standard Dytran LS-DYNA (default)

= 2 Flanagan-Belytschko integration

= 3 Flanagan-Belytschko with exact volume integration

= 4 stiffness form of type 2 (Flanagan-Belytschko)

= 5 stiffness form of type 3 (Flanagan-Belytschko)

= 6 Belytschko-Bindeman assumed strain co-rotational stiffness form for solid elements only.

DYHRGQH Real ≥ 0.0, SOL 700 only

Default hourglass coefficient, QH. Values of QH that exceed .15 may cause instabilities.

DYIEVERP Integer, Default = 0, SOL 700 d3plot output only

Every plot state for "d3plot" database is written to a separate file. This option will limit the database to 1000 states:

EQ.0 more than one state can be on each plotfile,

EQ.1 one state only on each plotfile.

DYINISTEP* Real ≥ 0, Default=1.0E-5 if parameter is not entered, SOL 700 only

Specifies the initial time step. If not given, the value will be calculated automatically, unless PARAM,DYENDTIM=-1, in which case the initial time step is equal to DT as specified on a TSTEP or TSTEPNL entry. This parameter may be entered in the Bulk Data or RC file. If entered in an RC file, it must be truncated to 8 characters, param,dyinste,value.
*Large field parameter only.

DYLDKND Integer, Default = 0, SOL 700 only

Stress-strains curves for the entire input file are defined as follows:

0 = Engineering stress-strain

1 = True stress and strain

2 = True stress and plastic strain

3 = Plastic modulus and true stress

DYMATS1 Integer, Default = 1, SOL 700 only

If DYMATS1=0, MD Nastran MATS1 entries will be translated directly to Dytran LS-DYNA. If DYMATS1=1, MD Nastran MATS1 entries will be translated to the Dytran Bulk Data entry YLDVM. The matching TABLES1 entry if present will be converted to TABLED1 as required by DYTRAN.

Note:	PARAM,DYLDKND determines the type of stress-strain curve entered. If desired, the values D and P in the Cowper Symonds strain-rate equation can be entered with PARAM,DYCOWPRD,D and PARAM,DYCOWPRP,P respectively.

If DYMATS1=-1, all MATS1 and TABLES1 entries will be skipped.

DYMAXINT Integer, Default = 0, SOL 700 d3plot output only

Number of shell integration points written to the binary database. If the default value of 3 is used then results are output for the outermost (top) and innermost (bottom) integration points together with results for the neutral axis. If MAXINT is set to 3 and the element has 1 integration point then all three results will be the same. If a value other than 3 is used then results for the first MAXINT integration points in the element will be output.

Note:	If the element has an even number of integration points and MAXINT is not set to 3 then you will not get mid-surface result.

DYMAXSTEP* Real > 0.0, Default = 1.e20, SOL 700 only

Defines the maximum allowable time step. If the stable time step calculated is greater than MAXSTEP, the time step is set to MAXSTEP.

*Large field parameter only.

DYMINSTEP* Real $\geq$ 0.0, Default = 1.0E-9 if parameter is not entered, SOL 700 only

Defines the minimum time step that causes the analysis to terminate. When the elements become very distorted, in a poorly designed mesh for example, or when they have endured a very large distortion, the time step may drop dramatically. The analysis continues, however, a lot of computer resources may be wasted. This option allows you to specify a minimum time step that causes the analysis to terminate. This parameter may be entered in ithe Bulk Data or an RC file. If entered in an RC file, it must be truncated to 8 characters, param,dyminstre,value.

*Large field parameter only.

DYNAMES Integer, Default = 0, SOL 700 only

If DYNAMES=0, Dytran LS-DYNA output names will be of the form jid.dytr.d3plot, jid.dytr.d3hsp etc.

If DYNAMES=1, Dytran LS-DYNA output names will be of the form d3plot, d3hsp, etc.

DYNEIPH Integer, Default = 0, SOL 700 d3plot output only

Number of additional integration point history variables written to the binary database for solid elements. The integration point data is written in the same order that it is stored in memory-each material model has its own history variables that are stored. For user defined materials it is important to store the history data that is needed for plotting before the data which is not of interest.

DYNEIPS Integer, Default = 0, SOL 700 d3plot output only

Number of additional integration point history variables written to the binary database for both shell and thick shell elements for each integration point, see also PARAM,DYNEIPH.

DYNINTSL Integer, Default = 1, SOL 700 d3plot output only

Number of solid element integration points written to the Dytran LS-DYNA database. The default value is 1. For solids with multiple integration points NINTSLD may be set to 8. Currently, no other values for NINTSLD are allowed. For solids with multiple integration points, an average value is output if NINTSLD is set to 1.

DYNLOADS Integer, Default = 0, SOL 700 only

Determines whether dynamic loading is allowed in a SOL 700 analysis.

0 Dynamic loading is not allowed and the job will abort if any is found

1 Dynamic loading is allowed.

Note: Dynamic loading is not officially a portion of SOL 700 in releases prior to MD Nastran 2006. Certain dynamic loadings specified for MD Nastran SOL 109 or SOL 129 will not work properly with SOL 700 in releases prior to MD Nastran 2006. Most loadings created by MSC.Patran which use LOADSET/LSEQ will work properly. The user is cautioned to examine the jid.dytr.dat and jid.dytr.str files to make sure all dynamic loading intended for a particular analysis is correct.

DYN3THDT Integer, Default = 1, SOL 700 d3plot output only

Material energy write option for D3THDT database

EQ.1 off, energy is NOT written to D3THDT database,

EQ.2 on (default), energy is written to D3THDT database

DYPRMSPC Integer, Default = 1, SOL 700 only

Determines if permanent constraints (field 8 of GRID entry) is translated to SPC.

0 Permanent constraints are not translated to SCP.

1 Permanent constraints are translated to SCP.

Note: Some versions of MSC.Dytran-lsdyna have trouble with permanent constraints and issue a bogus error message in the d3hsp file. This is the case for MSC.Nastran 2005 r2.

DYRBE3 Integer, Default = 0, SOL 700 only

If DYRBE3=-1, MD Nastran RBE3 will be translated to a new MSC.Dytran RBE3D entry.

If DYRBE3=0, MD Nastran RBE3 will be translated as is to MSC.Dytran LS-DYNA

DYRBE3TY Integer, Default = 1, SOL 700 only

Determines if RBE3 is mapped to MPC to MSC.Dytran's RBE3D.

0 RBE3 is mapped to MSC.Dytran's RBE3 entry.

1 RBE3 is mapped to MPC's.

Note: When DYRBE3TY=1, DYNREB23 will be set to zero.

DYRLTFLG Integer, Default = 1, SOL 700 d3plot output only

Flag for including stress resultants in the shell Dytran LS-DYNA database:

EQ.1 include (default),

EQ.2 exclude

DYNRBE23 Integer, Default = 1, SOL 700 only

MSC.Dytran LS-DYNA sometimes experiences numerical errors when RBE2 or RBE3 are used with less than 6 degrees of freedom. This parameter allows the user to determine whether to use the degrees of freedom specified in the model or to use all six degrees of freedom for all RBE2 and RBE3 elements.

0 Use the degrees of freedom found in the model without change (the user needs to check that MSC.Dytran LS-DYNA successfully completed all specified time steps when using this option).

1 Change all RBE2 and RBE3 to use all 6 degrees of freedom even if fewer are specified.

DYNSPCF Default = NEW

PARAM,DYNSPCF,NEW requests that mass and damping coupled to ground be included in the SPCForce calculations for the linear dynamic solutions: SOLs 103, 107 through 112, 115, 118, 145, 146, and 200. OLD neglects these effects and gives the same SPCForce results obtained in versions prior to Version 68.

DYSHELLFORM* Integer, Default = 10, SOL 700 only

Sets the default for the shell formulation.

1 = Hughes-Liu

2 = Belytschko-Tsay (similar to Dytran's BLT)

3 = BCIZ triangular shell

4 = C0 triangular shell

5 = Belytschko-Tsay membrane

6 = S/R Hughes-Liu

7 = S/R co-rotational Hughes-Liu

8 = Belytschko-Leviathan shell

9 = Fully integrated Belytschko-Tsay membrane

10 = Belytscho-Wong-Chiang (similar to Dytran's KEYHOFF)

11 = Fast (co-rotational) Hughes-Liu

16 = Fully integrated shell element (very fast)

17 = Fully integrated DKT, triangular shell element

18 = Fully integrated linear DK quadrilateral / triangular shell

20 = Fully integrated linear assumed strain C0 shell

21 = Fully integrated linear assumed C0 shell (5 DOF)

22 = Linear shear panel element (3 DOF per node)

*Large field parameter only.

DYSHGE Integer, Default = 1, SOL 700 d3plot output only

Output shell hourglass energy density:

EQ.1 off (default), no hourglass energy written

EQ.2 on.

DYSHINP Integer, Default = 3, SOL 700 only

Specifies the number of integration points for SOL 700 shell elements (see DYSHELLFORM* to specify the type of shell to be used in the analysis.

DYSHTHICK* Character, Default = YES, SOL 700 only

Specifies whether or not the thickness of the shell changes with membrane straining.

YES = Shell thickness is modified according to the membrane strain.

NO = Shell thickness is constant

*Large field parameter only.

DYSIGFLG Integer, Default = 1, SOL 700 d3plot output only

Flag for including stress tensor in the shell Dytran LS-DYNA database:

EQ.1 include (default),

EQ.2 exclude

DYSTATIC Integer, Default = 0, SOL 700 only

If DYSTATIC=0, and if SOL 700, 106 is entered, static analysis will be simulated using dynamic relaxation. This option requires the use of Bulk Data entry DAMPGBL. The use of dynamic relaxation frequently adds significant damping to the model which in some cases can change the static response compared to the use of small damping.

If DYSTATIC=1, and if SOL 700, 106 is entered, static analysis will be simulated using a slow buildup of the loading. It is not necessary to add excess damping with this approach, but run times may be longer than if dynamic relaxation is used. PARAM,DYENDTIME is required for DYSTATIC=1 unless the default end time of 1.00 is acceptable.

If DYSTATIC=2, and if SOL 700, 106 is entered, static analysis will be simulated using a slow buildup of the loading linearly increasing from zero to full value over a period of time defined by PARAM,DYBLDTIM. The load will be held at full value between the time defined by PARAM,DYBLDTIM and the time defined by the time defined by PARAM,DYENDTIME and / or Case Control command, ENDTIME. For this option, damping greater than what the structure actually has is frequently employed, but not as much damping as for the dynamic relaxation option (DYSTATIC=0).

DYSTEPFCT, Real, Default = 0.9, SOL 700 only

VALUE Time step safety factor. Used in conjunction with PARAM,SCALEMASS. The mass will not be increased above values that cause the timestep to be VALUE*DTMIN where DTMIN is provided on PARAM,SCALEMAS.

DYSTEPFCTL* Real > 0, Default = 0.9, SOL 700 only

Defines a scale factor to be used on the internally calculated time step.

Note: In the future we will add DYSTEPFCT*, once the Euler elements become available in SOL 700.

*Large field parameter only.

DYSTRFLG Integer, Default = 0, SOL 700 d3plot output only

Set to 1 to dump strain tensors for solid, shell and thick shell elements for plotting by LS-PREPOST and ASCII file ELOUT. For shell and thick shell elements two tensors are written, one at the innermost and one at the outermost integration point. For solid elements a single strain tensor is written.

DYSTSSZ Integer, Default = 1, SOL 700 d3plot output only

Output shell element time step, mass, or added mass:

EQ.1 off (default),

EQ.2 output time step size,

EQ.3 output mass, added mass, or time step size.

DYTERMNENDMAS* Real $\geq$ 0.0, Default = 1.e20, SOL 700 only

Percent change in the total mass for termination of calculation. This option is relevant if and only if mass scaling is used to limit the minimum time step size. See DYSTEPDT2MS*.

*Large field parameter only.

DYTSTEPDT2MS* Real, Default = 0.0, SOL 700 only

Time step size for mass scaled solutions, DT2MS.

Positive values are for quasi-static analyses or time history analyses where the inertial effects are insignificant. If negative, fact* | DT2MS | is the minimum time step size permitted and mass scaling is done if and only if it is necessary to meet the Courant time step size criterion. The value of fact is specified on PARAM,DYSTEPFCTL*. This mass scaling option can be used in transient analyses if the mass increases remain insignificant. See PARAM,DYTERMNENDMAS*.

*Large field parameter only.

DYTSTEPERODE* Integer $\geq$ 0, SOL 700 only

Flag determining the behavior when TSMIN is reached (see PARAM,DYTERMNDTMIN).

= 0 The calculation will terminate

= 1 The element(s) with this time step will be eliminated

*Large field parameter only.

ENFMOTN Default = ABS

This parameter is designed for use with the SPC/SPCD method of enforced motion specification in SOLs 108, 109, 111, 112, 146, and 200. The default value of ABS implies that the results of the analysis represent absolute motion of the model. If the value is specified as REL, then the results represent motion relative to the enforced motion of the base. In the case of modal dynamic analysis (SOL 111 and SOL 112), this latter scenario is equivalent to employing the large mass approach and excluding the rigid body modes from the analysis.

EPPRT Replaced by the EPSSING Keyword on the AUTOSPC Case Control command.

EPSILONT Default = SECANT

In nonlinear statics (SOL 106), thermal loads are computed using the secant method:

$$\varepsilon_T = \alpha_{\text{load}}(T_{\text{load}} - T_{\text{ref}}) - \alpha_{\text{init}}(T_{\text{init}} - T_{\text{ref}})$$

If the parameter is set to INTEGRAL, thermal loads are computed using the integral method:

$$\varepsilon_T = \int_{T_{\text{init}}}^{T_{\text{load}}} \alpha(T)\,dT$$

This parameter only applies to SOL 106, and only applies to CQUAD4, CTRIA3, CQUADR, and CTRIAR elements.

EPZERO Replaced by the EPS Keyword on the AUTOSPC Case Control command.

ERROR Default = -1

For SOLs 111-112, when the constraint modes have non-zero generalized force the parameter ERROR set to 0 causes the fatal error exit to be branched over and the dynamic response is computed and output. This option is intended for model checkout work, where bad results are better for diagnostic work than no results at all. This parameter is a traditional feature also used in other, similar circumstances.

EST Replaced by the ELSUM Case Control command.

EXTDR, Default = NO. See EXTOUT

EXTDROUT, Default = 31. See EXTOUT

EXTDRUNT Default = NO

EXTOUT When inputting the matrices for a reduced external superelement (SEBULK, CSUPER), there are four options that can be selected using the parameter EXTOUT. EXTOUT must be placed in the Case Control Section above any subcase or in the main Bulk Data Section. The options for Step 1 (see the table at the end of this discussion) are as follows:

If EXTOUT is set to MATRIXDB, the reduced structural matrices and loading are stored on the database.

If EXTOUT is set to DMIGDB, the reduced structural matrices and loading are stored on the database in a format which allows automatic connection to the analysis model if the identification numbers of the reduction grid points and scalar points are the same as the grid points and scalar points used in the analysis model.

If EXTOUT is set to DMIGOP2, the reduced structural matrices and loading are written in OUTPUT2 format to a tape unit specified by the parameter EXTUNIT (Default is 30). The storage format is the same as the DMIGDB option and allows automatic connection to the analysis model if the identification numbers of the reduction grid points and scalar points are the same as the grid points and scalar points used in the analysis model. The output unit can be assigned to a specific file by using an ASSIGN command in the File Management Section.

If EXTOUT is set to DMIGPCH, the reduced structural matrices and loading are output on the punch file (.pch) in DMIG format.

The procedure for accessing the external superelement information depends on the option used to output the external superelement in Step 1. The methods are as follows:

1. If EXTOUT was MATRIXDB or DMIGDB in Step 1, use the following commands in the File Management Section:

   ASSIGN SEXXX='step1.MASTER'
   DBLOCATE DATABLK=(EXTDB)CONVERT(SEID=xxx), LOGICAL=SEXXX

   where

   step1.MASTER is the database from the Step 1.

   xxx is the superelement identification number given to the partitioned Bulk Data Section for the external superelement.

2. If EXTOUT was DMIGOP2 in Step 1, then use the following commands in the File Management Section:

   ASSIGN INPUTT2='step1_output2_file',UNIT=extunit

   where:

   step1_output2_file is the OUTPUT2 file from Step 1.

   extunit is the unit number specified by the parameter EXTUNIT (default=30).

3. If EXTOUT was DMIGPCH in Step 1, then include the punch file from step one in the partitioned Bulk Data Section. In addition, add the following Case Control commands in the subcase for the external superelement:

   K2GG=KAAX
   P2G=PAX

   The SEBULK entry defining the superelement as an external superelement and the EXTRN entry in the partitioned Bulk Data Section should not be specified.

If data recovery is desired for the external component in SOLs 101, 103, and 107 through 112, there are three methods to transmit the displacements of the reduced model to the external full model. The method is selected by the parameter EXTDROUT in the partitioned Bulk Data Section. The options are as follows:

1. EXTDROUT set to MATRIXDB. The displacements of the reduced component model are stored directly on the database. The sequencing of the displacement degrees-of-freedom corresponds to the sequencing in the reduced model.
2. EXTDROUT set to DMIGDB. The displacements of the reduced model are stored on the database in a format which allows automatic connection to the reduced component model if the reduction grid points and scalar points are the same grid points and scalar points used in the analysis model. This option can only be used if EXTOUT was set to DMIGDB or DMIGOP2.
3. EXTDROUT set to DMIGOP2. The same as EXTDROUT set to DMIGDB except that the displacements of the reduced model are written in OUTPUT2 format to a tape unit specified by parameter EXTDRUNT (default=31). The output unit can be assigned to a specific file by using an ASSIGN command in the File Management Section. This option can only be used if EXTOUT was set to DMIGDB or DMIGOP2.

Data recovery for the external component is limited to SOLs 101 and 103 and 107 through 112. Data recovery is accomplished using a restart procedure from the data base created in step one and setting parameter EXTDR to YES. The method on inputting the reduced displacements into the component model depends on the method used to output the external component in Step 2. The input methods are as follows:

1. If EXTDROUT was MATRIXDB or DMIGDB in Step 2, then add the following commands in the File Management Section:

ASSIGN SEXX='step1.MASTER'
RESTART LOGICAL=SEXX
ASSIGN SEYYY='step2.MASTER'
DBLOCATE DATABLK=(EXTDB) WHERE(SEID=yyy),
LOGICAL=SEYYY

where:

step1.MASTER is the database from the Step 1.

step2.MASTER is the database from the Step 2.

yyy is the superelement identification number given to the partitioned Bulk Data Section for the external superelement in Step 2.

2. If EXTDROUT was DMIGOP2 in Step 2, then add the following commands in the File Management Section:

   ASSIGN INPUTT2='step2_output2_file',UNIT=extdrunt

   where:

   step2_output2_file is the OUTPUT2 file from Step 2.

   extdrunt is the unit number specified by the parameter EXTDRUNT (default=31).

For SOL 101, the Case Control structure must match the system model subcase structure in the numbers of loading conditions. The loading used in step one to generate the loads transmitted to the analysis model must also be specified in this step. If the analysis model had more loading conditions than the component model, then the loadings defined in Step 1 must be specified first.

For SOL 103 and 107 through 112, the Case Control structure must match the analysis model subcase structure in the number of eigenvalue extractions, FREQ / DLOAD or TSTEP / DLOAD subcases.

Step 1 Create External SE	Step 2 Perform Analysis	Step 3 Data Recovery for External SE
PARAM,EXTOUT, MATRIXDB	a. ASSIGN SEXX='step1.MASTER' DBLOCATE DATABLK=(EXTDB), CONVERT (SEID=xxx), LOGICAL=SEXX b. PARAM,EXTDROUT,MATRIXDB	a. ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB), WHERE(SEID=YYY), LOGICAL=SEYYY b. PARAM,EXTDR,YES
PARAM,EXTOUT, DMIGDB	a. ASSIGN SEXX='step1.MASTER' DBLOCATE DATABLK=(EXTDB), CONVERT (SEID=xxx), LOGICAL=SEXX b. PARAM,EXTDROUT,MATRIXDB or PARAM,EXTDROUT,DMIGDB or PARAM,EXTDROUT,DMIGOP2	a. ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB) WHERE(SEID=YYY) LOGICAL=SEYYY ASSIGN INPUTT2= 'step2_output_file', UNIT=Extdrunt b. PARAM,EXTDR,YES
PARAM,EXTOUT, DMIGOP2	a. ASSIGN INPUTT2='step1_output2_file', Unit = extunit b. PARAM,EXTDROUT,MATRIXDB or PARAM,EXTDROUT,DMIGDB or PARAM,EXTDROUT,DMIGOP2	a. ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB), WHERE(SEID=YYY), LOGICAL=SEYYY ASSIGN INPUTT2='step2_output_file', UNIT=Exdrunt b. PARAM,EXTDR,YES
PARAM,EXTOUT, DMIGPCH	Include the .PCH file in partitioned Bulk Data Section and for external SE subcase a. K2GG=KAAX PG=PAX	

EXTRCV Default = 0

EXTRCV > 0 indicates that data recovery is to be performed on an external superelement. In this type of run, the database for the external superelement must be attached as the primary database (“**Database Concepts**” on page 513 of the *MSC.Nastran Reference Guide*), and the database that contains the solution vectors, superelement map, and external/internal grid point equivalence table for its downstream superelement must be attached via the DBLOCATE statements. The value of EXTRCV must also be specified in the CONVERT clause of the DBLOCATE statement for the EMAP data block.

The following example shows the DBLOCATE statements for external superelement data recovery in SOL 101.

EXTUNIT Default = 31. See EXTOUT

FACTOR Default = 10000

See OLDSEQ.

FIXEDB Default = 0

FIXEDB is used to reduce the cost of superelement checkout.

FIXEDB = -2 (SOL 101 only) is used on the initial runs when the user suspects that the superelement may contain errors and that only operations necessary for fixed-boundary solutions need be performed. In particular, the generation of the $[G_{oa}]$ matrix is branched over in the SEKR operation and $[P_a]$ is not generated in the SELR operation. These operations typically result in 50% of the reduction cost and are not needed in the fixed-boundary data recovery operations described in the next paragraph. After this operation has been completed, the keyword SELANG will appear in the database dictionary, indicating that the $[P_a]$ stored there is incomplete, and should not be summed into the downstream superelement, because System Fatal Message 4252 will be issued.

FlXEDB ≤ -1 (SOLs 101 and 103 only) allows uncoupled solutions for any superelement with conventional output requests. This output may be obtained as soon as the superelement is successfully generated and reduced and does not require that the entire model be assembled. In superelement statics, the solution is the component due to the

$$\{u_o^o\}$$

vector, i.e., a fixed-boundary solution. In superelement modes, the solution is the uncoupled eigenvectors of the component. If PARAM,FIXEDB,-1 is specified in the Bulk Data or in the residual structure subcase, the modes of the residual structure will not be computed. For a printout or plotting of the component mode eigenvectors it is recommended that PARAM,FIXEDB,-1 be specified in the Bulk Data Section or above the subcase level in Case Control. If the modes of the residual structure are desired, then PARAM,FlXEDB,0 should be specified in the residual structure subcase. Exterior degrees-of-freedom listed on SECSETi and SESUP entries are free, and those on SEBSETi degrees-of-freedom are fixed. Data recovery for the residual structure should not be requested for this option.

FlXEDB = +1 (SOL 101 only) is used after the superelement has been proven valid. In the SEKR and SELR operations, it provides a branch over all operations already completed in the SEKR and SELR phases and completes the generation of the $[G_{oa}]$ matrix and the boundary stiffness and load matrices. It is also a method to recover the factor of the $[K_{oo}]$ matrix if the run aborted while computing $[G_{oa}]$.

FKSYMFAC Default = 0.024

FKSYMFAC controls the symmetrization of the follower force stiffness in SOL 106. If FKSYMFAC = 1.0 is specified, the follower force stiffness K_f is symmetrized as:

$$K_{fs} = \frac{1}{2}(K_f + K_f^T)$$

and the symmetric part K_{fs} is used for efficiency. If FKSYMFAC= 0. is specified, the original follower force stiffness K_f is used. If a value of 0. < FKSYMFAC < 1. is specified, the non-symmetric part of the follower force stiffness is calculated as:

$$K_{fn} = K_f - K_{fs}$$

and the ratio of unsymmetry:

$$r = \frac{\|K_{fn}\|}{\|K_f\|}$$

is compared with the user specified value of FKSYMFAC. The norm $\|\ .\ \|$ is the absolute maximum number of the matrix.

If r < FKSYMFAC, the symmetric stiffness K_{fs} is used.

If r > FKSYMFAC, the original unsymmetric stiffness K_f is used.

For most cases, the symmetrized follower force stiffness will give sufficiently accurate answers. If the influence of the non-symmetric part of the follower force stiffness is important, a value of FKSYMFAC=1.e-9 is recommended.

The parameter FKSYMFAC is applicable to SOL 106 only, all other solution sequences symmetrize the follower force stiffness. See parameter FOLLOWK for a list of solution sequences which calculate the follower force stiffness.

FLEXINCR Default = 0

In SOL 144, a value of YES will cause the TRIM subcases to be ignored. Instead, the TRIM Bulk Data will be used to obtain the set of Mach, Dynamic pressure and symmetry values for Unit Solutions (Flexible Increments). These data can be archived in the aeroelastic database for subsequent reuse. (Flexible Increments are always computed. This param merely avoids the TRIM subcase if these increments are all that is required.)

FLUIDMP Replaced by options on the Case Control command "**FLSPOUT**" on page 310.

FLUIDSE Default = 0

PARAM,FLUIDSE,seidf specifies a special superelement reserved for fluid elements. Frequency dependent fluid elements must still be in the residual. The newer partitioned superelements are not supported.

FOLLOWK Default = YES

In SOLs 103, 105, 106, 107, 108, 109, 110, 111, 112, 115, and 116, FOLLOWK=YES (default) requests the inclusion of follower force stiffness in the differential stiffness. FOLLOWK=NO requests that the follower force stiffness not be included. For FOLLOWK=YES in SOLs 103, 105, 107, 108, 109, 110, 111, 112, 115, and 116, a separate static subcase is required and the STATSUB command is also required in the eigenvalue subcase. In nonlinear analysis (SOL 106), the follower force is included if PARAM,LGDISP,1 is specified. FOLLOWK is ignored in SOL 106 if LGDISP is not specified.

FRQDEPO Default = NO

By default, frequency-dependent elements cannot be connected to o-set degrees-of-freedom. PARAM,FRQDEPO,YES allows frequency-dependent elements to be connected to o-set degrees-of-freedom. However, results may not be reliable.

FZERO Default = 1.0E-4

See AUTOSPRT.

G, Default = 0.0

GFL G and GFL specify the uniform structural and fluid-damping coefficient in the formulation of dynamics problems. In coupled fluid-structure analysis, G is applied to the structural portion of the model and GFL to the fluid portion of the model. To obtain the value for the parameter G or GFL, multiply the critical damping ratio, C/C_0, by 2.0. PARAM,G and GFL are not recommended for use in hydroelastic or heat-transfer problems. If PARAM,G (or GFL) is used in transient analysis, PARAM,W3 (or W3FL) must be greater than zero or PARAM,G (or GFL) will be ignored. See "**Formulation of Dynamic Equations in SubDMAP GMA**" on page 429 of the *MSC.Nastran Reference Guide*.

GEOMU Default = 40

See POST=0.

GPECT Default = -1

GPECT controls the printout of all elements connected to each grid point. GPECT=+1 requests the printout. In superelement analysis, the list is printed if PARAM,CHECKOUT,YES is specified or the SEMG or SEALL Case Control command selects the superelement. GPECT=-1 suppresses the printout.

GRDPNT Default = -1

GRDPNT>-1 will cause the grid point weight generator to be executed. The default value (GRDPNT=-1) suppresses the computation and output of this data. GRDPNT specifies the identification number of the grid point to be used as a reference point. If GRDPNT=0 or is not a defined grid point, the reference point is taken as the origin of the basic coordinate system. All fluid-related masses and masses on scalar points are ignored. The following weight and balance information is automatically printed following the execution of the grid point weight generator.

- Reference point.
- Rigid body mass matrix [MO] relative to the reference point in the basic coordinate system.
- Transformation matrix [S] from the basic coordinate system to principal mass axes.
- Principal masses (mass) and associated centers of gravity (X-C.G., Y-C.G., Z-C.G.).
- Inertia matrix I(S) about the center of gravity relative to the principal mass axes. Note: Change the signs of the off-diagonal terms to produce the “inertia tensor.”
- Principal inertias I(Q) about the center of gravity.
- Transformation matrix [Q] between S-axes and Q-axes. The columns of [Q] are the unit direction vectors for the corresponding principal inertias.

In superelement static or geometric nonlinear analysis, GRDPNT> -1 also specifies the grid point to be used in computing resultants, in the basic coordinate system, of external loads and single point constraint forces applied to each superelement. If GRDPNT is not a grid point (including the default value of -1), then the resultants are computed about the origin of the basic coordinate system. In superelement analysis, weights and resultants are computed for each superelement without the effects of its upstream superelements.

For the CTRIAX6, CTRIAX, and CQUADX elements, the masses and inertias are reported for the entire model of revolution but the center of gravity is reported for the cross section in the x-z plane.

GUSTAERO Default = 1

If gust loads are to be computed, for example on restart, set GUSTAERO to -1. The default is recommended if no gust loads are to be computed after the flutter analysis.

GYROAVG Default = 0

Used to specify one of two formulations for frequency response analysis using the rotor dynamic capability. The default is to determine any frequency-dependent terms for each frequency. This option activates the frequency-dependent looping option. Setting the value < 0 uses an ‘average’ frequency formulation. This option avoids using the frequency-dependent looping and results in a shorter execution time. For this option, PARAM,WR3 and PARAM,WR4 must be specified to include rotor damping.

HEATCMD Character*16, Default=nastran

Name of a command to run MD Nastran SOL 600 thermal contact runs. MD Nastran first sets up an MSC.Marc run to determine the thermal contact conditions which are output in a file named jid.nthcnt. Next, MD Nastran converts these to standard MD Nastran thermal elements, and finally spawns a second MD Nastran job from the primary MD Nastran job. The command to run the second MD Nastran job is provided using this parameter. For example, if nast2005t1 is desired, enter CMD=nast2005t1. If the command "Nastran" is desired, either leave the parameter out or enter "nastran". The MD Nastran run to be spawned will have the form:

CMD jid.nast.dat rcf=RCF

Where file RCF depends on PARAM,MARHEATM

Remarks:
1. See PARAM,MRSPAWN2 for structural analysis.
2. CMD will be converted to lower case regardless of the case entered.

HEATSTAT Default = NO

In SOL 101, if PARAM,HEATSTAT,YES is entered, then temperatures are computed in a linear steady state heat transfer and then applied as thermal loads in a subsequent thermal stress analysis. Two subcases are required. The first defines the temperature loads, boundary conditions, and output requests for the heat transfer analysis and the second subcase defines the thermal loads, boundary conditions, and output requests for the thermal stress analysis. Thermal loads in the second subcase are requested through the command

TEMP(LOAD) = Heat Transfer Subcase ID

If this default is not acceptable, then in heat transfer subcase add the Case Control word TSTRU=SID and in structures subcase here

TEMP(LOAD) = SID

See the Case Control command, "**TSTRU**" on page 519.
PARAM,NESET is no longer used. HEATSTAT not supported for p-elements.

HFREQ, Default = 1.+30

HFREQFL The parameters LFREQ, HFREQ, LFREQFL, and HFREQFL specify the frequency range in cycles per unit time of the modes to be used in the modal formulations. (LFREQ and LFREQFL are the lower limits and HFREQ and HFREQFL are the upper limits.) In coupled fluid-structure analysis, HFREQ and LFREQ are applied to the structural portion of the model and HFREQFL and LFREQFL are applied to fluid portion of the model. The default for HFREQ and HFREQFL will usually include all vectors computed. Related parameters are LMODES and LMODESFL.

Note: If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

ICOPT Default = 1

Parameter ICOPT works together with the NLIC Case Control Command for SOL 400 only. The user input loads may or may not be in equilibrium with the initial condition. If ICOPT=0, MD Nastran will compute the initial acceleration based on user's inputs. Otherwise, it will be assumed that the initial acceleration is null. In other words, when ICOPT=1 (the default), it is assumed the whole structure is in equilibrium automatically. Theoretically, ICOPT=0 gives better solution. However, due to that the matrix is highly singular, a large amount CPU time may be required and the accuracy of the result may be in doubt for the solution with ICOPT=0.

IFP Default = value of PARAM,DBALL.

See DBALL.

IFTM Default = 0

IFTM specifies the method for Inverse Fourier Transformation in SOLs 111 and 146. See the *MSC.Nastran Aeroelastic Analysis User's Guide* for further discussion. Permissible values are 0, 1, and 2. The default value is recommended.

0: constant (default)
1: piecewise linear
2: cubic spline

INREL Default = 0

INREL controls the calculation of inertia relief or enforced acceleration in linear static analysis, buckling analysis, and differential stiffness in dynamic analysis. INREL = -1 or -2 requests that inertia relief or enforced acceleration be performed.

Enforced accelerations, if desired, are input on the DMlG,UACCEL Bulk Data entry. (See Sections 9.5.6 and 9.4.11 of the *MSC.Nastran Reference Manual* for the theoretical basis.)

INREL=-1:

SUPORT or SUPORT1 entries are required on one or more grid points in the Bulk Data Section which restrain rigid body motion. The total number of degrees-of-freedom specified on SUPORT and SUPORT1 entries must be less than or equal to six.

In SOL 105, SUPORT1, not SUPORT, Bulk Data entries must be used to define the supported degrees-of-freedom and the SUPORT1 Case Control command may only be specified in a separate static subcase.

Loads due to unit rigid body accelerations at the point referenced by PARAM,GRDPNT are computed and then appended to the external loads. If PARAM,GRDPNT is specified in superelement analysis, then the point must be interior to the residual structure and exterior to all superelements.

INREL=-2:

The value of PARAM,INREL,-2 provides an inertia relief analysis in INREL=-2 without the need for a SUPORTi entry. Remove all such entries. This method leads to indeterminate matrices which are not suppported by buckling. If attempted the solution will fail.

INRLM — Replaced by the INRLOD Keyword on the RESVEC Case Control command.

IRES — Default = -1

IRES=1 requests that the residual load vectors RULV and RUOV be output in all solution sequences. In superelement analysis, the parameters PRPA and PRPJ may also be used to request output of the partial load vectors $\{P_a\}$ and $\{P_j\}$, respectively. In geometric nonlinear analysis, PARAM,lRES,1 will cause the printing of the residual vector

$$\{\Delta P_f\} = [K_{ff}]\left\{u_f^{n+1} - u_f\right\} + \{F_f\} - \{P_f\}$$

ITAPE — Default = -1

ITAPE specifies the output status of the DSCMR matrix in SOLs 101, 103, and 105; and the DSCMCOL table and the DSCM2 matrix in SOL 200. (See the OUTPUT2 and OUTPUT4 module descriptions in the *MD Nastran 2006 DMAP Programmer's Guide*.)

IUNIT — Default = 11

IUNIT specifies the FORTRAN unit number on which the DSCMR matrix in Design Sensitivity SOLs 101, 103, and 105 and the DSCMCOL table and the DSCM2 matrix in SOL 200 will be written. (See the OUTPUT2 and OUTPUT4 module descriptions in the *MD Nastran 2006 DMAP Programmer's Guide*.)

KDAMP, — Default = 1

KDAMPFL — If KDAMP or KDAMPFL is set to -1, viscous modal damping is entered into the complex stiffness matrix as structural damping. In coupled fluid-structure analysis, KDAMP is applied to the structural portion of the model and KDAMPFL to the fluid portion of the model. See "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.

KDIAG Default = -1.0 (SOLs 106 and 153 only)

In nonlinear static analysis, KDIAG may be used to eliminate spurious mechanisms and singularities in the nonlinear stiffness matrix. The absolute value of KDIAG will be added to some or all of the diagonal terms in the nonlinear stiffness matrix as follows:

If KDIAG < 0.0, then add the absolute value of KDIAG to the null diagonal terms only. (Default)

If KDIAG = 0.0, then no action is taken.

If KDIAG > 0.0, then add the value of KDIAG to all diagonal terms.

K6ROT Default = 100.

K6ROT specifies the scaling factor of the penalty stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements. The contribution of the penalty term to the strain energy functional is

$$\Pi_p = 10^{-6}\text{K6ROT}\frac{1}{2}G\int_A (\Theta_z - \Omega_z)^2\, t dA$$

with

$$\Omega_z = \frac{1}{2}\left(\frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y}\right)$$

where A is the area of the shell element, t is the shell thickness, G is the in plane shear modulus, see the MID1 material identification number on the PSHELL Bulk Data entry. The in plane displacements u_x, u_y and the normal rotation Θ_z are shown in **Figure 5-1**. The normal rotation has no physical meaning and should be ignored. The penalty stiffness removes the singularity in the normal rotation. A higher value than K6ROT=100. is not recommended because unwanted stiffening effects may occur. If K6ROT=0. is specified, the singularity can be suppressed with the parameter AUTOSPC.

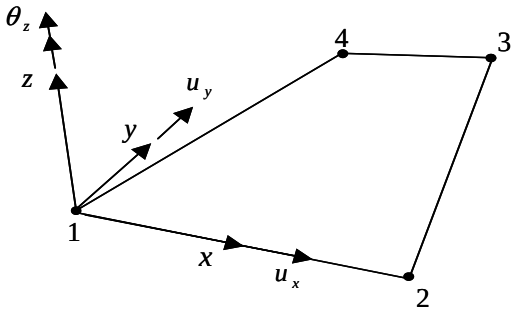


Figure 5-1 In plane displacements u_x, u_y and normal rotation Θ_z

LANGLE

$$\text{Default} = \begin{cases} 1 (\text{for all SOLs except 400}) \\ 2 (\text{for SOL 400}) \end{cases}$$

LANGLE specifies the method for processing large rotations in nonlinear analysis. By default, large rotations are computed with the gimbal angle method in nonlinear analyses SOLs 106, 129, 153, and 159 with geometric nonlinearity (PARAM,LGDISP,1). If PARAM,LANGLE,2 is specified, then they are computed with the Rotation Vector method. The value of LANGLE cannot be changed in a subsequent restart.

LFREQ, Default = 0.0

LFREQFL See HFREQ,

If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

LGDISP Default = -1

If LGDlSP = 1, all the nonlinear element types that have a large displacement capability in SOLs 106, 129, 153, 159, 400, and 600 (see **Table 3-1** in the *MSC.Nastran Reference Guide*, under “Geometric Nonlinear”) will be assumed to have large displacement effects (updated element coordinates and follower forces). If LGDlSP = -1, then no large displacement effects will be considered.

If LGDISP = 2, then follower force effects will be ignored but large displacement effects will be considered.

If LGDISP ≥ 0, then the differential stiffness is computed for the linear elements and added to the differential stiffness of the nonlinear elements.

LMFACT — LMFACT and PENFN are the scale factor and penalty function for the Lagrange rigid elements. For Lagrange rigid elements, please see Case Control command, RIGID. The purpose of LMFACT and PENFN is to make the values of stiffness matrix of the Lagrange rigid elements about the same relative magnitude as those of the other elements in the model. Too small a value will produce inaccurate results and too large a value will produce numerical difficulties. The same value is usually assigned to both LMFACT and PENFN. Under special requirement, user may assign different values for LMFACT and PENFN. For example, if PENFN=0.0 and LMFACT $\neq$ 0.0, then the solution method for the rigid elements become the pure Lagrange multiplier method instead of the augmented Lagrangian method. However, user must exercise caution if different values are assigned to LMFACT and PENFN. The default value is 1.0e+5 for all solution sequences except SOL 400. For nonlinear solution sequences SOL 400, MD Nastran will compute the appropriate default value in the nonlinear iterations.

LMODES, — Default = 0

LMODESFL — LMODES and LMODESFL are the number of lowest modes to use in a modal formulation. In coupled fluid-structure analysis, LMODES specifies the lowest modes of the structural portion of the model and LMODESFL the modes of the fluid portion of the model. If LMODES (or LMODESFL) = 0, the retained modes are determined by the parameters LFREQ and HFREQ (or LFREQFL and HFREQFL).

In SOL 103, LMODES may be used to reduce the number of eigenvectors to be processed in data recovery which may significantly reduce the CPU and storage costs.

Note: If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

LOADU Default = -1

See POST=0.

LOOPID Default = 0

LOOPID defines the desired loop number for initial conditions in a restart of SOLs 106, 129, 153, and 159. By default in SOLs 106 and 153 the restart proceeds from the last loop ID of the subcase defined by SUBCASID or SUBID. In SOLs 106, and 153 PARAM,SUBID or SUBCASID may also be specified.

LSTRN Replaced by the STRAIN Case Control command.

MACH Default = 0.0

Mach number. If more than one Mach number was used to compute aerodynamic matrices, the one closest to MACH will be used in dynamic aeroelastic response analysis. The default causes the matrices computed at the lowest MACH number to be used.

MARBATCH Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Specifies whether Marc will be spawned from MD Nastran in the "batch" mode or not.

0 Marc will be spawned suing batch=no.
1 Marc will be spawned using batch=yes.

Note: PARAM,MARCBATCH,0 requires PARAM,MARCTEMP,1 (which is the default). This combination of parameters will place the Marc log file in the MD Nastran log file.

MARC3D Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MARC3D determines whether to execute the MSC.Marc translator inside MD Nastran or not and is an alternative to the SOL 600 Executive Control statement.

MARC3D=0, MSC.Marc translator is not executed.

MARC3D=1, MSC.Marc translator is executed.

Not required if SOL 600 Executive Control is used.

Note: This parameter must be placed in the CASE CONTROL above any subcases.

MARCASUM Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

MARCASUM=-1 for nonlinear analyses and 1 for linear analysis. MSC.Marc's assumed strain formulation is not used.

MARCASUM=1, MSC.Marc's assumed strain formulation is used for plane stress, plane strain and solid elements (MSC.Marc types 3, 11 and 7). The assumed strain formulation improves the bending behavior of these elements and uses an enriched set of interpolation functions. Assumed strain should be off for analyses with a significant amount of plasticity. In determining the type of analysis (linear or nonlinear) for defaults of this parameter, the SOL 600,ID Executive statement is used. If ID is 106 or 129, the analysis is considered to be nonlinear and the default is -1. If ID is any other value, the analysis is considered to be linear and the default is 1. For nonlinear analyses without plasticity, this parameter should be turned on for models with solid elements.

MARCAUTO Integer, Default = leave out parameter, MD Nastran Implicit Nonlinear (SOL 600) only

Determines which MSC.Marc's increment option is used.

MARCAUTO=1, NLPARM entries will be translated to MSC.Marc's AUTO STEP option. If contact is present, the number of steps (NINC) is less than 100, it will be reset to 100. MSC.Marc will adaptively reduce the number of steps if possible, however, this option forces the first step to be 1% of the total time. If the first step is too large, experience has shown that convergence problems much later in time might result. To start with a different initial time step, see options 999 or -999.

MARCAUTO=-1, NLPARM entries will be translated to MSC.Marc's AUTO INCREMENT option. If contact is present, the number of steps is automatically set to 100. It has been found that certain difficult contact problems which fail using the AUTO STEP option run successfully using AUTO INCREMENT. This option is not available if the only "loading" is rigid contact or velocity control.

MARCAUTO=-2, NLPARM entries will be translated to MSC.Marc's AUTO LOAD option with no adjustment in the number of steps. Use of the option is not recommended. This option is not available if the only "loading" is rigid contact or velocity control.

MARCAUTO=999, MSC.Marc's AUTO STEP option will be used with no adjustment in the number of steps whether or not contact is present. This option is not available if the only "loading" is rigid contact or velocity control.

MARCAUTO=-999, MSC.Marc's AUTO INCREMENT option will be used with no adjustment in the number of steps whether or not contact is present. This option is not available if the only "loading" is rigid contact or velocity control.

See PARAM,MARCITER for a similar option. Do not use both MARCAUTO and MARCITER parameters.

MARCAXEL Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Allows a combination of axisymmetric and plane stress elements for 2D analyses. This analysis technique is sometimes used for approximate turbine disk / blade analysis.

0 The combination, if present in the input data will cause a Severe Warning and MSC.Marc will not be spawned.

1 Combination is allowed and all CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR elements will be considered to be plane stress and mate with CTRIAX6 elements.

MARCBATCH Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Specifies whether MSC.Marc will be spawned from MD Nastran in the "batch" mode or not.

0 Marc will be spawned using batch=no

1 Marc will be spawned using batch=yes

Note: PARAM,MARCBATCH,0 requires PARAM,MARCTEMP,1 (which is the default). This combination of parameters will place the MSC.Marc log file in the MD Nastran log file.

MARCBEAM Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

MARCBEAM=0, CBEAM entries will be mapped to MSC.Marc element type 14 for all cross sections specified using PBEAML. Full plasticity is available for all such cross section shapes with this option. CBEAM cross section specified using PBEAM will be mapped to MSC.Marc element 98 and remain elastic.

MARCBEAM=1, all CBEAM entries will be mapped to MSC.Marc element type 98 and remain elastic regardless of whether the cross section is specified using PBEAM or PBEAML.

MARCBUSH Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only Determines whether various fields on CBUSH are ignored or not.

0 Fields G0, X1, X2, X3 and / or CID cannot presently be converted to MSC.Marc and the job will terminate with an appropriate message.

1 Fields G1, X1, X2, X3 and / or CID will be ignored and the translation will proceed as fi these fields were blank.

Note: This parameter can be added to RC files if desired.

MARCCBAR Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only Specifies whether CBAR will be replaced by CBEAM for SOL 600 and 700.

0 CBAR is not replaced by CBEAM

1 CBAR is replaced by CBEAM (PBAR is replaced by PBEAM, PBARL is replaced by PBEAML)

Note: Use of this parameter is not usually required but might be beneficial in combination with PARAM,MSPEEDSE,1 to speed up translation of models with a large number of CBAR elements particularly when a there are large number of PBAR entries or PBARL entries.

MARCCENT Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Controls where the element output is generated.

MARCCENT=0, element output from MSC.Marc will be generated for each integration point.

MARCCENT=1, element output from MSC.Marc will be generated at the center of each element only. This option saves disk space and computer time, but may not catch the maximum stresses or strains. It might be sufficient for models with very fine grid spacing. (For shells, the output at the top and bottom surfaces will still be obtained.)

MARCCON2 Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If entered, the integer value entered is the second value on MSC.Marc's CONTACT second entry representing the maximum number of entities to be created for any contact surface. No longer required for MSC.Nastran 2005 r2.

MARCCON3 — Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If entered, the integer value entered is the third value on MSC.Marc's CONTACT second entry representing the maximum number of nodes that lie on the periphery of any deformable contact surface. No longer required for MSC.Nastran 2005 r2.

MARCCPY — If MARCCPY is specified, MSC.Marc files will be copied to MD Nastran output files and / or deleted according to the option (0, 1, or 2) shown below.

MARCCPY Option	Copy MSC.Marc Output Files to MD Nastran Output Files	Delete MSC.Marc Input & Output Files
0 (default)	No	No
1	Yes	Yes
2	Yes	No
3	No	Yes

If MARCCPY is 1 or 2, the out and log files will be copied as produced by MSC.Marc. If MARCCPY is -1 or -2 the actions as shown above for +1 or +2 will occur, and MSC.Marc-type test will be converted to MD Nastran-type text using and ASCII file named marcfilt.txt which must be located in the same directory where the MD Nastran input resides or in the same directory where the MD Nastran executable resides.

The following MSC.Marc files are potentially affected by the MARCCPY option:

MSC.Marc Output File	MD Nastran Output Copied to	MARCCPY
name.marc.out	name.f06	1, 2, -1, -2
name.marc.log	name.log	1, 2, -1, -2

MSC.Marc Output File	MD Nastran Output Copied to	MARCCPY
name.marc.t16	not copied, will remain if produced	
name.op2, fort.11, or ftn11	not copied, will remain if produced	

MARCDEF Integer, Default = 2, MD Nastran Implicit Nonlinear (SOL 600) only

MARCDEF=0, SOL 600 default options for MSC.Marc will be set to values determined to be best for MD Nastran-type problems (for MARCDEF=0, MSC.Marc's SHELL SECT parameter will be set to 11 if the value of MARCDEF is zero.

MARCDEF=1, default values will be set to current MSC.Marc standard (Mentat) values.

MARCDEF=2, default values will be set to "improved" MSC.Marc default values agreed on by the MSC.Marc and MD Nastran development groups.

Default values affect the following MSC.Marc data options and fields:

MARCDEF	Field	Value
0 - control	2	10
0 - Auto Step	5	0.01*max time
0 - Auto Step	8	10
0 - Auto Step	10	1
1 - control	2	3
1 - Auto Step	5	1.0E-5*max time
1 - Auto Step	8	5
1 - Auto Step	10	0
2 - control	2	10
2 - Auto Step	5	1.0E-3*max time
2 - Auto Step	8	5

MARCDEF	Field	Value
2 - Auto Step	10	1

Note: For MARCDEF=0, the first three values were found to provide better convergence and the last (auto step 10) allows snap-through solution to converge correctly without having to use arc-length methods. This parameter can be set in the system-wide rc file as well as the user's rc file or the local rc file (same directory as the MD Nastran input data to provide the selected set of defaults for all runs if so desired. If the parameter is entered in the MD Nastran input data file, it will override any parameters set in any of the rc files.

MARCDILT — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
If omitted, SOL 600 determines the value.

If MARCDILT=0, constant dilitation is not used.

If MARCDILT=1, constant dilitation formulation is used for solids, axisymmetric, and plane strain elements (MSC.Marc element types 7, 10, 11, 19 and 20) if the model includes any of these element types. For elastic-plastic and creep analysis this formulation is usually too stiff when constant dilitation is not used. MARCDILT=1 and MARCASUM=1 should not both be used.

MARCDIS2 — Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If entered, this integer value entered here is the second value on MSC.Marc's DIST LOADS ("parameter" Section 2 of MSC.Marc's Volume C Program Input) entry representing the maximum number of different lists of distributed loads.

MARCDIS3 — Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If entered, the integer value entered here is the third value on MSC.Marc's DIST LOADS ("parameter" Section 2 of MSC.Marc's Volume C Program Input) entry representing the maximum number of elements in any particular distributed loads list.

MARCDIS4 Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value If entered, the integer value entered here is the fourth value on MSC.Marc's DIST LOADS ("parameter" Section 2 of MSC.Marc's Volume C Program Input) entry representing the maximum number of nodes with point loads applied.

MARCDMIG, Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

N If matrices or loads are entered using K2GG, M2GG, B2GG, K2PP, M2PP, B2PP, P2G in the MD Nastran Case Control Section, they will be translated to MSC.Marc as follows depending on the value of N:

N=0 All DMIG's in the MD Nastran file (and include files) will be placed in the MSC.Marc input file whether used or not.

N>0 All DMIG's in the MD Nastran file (and include files) will be placed on a new file named dmigxxxx.dmi where xxxx is the value of N. This new file will be "included" in MSC.Marc using a MSC.Marc include statement. For example, if N=100 the file name will be dmig100.dmi if N=25765 the file is dmig25765.dmi. N must not exceed a value of 999999.

Note: This parameter is ignored for External Superelements (if the MESUPER Bulk Data entry is present).

MARCEKND Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

ID Selects the type of strain results to be placed in a MD Nastran op2 file (if a request for an op2 file is made). ID can take the following values:

MARCEKND=0, Total strains will be processed

MARCEKND=1, Plastic strains will be processed

MARCEKND=2, Elastic strains will be processed

For creep analyses, creep strain is output if a request for strain output is made.

MARCEXIT Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MARCEXIT=0, if one of the COPYR options on the SOL 600 statement is specified, MD Nastran will process these options and then a DMAP exit will occur.

MARCEXIT=1, the COPYR options will be processes and MD Nastran will not exit.

MARCFILi Character, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

Name Name a file name limited to 8 characters (16 characters if param*is used) used in conjunction with ne of the CONTINUE options on the SOL 600 statement. For example, if CONTINUE=1 on the SOL 600 statement and PARAM,MARCFIL1,DMIG44 is entered, friction stiffness and possibly damping) matrices are created in DMIG format by MSC.Marc and placed on file DMIG44. The various CONTINUE options use the following MARCFILi entries:

Continue Option	MARCFILi	SOL Executed	K2GG/K2PP
1.	MARCFIL1	107	K2GG
2.	MARCFIL2	107	K2GG
3.	MARCFIL3	111	K2PP
4.	MARCFIL1	112	K2PP
5.	MARCFIL1	*	*
6.	MARCFIL1	110	K2GG
7.	MARCFIL1	103	K2GG

	Note: For CONTINUE=5, the ninth field of the MDMIOUT entry is used to determine the solution. K2GG or K2PP will be selected according to the above rules. The files should normally have the extension .dmi appended, for example test1.dmi . If the name including the dmi extension exceeds 8 characters, use the wide field form of the parameter and code in fixed field (not free field). The name should be entered in lower case. For example, $2345678 234567890123456 234567890123456 PARM* MARCFIL1 longname.dmi
MARCFRIC	Real, Default = 0.0, MD Nastran Implicit Nonlinear (SOL 600) only
Value	When the Case Control command, BCONTACT = ALL is specified, no other 3D contact data is required in the input file, except that the Coulomb coefficient of friction may be entered using the value of this parameter. Do not enter this entry if contact surfaces are specified in the Bulk Data.
MARCGAPD,	Real, no Default, MD Nastran Implicit Nonlinear (SOL 600) only
D	Depending on the value of PARAM,MARCGAPP, enter the gap closure distance for fixed direction gaps or the minimum distance between end points for the true distance gap. If $d > 0$, the two end points are never closer than a distance $\|d\|$ apart. If $d < 0$, the two end points are never farther apart than $\|d\|$.
MARCGAPN	Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only
ID	ID of gap element for which the immediately following PARAM,MARCGAPP and PARAM,MARCGAPD apply. Unlike most other parameters, several sequences of parameters MARCGAPN, MARCGAPP and MARCGAPD may be entered to specify values for all gap elements. If no MARCGFAPN is entered, the values entered for MARCGAPP and MARCGAPD will be used for all gaps in the model.
MARCGAPP	Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only MARCGAPP=0, MD Nastran gap elements will be translated to MSC.Marc fixed gap elements.

MARCGAPP=1, MD Nastran gap elements will be translated to MSC.Marc True Distance gaps.

MARCHEAT Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600 only)
Determines whether the analysis involves heat transfer or not.

0 Analysis is a structural analysis without heat transfer

1 Analysis is heat transfer (no structural analysis)

2 Analysis is coupled heat / structural analysis (not presently available)

Note: PARAM,MARCHEAT,1 is the same as using SOL 600,153, or SOL 600,159.

MARCHOST Character, no Default, MD Nastran Implicit Nonlinear (SOL 600 only)

Determines the name of a hostfile to be used with SOL 600 parallel runs. If this parameter is missing, no host file is used and the parallel run will run on one machine. That machine may have several processors and as many processors as specified on the PARAMARC Bulk Data entry will be used. IF PARAM,MARCHOST,Name is specified, the hostfile must be generated by the user in a format acceptable to MSC.Marc (see the *MSC.Marc and MSC.Marc Mentat Installation and Operations Guide*). Each line of the hostfile normally lists how many processors are used on each machine. If PARAM*,MARCHOST is entered, the name is limited to 16 characters (all lower case).

MARCIAMN Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600 only)

MARCIAMN=0, MD Nastran is directed to spawn MSC.Marc (as specified by the SOL 600 Executive Control statement or PARAM,MARCRUN), using a full version of MSC.Marc. Standard MSC.Marc licensing is required.

MARCIAMN=1, a special version of MSC.Marc is spawned by MD Nastran. This version of MSC.Marc may have certain features that are not available in the full version. MSC.Marc will be spawned from MD Nastran with the additional command line switch - iam nanl. The licensing for both MD Nastran and MSC.Marc reflect this situation. This option applies only to MSC.Marc version 2003 or later. If PARAM,MARCVERS points to a MSC.Marc version earlier than 2003, MARCIAMN will be set to zero and a full version of MSC.Marc is required.

The parameter may be set in the system-wide rc, the user rc file or as an environmental variable using NASM_IAMN=0 or 1 (similar to the way values on the SOL 600 entry are set).

MARCITER — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only. Used to control fixed time stepping in SOL 600.

MARCITER=0, fixed time steps or auto time steps will be controlled by PARAM,MARCAUTO.

MARCITER=N, allows fixed time steps to be used without needing to set the maximum and minimum times to nearly the same value (using MSC.Marc's AUTO STEP option). This parameter triggers true fixed time stepping with the other advantages AUTO STEP has over methods such as AUTO LOAD. For example, it uses better numerical damping. If this parameter is entered with a positive integer (N), a value of 2 is placed in the AUTO STEP field 9 and N is placed in field 7.

MARCITER=-1, this option is similar to PARAM,MARCITER,N (fixed time stepping will be used) except that the time comes from the NLPARAM or TSTEPNL entry. This option is not available if the only "loading" is rigid contact or velocity control. See PARAM,MARCAUTO for a similar option. Do not use both MARCAUTO and MARCITER parameters.

MARCLOWE — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Used in conjunction with superelement matrices created by MSC.Marc.

MARCLOWE=0, standard modulus values for all materials will be used.

MARCLOWE=1, all modulus of elasticity values will be changed to 1.0E-9 for the second MD Nastran run (when MD Nastran spawns another MD Nastran run using the SOL 600 continue option. This option is sometimes necessary for cases where MSC.Marc creates a superelement or substructure stiffness matrix but does not create a mass matrix. In this case, the second MD Nastran run will create the mass matrix using standard elements, density and other concentrated and distributed masses but the stiffness created by MD Nastran will be very low. Essentially the entire stiffness of the model will come from the stiffness matrices created by MSC.Marc.

MARCLUMP — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MARCLUMP=0, lumped mass will be used for SOL 600 transient dynamics or eigenvalue problems.

MARCLUMP=1, consistent mass will be used for SOL 600 transient dynamics or eigenvalue problems.

MARCMAT2 — Real, Default = -1.0, MD Nastran Implicit Nonlinear (SOL 600) only

Used if g33 = 0.0 on MAT2 entries. MSC.Marc will diverge if g33 = 0.0 for MAT2 entries. If the value entered is positive, the value is a multiplier of g11 and g22 to calculate g33 as follows:
g33 = marcmat2*(g11 + g22)

MARCMAT3 — Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

Used if MD Nastran has generated MAT2 from PCOMP and the MID of MAT2 is 30000001 is greater corresponding to MID3 for PSHELL.
If the value entered is 0, this entry is ignored regardless of the MAT2 MID value.
If the value entered is 1 the MAT2 entry will be mapped to MSC.Marc's ANISOTROPIC entry such that all Cij are zero except the following:
C55=g11
C56=g12
C66=g22

If the value entered is 2 the MAT2 entry will be mapped to MSC.Marc's ANISOTROPIC entry such that all Cij are zero except the following:
C44=g11
C45=g12
C55=g22
C66=g22
If the value entered is 11 the MAT2 entry will be mapped to MSC.Marc's ANISOTROPIC entry such that all Cij are zero except the following:
C55=g11
C56=g12
C66=g22
If the value entered is 12 the MAT2 entry will be mapped to MSC.Marc's ANISOTROPIC entry such that all Cij are zero except the following:
C44=g11
C45=g12
C55=g22
C66=g22
Note: This entry is ignored unless the MAT2 MID is greater then 30000000.

MARCMEM — Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If entered, the integer value entered here is the second field on MSC.Marc's SIZING entry (MAXALL) and is the main memory specification for memory in MSC.Marc. This value is entered in MW (the program multiplies it by 1,000,000). For example, if a value of 350 is entered, the number of 350000000 will be placed in the second field of the SIZING entry. This value is not used in MSC.Nastran 2005 r3 and subsequent versions.

MARCMNF — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Controls creation of an MSC.Adams MNF file by MSC.Marc for eigenvalue analysis when MSC.Marc is spawned from SOL 600.

0 MNF file will not be created.

1 MNF file will be created (for MSC.Marc 2003, the MNF file is located in the .t19 file, so PARAM,MARCT19,1 must also be specified).

Note: It is not necessary to use this parameter if the MDMIOUT Bulk Data entry is used to request an MSC.Adams MNF file.

MARCMPCC Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether SOL 600 converts MPCs to stiff beams or not.

0 MPCs are not converted to stiff beams.

1 MPC's are converted to stiff beams.

MARCND99 Integer, default-see below. MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether a set in the MSC.Marc input file to output all nodal quantities will be generated or not. If MARCND99=1, all Case Control nodal output requests must have the PLOT qualifier or the job may fail.

-1 A set named ND999999 will be generated to output all nodes for at least one type of nodal output. This is the default of all Case Control nodal requests do not have (plot).

1 The set will not be generated. If all nodal Case Control requests have (plot) such as DISP(PLOT)=ALL, ACCEL(PLOT)=ALL, etc. the default is 1 even if the parameter is not entered.

MARCNOER Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines action to take when unsupported features are encountered.

MARCNOER=0, the internal MSC.Marc translator will stop and generate FATAL ERRORs if unsupported features in MSC.Marc or in the internal translator are encountered.

MARCNOER=1, if unsupported features are encountered, they are ignored, no FATAL ERROR messages are issued and if requested, MSC.Marc will be executed.

MARCOFFT — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Controls whether Nodal Temperatures are specified at the original or offset grid point or both grid points for beams and shells with offsets.

0 Temperatures are applied both at the original grid point and at the offset grid point.

1 Temperatures are applied at the original grid point only.

2 Temperatures are applied at the offset grid point only.

-1 Temperature loading is not altered in any way from the MD Nastran input.

Note: Processing time can increase significantly if beam or shell offsets are present and param,marcoftt is zero or greater unless PARAM,MOFFOCRE,1 is used.

MARCONLY — Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether to run MD Nastran to completion or not.

MARCONLY=0, runs MD Nastran to completion.

If parameter MARCONLY=1, MD Nastran will be stopped after the IFP and all results are computed by MSC.Marc.

MARCONTF — Character, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

Name — Name of a file name limited to 8 characters (16 characters if param* is used) used in conjunction with one of the CONTINUE options on the SOL 600 statement. If entered this file will be used as the input file for the second MD Nastran execution (after MSC.Marc has finished). If specified, this file will be used instead of automatically creating a file named jid.nast.dat from the original jid.dat input. This option allows more versatility in achieving exactly what is desired in the MD Nastran continuation run input at the expense of additional input data preparation.

MARCOOCC Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MARCOOCC=0, standard (pre-MSC.Nastran 2005 r2) MSC.Marc memory management will be used. If the memory is sufficient the decomposition will be in core. If the memory requirements are too large, an out of core solution will take place.

MARCOOCC=1, an out of core solution will be forced if solvers 0, 4 or 8 are used. This option triggers MSC.Marc parameter, OOC (without any other characters).

MARCOOCC=2, an out of core solution will be forced if solver 0, 4 or 8 are used - available starting with MSC.Marc 2005 r2. This option generates MSC.Marc parameter OOC,0,1.

MARCOPT MD Nastran Implicit Nonlinear (SOL 600) only

Determines which bandwidth optimizer is to be used.

MARCOPT=0, MSC.Marc does not optimize the bandwidth.

MARCOPT=2, Cuthill-McKee bandwidth optimization is used

MARCOPT=5, external user-supplied bandwidth optimization is used

MARCOPT=9, Sloan bandwidth optimization is used. (Default)

MARCOPT=10, minimum degree bandwidth optimization is used (only available with the sparse solver)

MARCOPT=11, Metis nested dissection algorithm (only available with multifrontal direct sparse solver)

MARCOPT=-9999, Set MARCOPT to -9999 if the OPTIMIZE entry is not wanted in the MSC.Marc file for example with use by the iterative solver.

MARCOSET Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether SOL 600 set names will be standard sets or “open sets” for nodes and elements. The standard MSC.Marc sets are:

```
DEFINE, ELEMENT, SET, PR00001
DEFINE, NODE, SET, ND001
```

The MSC.Marc open sets (OSET) are:

```
DEFINE, ELEMENT, OSET, PR00001
DEFINE, NODE, OSET, ND001
```

0 Standard sets are defined.

1 Open sets are defined.

Note: For Parallel (DDM) analyses, it is sometimes necessary to set marcoset=1 for MSC.Marc versions starting with 2005 r3.

MARCOTIM Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only and is mapped to MSC.Marc’s POST 2nd line 9th field.

Determines if MSC.Marc is to be processed at selective or at all output times.

MARCOTIM=0 or 1, MSC.Marc output data will be processed by MD Nastran at all converged output times. This option is similar to INTOUT=YES on the NLPARM entry.

MARCOTIM=2, MSC.Marc output data will be processed by MSC.Marc only at times near 1.0, 2.0, 3.0, etc. For this option, no additional output times will be available in the MSC.Marc .t16 and / or .t19 output files. This option is similar to INTOUT=NO on the NLPARM entry.

MARCOTIM=N, MSC.Marc output data will be processed by MSC.Marc every Nth increment for the .t16 and / or .t19 output files. (N > 2)

MARCOUTR Integer, Default = 1 if MSC.Marc single file parallel input is used, Default = 0 if MSC.Marc multiple file inputs are used, see Note, MD Nastran Implicit Nonlinear (SOL 600) only

Determines how MSC.Marc t16 file results will be handled for SOL 600 parallel processing.

0 Multiple t16 files, one for each domain will be produced.

1 A single t16 file will be produced by MSC.Marc. This option requires MSC.Marc 2005 and the parallel run made using the “single file” input (PARAMARC KIND=0).

Note: Whether single file or multiple MSC.Marc inputs are used for parallel processing is determined by the PARAMARC Bulk Data entry.

MARCPARR Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Controls options for splitting an MSC.Marc file into parts for DDM.

MARCPARR=0, all MSC.Marc files will be created during this run.

MARCPARR=1, MD Nastran will be stopped after the single-processor file has been created and before DDM files are created. If desired, all files may be copied to a backup directory for use with MARCPARR=2.

MARCPARR=2, the parallel files will be created starting with the single processor file created using the MARCPARR=1 option.

MARCPARR=3, same as MARCPARR=2 except the debug option MARCBUG=1 is turned on.

MARCPINN Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

0 If MARCPINN=0, pin flags will be included by created new nodes and appropriate MPC’s by the translator in MD Nastran. This option is available starting with MSC.Nastran 2005 r2.

1 If MARCPINN is 1, pin flags will be ignored and the translator will continue.

2 If MARCPINN is 2, a severe warning will be issued and MSC.Marc will not run.

MARCPLAS — Integer, Default = 3 if there is plasticity in the model. MD Nastran Implicit Nonlinear (SOL 600) only

n — This parameter effects the value of MSC.Marc's PLASTICITY parameter. The value of n can range from 0 to 6 (0 is the same as 3).

Enter 1 for additive decomposition using the mean normal method; small strain formulation.

Enter 2 for additive decomposition using the radial return method; small strain formulation.

Enter 3 for additive decomposition using the mean normal method; large strain formulation using the updated Lagrange procedure.

Enter 4 for additive decomposition using the radial return method; large strain formulation during the updated Lagrange procedure.

Enter 5 for multiplicative decomposition (FeFp) using the radial return method and the three field variational principle; large strain formulation using the updated Lagrange procedure.

Enter 6 for MSC.Marc element type 3 and 26 (plane stress), 18 and 30 (membrane) using multiplicative decomposition with the radial return method; large strain formulation using the updated Lagrange procedure.

MARCPOS — Integer, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether to terminate MSC.Marc if a non-positive definite matrix is attempted.

MARCPOS=0, the run will terminate if a non-positive definite matrix decomposition is encountered.

MARCPOS=1, non-positive definite matrices will be solved.

MARCPOST — Integer, Default, if omitted, =9, MD Nastran Implicit Nonlinear (SOL 600) only

Determines the format for the .t16 and .t19 files.

MARCPOST=-1, the .t16 and .t19 files will be created using whatever version of MSC.Marc is executed.

MARCPOST=0, the .t16 and .t19 files will be created using MSC.Marc 2000 formats.

MARCPOST=1, the .t16 and .t19 files will be created using MSC.Marc K2 formats.

MARCPOST=3, the .t16 and .t19 files will be created using MSC.Marc K3 formats.

MARCPOST=4, the .t16 and .t19 files will be created using MSC.Marc K4 formats.

MARCPOST=5, the .t16 and .t19 files will be created using MSC.Marc K5 formats.

MARCPOST=6, the .t16 and .t19 files will be created using MSC.Marc K6 formats.

MARCPOST=7, the .t16 and .t19 files will be created using MSC.Marc K7 formats.

MARCPOST=8, the .t16 and .t19 files will be created using MSC.Marc K8 formats.

MARCPOST=9, the .t16 and .t19 files will be created using MSC.Marc 2000 formats.

MARCPOST=10, the .t16 and .t19 files will be created using MSC.Marc 2001 formats.

MARCPOST=11, the .t16 and .t19 files will be created using MSC.Marc 2003 formats.

MARCPOST=12, the .t16 and .t19 files will be created using MSC.Marc 2005 formats.

Note: It is suggested that a small test case be executed and tested with your postprocessor to determine what version is necessary for your postprocessor.

MARCPR99 Integer, Default-see below. MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether a set in the MSC.Marc input file to output all elemental quantities will be generated or not. If MARCPR99=1, all Case Control element output requests must have the PLOT qualifier or the job may fail.

-1 A set named PR999999 will be generated to output all elements for at least one type of element output. This is the default if all elemental Case Control requests do not have (plot).

1 The set will not be generated. If all elemental Case Control requests have (plot) such as STRESS(PLOT)=ALL, STRAIN(PLOT)=ALL, etc. The default is 1 even if the parameter is not entered.

MARCPRN MD Nastran Implicit Nonlinear (SOL 600) only

Controls the amount of contact information printed.

For MARCPRN=0, detailed contact information is not printed.

MARCPRN=1, detailed contact information is printed (this is equivalent to MSC.Marc parameter PRINT,2,8).

MARCPRN=2, somewhat less detailed contact information is printed (this is equivalent to MSC.Marc parameter PRINT,2). Print constraint matrices associated with MPC's, RBAR, RBE2, RBE3 and the formable to deformable contact.

MARCPRN=5, MSC.Marc print option. PRINT,5 will be used. Prints messages when changes in contact status occur.

MARCPRN=25, MSC.Marc print options. PRINT,2,5 will be used.

MARCPRN=258, MSC.Marc print options. PRINT,2,5,8 will be used. In addition to 2 and 5, also prints the displacement and reaction forces in the local coordinate system associated with formable to rigid contact.

For other print options, use the MARCIN Bulk Data entry.

MARCPRNG Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether geometry is printed in the MSC.Marc .out file.

0 Most geometry printing is suppressed.

1 All geometry is printed.

MARCPRNH Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether nodal stress and strain output is printed in the MSC.Marc .out file.

0 Nodal, stress and strain printing is suppressed.

1 Nodal, stress and strain printing will occur if MD Nastran Case Control options request it specifically or by default.

MARCPROG Character, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

prg "prg" is the name of a program to be executed instead of MSC.Marc. The program may be any program desired and must already be fully compiled on the computer system being run. prg is limited to 8 characters, however 16 characters can be used if PARAM* is entered. "prg" must be a compiled program, not a script or batch file. The name of the program must be in lower case (if not, it will be converted to lower case).

MARCPTH MD Nastran Implicit Nonlinear (SOL 600) only

Defines the path for MD Nastran Implicit Nonlinear (SOL 600).

MARCPTH=0, MSC.Marc is run from inside MD Nastran using the command: run_marc -jid nasmarc. (Default)

MARCPTH=1, a file named marcrun.pth contains the entire path where the desired MSC.Marc run script (or batch file) is located.

MARCPTH=2, the directory where the MSC.Marc run script is located is on the PATH, and MSC.Marc will be executed using the command: run_marc -jid jid.marc -v no

MARCPTH=3, the complete command line to run MSC.Marc is specified as one line in file marc.pth which must be located in the same directory as the MD Nastran input data. An example would be:
/mycomputer/marc200x/tools/run_marc -jid name.marc -v no

MARCRACC Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

SOL 600 does not support RACC on the RFORCE entry. The default is to fatal a job when RACC is not zero or blank. This parameter may be used to set it to zero and continue the analysis.

0 If RACC is not zero or blank, the job will terminate with an appropriate message.

1 If RACC is not zero or blank, it will be set to zero internally and the job will continue.

2 If RACC is not zero or blank, Coriolis loading and values (C1, C2, C3) will be placed in MSC.Marc ROTATION A 3rd datablock fields 4-6 as follows:
RR=SQRT(R1**2+R2**2+R3**2)
C1=RACC*R1/RR
C2=RACC*R3/RR
C3=RACC*R3/RR

Where RACC, R1, R2, R3 are described on the RFORCE entry.

Note: See PARAM,MARCRCID for related SOL 600 RFORCE uses.

MARCRBAL Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

This parameter is used only for eigenvalue analysis in the MD Nastran-MSC.Marc interface where natural frequencies or buckling modes need to be calculated using the deformed geometry from a nonlinear analysis. The parameter is only necessary if the last nonlinear increment created a non-positive definite matrix. When MARCRBAL=1 is set, the system will be rebalanced and a positive-definite matrix is assured. Do not use this parameter unless it is known that a non-positive definite system occurs just prior to eigenvalue analysis.

MARCRBAR Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only.

Determines how RBAR is treated.

MARCRBAR=0, MD Nastran RBARs will be simulated using MSC.Marc's Servo Link. This option is best only for small incremental deformation and rotations.

MARCRBAR=1, RBARs will be converted to RBE2 with GN=GA, CM=123456, GM1=GB. The MARCRBE2 option specified will then be used to translate the converted RBARs to MSC.Marc.

MARCRBE2 Integer, Default = 3 for Version 2003 and subsequent versions and 1 for prior versions, MD Nastran Implicit Nonlinear (SOL 600) only

ID Determines the type of RBE2 used.

MARCRBE2=0, MD Nastran RBE2s will be simulated using MSC.Marc's Servo Link. This option is best only for small incremental deformations and rotations.

MARCRBE2=1, MD Nastran RBE2s will be simulated using MSC.Marc's TYING type 80 for translation and Servo Link for rotations. This option is capable of larger incremental deformations but requires small rotational increments. The MARCRBE2=1 option is only available if all 6 DOF's are specified in the CM field (4) of the RBE2 entry.

MARCRBE2=2, MD Nastran RBE2s will be simulated using the new RBE2 element introduced into MSC.Marc Version 2003 for a 2D analysis.

MARCRBE2=3, MD Nastran RBE2s will be simulated using the new RBE2 element introduced into MSC.Marc Version 2003 for a 3D analysis.

Note: If RBAR, RROD or RTRPLT elements are found in the model and if MARCRBE2=2 or 3, these elements will be converted to equivalent RBE2's and used with the new MSC.Marc RBE2 element during the MSC.Marc execution (thus providing higher accuracy for large deformations and / or rotations).

MARCRBE3 Integer, Default = 3 for Version 2003 and subsequent versions, 0 for prior versions, MD Nastran Implicit Nonlinear (SOL 600) only

ID Determines the type of RBE3 used.

MARCRBE3=0, MD Nastran RBE3s will be simulated using MSC.Marc's Servo Link. This option is best only for small incremental deformations and rotations (see option 4 for a similar alternative).

MARCRBE3=2, MD Nastran RBE3s will be simulated using the new MSC.Marc RBE3 element introduced into MSC.Marc Version 2003 for a 2D analysis.

MARCRBE3=3, MD Nastran RBE3s will be simulated using the new MSC.Marc RBE3 element introduced into MSC.Marc Version 2003 for a 3D analysis.

MARCRBE4=4, Same as MARCRBE3=0 except that all MPC's due to RBE3 will be placed ahead of all other MPC's. This option might improve the MSC.Marc solution for versions where MSC.Marc has not implemented AUTOMSET logic (those versions prior to MSC.Nastran 2005 r3).

MARCRCID Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
SOL 600 does not fully support CID on the RFORCE entry. The default is to fatal a job when CID is not zero or blank. This parameter may be used to continue the analysis in a manner that can usually emulate the use of CID:

0 If CID is not zero or blank, the job will terminate with an appropriate message

1 If CID is not zero or blank, A, R1, R2, R3 are used in the following manner.

The direction cosines of the rotation angle (N1,N2,N3) are defined as follows:
RR=SQRT(R1**2+R2**2+R3**2)
N1=R1/RR
N2=R2/RR
N3=R3/RR

The magnitude of the angular velocity (V) is defined as
V=A*RR

Note: See PARAM,MARCRACC for related SOL 600 RFORCE uses.

MARCREVR Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Specifies that all rigid surfaces need to be modified.

MARCREVR=0, rigid contact surfaces are correct as entered and no changes are made by the translator.

MARCREVR=1, all rigid surfaces are entered backwards and will be reversed.

MARCSAME Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether SOL 600 runs with multiple subcase having the same LOAD ID in more than one subcase will be processed or not. SOL 600 will usually run under such circumstances but may get the wrong results. **Do not use this parameter if the loading contains enforced displacements or the results may be incorrect.**

0 The job will be aborted before MSC.Marc is spawned with a "Severe Warning" message.

1 The job will run to completion (if there are no other errors) and a standard Warning message will be issued.

It is recommended that if the same loads are to be used in multiple subcases that each subcase have a different LOAD ID. A typical file setup for SOL 600 should be setup in the following manner:

```
SOL 600,NLSTATIC PATH=1 STOP=1
CEND
DISP=ALL
STRESS=ALL
SPC=123
TEMP(INIT)=33
SUBCASE 1
LOAD=100
SUBCASE 2
LOAD=200 $ Do not use LOAD=100 as it was used in subcase 1 even though
$$$$$$$$$$$$$$$$$$$$the loads are the same
TEMP(LOAD)=300
BEGIN BULK
PARAM,MARCSAME,1
LOAD, 100, 1., 1.0, 1000, 1.0, 2000
LOAD, 200, 1., 1.0, 1000, 1.0, 2000
PLOAD4, 1000, 10, 20.0
PLOAD4, 2000, 20, 25.0
(Other Bulk Data entries)
ENDDATA
```

MARCSETT Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MARCSETT=0, the current environment is not printed.

MARCSETT=1, the current environment is printed in the .f06 file. A user program named eodenv.f must be compiled, linked and placed in the input file directory. The contents of eodevn.f resembles the following:

```
Program eodenv
call system("set")
stop
end
```

MARCSINC — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

This parameter controls how often a spline file is written if the spline (analytical contact for deformable bodies) is requested. If this parameter is not entered or if it is 0 or -1, then a file is not written. If N is greater or equal to 1, then every nth time step is written. Spline files have the extension *.mfd.

MARCSIZ3 — Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If entered, the integer value entered here is the third value on MSC.Marc's SIZING entry representing the maximum number of elements.

MARCSIZ4 — Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If entered, the integer value entered here is the fourth value on MSC.Marc's SIZING entry representing the maximum number of grid points.

MARCSIZ5 — Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If entered, the integer value entered here is the fifth value on MSC.Marc's SIZING entry representing the maximum number of constrained degrees-of-freedom.

MARCSIZ6 Integer, Default = Program determines value, MD Nastran Implicit Nonlinear (SOL 600) only

Value If entered, the integer value entered here is the sixth value on MSC.Marc's SIZING entry representing the maximum number of elements in the largest list of distributed loads (the internal MSC.Marc to MD Nastran translator generates these one at a time, so this value is normally 1).

MARCSLHT Integer, Default = 5, MD Nastran Implicit Nonlinear (SOL 600) only

Number of layers through the shell thickness used to integrate shell and beam elements. For linear behavior, N=1 is sufficient. For most plasticity problems, N=5 is adequate. For extremely nonlinear plasticity problems N=11 should be used. SOL 600 requires that N be 5 or larger. If N is entered with a positive value less than 5, SOL 600 will set it to 5. To use values smaller than 5, enter N as a negative number. The absolute value will be used, however MSC.Marc might fail with error code 13.

Note: Use of PARAM,MARCDEF can effect the value of MSC.Marc's SHELL SECT parameter if PARAM,MARCSLHT is not entered. To eliminate SHELL SECT from the MSC.Marc file set N to -9999.

MARCSOLV Integer, Default = 8, MD Nastran Implicit Nonlinear (SOL 600) only

Determines the solver to use for MSC.Marc.

IMARCSOLV=0, the profile direct solver will be used (see MSC.Marc documentation for additional solver details)

MARCSOLV=2, the Sparse Iterative Solver will be used.

MARCSOLV=4, the Direct Sparse Solver will be used.

MARCSOLV=6, a hardware-provided solver will be used.

MARCSOLV=8, a sparse solver similar to the one used by MD Nastran will be used (Default)

MARCSOLV=9, the CASI solver will be used.

Note: If any NLSTRAT entries are entered, the solver type must be specified using the IOLSVER option of NLSTRAT rather than this parameter.

MARCSPCD — Integer, Default = -1, MD Nastran Implicit Nonlinear (SOL 600) only

Controls how enforced displacements using SPC and/or SPCD are applied. This parameter must be entered in the Case Control and may be entered for each subcase. If entered above the subcases, it applies to all subcases not over-ridden by a MARCSPCD parameter with a different value.

MARCSPCD=1, SPC's and SPCD's are applied as total values

MARCSPCD=-1, SPC's and SPCD's are applied as incremental values

Note: -1 was the default for MSC.Nastran 2005.

MARCSTIFF — Real, Default = 1.0 MD Nastran Implicit Nonlinear (SOL 600) only

Time — This parameter specifies what time matrices entered using PARAM,MARCFILi will be used in a MD Nastran solution. The file may contain matrices at several times, but only the matrices specified by the parameter will be used. This parameter is not usually used, MRMTXNAM,NAME is used instead.

MARCSUMY — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines if the summary of maximum values is to be printed.

MARCSUMY=0, a summary of maximum displacements, stresses and strains will be printed in the MSC.Marc output file.

MARCSUMY=-1, the summary of maximum values is not output.

MARCT16 — Integer, Default = 2, MD Nastran Implicit Nonlinear (SOL 600) only

Controls generation of a MSC.Marc t16 file. MARCT16=0, MSC.Marc does not generate a .t16 output file.

MARCT16=-1 or 0 does not generate a t16 file.

Parameter omitted generates a t16 file. All entries are controlled by the MARCOUT Bulk Data entry, or if MARCOUT is not specified, by the default shown in option 2 below.

MARCT16=1 generates a .t16 output file with the following post codes (default until version 2005 r2):

11 11,1 11,N 12 12,1 12,N 13 13,1 13,N 14 14,1 14,N
15, 15,1 15,N 16 16,1 16,N 17 17,1 17,N 18 18,1 18,N
7 7,1 7,N 27 27,1 27,N 301 301,1 301,N 321 321,1 321,N
341 341,1 341,N 401 401,1 401,N
Nodal: 1, 2, 34, 35, 36, 37, 38, 39

MARCT16=2 generates a .t16 output file with the following post codes (default starting version 2005 r2):

301, 301,1 301,N 341 341,1 341,N 47
Nodal: 1, 2, 34, 35, 36, 37, 38, 39

MARCT16=3, generates a .t16 output file with the following post codes (op2, xdb, f06, punch results cannot be made using this option):

301, 341 47
Nodal: 1, 2, 5, 35, 37, 38, 39

MARCT16=4, generates a .t16 output file with the following post codes (op2, xdb, f06, punch results cannot be made using this option):

301 341 47
Nodal: 1

This entry is used as an easy way to control which results are placed on MSC.Marc's op2 file. All entries can be overridden using the MARCOUT Bulk Data entry. For MARCOUT values of 1 and larger, MARCOUT should be omitted for the input file.

The frequency of output is controlled by the NLPARM or TSTEPNL Bulk Data entries (variables INTOUT and NO respectively).

Items such as 341,1 and 341,N designate stresses at the bottom and top surfaces (for applicable elements). Items such as 341 designate stress at mid-thickness.

Consult MSC.Marc Volume C documentation for the meaning of the above blocks. Option 1 provides most of the structural output anyone might want, option 2 provides total strain, Cauchy stress, displacement and contact information at the top, center and bottom of surfaces. Option 3 provides the information of option 2 but only at the center (not at the top and bottom).

It is necessary to generate a t16 file in order to produce op2, xdb, f06 or punch results.

Op2, xdb, punch and f06 results can only be created using option 1 and 2 although option 0 can also be used if the selected outputs are the same as option 1 or 2.

MARCT19 Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MARCT19=0, MSC.Marc does not generate a .t19 output file.

MARCT19=1, generates a .t19 output file.

MARCTABL Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines if contact table is to be generated.

MARCTABL=0, contact tables will be generated for the main MSC.Marc input (phase 0) and for each subcase if specified by the user (Default). When marctable=0, each subcase may have a BCONTACT Case Control command and a matching BCTABLE ID entry. In addition, MSC.Marc's "Phase 0" entry is supported by entering a BCTABLE with an ID of zero (or 1,000,000). Each separate BCTABLE will reference the BCBODY entries defined which, in turn, reference BSURF entries.

If MARCTABL=1, no contact tables will be generated and all contact bodies (if any) will be placed in the main input data section. Contact will thus be the same for all subcases. When MARCTABL=1, there must only be one BCTABLE entry in the file. There must only be one BCONTACT command in the Case Control and it must be above all subcases. The BCONTACT and BCTABLE entry must have the same ID. The BCTABLE entry can reference several BCBODY entries which, in turn, reference BSURF entries.

MARCTEDF Character*8, no Default, MD Nastran Implicit Nonlinear (SOL 600) only
Enter the MSC.Marc nthcnt file name without extension. Use this option only if PARAM,MARCTEDN,1 is entered (the file name without extension is limited to 8 characters). The characters ".nthcnt" will automatically be appended to the name specified.

MARCTEDN Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether a thermal contact analysis will use the MSC.Marc jid.nthcnt file generated by this run or use one input by the user.

0 Analysis uses the MSC.Marc jid.marc.nthcnt file generated in this run.

1 Analysis uses a MSC.Marc nthcnt file generated by the user or in a previous run.

Note: If MARCTEDN=1, PARAM,MARCTEDF below must be entered to specify the file name.

MARCTEMP Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

MARCTEMP=0, the scratch files produced by MSC.Marc will be in the same directory as the MD Nastran input file.

MARCTEMP=1, the scratch files produced by MSC.Marc will be in the same directory as the MD Nastran scratch files.

Note: The MSC.Marc scratch files cannot be split.

MARCTIEC Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

MARCTIEC=1, Transient Time Integration Error Check for MSC.Marc's AUTO STEP method. A value of 1 turns the check on.

MARCTIEC=0, a value of 0 turns the check off. Turn the check off to match MSC.Marc results for version prior to MSC.Marc 2003 r1.

MARCTOL Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines the method of convergence tolerance.

If parameter MARCTOL=0, convergence tolerances are based on residuals (loads). If parameters MARCTOL and MARCTVL are not entered, the tolerances are determined by TSTEPNL and / or NLPARM Bulk Data entries. However, parameters MARCTOL and MARCTVL provide extra control over these convergence tolerances particularly in the case where TSTEPNL or NLPARM specify more than one convergence type (such as load and energy).

MARCTOL=1, convergence tolerances are based on displacement.

MARCTOL=2, convergence tolerances are based on strain energy.

MARCTOL=4, convergence is achieved when either residual or displacement satisfies the criteria.

MARCTOL=5, convergence is achieved when both residual and displacement satisfies the criteria.

MARCTVL — Real, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If parameter MARCTVL is entered, it must have a real value. The value entered is the convergence tolerance used by MSC.Marc (see PARAM,**MARCTOL**). If parameters MARCTOL and MARCTVL are not entered, the tolerances are determined by TSTEPNL and/or NLPARM Bulk Data entries. However, parameters MARCTOL and MARCTVL provide extra control over these convergence tolerances particularly in the case where TSTEPNL or NLPARM specify more than one convergence type (such as load and energy).

MARCUSUB — Character, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

chr — chr is the name of a “user subroutine” to be included in the MSC.Marc run. chr is limited to 8 characters without the .f extension. This file must be located in the same directory as the MD Nastran input data. The subroutine name must have all lowercase letters. chr should be in lower case to prevent confusion. (MD Nastran will convert the file name to uppercase, but it will be reconverted to lower case when MSC.Marc is spawned.) Any user subroutine available to MSC.Marc may be specified. Multiple user subroutines must be combined into one file.

Restriction: The computer must have a Fortran compiler and Linker and the Fortran compiler must be the same as used to create the original MSC.Marc executable (see the MSC.Marc installation manual). The MSC.Marc input file does not as yet call out user subroutines, so manual editing of the MSC.Marc input file may be necessary in some cases to invoke them. Existing regular MSC.Marc subroutines can be modified and handled in the same manner if available to you.

Note: If more than one user subroutine is required, all should be combined into one file before execution.

MARCVERS — Integer, MD Nastran Implicit Nonlinear (SOL 600) only

ID — Default to latest version available. The version of MSC.Marc (or MSC.Marc contact subroutines) that are used in the analysis. Valid values for ID are 2000, 2001, 2003 and 2005. If this parameter is omitted, the defaults are 2005 for MD Nastran Implicit Nonlinear (SOL 600). It should be noted that not all options are available for ID less then 2005.

MARCWDIS — Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether MSC.Marc parameter section DIST LOADS Is written or not. If any of the three values for DIST LOADS are entered (see PARAM,MARCDIS2 PARAM,MARCDIS3 PARAM,MARCDIS4) it will be written. If PARAM,MARCWDIS,1 is entered, it will be written. If PARAM,MARCWDIS,-1 is entered, it will not be written.

Caution: MD Nastran cannot estimate these values very well and produces overly conservative numbers that sometimes leads to failure of the MSC.Marc run due to lack of memory. We suggest that the user should use this parameter sparingly and enter the MARCDIS2, MARCDIS3 and MARCDIS4 values for best MSC.Marc memory usage.

MARELSTO — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether MSC.Marc's parameter ELSTO will be created.

-1 ELSTO will not be created.

0 ELSTO will be created only for large models as determined by MD Nastran. The value of ELSTO will be 40960 except for parallel runs (where a PARAMARC entry exists in the input file) in which case ELSTO will not be created.

>0 ELSTO will be created with the value specified used as the MSC.Marc ELSTO parameter whether or not the run uses parallel processing.

MARGPFEL Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Used to determine if MSC.Marc GRID FORCE output will occur by element, by node, or both ways.

-1 Output is by element only.

0 Output is by node only.

1 Output is by both element and node.

Note:

1. This parameter effects the contents of the MSC.Marc jid.marc.grd file. Option 0 or 1 is required for GPFORCE output in op2, xdb, punch, and / or f06 files.
2. Use of this parameter requires MD Nastran and MSC.Marc 2005 r3 and beyond.
3. This parameter can be set in rc files.

MARGPFOR Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Used to determine whether GPFORCE is active for SOL 600. A Case Control request GPFORCE must be specified to obtain grid point forces (see note 4). The grid point forces are output from MSC.Marc on a file named jid.marc.grd. The t16op2 program reads this file, puts the data on the f11 file along with displacements, stresses, etc. After t16op2 finishes, the f11 file is brought into DBALL, from which DMAP generated on the fly can produce op2, xdb, punch, or f06 output.

-1 GPFORCE is ignored ((this is what happens for all version prior to 2005 r3).

0 GPFORCE is output for the last time in the last subcase only.

1 GPFORCE is output at all times given in the .sts file.

N GPFORCE is output for times 1, 1+N, 1+2N, etc.

Note:

1. This parameter can be set in rc files.
2. Use of this parameter requires MD Nastran and MSC.Marc r3 and beyond.
3. GPFORCE in MSC.Marc is a new development for the MSC.Nastran r3 release. Output is available only in the new MSC.Marc jid.marc.gid file.
4. The Case Control GPFORCE request must be above all subcases, or the same within all subcases.
5. The jid.marc.grd file can become very large if option 0 or N is not used.

MARHEATM Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether a file named heatm.rc is necessary to run the second phase of SOL 600 heat transfer initial contact job.

15 heatm.rc is not required. Defaults will be used. The defaults are scr=yes batch=no mem=80mw

16 A heatm.rc file will be supplied by the user in the same directory as the original MD Nastran input file. The heatm.rc can contain any information used by other rc files except that batch=no. If the original MD Nastran input file is named jid.dat (or jid.bdf) and out=jid is specified, the final output will be in files such as jid.f06, jid.op2, jid.xdb. If out=jid is not specified the final output will be in files such as jid.nast.f06, jid.nast.op2, jid.nast.xdb.

17 heatm.rc is not required. Defaults will be used. The defaults are scr=yes batch=no mem=80mw

18 A heatm.rc file will be supplied by the user in the same directory as the original MD Nastran input file. The heatm.rc file can contain any information used by other rc files except that batch=no is required.

MARIBOOC Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MARIBOOC=0, incremental backup data will be stored in memory.

MARIBOOC=1, for large problems, incremental backup data will be stored on disk. This option triggers MSC.Marc's parameter IBOOC.

MARMPCHK Integer, Default = -1, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether MSC.Marc parameter MPC-CHECK is written.

-1 MPC-CHECK is not written.

N MPC-CHECK,N is written. Please consult MSC.Marc Documentation regarding the meaning of MPC-CHECK and the value of N to use (currently MPC-CHECK,2 is a popular option.

Note: This parameter can be set in rc files.

MARMPICH Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether MPICH for Marc parallel processing on Windows 32-bit systems has been installed or not.

0 MPICH has been installed

1 MPICH has not been installed and will be installed as part of this MD Nastran run. The installation requires a "data file" in the same directory as the MD Nastran input named mpich. dat with the following three lines starting in column 1:
PC Login Name (Name you use to login to the PC)
Domain Name (if not part of a domain, enter local)
Password (the password you use to login to your PC)

This needs to be done only once unless the MD Nastran or MSC.Marc versions are changed, the login name, domain or password is changed. All subsequent jobs should use MARMPICH=0 or omit this parameter.

MARMTLCK Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether a check of various property-material combinations for SOL 600 will be made or not.

0 The checks will not be made.

1 Check will be made. These take extra computer processing time and for most models are not required. The user should turn on these check if he is in doubt if any property-type of material combinations entered into the model may be in error. Current check made are for the following illegal combination.
PSOLID / MATHE (model should b e PLSOLID / MATHE)
PSOLID / MATHP (model should be PLSOLID / MATHP)

MARPLANE Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

For composite structures described using PCOMP, together with CQUAD4, CQUADR, etc., it is not possible to tell whether a standard 3D shell or a plane strain shell has been modeled. If MARPLANE is set to 1, such composite models will be assumed to be plane strain (as if a PLPLANE property had been entered rather than PCOMP).

MARNOSET Character, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

Name If entered, this parameter will not write out a set with the specified name. This is useful, when portion of the model specify sets that are not actually used in the present analysis. Up to 20 of these can be specified.

MAROFSET Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only
Determines how beam and shell offsets are applied.

0 Extra grids and rigid elements will be created to model the offsets (this was the only capability available until MSC.Nastran 2005 r2.

1 MSC.Marc will automatically handle offsets for beam and shell elements. No extra grids or elements will be created. The offsets will be found in MSC.Marc's GEOMETRY data.

2 MSC.Marc will automatically handle offsets for beam elements only.

3 MSC.Marc will automatically handle offsets for shell elements only.

Note: If MAROFSET is 1 or 2, the beam orientation can be specified using the CBAR/CBEAM "BIT" flag. It is suggested that only combinations, GGG or BGG be used.

MARUPDAT Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

If this parameter is omitted, the Updated Lagrange method will be used if plasticity is involved. Ensure that all elements in the analysis are capable of using the Updated Lagrange method. If not, enter PARAM,MARUPDAT,-1. The Updated Lagrange Method is more accurate for many problems and also runs faster for some problems.

MARUPDATE=-1, the Total Lagrange solution procedure will be used when MSC.Marc is executed from MD Nastran.

MARUPDAT=1, the Updated Lagrange solution procedure will be used when MSC.Marc is executed from MD Nastran. This corresponds to Marc parameter update.

MARUPDAT=2, the updated Lagrange solution with large rotations for beam elements. This corresponds to Marc parameter (see Marc VOL C documentation for details): update,0,2

MARUPDAT=3, the following Marc parameter will be set, in which LARGE DISP need not be specified elsewhere (see Marc VOL C documentation for details): update,0,2,1

MARCUPDAT=4, the following Marc parameter will be set, in which LARGE DISP need not be specified elsewhere. (see Marc VOL C documentation for details): update,0,,1

MARCUPDAT=5, the following Marc parameter will be set (see marc VOL C documentation for details): update,0,1

MAUTOSPC Integer, Default = -1, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether Marc AUTOSPC will be added (this is considered in beta test for MSC.Nastran 2005 r3.

-1 Do not add AUTOSPC

1 Add AUTOSPC to Marc's parameter's section. It will remain throughout the run.

2000 Turn on AUTOSPC by adding the integer 1 to Marc's model definition SOLVER option

2001 Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 1

2002 Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 2

.....

2999 Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 999

3000 Turn off AUTOSPC by adding the integer 1 to Marc's model definition SOLVER option

3001 Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 1

3002 Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 2

.....

3999 Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 999

Note: Only options -1 and 1 are available in MSC.Nastran 2005 r3.

MAXLP Default = 5

MAXLP specifies the maximum number of iterations for element relaxation and material point subincrement processes in SOLs 129 and 159. MAXLP is 10 in SOLs 106 and 153 and cannot be changed by the user.

MAXRATIO Default = 1.E7

The ratios of terms on the diagonal of the stiffness matrix to the corresponding terms on the diagonal of the triangular factor are computed. If, for any row, this ratio is greater than MAXRATIO, the matrix will be considered to be nearly singular (having mechanisms). If any diagonal terms of the factor are negative, the stiffness matrix is considered implausible (non-positive definite). The ratios greater than MAXRATIO and less than zero and their associated external grid identities will be printed out. The program will then take appropriate action as directed by the parameter BAILOUT.

By default, in the superelement solution sequences the program will terminate processing for that superelement. A negative value for BAILOUT directs the program to continue processing the superelement. Although forcing the program to continue with near-singularities is a useful modeling checkout technique, it may lead to solutions of poor quality or fatal messages later in the run. It is recommended that the default values be used for production runs. A related parameter is ERROR.

The value -1 of BAILOUT causes the program to continue processing with near singularities and a zero value will cause the program to exit if near singularities are detected.

In SOLs 101 through 200 when PARAM,CHECKOUT,YES is specified, PARAM,MAXRATIO sets the tolerance for detecting multipoint constraint equations with poor linear independence. (See "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.)

BAILOUT and MAXRATIO may be specified in the Case Control Section in order to provide different values between superelements but not between different boundary conditions.

MCONTACT Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

N This parameter allows the user to create a set of contact MPC's which remain constant throughout the analysis. A second MSC.Marc analysis is spawned using the contact MPC's and the original contact definitions are removed. The value of N is used to determine when to pick the contact MPC's. For N=0, MPC's are determined during MSC.Marc's increment 0. For N>0, MPC's are determined at the end of increment N. For N<0, the first set of contact MPC's encountered in the MSC.Marc run after increment zero will be used and the initial MSC.Marc run will execute for abs(N) increments.

This option is useful in certain circumstances. For example, spot welds where the weld beads are modeled as solid elements and placed between plate elements and the connections occur only due to contact, or to connect two parts with mesh mismatches using contact. Use of this parameter causes MD Nastran to spawn MSC.Marc once to determine the requested contact MPC's, and then to either spawn MSC.Marc again with contact definitions removed and the contact MPC's active or to spawn another MD Nastran nonlinear run using the contact MPC's. Whether to spawn MSC.Marc or MD Nastran for the second run is determined by PARAM,RCONTACT. If PARAM,RCONTACT is omitted but PARAM,MCONTACT is entered, MSC.Marc will be spawned for the second run. For MSC.Nastran 2005 r2, N must equal 0.

MDUMLOAD Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MDUMLOAD=0, for subcases with no applied loads and enforced displacement or velocity of rigid contact bodies, a small magnitude dummy load will be created. Without these dummy loads, MD Nastran becomes confused and does not produce the correct contact information. If there is only one subcase, the dummy loads are not necessary.

MDUMLOAD=1, dummy loads will not be produced for any subcase and rigid contact with enforced motion may be incorrectly described.

MECHFIL Default = 1.E-6

Criterion for discarding massless mechanism modes with small generalized mass. A smaller value will result in more marginal constraint modes being retained.

MECHFIX Default = AUTO

Control for fixing the massless mechanism problem. The new capability is provided automatically for the default of this parameter, listed above. The new capability is executed only when the eigensolution does not provide answers because of symptoms consistent with the presence of massless mechanisms. If MECHFIX is set to YES, the constraint modes are removed before attempting an eigensolution. When set to NO, the new capability is blocked, and the eigensolution uses the pre MSC.Nastran 2001 rules, i.e., three failed shifts and a fatal exit. See also "MMFIL" on page 768.

MECHPRT Default = NO

For SOL 103 only, if massless mechanisms are found the constraint modes are printed with a format similar to eigenvectors when this parameter is set to YES. They are labeled CONSTRAINT MODES, and are numbered sequentially. Grid points with only zero values in a mode are not printed. This parameter should be used when performing initial checkout of a model and a goal is to remove all massless mechanisms before starting production analysis. The number of each "mode" matches the corresponding GID,C pair in the high ratio message. If there are many (thousands) of such modes the output file will be large. There is no method to plot these shapes at present.

MESH Default = NO

If MESH=YES is specified, then a summary of the shading status and the subelement mesh for each CHBDYi element referencing a VIEW Bulk Data entry is printed.

METHCMRS Default = 0

In dynamic analysis (SOLs 107, 108, 109, 110, 111, 112, 145, 146, and 200), METHCMRS specifies the set identification number of an EIGR or EIGRL entry to be used in the calculation of the normal modes on the v-set of the residual structure.

By default, the residual structure v-set normal modes will be computed based on the METHOD Case Control command selection as long as q-set is present.

MEXTRNOD Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether extra grids will be added to SOL 600 parallel analyses.

0 Extra grids will not be added.

1 Extra grids will be added so that all grids from 1 to the highest grid are defined this was necessary for certain version of MSC.Marc prior to the 2005 version. All extra grids that are added have coordinates of 0.0 in all three directions.

MFEA5701 Integer, Default = 1,MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether feature 5701 is written to the MSC.Marc file. When this feature is set to 1, it disables default rotation checking set to 0.001, which was initially set for early versions of MSC.Marc when RBE2 and similar elements were added because the 0.001 value proved to be responsible for convergence problems. For models that need rotation checking, you can enter the value using the NLSTRAT variable RLROTT. We recommend always using MFEA5701,1 to turn off the default 0.001 checking value in MSC.Marc whether or not the value is entered using NLSTRAT RLROTT.

0 Rigid rotation of 0.001 is turned on.

1 Rigid rotation of 0.001 is turned off.

MHEATSHL Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether a membrane or thick shell element formulation is used for heat transfer. This parameter can be overridden by individual PSHELL entries. In the current release, the membrane elements should not be used for thermal contact analysis.

-1 All “quad” elements in the model will have membrane capability regardless of MIDi values on the PSHELL entries.

0 All “quad” elements in the model will have membrane capability.

1 All “quad” elements in the model will have thick shell capability.

2 Shells use 2 dof per node, linear variation of temperature through thickness

3 Shells use 3 dof per node, quadratic variation of temperature through thickness

4 Shells use n+1 dof per node, linear variation of temperature through each layer.

5 Shells use 2*n+1 dof per node with quadratic variation of temperature through each layer.

Note: Membrane capability in heat transfer means that the temperature is constant throughout the thickness, The MHEATSHL=0 option can be overridden by entering a non-blank value for MID2, MID3 and/or MID4 on an applicable PSHELL entry in which case the MHEATSHL=1 option will be used if no MHEATSHL parameter is entered.

MHEATUNT Integer, Default = 2, MD Nastran Implicit Nonlinear (SOL 600) only

Specifies the units for heat transfer using SOL 600.

0 SI mm units used

1 SI m units used

2 US units used

Note: This parameter is used by MSC.Marc's ISIOTROPIC (heat transfer) third datablock, fourth field.

MHOUBOLT Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

If MHOUBOLT=0, SOL 600 transient dynamics will use the single step Houbolt numerical integration method.

MHOUBOLT=1, SOL 600 transient dynamics will use the Newmarc Beta numerical integration method.

MHOUBOLT=2, SOL 600 transient dynamics will use the standard Houbolt numerical integration method.

MINIGOA Default = No

Allows for the reduction in the amount of disk space used when using superelements. When this parameter is set to YES, the transformation matrix GOA will contain terms only for the degrees-of-freedom in the U5 (USET, USET1, SEUSET, SEUSET1) set. This can allow for a significant reduction in the amount of disk space used by the database. The limitation of using this approach is that data recovery will be available only for these degrees-of-freedom and elements connected to them.

MINRECCC Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

N Minimum number of iterations per load step. This is the same as MINREC on the NLSTRAT entry. If no other NLSTRAT values are entered, it is easier to enter this parameter. The value can range from 0 to 9. For certain problems, the value should be 2 or greater or accuracy will be poor.

MLSTRAIN Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only
Corresponds to MSC.Marc's "LARGE STRA" parameter

0 Not used.

1 Automatically selects the best options for a large strain analysis based on the element type (see table below).

Note: This parameter is active for MD Nastran and MSC.Marc versions of 2005 r3 or beyond.

Element Type / Material Model	1-Dimensional	Plane Stress or Membranes or Shell Elements	Plane Strain or Alisymmetric, or 3-Dimensional Displacement Form	Plane Strain or Axisymmetric, or 3-Dimensional Hermann Form
Conventional elastic-plastic	Updated Lagrange additive plkasticity; no finite strain	Updated Lagrange additive plkasticity; includes finite strain	Updated Lagrange additive plkasticity; includes finite strain utilized constant strain	Updated Lagrange multiplicative plasticity; includes finite strain
Mooney, Ogden, Gent, or Arruda-Boyce	Total Lagrange	Total Lagrange	Updated Lagrange	Updated Lagrange
Foam	Total Lagrange	Total Lagrange	Updated Lagrange	Updated Lagrange; incompressibility neglected

MMAT2ANI Integer, Default = 2, MD Nastran Implicit Nonlinear (SOL 600) only

Determines how MAT2 will be mapped to MSC.Marc

0 MAT2 will be mapped to MSC.Marc's ORTHOTROPIC option.

1 MAT2 will be mapped to MSC.Marc's ANISOTROPIC option.

2 MAT2 will be mapped as explained in the following note.

Note: The default, MMAT2ANI=2, maps MAT2 to MSC.Marc's ORTHOTROPIC option if G13 and G23 are both zero or blank and to MSC.Marc's ANISOTROPIC option if G13 and / or G23 are non-zero.

MMFIL Default = 1.e-10

Filter value used to distinguish between massless mechanism modes and rigid body modes. A smaller value may discard rigid body modes. The default value has been effective on all problems solved to date.

MODACC

Default = -1

MODACC = 0 selects the mode acceleration method for data recovery in dynamic analysis. See "**Formulation of Dynamic Equations in SubDMAP GMA**" on page 429 of the *MSC.Nastran Reference Guide* for further discussion. If PARAM,SPARSEDR,NO is specified, then PARAM,DDRMM,-1 must also be specified.

MODACC = 1 is the same as MODACC = 0 except if SUPORTi entries are present then the displacements are corrected for rigid body motion and the local flexibility at the SUPORTi degrees-of-freedom.

MODACC $\geq$ 0 is not recommended for use in hydroelastic problems.

MODEL

Default = 0

This parameter also allows several models to be stored in the same graphics database created by PARAM,POST,0.

MOFFCORE

Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

Determines how memory for PARAM,MARCOFFFT above is to be allocated (for increased speed).

-1 Additional memory is not allocated.
1 Additional memory is allocated if available.

MOP2TITL

Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only
Determines how titles are placed on the 146 word record for op2 output records generated by SOL 600.

1 Standard titles will not be placed, however titles of the form "CQUAD4 STRESS FROM MARC", and similar titles for other element types, strains, displacements, etc. will be written. This option is useful for certain postprocessors.

-1 Standard MD Nastran title, subtitles will be placed on op2 files generated by SOL 600. This option is useful for postprocessors that require SOL 600 op2 data to be exactly in the same format as that generated by other MD Nastran solution sequences.

Note: The SOL 600 op2 file follows that of SOL 109 as closely as possible.

MPCX Default = 0

See OLDSEQ.

MRAFFLOR Integer, Default N = 0, MD Nastran Implicit Nonlinear (SOL 600) only

N If N=0, a new AF_flowmat file containing temperature-dependent stress-strain curves will be generated during the current MD Nastran execution and also used in the spawned MSC.Marc run. If N=1, an existing AF_flowmat file will be used. The name of the file is always determined by the value of PARAM,MRAFFLOW, but PARAM,MRAFFLOT determines if other characters are added.

MRAFFLOT Integer, Default N = 0, MD Nastran Implicit Nonlinear (SOL 600) only

N If N=0, the file name as specified using PARAM,MRAFFLOW,Name will be used with no changes except that all characters will be in lower case and the extension ".mat" will be added. If N=1, the characters "asm_" will be added at the beginning of Name, the first character of Name will be upper case (the other characters of Name will be lower case) and the extension ".mat" will be added. This will make the AF_flowmat file name compatible with many names in MSC.Marc's AF_flowmat directory.

MRAFFLOW	Character, no Default, MD Nastran Implicit Nonlinear (SOL 600) only
Name	Name of a file containing temperature dependent stress versus plastic strain curves in MSC.Marc's AF_flowmat format. This file can be generated from the current MD Nastran run using TABLEST and TABLES1 entries or a pre-existing file can be used depending on the value of PARAM,MRAFFLOR. The extension ".mat" will be added to Name. If this is a new file, it will be saved in the directory from which the MD Nastran execution is submitted. If a pre-existing file is to be used, it can either be located in the directory where the MD Nastran execution is submitted or in the MSC.Marc AF_flowmat directory.
MRALIAS ID	Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only. This parameter is not usually used.
MALIAS02 MALIAS03, etc.	The purpose of the parameter is to map the MSC.Marc element type selected by the internal MSC.Marc translator in MD Nastran to a different type. For example, if the element type 75 for CQUAD4 is normally used, a mapping to MSC.Marc element type 139 could be made. ID is a 6 digit number. The left 3 digits are the element type normally selected by the translator and the right 3 digits are the element type to be mapped. In the above example, element type 75 is to be mapped to 139. The user would enter ID=075139. If element type 165 is to be mapped to element type 1 (which is not a real case), ID=165001. Consult MSC.Marc Volume B for a list of elements and their meaning. The user is responsible for ensuring that the mapping selected is proper. There is a limit of 18 aliases that may be entered in any model. Since MD Nastran can only accept one parameter with a given name, the second alias should be named PARAM,MALIAS02 and the third PARAM,MALIAS03, etc. All original element types mapped must actually exist in the model.

Note: If you use one of the MRALIAS parameters, certain "parameters" in the MSC.Marc file may no longer be correct. For example, an element originally capable of using the updated Lagrange method may be aliased to one that must use the total Lagrange method. Such conditions are not checked by the translator when you use alias and you will need to make modifications to the MSC.Marc input file yourself to reflect them. To resolve this use PARAM,MLSTRAIN.

MRALLOCG Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

The value entered here is the amount of memory (MB) allocated for general MSC.Marc memory when MSC.Marc is spawned from MD Nastran. It specifies the initial allocation of "general memory". This is used for storing element stiffness matrices and part or all of the matrix solver workspace among other things. Please note that element data like stresses and strains are not part of the general memory. Solvers 6, 8, and 9 use the main part of the workspace in separate memory. Initial allocation of the general memory can be used for avoiding reallocation (increase of the workspace). For parallel processing the amount specified is the total for the job. It is divided by the number of domains used.

MRALLOCS Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

The value entered here is the amount of memory (MB) allocated for MSC.Marc solver memory when MSC.Marc is spawned from MD Nastran. It specifies the initial allocation of memory for solver 8. By giving a value that is more than the maximum used during the run, one avoids that the solver workspace is increased (reallocated). This can be particularly useful for large contact jobs, where additional memory may be allocated due to contact. If the given workspace is less than what is needed, it is automatically increased. This option is only for use with solver type 8. No check is done to see if solver type 8 is used in the job. For parallel processing the amount specified is the total for the job. It is divided by the number of domains used.

Note: The above features are only used for certain problems, even though all are included with the default option, they have not effect for problems that do not use them. For example, feature,5801 has no effect on models that do not use MSC.Marc element type 140, feature 601 has no effect on models that do not have contact, etc.

MRBEAMB — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MRBEAMB=0, write equivalent radius for all beams (see PARAM,BEAMBEA) whether beam-beam contact is anticipated or not. The equivalent radius is the 7th field of MSC.Marc's GEOMETRY values for beam type elements.

MRBEAMB=-1, do not write equivalent radius (7th field is blank). This might be necessary for versions of MSC.Marc earlier than 2003.

MRBEPARM, — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

IJK — Provides settings for MSC.Marc's RBE parameter. If PARAM,MRBEPARM parameter is entered, MSC.Marc's RBE parameter will be set using this IJK If PARAM<MRBEAMPM is not entered, PARAM,MARCRBE2 or PARAM,MARCRBE3 can be used to set MSC.Marc's RBE parameter.

IJK IJK is a combination of three variables for example 311.

Descriptions for individual entries are as follows:

I Enter 3 or 6 to control the number of degrees-of-freedom. For the dependent grid (reference grid) of each rbe2 or rbe3. The independent grids can have 3 or 6 dof and can be different than what is specified by I.

J Enter 1 to use large displacement formulation of rbe2. Enter 3 to deactivate automatic convergence test for rbe2.

K Enter 1 to use large displacement formulation of rbe3. Enter 2 to activate non-normalized rotation constraint coefficient for rbe3.

Note: If IJK values other than specified above are entered, IJK will be set to zero and the parameter will not be used. This parameter should not be entered unless there are rbe2's or rbe3's in the model and they are to be used as such in MSC.Marc (rather than mpc or stiff beams).

MRBIGMEM — Integer, Default N = 0, MD Nastran Implicit Nonlinear (SOL 600) only

N — If N=0, memory allocations during loads translation phases are sized for computers with limited memory and swap space (paging space). Some large problems and / or unusual problems may not run. If this happens, use a newer modern computer with lots of memory and disk space (and lots of swap space or gaping space) and set N=1. Larger memory allocations will then be available. This parameter is not usually required unless the available memory is extremely small.

MRBUKMTH — Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

MRBUKMTH=1, buckling modes will be computed using the Inverse Power method.

MRBUKMTH=2, buckling modes will be computed using the Lanczos method. Matrices must be positive-definite for this option.

MRBUKMTH=3, use Lanczos if EIGRL is specified, Inverse Power if EIGB is specified.

Note: MRBUKMTH must be specified if EIGR or EIGRL is used for buckling unless SOL 600,105 is the solution sequence. An alternative is to use ANALYSIS=BUCK. In SOL 600, it is not possible to compute natural frequencies and buckling modes in the same run. If ANALYSIS=BUCK is specified anywhere in the Case Control and if PARAM,MRBUKMTH is omitted, param,mrbukmth=3 will be set automatically.

MRC2DADD — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Allows an offset to be added to all coordinates for 2D analyses so that X and Y will always be positive.

0 Offsets will not be added.

1 Offsets will be determined so that all MSC.Marc X and Y coordinates are positive (will exceed 0.1).

MRCOMPOS Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Specifies whether a dummy PCOMP entry has been input for solid elements which are not actually composites. In the versions up to 2005 r2, this was necessary if a coordinate system for PSOLID was specified.

0 Extra dummy PCOMP entries have not been added.

1 Extra dummy PCOMP entries have been added.

Note: The addition of dummy PCOMP entries is not necessary for versions after 2005 r2.

MRCONRES Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

For restart runs, MSC.Marc will produce “continuous” results files (t16/t19) which contain the results of the original run(s) as well as the results of the restart run. If MD Nastran postprocessing is requested to generate op2, xdb, etc. files, they will also contain the results from prior runs as well as the restart run.

MRCONRES=1, output results files will contain the results of the restart run only.

MRCONVER Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines version of MSC.Marc to use for CONTACT and CONTACT TABLE for structural analysis.

0 Uses enhanced MSC.Marc version 9 (MSC.Marc 2001 with added fields)

11 Users MSC.Marc version 11 (MSC.Marc 2005)

Note: This parameter only affects contact and only options 0 and 11 are available. It is not needed unless BIAS is needed in one or more contact tables. This parameter can be set in an RC file.

MRDISCMB Integer, Default = 0 without gravity, =1 with gravity, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether distributed loads, such as pressure, are combined in the MSC.Marc input file into one list if the magnitude of the distributed loads are the same. This parameter applies to MSC.Nastran 2005 only. For previous MSC.Nastran versions, MRDISCMB was 1. Option 0 save MSC.Marc memory and processing time.

0 Distributed loads with the same magnitude are combined.

1 All distributed loads are input individually.

Caution: This parameter should be used with caution. For multiple load cases, MRDISCMB=0 may produce the wrong results particularly if gravity loading is present.

MREIGMTH Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

MREIGMTH=1, eigenvalue analysis will be done in MSC.Marc using the Lanczos method.

MREIGMTH=0, eigenvalue analysis will be done in MSC.Marc using the inverse power sweep with double eigenvalue extraction.

MREIGMTH=3, eigenvalue analysis will be done in MSC.Marc using the inverse power sweep with single eigenvalue extraction.

MREL1103 Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MREL1103=0 maps CQUAD4 to MSC.Marc's element type 11 for plane strain problems.

MREL1103=3 maps CQUAD4 to MSC.Marc's element type 3 for plane stress problems.

MRELRB Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

If MRELRB is 0, and if BCMOVE with the release option is specified, IDRBODY (see the BCMOVE entry) will refer to the actual IDs of BCBODY entries.

If MRELRB is 1 and the release option of BCMOVE is used, IRDBODY (see the BCMOVE entry) will be in the order of occurrence of the BCBODY entries in the sorted MD Nastran Bulk Data file. For example, if there are two BCBODY entries with ID 12 and 22, the MRELRB=1 option means that you should specify IRDBODY on the BCMOVE entry using values of 1 and 2. If MRELRB=0, the IRDBODY values should be 12 and 22.

MRENUELE Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MRENUELE=0, element IDs will not be renumbered when translated to MSC.Marc.

MRENUELE=1, element IDs will be consecutively renumbered starting with one. This is sometimes required for MSC.Marc parallel processing (ddm).

MRENUMMT Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MRENUMMT=0, materials will not be renumbered when translated to MSC.Marc. Materials IDs should not exceed 399,999 for MSC.Marc versions up to and including MSC.Marc 2003 r2 or MSC.Marc might fail with a core dump on some computers.

MRENUMMT=1, an attempt to renumber materials sequentially starting with 1 will be made. This will prevent core dumps for material IDs exceeding 399,999 but there are cases where this may not work properly. If this option is used, the MSC.Marc input file should be examined carefully. This parameter is no longer required for MSC.Nastran 2005 and subsequent releases.

MRESTALL Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

Controls rotational restraints for solid element-only models.

0 SPCs for DOFs 4-6 will be ignored if entered in the MD Nastran file

1 SPCs for DOFs 4-6 will be included if entered in the MD Nastran file.

	Note: This option might produce an input-data error in the MSC.Marc run but is sometimes required if RBEs or other special items are included in the model.
MRESULTS	Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only If MRESULTS is set to 3, postprocessing of a previously-generated MSC.Marc t16 file to a results-only op2 file (normally called an f11 file) will be accomplished. OUTR=f11 and STOP=3 should also be set on the SOL 600 command line. This capability is available starting with MSC.Nastran 2004.1.0. If MRESULTS=0 the t16 file from the current job will be processed if requested by OUTR options on the SOL 600 entry.
MRFINITE	Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only Controls MSC.Marc's FINITE parameter. This parameter is available starting in MSC.Nastran 2005 r2. If entered, with a value of 1, MSC.Marc's FINITE option will be employed. If this option is entered, parameters MRFOLOW1, MRFOLOW3 and MARUPDAT should also be entered. Other parameters to be considered with this one are MARCDILT, MARCASUM and LGDISP.
MRFOLLOW	Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only MRFOLLOW=1, FORCE1, FORCE2, MOMENT1, MOMENT2 will act as non-follower forces. This option must be entered if MSC.Marc versions prior to 2003 r1 are to be used. MRFOLLOW=0, follower forces entered using FORCE1, FORCE2, MOMENT1, MOMENT2 will be mapped to MSC.Marc's new follower force option available starting with MSC.Marc version 2003r1. MRFOLLOW=-1, follower forces will be turned off even if requested to be on using other options. This is sometimes necessary for multiple load cases where pressures are applied to different elements in the different load cases.

MRFOLLO2 Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Controls whether MSC.Marc's follow for, 2 is used when multiple subcases are present. This option is available starting with MSC.Nastran 2005.

0 MSC.Marc's follow for, 2 will not be used when multiple subcases are present

1 MSC.Marc's follow for, 2 will be used when multiple subcases are present

MRFOLOW1 Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only
Controls the second field of MSC.Marc's follow for, N, M, L parameter (in this case, value N). This parameter is available starting in MSC.Nastran 2005 r2. If entered, options 1, 2, 3 and -1 are currently available. If this option is entered, parameters MRFOLOW3, MR FOLOW4 and MARUPDAT should also be entered. Other parameters to be considered with this one are MRFINITE, MARCDILT, MARCASUM and LGDISP.

Enter 0 if follower force due to distributed loads (pressure) is not to be considered.

Enter 1 if follower force stiffness due to distributed loads is to be included. (Default)

Enter 2 if follower force stiffness due to distributed loads is to be included.

Enter 3 if the follower force for distributed loads is based upon the displacement at the beginning of the increment, as opposed to the last iteration.

Enter -1 if the undeformed geometry is required but total values of distributed loads are to be used (no presently available)

MRFOLOW3 Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only
Controls the third field of MSC.Marc's follow for N, M, L parameter (in this case, value M). This parameter is available starting in MSC.Nastran 2005 r2. If entered, options 0 (incremental loads) or 1 (total loads) are currently available. Please not that total loads are not presently supported elsewhere in SOL 600, so unless loads are input by hand, this option should not normally be set to 1. If this option is entered, parameters MRFOLOW1, MRFOLOW4 and MARUPDAT should also be entered. Other parameters to be considered with this one are MRFINITE, MARCDILT, MARCASUM and LGDISP.

MRFOLOW4 Integer, no Default, MD Nastran Implicit Nonlinear (SOL 600) only
Controls the fourth field of MSC.Marc's follow for N, M, L parameter (in this case, value L). This parameter is available starting in MSC.Nastran 2005 r2. If entered, options 0 (ignore follow forces for point loads) and 1 (use follower forces for point loads) are currently available. If this option is entered, parameters MRFOLOW1, MRFOLOW3, and MARUPDAT should also be entered. Other parameters to be considered with this one are MRFINITE, MARCDILT, MARCASUM and LGDISP.

Enter 0 if follower force for point loads is not required (Default).

Enter 1 if follower force for point loads is to be considered.

MRGAPUSE Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether MD Nastran CGAP elements will be approximated as MSC.Marc gap elements. The default is to fatal SOL 600 analyses if GAPS are found in the model.

0 Do not translate MD Nastran models using MD Nastran CGAP elements

1 Translate MD Nastran models using CGAP elements. MSC.Marc gap elements are quite different than MD Nastran elements and usually can't be translated. In a few limited cases the MD Nastran and MSC.Marc gap elements are equivalent. It is up to the user to determine whether the gap MD Nastran elements can be used with SOL 600 or not. It is suggested that the user read MSC.Marc Volume A and C and run small test models to access each particular use of gaps. If gaps can be used, set PARAM,MRGAPUSE,12 and re-run the analysis. See also **MARCGAPP**, **MARCGAPN**, and **MARCGAPD**,.

MRHYPMID — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MRHYPMID=0, hyperelastic models with mid-side nodes will be translated to MSC.Marc including the mid-side nodes. Such models might run quite slowly.

MRHYPMID=1, hyperelastic models with mid-side nodes will be translated to MSC.Marc leaving out the mid-side nodes. These models will normally run faster, but the displacements of the mid-side nodes will be zero and thus plots might look strange unless the model without mid-side nodes is read into the postprocessor using the .marc.dat or .t16 files.

MRMAT8E3 — Real, Default = 0.1, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If solid composites are modeled using MAT8, the third modulus, E3 is not defined. E3 is calculated as E3=value * E1.

MRMAT8A3 — Real, Default = 1.0, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If solid composites are modeled using MAT8, the third coefficient of thermal expansion, ALPHA3 is not defined. ALPHA3 is calculated as ALPHA3=value * ALPHA1.

MRMAT8N3 — Real, Default = 1.0, MD Nastran Implicit Nonlinear (SOL 600) only

Value — If solid composites are modeled using MAT8, the NU23 is not defined. NU23 is calculated as NU23=value * NU12.

MRMAT8N1 Real, Default = 0.5, MD Nastran Implicit Nonlinear (SOL 600) only

Value If solid composites are modeled using MAT8, NU31 is not defined. NU31 will be calculated as NU31=value * NU12.

MRMAXISZ Integer, Default = Value in MSC.Marc include file in tools directory. MD Nastran Implicit Nonlinear (SOL 600) only
If this value is entered, the integer value will be used on the command line to run MSC.Marc as -maxsize N where N is the integer entered.

MRMAXMEM Integer, Default = unlimited, MD Nastran Implicit Nonlinear (SOL 600) only

Value Specifies the maximum memory that can be used in SOL 600 for the MSC.Marc execution portion of a SOL 600 run. Values are expressed in Mega Words (MW). For example, if the value 100 is entered, the memory will be limited to 100MW (do not enter the characters MW).

If this value is entered, the integer value will be used on the command line to run MSC.Marc as -ml N where N is the integer entered here.

MRMAXNUM Integer, Default = Value in MSC.Marc include file in tools directory. MD Nastran Implicit Nonlinear (SOL 600) only
If this value is entered, the integer value will be used on the command line to run MSC.Marc as -maxnum N where N is the integer entered.

MRMEMSUM Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

If mrmemsum=1, a summary of memory used by the internal MD Nastran-to-MSC.Marc translator will be printed in the f06 file. Each line will contain four numbers (all are in 4-byte words). The first number is the current memory request, the second the current memory (in addition to standard MD Nastran open core), the third is the memory limit with zero meaning no memory limit, and the fourth is the high water memory used so far by the MD Nastran-to-MSC.Marc translator and / or the t160p2 results conversion.

This parameter must be entered in the Case Control.

MRMTXKGG Character, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

NAME The NAME will be used for K2GG entries if entered. This parameter is similar to MRMTXNAM. Either MRMTXKGG or MRMTXNAM can usually be entered. The proper Case Control K2GG=NAME or K2PP=NAME will be selected automatically by SOL 600 as follows: If the continue option involves dynamic response analysis, K2PP will be used. If the continue option involves eigenvalue extraction or static analysis, K2GG will be used.

MRMTXNAM Character, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

NAME The NAME (field 2 of the DMIG entry) that will be used for DMIG values in a spawned MD Nastran execution. For example, a common name used frequently is K2XX. The Case Control command K2PP=Name will be added (in this example K2PP=K2XX will be added) at the end of the Case Control of the spawned job. DMIG entries with other names may exist on the file, but only those with NAME will be used in the spawned execution. Either MRMTXKGG or MRMTXNAM can usually be entered. The proper Case Control K2GG=NAME or K2PP=NAME will be selected automatically by SOL 600 as follows: If the continue option involves dynamic response analysis, K2PP will be used. If the continue option involves eigenvalue extraction or static analysis, K2GG will be used.

MRNOCOMP Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether n-layer composite will be created if CORDM is defined on the PSOLID entry. This allows output in material coordinate systems.

-1 Composite are not created (automatically activated for brake squeal)

1 1-layer composites are created.

N N-layer composites are created.

MRNOCOR — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MRNOCOR=0, MSC.Marc parameters ELASTICITY, PLASTICITY, UPDATE, LARGE DISP, CONSTANT DILITATION will be automatically adjusted as recommended by the MSC.Marc developers.

MRNOCOR=1, the above parameters will be adjusted as determined to be the most consistent in correlation between MD Nastran and MSC.Marc results for similar problems. This parameter should be entered for buckling problems without plasticity. PARAM,LGDISP,1 should also be entered.

MRNOECHO — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether various outputs are placed in the MSC.Marc file

0 Output is not suppressed (unless other entries are made to suppress it)

1 Suppress echo of nodes and element lists

2 Suppress echo of boundary conditions

3 Suppress echo of nurbs data

Note: Enter any combination to suppress whatever is desired. To suppress all items, enter 123.

MRORINTS — Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

Controls orientation type for all solid elements in model (see MSC.Marc Volume C documentation, ORIENTATION option). This option is available starting with MSC.Nastran 2005.

1 Edge 1-2
2 Edge 2-3
3 Edge 3-4
4 Edge 3-1
5 Edge 4-1
6 XY Plane
7 YZ Plane
8 ZX Plane
9 XU Plane
10 YU Plane
11 ZU Plane
12 UU Plane
13 UORIENT Define transformation matrix with orient.f user subroutine
14 3D ANIO

Note: For solid composites, it is necessary to use the PSOLID entry as well as the PCOMP entry. The PSOLID entry should normally have an entry in field 4 which specifies a CORDi entry to use for the material alignment direction.

MROUTLAY — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Indicates which shell or composite layers are to be output using the MARCOUT Bulk Data entry or by default. Options are as follows:

n

-N = Layers 1 and abs(N) will be output
0 = Top and Bottom layers only will be output (the bottom layer is always 1 and the top is 11 by default or the value of PARAM,MARCSLHT if it is in the input file.
N = Layers 1 through N will be output
9999 = Stresses at the element center only will be output (top and bottom are not output) This option produces output with the assumption that the element has constant stress and strain throughout the element.

MRPLOAD4 Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether continuation entry for PLOAD4 will cause the run to stop or if the continuation line(s) are to be ignored.

0 The job will stop and a “Severe Warning” message will be issued.

1 The job will continue and the continuation lines will be ignored. A warning message will be issued for the first few such entries.

MRPRSFAC Real, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Factor by which pressure loads are scaled for SOL 600. Prior to MD Nastran 2006, MSC.Marc was not capable of handling different pressure applied to different corners of a surface. In other words, different values of p1, p2, p3, p4 on the PLOAD4 entry could not be handled. Approximations can be made by assuming the pressure is uniform over the surface if the surface is small enough. For SOL 600, the pressure (P) used by MSC.Marc is calculated using the following:

If p1 is not blank (or zero) regardless of the value of mrprsfac
P = p1

If mrprsfac is positive and p1 is blank or zero and one or more of p2, p3, or p4 are not zero or blank,
P=mrprsfac*(p1+p2+p3+p4) (This default if p2 or p3 or p4 are not zero or blank)

If mrprsfac is zero or negative,
If none of p1, p2, p3, p4 are blank or zero
P=0.25*(p1+p2+p3+p4)
Otherwise,
p=(+/-)pp where pp is the value largest absolute value of p1, p2, p3, p4 and P will have the proper associated sign.

Note: The default for mrpsfac varies from element to element. For each element it is 1.0 divided by the number of Pi defined. For example, if only one of p1,p2,p3,p4 is defined, the default mrprsfac is 1.0. If two of p1,p2,p3,p4 are defined, the default for mrpsfac is 0.5. If three are defined the default is 0.33333 and if all four are defined the default is 0.25. When PARAM,MRPRSFAC is entered, the value is the same for all elements with pressure specified by pload4.

MRPSHELL — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

Used to control shell property specifications for MSC.Marc in SOL 600

0 Classical GEOMETRY options will be used for all shell elements.

1 The MSC.Marc PSHELL option will be used, available starting in MSC.Nastran 2005 r3 (can only be used with CQUAD4 and CTRIA3).

Note: The default is 0 for MSC.Nastran 2005 r3; this option was not available in earlier versions.

Important: If MRPSHELL=1, shell elements will use MSC.Marc's new PSHELL option. No materials used by PSHELL may be referenced by other types of elements that this option has used. Models can only contain CQUAD4 and / or CTRIA3 elements. If CQUAD8, CTRIA6, CQUADR, CTRIAR elements are in a model with MRPSHELL=1, the job will terminate with an error message.

MRRCFILE — Character, no Default, MD Nastran Implicit Nonlinear (SOL 600) only

RCF — Name of RCF file name (limited to 8 characters) used in conjunction with another MD Nastran run spawned from an original MD Nastran run as specified by the CONTINUE option on the SOL 600 command. The RCF file may contain any information required (such as scratch=yes, exe=, etc.) as discussed in "**The nastran Command**" on page 2 of this guide. This rcf file does not have to use the same options as the primary rcf file and should normally set batch=no as one of the options.

MRRELNOD — Integer, Default = -1, Controls enforced displacements using SPCD (see **MARCSPCD**), MD Nastran Implicit Nonlinear (SOL 600) only

MRRELNOD=0, For multiple subcases with SPCDs, the SPCDs from the previous subcase will be released at the start of the current subcase.

MRRELNOD=1, SPC's and SPCD's from the previous subcase, not specified again in the current subcase, will be released gradually during the current subcase.

MRRELNOD = -1, SPC's and SPCD's from the previous subcase, not specified again in the current subcase, will be subtracted out. For example, if the previous subcase applied a SPCD of 0.1 to a particular dof, the current subcase will apply -1.0. This will bring the displacement of that dof to zero for the current subcase.

Note: -1 was the default in MSC.Nastran 2005 even though the *MSC.Nastran 2005 Quick Reference Guide* stated it was zero.

MRSETNA1 — Integer, Default = Program calculated, MD Nastran Implicit Nonlinear (SOL 600) only

N — If this parameter is entered with N>0, the value entered will be used in MSC.Marc's SETNAME parameter section as the first value of SETNAME,N,M which is an undocumented MSC.Marc option. N is the number of sets and M is the largest number of items in any set. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. Both MRSETNA1 and MRSETNA2 must be included for either to take effect. This option is no longer required for MSC.Nastran 2005 r2 and subsequent releases.

MRSETNA2 — Integer, Default = Program calculated, MD Nastran Implicit Nonlinear (SOL 600) only

M	If this parameter is entered with M>0, the value entered will be used in MSC.Marc's SETNAME parameter section as the second value of SETNAME,N,M which is an undocumented MSC.Marc option. N is the number of sets and M is the largest number of items in any set. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. Both MRSETNA1 and MRSETNA2 must be included for either to take effect. This option is no longer required for MSC.Nastran 2005 r2 and subsequenct releases.
MRSETNAM	Integer, Default = Program calculated, MD Nastran Implicit Nonlinear (SOL 600) only
N	If this parameter is entered with N>0, the value entered will be used in MSC.Marc's SETNAME parameter section. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. To get around a MSC.Marc bug, if a computed setname value is large, MD Nastran will normally use the undocumented form (see MRSETNA1). To prevent this, set N to -1. This option is no longer required for MSC.Nastran 2005 r2 and subsequenct releases.
MRSPAWN2	Character, Default = Nastran, MD Nastran Implicit Nonlinear (SOL 600) only
CMD	Name a command to run MD Nastran (limited to 8 characters single field or 16 for large field) used in conjunction with the CONTINUE options on the SOL 600 command. The first 3 characters (nas) are automatically added to the string entered here. If nast2005t1 is desired, enter CMD=t2005t1. If nastran is desired, either leave the parameter out or enter "tran". The MD Nastran run to be spawned will have the form: CMD jid.nast.dat rcf=RCF where file RCF is provided by PARAM,MRRCFILE,RCF. As an example, if CMD is nastran, jid is myjob (original file myjob.dat) and RCF=nast.rc, the spawned run will execute using: nastran myjob.nast.dat rcf=nast.rc

Remarks:
1. See PARAM*,HEATCMD for SOL 600 thermal contact heat transfer analysis.
2 CMD will be converted to lower case regardless of the case entered.

MRSPRING — Real, Default = 0.0, MD Nastran Implicit Nonlinear (SOL 600) only

Specifies a stiffness value to be added to the main diagonal of each translational term of the stiffness matrix. This option is useful in nonlinear static analysis with 3D contact of two or more separate structures. Some of the pieces may not be grounded until contact occurs. By adding a small spring to ground, such as K=1.0, these pieces are stabilized until contact occurs. This option applies to MD Nastran Implicit Nonlinear (SOL 600) only and generates SPRINGS in the MSC.Marc input file for all nodes in the model and all three translational degrees-of-freedom.

MRT16OP2 — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MRT16OP2=0, for Windows systems only, if mrt16op2=0 the external t16op2 program is located on the path.

MRT16OP2=1, for Windows system only, the external t16op2 program is located in the installation directory for the special version of MSC.Marc that comes with MD Nastran ("MSC_BASE"\marc\t16op2.exe - where "MSC_BASE" is the base installation directory for MD Nastran.

MRT16STP — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Enter in Case Control at the subcase level.

N — Number of load increments to put on the t16/t19 files for each subcase when the AUTO STEP method is used. The default of 0 puts all adaptive increments on the t16/t19 file. If a value of N is entered, load steps for times=tmax/N will always be introduced into the auto stepping process and the t16/t19 files will have outputs only at zero and those times. This corresponds to field 1 of MSC.Marc's AUTO STEP 2nd option and can also be set using the NLAUTO option. If this is the only non-default NLAUTO variable to set, it is more easily accomplished using this parameter. It is suggested that this parameter always be used for large models and that N be 10 or greater, otherwise the size of the t16/t19 files may become very large. This is especially important for Windows systems, which presently has a 4GB limit in converting t16 to op2 files due to compiler limitations.

MRTABLS1 — Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MD Nastran TABLES1 stress-strain curves are converted to MSC.Marc WORK HARD stress-plastic strain curves according to the following formulas for the value entered for MRTABLS1. In the formulas s is the stress entered for TABLES1, e is the strain entered in TABLES1, S is the MSC.Marc WORK HARD stress and E is the MSC.Marc WORK HARD Plastic strain. ey is the yield strain (sy/E) where sy is the yield stress. The first point of the MD Nastran curve will be skipped unless MRTABLS2 is set to 1.

0 S=s
E=e - ey

1 S=s(1+e)
E=ln(e+1)

2 S=s
E=e - s/E

3 S=s(1+e)
E=ln(1+e) - s/EE

4 S=s
E=e

5 S=s
E=ln(1+e)

6 S=s
E=ln(1+e-s/E)

7 S=s(1+e)
E=1n(1+e)-S/EE

Note: This parameter (like any other parameter) can only be entered once in an input file.

MRTABLS2 Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

If MRTABLS2=0, MD Nastran TABLES1 stress-strain curves are converted to MSC.Marc WORK HARD stress-plastic strain curves starting with the yield point. The first point will be skipped.

Note: This parameter (like any other parameter) can only be entered once in an input file.

MRTIMING Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

If MRTIMING is 1, timing summaries for various portions of the internal MSC.Marc translator will be provided in the f06 and log files.

MRTSHEAR Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MRTSHEAR=0, Parabolic shear deformation is not included in the formulation of beam and shell elements.

MRTSHEAR=1, Parabolic shear deformation is included in the formulation of beam and shell elements.

MRV09V11 Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether certain Marc “features” which are default in Marc “version 11” are added for SOL 600-generated models that use “version 9”.

-1 Do not add the features

1 Add the following features:
feature,4703 to speed up DDm jobs 9for 1-processor jobs, it has no effect)
feature,5701 to disable old rigid rotation checking which was too stringent
feature,601 to improve contact
feature,5301 to improve deformable-deformable contact
feature,3201 to improve contact friction types 6 and 7‘
feature,5601 to improve thickness updating when the updated Lagrange method is used
feature5801 to improve in-plane bending of Marc element type 140
feature,6001 to improve concrete cracking analysis

Note: The above features are only used for certain problems, even though all are included with the default option, they have no effect on models that do not use Marc element type 140, feature 601 has no effect on models that do not have contact, etc.

MSIZOVRD Integer, Default = -1, for small models and +1 for large models, MD Nastran Implicit Nonlinear (SOL 600) only

Determines whether Marc SIZING values for large models will be updated or not. Nastran includes Marc directories with a file named “include” in the tools directory which includes a line MAXNUM=N, where N is some value like 1000000. If the maximum number of nodes or elements in the model exceeds N, memory overwrites or job aborts are possible unless either N is set larger than the actual max node or element number in the model or the values are specified on the SIZING entry (field 3 for max element and field 4 for max node).

-1 The sizing entry will not be updated (either the model is not large, MAXNUM in the include file has been updated or parameters such as MARCSIZ3 and/or MARCSIZ4 have been entered into the run to provide values that are large enough.

1 If the number of nodes or elements in the model exceeds 1,000,000 the sizing entry will be updated to the max number of nodes and elements actually in the model. Extra nodes and or elements to account for welds, pinned members, Herrmann elements, etc. will be included.

Note: For PARAM,MSIZOVRD large models are considered to be those with more than 1,000,000 nodes and/or elements.

MSOLMEM, MBYTE — Integer, Default = Program determined value, MD Nastran Implicit Nonlinear (SOL 600) only

If entered, the integer value entered here is the 8th field of MSC.Marc's SOLVER option, and is the maximum memory in Mega Bytes that can be used by MSC.Marc's solver types 6 (hardware provided direct) and 8 (multi-frontal direct, which is the default solver for SOL 600) before going out of core. This parameter is the same as the MBYTE field on the NLSTRAT entry any may be easier to enter as a parameter if no other NLSTRAT values are needed.

MSPEEDCW — Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether CWELD elements will be translated to MSC.Marc in core (for increased speed) or out of core. This parameter is needed if many CWELD elements are present in the model to avoid large translation times.

0 Processed out of core

1 Process in core.

MSPEEDOU — Integer, Default = 1, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether speed enhancements are activated for the t16op2 conversion.

0 Speed enhancements are not activated.

1 Speed enhancements are activated which will place certain scratch data in memory. For large models make sure enough memory is available (if PARAM,MSPEEDSE and/or PARAM,MSPEEDP4 are used, there is probably sufficient memory unless there is a large number of output "time" points.

MSPEEDP4 Integer, Default = -1, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether PLOAD4 entries will be translated to MSC.Marc in core (for increased speed) or out of core.

1 PLOAD4 will be processed in core.

-1 PLOAD4 will be processed out of core.

Note: MSPEEDP4=-1 was the only choice in all MSC.Nastran versions prior to MSC.Nastran 2005 r2. The MSPEEDP4=1 option may require more memory than is available on certain computers for large models. Do not use if CWELD elements are present.

MSPEEDSE Integer, Default = -1, MD Nastran Implicit Nonlinear (SOL 600) only
Determines whether speed enhancements are activated using extra memory and/or special low level I/O routines.

1 Solid elements 2D elements data will be processed in core.

2 All elements will be processed in core.

3 2D and 3D elements will be processed in core, 1D elements will be processed using bioxxx (low level direct access routines used by gino).

-1 No speed enhancements will be activated.

Note: MSPEEDSE=-1 was the only choice in all MSC.Nastran versions prior to MSC.Nastran 2005 r2. The MSPEEDSE=1 option may require more memory than is available on certain computers for large models.

MUSBKEEP Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

MUSBKEEP=0, if user subroutines are included, they are compiled and linked to form a new version of MSC.Marc if MUSBKEEP=0, the new version of MSC.Marc will be deleted at the end of the run.

MUSBKEEP=1, the new MSC.Marc executable will be saved on disk in the same directory as the MD Nastran input file. Its name will be the name used in PARAM,MARCUSUB with the extension marc.

NASPRT Default = 0

NASPRT specifies how often data recovery is performed and printed in SOL 200. By default, SOL 200, in addition to performing an analysis and optimization, will perform full data recovery operations at the first design cycle and upon completion of the last design cycle.

If NASPRT > 0, then data recovery operations are performed at the first design cycle; at every design cycle that is a multiple of NASPRT; and the last design cycle. For example, if PARAM,NASPRT,2 and the maximum number of design iterations requested is 5, then data recovery is obtained at design iterations 1, 2, 4, and 5.

If NASPRT < 0, then no data recovery operations are performed.

NDAMP Default = 0.01 for SOLs 129 and 159, -0.05 for SOL 400

In SOLs 129 and 159, numerical damping may be specified for the METHOD="ADAPT" on the TSTEPNL entry (two-point integration) in order to achieve numerical stability. A value of zero requests no numerical damping. The recommended range of values is from 0.0 to 0.1.
In SOL 400, numerical damping may be specified for all METHODs on the TSTEPNL Bulk Data entry. When value < 0.0, numerical damping is introduced into the system. The recommended range of values is from 0.0 to -0.33333.

NINTPTS Default = 10

NlNTPTS requests interpolation over the NlNTPTS elements closest to each grid point. NlNTPTS=0 requests interpolation over all elements, which can be computationally intensive. Related parameters include BlGER, CURV, NUMOUT, OG, OUTOPT, S1G, S1M, S1AG and S1AM.

NLAYERS Default = 5 (minimum=1, maximum=12) (SOLs 106, 129, 153, and 159)

NLAYERS is used in material nonlinear analysis only and is the number of layers used to integrate through the thickness of CQUAD4 and CTRlA3 elements with isotropic material properties. Set NLAYERS=1 for efficiency if no bending is selected (MID2=0 or -1 on all PSHELL data entries). Do not specify NLAYERS=1 if MID2 is greater than zero. A larger value of NLAYERS will give greater accuracy at the cost of computing time and storage requirements.

NLHTLS Default = 0

See Remarks under Case Control command, "**TSTRU**" on page 519.

NLMAX Default = 60

The number of suspected mm's is determined from the number of high ratio messages. If this number exceeds NLMAX the number of trial mm's is reduced to NLMAX. This is a means to avoid an expensive debug run when there may be thousands of mm's due to systematic modeling error, such as having CONM2 elements on many grid points for which structural elements have been left out through oversight. The value of this parameter may be increased on initial debug runs where it is possible that there are many high ratio DOFs, and you'd rather see them all at once rather than on successive runs where you remove only a part of them at one time.

NLMIN Default = 10

If there are only one or a few high ratio DOFs there may be more mm's present. More trial mm vectors are used, and those that do not indicate true problems are discarded. A smaller value could be considered on a stable model undergoing small modeling changes.

NLPACK

NLPACK is used in the nonlinear solution sequence SOL 400 only. The SOL 400 will pack output data for NLPACK output time steps and restart data for the last time step as a single data package. For example, if NLPACK=100 (the default), then one data package have output data for 100 output time steps and restart data for the last time step. Later usage such as restart or initial condition for later step can be performed only at data package boundaries. If NLPACK= -1, all output data for a STEP and restart data for end of the STEP are grouped into a single package. In this case, the restart can be performed only at STEP boundaries. NLPACK=0 is illegal. If NLPACK=1, each package of data on the database includes output data for one output time step and restart data. In this case, restart can be performed at every output time step. Please note that the output time step is controlled by the NO field on the TSTEPNL bulk data entry.

NLTOL

$$\text{Default} = \begin{cases} 2 (\text{SOL 106 and SOL 400}) \\ 0 (\text{SOL 153, nonlinear heat transfer}) \end{cases}$$

NLTOL establishes defaults for the CONV, EPSU, EPSP and EPSW fields of NLPARM Bulk Data entry for nonlinear static and heat transfer analysis according to the following table:

NLTOL	Level of Accuracy
0	Very high
1	High
2	Engineering Design
3	Preliminary Design

See Remark **16.** of the NLPARM entry for further details and corresponding default NLPARM field values.

NMLOOP

Default = 0

In SOLs 106 and 153, nonlinear static analysis, normal modes will be computed with the updated nonlinear stiffness if PARAM,NMLOOP is not equal to zero. The nonlinear normal modes will be computed at the last iteration loop of each subcase in which a METHOD command appears.

NOCOMPS Default = +1

NOCOMPS controls the computation and printout of composite element ply stresses, strains and failure indices. If NOCOMPS = 1, composite element ply stresses, strains and failure indices are printed. If NOCOMPS = 0, the same quantities plus element stresses and strains for the equivalent homogeneous element are printed. If NOCOMPS=-1, only element stresses and strains are printed. Composite ply stress recovery is not available for complex stresses.

Homogenous stresses are based upon a smeared representation of the laminate's properties and in general will be incorrect. Element strains are correct however.

NODCMP Default = 0

For some type of nonlinear transient problems, including heat transfer, the decomposition of the solution matrix is not required. In order to increase efficiency, NODCMP is created to determine whether the solution matrix will be decomposed. If NODCMP=0, the solution matrix will be decomposed. If NODCMP=1, the solution matrix will NOT be decomposed. In this case, MAXBIS and DJUST on the Bulk Data entry, NLPARM, must be set zero on the Bulk Data entry. NODCMP is available for SOL 129 and 159 only.

NOELOF Default = -1

If NOELOF > 0, then the grid point forces (GPFORCE Case Control command) are computed along the edges of the two-dimensional elements. The default value will suppress this output.

NOELOP Default = -1

If NOELOP > 0, then the sum of the grid point forces (GPFORCE Case Control command) are computed parallel to the edges of adjacent elements. The effect of CBAR element offsets is not considered in the calculation of the forces. The default value will suppress this output.

NOGPF Default = 1

NOGPF controls the printout of grid point forces (GPFORCE Case Control command). NOGPF > 0 specifies that the grid point forces are to be printed. NOGPF < 0 suppresses the printing of the grid point forces.

NOMSGSTR Default = 0

If NOMSGSTR = -1, the MSGSTRES module will be skipped even though Bulk Data entries provided for it. See Part III of the *MSGMESH Analyst Guide* for a discussion of MSGSTRESS.

NONCUP Default = -1

In SOL 112, NONCUP selects either a coupled or noncoupled solution algorithm in modal transient response analysis. See "**Transient Response in SubDMAPs SEDTRAN and SEMTRAN**" on page 467 of the *MSC.Nastran Reference Guide*. By default, the noncoupled solution algorithm is selected unless the dynamic matrices KHH, MHH, or BHH have off-diagonal terms. NONCUP=1 requests the coupled algorithm and -2 the uncoupled algorithm regardless of the existence of off-diagonal terms in the dynamic matrices. User Information Message 5222 indicates which algorithm is used in the analysis.

NQSET Default = 0

If NQSET > 0, and the PARAM entry is in Case Control, all part superelements that do not contain QSET entries, or are not referenced by SENQSET entries in the main Bulk Data Section, have NQSET generalized coordinates assigned to them. These QSET variables are functionally equivalent to those generated by SENQSET entries.

NUMOUT
NUMOUT1
NUMOUT2 See S1.

OELMSET Integer; Default = 0

Identification number of a Case Control command SET definition. The members of the specified SET represent the identification numbers of the finite elements that are to be retained in the "reduced" op2 file element connection data block.

OG Default=0

See CURV.

OGEOM Default = YES

See POST < 0.

OGRDOPT Integer; Default = 1

Selects the method used to create the set of grid points retained in the reduced grid point geometry data block. The default simply uses the set of grid point IDs listed in the OGRDSET case control SET. Set consistency is checked. OGRDOPT=2 uses the list of grid point IDs that are connected to elements in the OELMSET case control SET. OGRDOPT=3 merges the contents of the OGRDSET case control SET with the contents of the grid point list connected to the elements in the OELMSET case control SET. There is no consistency check for OGRDOPT=2 or OGRDOPT=3. OGRDOPT=0 turns the SET consistency check off altogether. For this case, the grid points retained are those specified in the OGRDSET SET and the elements retained are those specified in the OELMSET SET.

OGRDSET Integer; Default = 0

Identification number of a case control command SET definition. The members of the specified SET represent the identification numbers of the grid points that are to be retained in the "reduced" op2 file grid geometry data block.

OLDSEQ Default = { -1 for non-iterative-distributed-parallel solutions; 5 for iterative solutions using distributed parallel methods (NASTRAN ITER=1 and system(231)>0); 6 if SUPER=2 }

OLDSEQ selects from the following options for resequencing:

-1 No resequencing is performed.

1 Use the active / passive option.

2 Use the band option.

3 For the active / passive and the band option select the option giving the lowest RMS value of the active columns for each group of grid points.

4 Use the wavefront (Levy) option.

5 Use the Gibbs-King option even if the CPU estimate is higher than for nonsequencing.

6 Use the automatic nested dissection option even if the CPU estimate is higher than for no resequencing. See the following SUPER=2 description.

8 Semiautomatic selection. The program will compute estimates for two options that are suitable for the decomposition method selected by the PARALLEL and SPARSE keywords on the NASTRAN statement and select the option with the lowest estimate. The following table shows the suitable options for each decomposition method.

Decomposition Method	Suitable Options
regular	1 and 4
parallel	2 and 5
sparse	6 and 7

9 The extreme partitioning method is used to partition the model into domains

10 Currently not used.

11 The MSCMLV partitioning method is used to partition the model into domains

Note: The model partitioning options make sense only when running with the DOMAINSOLVER command in the Executive Control Section. For DOMAINSOLVER (PARTOPT=GRID), param,oldseq,9 is the default. For all other DOMAINSOLVER options, the default is param,oldseq,11.

Note: The wavefront option does not support superelement resequencing or starting nodes. Also note that the automatic nested dissection option uses starting nodes only to establish the root of the initial connectivity tree.

If the value of OLDSEQ is changed in superelement analysis, an SEALL=ALL restart is required.

PARAM,FACTOR is used to generate the sequenced identification number (SEQlD) on the SEQGP entry as follows:

SEQID=FACTOR*GRP+SEQ

where:

SEQ = generated sequence number

GRP = group sequence number

If GRP=0, use GRP(MAX)+1 where GRP(MAX) is the largest group sequence number in the database.

PARAM,MPCX controls whether the grid point connectivity created by the MPC, MPCADD, and MPCAX entries and / or the rigid element entries (e.g., RBAR) is considered during resequencing:

-1 Do not consider the connectivity of the MPC, MPCADD, MPCAX, or rigid element entries.

0 Consider the connectivity of the rigid element entries only. (Default).

>0 Consider the connectivity of the rigid element entries and the MPC, MPCADD, and MPCAX entries with the set identification number set to the value of this parameter.

PARAM,SEQOUT controls the output options as follows:

0 Do not generate any printed or punched output for the new sequence (default).

1 Print a table of the internal / external sequence in internal order.

2 Write the SEQGP entries to the PUNCH file.

3 Perform SEQOUT=1 and 2.

PARAM,START specifies the number of the grid points at the beginning of the input sequence. The input sequence will be the sorted order of the grid point numbers including the effect of any SEQGP entries input by the user. A single SEQGP entry can be input to select the starting point for the new sequence. Otherwise, the first point of lowest connectivity will be used as the starting point.

If PARAM,SUPER<0, all grid points from the connection table that are not part of the group currently being processed are deleted. This option provides for sequencing only the interior points of a superelement. If any superelements are present, the residual structure is not resequenced. If all of the grid points are in the residual structure, they are resequenced.

If PARAM,SUPER=0 or 1, all grid points in the connection table are considered. This option provides for the recognition of passive columns.

If PARAM,SUPER=2, then all points that are connected to multipoint constraints (via MPC entries) or rigid elements (e.g., the RBAR entry) are placed in a special group at the end of the sequence. This option also forces OLDSEQ=6 and may not be selected with other values of OLDSEQ. This option is intended primarily for models that have many active columns due to MPCs or rigid elements; e.g., a model with disjoint structures connected only by MPCs or rigid elements.

OMACHPR Default = NO

See POST<0.

OMAXR Default = 2 · BUFFSIZE

OMAXR specifies the maximum record length of data written by the OUTPUT2 module under PARAM,POST,<0 and PARAM,OPTEXIT,-4. BUFFSIZE is a machine-dependent value defined in the *MD Nastran Configuration and Operations Guide*. For further information, see the OMAXR parameter description under the OUTPUT2 module description in *MD Nastran 2006 DMAP Programmer's Guide*.

OMID Default = NO

To print or punch the results in the material coordinate system, set the parameter OMID to yes. Applicable to forces, strains, and stresses for CTRIA3, CQUAD4, CTRIA6, and CQUAD8. Other elements and outputs are not supported. This capability is not supported by pre-processors (xdb and op2 output are not changed) and grid point stress output that assumes output is in element coordinate system.

OMSGLVL Integer; Default = 0

Set consistency check error message severity flag. The default causes FATAL messages to be generated if the grid set is not consistent with the element-related grid point set and the job is terminated. If OMSGLVL=1, the FATAL messages are reduced to WARNINGS and the job is allowed to continue.

OPCHSET Integer; Default = 0

SET punch request flag. If OPCHSET=1, then the list of grid points used to reduce the grid point geometry data block will be punched in case control SET definition format.

OPGEOM Default = -1

OPGEOM < *1 prints the set definition for all degrees-of-freedom, including the aerodynamic degrees-of-freedom. OPGEOM is similar to the USETPRT parameter.

OPGTKG Default = -1

OPGTKG > -1 prints the matrix for the interpolation between the structural and aerodynamic degrees-of-freedom.

OPPHIB Default = -1

In the flutter (SOLs 145 and 200) and aeroelastic (SOLs 146 and 200) solution sequences, OPPHIB > -1 and a DISPLACEMENT request in the Case Control Section will output the real vibration modes with the structural displacement components transformed to the basic coordinate system.

OPPHIPA Default = -1

In the flutter (SOLs 145 and 200) and the dynamic aeroelastic (SOL 146) solution sequences, OPPHIPA > -1 and a DISPLACEMENT command in the Case Control Section will output the real vibration modes at all degrees-of-freedom, including the aerodynamic degrees-of-freedom in the global coordinate system. Use PARAM,OPPHIPB to output in the basic system.

OPTEXIT Default = 0

In SOL 200, especially during the checkout of the analysis model and the design optimization input data (design model), it may be desirable to exit the solution sequence at certain points before proceeding with full optimization. OPTEXIT may be set to values of 1 through 7 and -4. The DSAPRT Case Control command overrides the specification of PARAM,OPTEXIT,4, -4, or 7. The description of OPTEXlT values follow.

OPTEXIT Value	Description
0	Do not exit. Proceed with optimization.
1	Exit after the initialization of the analysis and design model but before finite element analysis begins.
2	Exit after finite element analysis and initial design response and shape basis vector processing.
3	Exit after design constraint evaluation and screening.
4	Exit after design sensitivity analysis and print the matrix of design sensitivity coefficients (DSCM2). This is equivalent to the DSAPRT (UNFORM,END=SENS) Case Control command.
-4	Exit after design sensitivity analysis and write the data blocks related to sensitivity coefficients (DSCM2 and DSCMCOL) to an external file using the OUTPUT2 and OUTPUT4 modules. This is equivalent to the DSAPRT (NOPRINT,EXPORT END=SENS) Case Control command. See related parameters ITAPE, IUNIT, and OMAXR.
5	Exit after the first approximate optimization of the design model.
6	Exit after the first update of the analysis model based on the first approximate optimization of the design model.
7	Compute and output design sensitivity coefficients at the end of normal program termination: hard convergence, soft convergence, or maximum design cycles. This is equivalent to the DSAPRT (UNFORM,START=LAST) Case Control command.

OPTION Default = ABS

See SCRSPEC.

OSWELM Default = -1

Offset for identification numbers of internally generated m-set constraint elements corresponding to CWELD elements with formats ELEMID and GRIDID. By default, OSWELM=-1, the numbering starts with SYSTEM(182) + 1. The default of system cell 182 is SYSTEM(182)=100,001,001. If the user defines OSWELM > 0, then the numbering starts with OSWELM + 1. For each CWELD element, a pair of RWELD constraint elements is generated if MSET=ON is specified, see the entry "**PWELD**" on page 2279 for an explanation.

OSWPPT Default = -1

Offset for internally generated grid identification numbers GA and GB in CWELD elements. By default, OSWPPT=-1, the numbering starts with SYSTEM(178) + 1. The default of system cell 178 is SYSTEM(178)=101,000,000. If the user provides OSWPPT > 0, then the numbering starts with OSWPPT + 1. For each CWELD element, a pair of grid points GA and GB is generated internally if the formats ELEMID or GRIDID are used and if no identification numbers for GA and GB are specified, see the entry "**CWELD**" on page 1350 for a definition of the formats.

OUGCORD See POST.

OUNIT1 Default = Value of OUNIT2

For PARAM,POST,-1 and -2 defines the unit that geometry data blocks are output to. See PARAM,POST. This parameter should not be specified after BEGIN SUPER.

OUNIT2 Default = 12

For PARAM,POST,-1 and -2 defines the unit that results data blocks are output to. See PARAM,POST. This parameter should not be specified after BEGIN SUPER.

OUTOPT Default = 0

See CURV.

PANELMP Replaced by a keyword on the FLSPOUT Case Control command.

PARTMEM Integer, Default = 10

Amount of memory for automatic parallel during partitioning (SEQP). Default is 10, increase value for large problems.

PATVER Default = 3.0

See POST = -1.

PDRMSG Default = 1

PDRMSG controls the printout of messages associated with deformed plots, including error messages. PDRMSG = 0 suppresses the printout. Contour values will not be displayed unless the default value is used.

PEDGEP Default = 0

Cubic edges of p-elements can be created with the FEEDGE Bulk Data entry by defining two vertex grids and two points in between. By default, the two points on an edge are moved to the parametric 1/3 and 2/3 locations of the edge. For PEDGEP = 1 the points are not moved. MSC.Patran V7 generates points so that adjacent edges are C1 continuous. These points should not be moved. Therefore, MSC.Patran generates a Bulk Data Section with PARAM,PEDGEP,1 if p-elements are in the model.

PENFN Default = 1.0e+5 (See LMFACT)

PH2OUT Default = 0

For nonlinear solution sequence, SOL 400, in addition to the regular phase III output, the user can also request the phase II output. This is useful when the run is terminated abnormally before the phase III outputs are formatted and printed. The phase II output consists of all outputs requested by the Case Control commands in the input file and prints in sort1 format. If PH2OUT, MD Nastran outputs phase III outputs only. This is the regular output. If PH2OUT=1, MD Nastran outputs phase II outputs only. In this case, there will be no output for the upstream superelements. If PH2OUT=3, MD Nastran outputs both phase II and phase III outputs. In this case, some of the outputs for the residual structure may be redundant.

PKRSP Default = -1

If PKRSP=0, the magnitude of the output quantities at the time of peak acceleration of the modal variables is output. The SRSS technique that is used for response spectra is described in "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide* 15. This option is available only for modal transient analysis.

PLTMSG Default = 1

PARAM,PLTMSG,0 suppresses messages associated with undeformed plot requests, including error messages.

POST Default = 1

If PARAM,POST,0, then the following parameters and discussion apply:

The data blocks often used for pre- and postprocessing will be stored in the database and also converted, by the DBC module (see *MD Nastran 2006 DMAP Programmer's Guide*), to a format suitable for processing by MSC.Patran, MSC/XL and MSC.Aries. These data blocks include input data related to geometry, connectivity, element and material properties, and static loads; they also include output data requested through the Case Control commands OLOAD, SPCF, DISP, VELO, ACCE, THERMAL, ELSTRESS, ELFORCE, FLUX, GPSTRESS, GPFORCE, ESE, GPSDCON, and ELSDCON.

By default, the data blocks are converted to a format suitable for MSC/XL. If MSC.Aries is to be used, then PARAM,DBCCONV,ARIES must be entered in order to obtain the proper format.

The converted data is written to logical FORTRAN units, which may be assigned to physical files in the File Management Section. The FORTRAN unit numbers are specified by the parameters GEOMU, POSTU, and LOADU. By default, all data is written to the logical FORTRAN unit indicated by GEOMU. If LOADU > 0, static load data may be diverted to another unit indicated by LOADU. If POSTU > 0, then output data requested with the Case Control commands listed above will be diverted to the logical unit indicated by POSTU. See "**Database Concepts**" on page 513 of the *MSC.Nastran Reference Guide* for the procedure for assigning physical files.

By default, if converted data already exists on the files indicated by GEOMU, POSTU, and LOADU, then the DBC module will overwrite the old data. If this is not desirable, then PARAM,DBCOVWRT,NO must be entered. The parameters MODEL and SOLID may be used to store more than one model and solution in the graphics database. These parameters are not supported by MSC.Patran.

PARAM,DBCDlAG > 0 requests the printing of various diagnostic messages from the DBC module (see *MD Nastran 2006 DMAP Programmer's Guide*) during the data conversion. By default, no messages are printed.

If PARAM,POST,<0, then the following parameters and discussion apply:

PARAM,POST,-1 outputs the appropriate files for the MSC.Patran NASPAT program. PARAM,POST,-2 outputs the appropriate files for the EDS I-DEAS® program. PARAM,POST,-4 outputs the files indicated below along with OPHIG for the MSC_NF interface by LMS International. PARAM,POST,-5 outputs the files indicated in the table below along with LAMA and OPHG1 for the FemTools interface by Dynamic Design Solutions. POST=-4 and -5 are intended for SOL 103 only. PARAM,POST,-6 outputs the files indicated below for EDS Unigraphics®.

An OUTPUT2 file for FORTRAN unit 12 in binary format is automatically created in the same directory and with the same name as the input file and with the extension ".op2". For example, if the input file is fender.dat then the OUTPUT2 file will be called fender.op2.

An ASSIGN statement is required in the FMS Section only if neutral file format is desired as follows:

ASSIGN OP2='filename of FORTRAN file' FORM

Geometry data blocks are output with PARAM,OGEOM,YES (except with PARAM,PATVER<3.0) and are written to a FORTRAN unit specified by PARAM,OUNIT1 (Default = OUNIT2) for POST = -1, -2, -4, and -6. PARAM,OUNIT2K (default = 91) specifies the unit number for KELM and KDICT with PARAM,POST,-5. PARAM,OUNIT2M (default = 92) specifies the unit number for MELM and MDICT with PARAM,POST,-5. See the following table for the specific geometry data blocks written for different values for POST.

See also the PARAM,POSTEXT description for additional data blocks written to the .op2 file.

POST					Geometry Data Block	Description
-1*	-2	-4	-5	-6		
YES	NO	NO	NO	NO	GEOM1S, GEOM1VU	Grid Point Definitions (Superelement)
NO	YES	YES	NO	YES	CSTM	Coordinate System Transformations
NO	YES	YES	NO	YES	GPL	Grid Point List
NO	YES	YES	NO	YES	GPDT	Grid Point Definitions
NO	YES	YES	NO	NO	EPT	Element Properties
NO	YES	YES	NO	NO	MPT	Material Properties
NO	YES	YES	NO	NO	GEOM2	Element Definitions
NO	YES	NO	NO	NO	GEOM3	Load Definitions
NO	YES	NO	NO	NO	GEOM4	Constraint Definitions
YES	NO	NO	NO	NO	DIT	Dynamic Table Input
YES	NO	NO	NO	NO	DYNAMICS	Dynamic Loads Definition
NO	NO	YES	YES	NO	KDICT	Element Stiffness Dictionary
NO	NO	YES	YES	NO	KELM	Element Stiffness Matrices
NO	NO	YES	YES	NO	MDICT	Element Mass Dictionary
NO	NO	YES	YES	NO	MELM	Element Mass Matrices
NO	NO	NO	NO	YES	ECTS	Element Connections
YES	NO	NO	NO	NO	VIEWTB	View Element Table
YES	NO	NO	NO	NO	EDOM	Design Model Input
YES	NO	NO	NO	NO	GEOM2S, GEOM2VU	Same as GEOM2 for superelements
YES	NO	NO	NO	NO	CSTMS	Same as CSTM for superelements
YES	NO	NO	NO	NO	EPTS	Same as EPT for superelements
YES	NO	NO	NO	NO	MPTS	Same as MPT for superelements

PARAM,OMACHPR,NO (default) selects the Version 68 (and 68.2) format for GPDT, CSTM, and GEOM1. PARAM,OMACHPR,YES selects the Version 69 format.

For PARAM,POST = -1 and -2, results data blocks are output to a FORTRAN unit specified by PARAM,OUNIT2 (Default = 12). This parameter is allowed to vary between superelements. In buckling solution sequence (SOL 105), a unique value of OUNIT2 should be specified for the buckling subcase. See also the related parameter OMAXR.

By default under PARAM,PATVER, ≥ 3.0, the displacements are output in the global coordinate system. To output in the basic coordinate system, specify PARAM,OUGCORD,BASIC. Under PARAM,PATVER,<3.0, the opposite is true.

PARAM,POST,-1: Results Data Blocks for MSC.Patran

By default, the following data blocks are output under PARAM,POST,-1. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file. PARAM,PATVER selects the appropriate version of MSC.Patran (Default=3.0).

PARAM,PATVER		Parameter Name	Case Control	Data Block Name	Description
<3.0	≥3.0				
YES	YES	OQG	SPCFORCE	OQG1	Forces of single-point constraint
YES	NO	OUG	DISP	OUGV1PAT	Displacements in the basic coordinate system
YES	YES	OUG	DISP	OUGV1	Displacements in the global coordinate system
YES	NO	OES	STRESS	OES1	Element stresses (linear elements only)
YES	NO	OEF	FORCE	OEF1	Element forces or heat flux (linear elements only)
YES	YES	OEE	STRAIN	OSTR1	Element strains
YES	YES	OGPS	GPSTRESS	OGS1	Grid point stresses
YES	YES	OESE	ESE	ONRGY1	Element strain energy
YES	YES	OGPF	GPFORCE	OGPFB1	Grid point force balance table
NO	YES	OEFX	FORCE	OEF1X	Element forces with intermediate (CBAR and CBEAM) station forces and forces on nonlinear elements
NO	YES	OESX	STRESS	OES1X	Element stresses with intermediate (CBAR and CBEAM) station stresses and stresses on nonlinear elements
NO	YES	OPG	OLOAD	OPG1	Applied static loads
NO	YES	OCMP	STRESS	OES1C	Ply stresses
NO	YES	OCMP	STRAIN	OSTR1C	Ply strains

PARAM,PATVER		Parameter Name	Case Control	Data Block Name	Description
<3.0	≥3.0				
NO	YES	none	DISP SPCFORCE FORCE STRESS STRAIN	OUPV1 OQP1 DOEF1 DOES1 DOSTR1	Scaled Response Spectra
			none	LAMA	Nonlinear Buckling
NO	YES	none	DISP OLOAD	OCRUG OCRPG	
NO	YES	none	NLSTRESS	OESNLXR	Nonlinear static stresses
NO	YES	none	BOUTPUT	OESNLBR	Slideline stresses
NO	YES	none	NLLOAD	OPNL1	Nonlinear loads
NO	YES	none	STRESS	OESNLXD	Nonlinear transient stresses
NO	YES	none	none	ERRORN	p-element error summary table

PARAM,POST,-2: Results Data Blocks for EDS I-DEAS®

By default, the following data blocks are output under PARAM,POST,-2. By default, the displacements are output in the basic coordinate system. To output in the global coordinate system, specify PARAM,OUGCORD,GLOBAL. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file.

PARAMeter Name	Case Control	Results Data Block Name	Description
OQG	SPCFORCE	OQG1	Forces of single-point constraint
OUG	DISPLACE	BOUGV1	Displacements in the basic coordinate system
		BOPHIG	Eigenvectors in the basic coordinate system
		OUGV1	Displacements in the global coordinate system
		TOUGV1	Grid point temperatures

PARAMeter Name	Case Control	Results Data Block Name	Description
OES	STRESS	OES1	Element stresses (linear elements only)
OEF	FORCE	OEF1	Element forces (linear elements only)
	FLUX	HOEF1	Element heat flux
OEE	STRAIN	OSTR1	Element strains
OESE	ESE	ONRGY1	Element strain energy
OCMP	STRESS	OEFIT	Failure indices
	STRESS	OES1C	Ply stresses
	STRAIN	OSTR1C	Ply strains
OUMU	ESE	LAMA	Eigenvalue summary
		ONRGY2	Element strain energy
OEFX	FORCE	OEF1X	Element forces (nonlinear elements only)
OESX	STRESS	OES1X	Element stresses (nonlinear elements only)
none	none	ODELBGPD	Shape optimization geometry changes

PARAM, POST, -4: Results Data blocks for LMS International/MSC_NF

By default, the following data blocks are output under PARAM,POST,-4. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file.

PARAMeter Name	Case Control	Data Block Name	Description
OUG	DISPLAC	OPHIG	Eigenvectors in the global coordinate system

PARAM, POST, -5: Results Data blocks for Dynamic Design Solutions / FemTools

By default, the following data blocks are output under PARAM,POST,-5. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file. PARAM,OUNIT2O (Default51) specifies the unit number of the OUTPUT2 file.

PARAMeter name	Case Control	Data Block Name	Descriptions
OUG	DISPLAC	OUGV1	Eigenvectors in the global coordinate system
		LAMA	Eigenvalue summary

PARAM, POST, -6: Results Data Blocks for EDS Unigraphics®

By default, the following data blocks are output under PARAM,POST,-6. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that displacements not be written to the OUTPUT2 file.

PARAMeter Name	Case Control	Data Block Name	Description
OQG	SPCFORCE	OQG1	Forces of single-point constraints
OUG	DISPLAC	OUGV1	Displacements
OES	STRESS	OES1	Element stresses
OEF	STRESS	OEF1	Element forces
OEE	STRAIN	OSTR1	Element strains
OESX	STRESS	OES1X	Element stresses with intermediate station stresses and stresses on nonlinear elements
OEFX	STRESS	OEF1X	Element forces with intermediate station forces and forces on nonlinear elements

PARAMeter Name	Case Control	Data Block Name	Description
OPG	OLOAD	OPG1	Applied static loads
none	none	DVPTAB	Designed property table
none	none	OPTPRMG	Optimization parameters
none	none	PROPO	Final element properties
none	none	DBCOPT	Optimization summary data

POSTU Default = -1

See POST=0.

POSTEXT Default = NO

Under PARAM,POST,-1 and -2, and if PARAM,POSTEXIT,YES is specified, then the following data blocks are also written to the .op2 file specified by PARAM,OUNIT2.

Data Block Name	PARAM POST	Description
FRL	-1, -2	Frequency response list (modal frequency response only).
BHH	-1, -2	Modal damping matrix (modal frequency response only).
KHH	-1, -2	Modal stiffness matrix (modal frequency response only).
BGPDT	-1	Basic grid point definition table.
PVT0	-1, -2	User parameter value table
CASECC	-1	Case Control table
EQEXIN(S)	-1	Equivalence external to internal grid ID table
CLAMA	-1, -2	Complex eigenvalue table
OEDE1	-1, -2	Element energy loss output table
OEKE1	-1, -2	Element kinetic energy output table
OUGV2	-1, -2	Displacement output table in SORT2
PSDF	-1, -2	Power spectral density table

Data Block Name	PARAM POST	Description
OGPWG	-1, -2	Grid point weight generator output table
TOL	-1, -2	Time output list
OPHSA	-1, -2	Solution set eigenvectors (modal frequency response only)
LAMA	-1	Eigenvalue summary table
ONRGY2	-1	Element kinetic energy (obsolete)
PSDFH	-1, -2	Power spectral density table for modal coordinates
DSCM2	-1, -2	Design sensitivity coefficient matrix
DSCMCOL	-1, -2	Design sensitivity parameters table

PREFDB Default = 1.0

See ACOUT.

PRGPST Replaced by the PRINT Keyword on the AUTOSPC Case Control command.

PRINT Default = YES

PARAM,PRINT,NO suppresses the automatic printing of the flutter summary in flutter analysis.

PROUT Default = -1

PARAM,PROUT,-1 suppresses execution and printout from the ELTPRT module. PARAM,PROUT,-1 prints a list of all elements sorted on EID and summary tables giving the range of element identification numbers for each element type.

PRPA Default = 1.0E37

PRPJ PRPA and PRPJ control the printout of intermediate load matrices for diagnostic purposes during superelement assembly. If the value of PRPA (or PRPJ) is positive, all terms larger in magnitude than the value are printed. If the value of PRPA (or PRPJ) is negative, all terms smaller in magnitude than the value are printed. The default value requests no printout. PARAM,IRES,1 must be present for these parameters to be effective. The PA matrix contains the internal loads transmitted to the downstream superelement. The PJ matrix contains external loads applied on the superelement; that is, it has the same content as the data produced by the Case Control command OLOAD. All of this data may be obtained on restart using the SELR Case Control command option. A related parameter is IRES.

PRPHIVZ Default = 1.0E37

PRPHIVZ controls the printout of the PHlVZ matrix that contains the component mode eigenvectors of the model. It includes all degrees-of-freedom with motion except the m-set, eliminated for multipoint constraints. The FAPPROX matrix contains the square root of the diagonal terms of the generalized stiffness matrix divided by 2π. For fixed-boundary solutions, it is a good approximation for the natural frequencies of the component. For free- or mixed-boundary solutions, it is of a lower value than the natural frequencies. All terms larger than PRPHlVZ in both matrices will be printed (i.e., PARAM,PRPHIVZ,0.0 causes all terms to be printed).

PRTMAXIM Default = NO

PRTMAXIM controls the printout of the maximums of applied loads, single-point forces of constraint, multipoint forces of constraint, and displacements. The printouts are titled “MAXIMUM APPLIED LOADS”, “MAXIMUM SPCFORCES”, “MAXIMUM MPCFORCES”, and “MAXIMUM DISPLACEMENTS”.

PRTRESLT Default = YES

PRTRESLT controls the printout of the resultants of applied loads, single-point forces of constraint, and multipoint forces of constraint. The printouts are titled “OLOAD RESULTANTS”, “SPCFORCE RESULTANTS”, and “MPCFORCE RESULTANTS”.

PVALINIT Default = 1

Starting p-value in a p-adaptive analysis restart.

Q Default = 0.0

Q specifies the dynamic pressure. Q must be specified in aeroelastic response analysis (SOLs 146), and the default value will cause a User Fatal Message.

RCONTACT Integer, Default = 0, MD Nastran Implicit Nonlinear (SOL 600) only

If RCONTACT=0 and PARAM,MCONTACT has been entered, a second MSC.Marc analysis as described in the discussion of PARAM,MCONTACT is started from the primary MD Nastran run.

If RCONTACT=1 and PARAM,MCONTACT has been entered, a second MD Nastran analysis as described in the discussion of PARAM,MCONTACT is started from the primary MD Nastran run.

RESLTOPT Default = 8

RESLTOPT's default value provides component-level force summary output for model checkout (PARAM, CHECKOUT, YES), loads generation (OLOAD output), and SPC and MPC forces. Setting RESLTOPT to a value of 1 produces abbreviated output formats only.

RESVEC This parameter and the related parameters RESVINER, RESVSO, RESVSE, and RESVSLI are obsolete or replaced by options on the RESVEC Case Control command.

RMSINT Default = LINEAR for the trapezoidal approximation.

RMSINT specifies the interpolation method for numerical integration when computing both RMS (Root Mean Square) and N0 (Number of Zero Crossings or Mean Frequency) from PSDF (Power Spectral Density Function). RMSINT = LINEAR requests the trapezoidal approximation, which is the existing MD Nastran approach. RMSINT = LOGLOG requests the Log-Log interpolation.

RSPECTRA Default = -1

RSPECTRA = 0 requests that response spectra be calculated for transient analysis. See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide* for a discussion of this capability. Response spectra will be calculated for any superelements or the residual structure for which other output requests are present in the same run. The requirements for the other output requests are also in "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*. Any punch data produced is sent to the standard MD Nastran PUNCH file. Related parameters are TABID and RSPRINT.

RSPRINT Default = 0

RSPRINT controls the printout of tabulated values of response spectra. RSPRINT = -1 suppresses the printout. The related parameter is RSPECTRA.

S1 Default = -1

S1G
S1M

The MAXMIN Case Control command offers more features with much greater efficiency. PARAM,S1i,+1 requests the sorting and / or filtering of the element stresses selected on the DTI,INDTA entry. Stresses in the element coordinate systems (S1), at grid points (S1G), and / or in material coordinate systems (S1M) based on the parameters BIGER, NUMOUT, SRTOPT, and SRTELTYP may be requested. The S1G and S1M options also require the presence of PARAM,CURV,1.

Parameter	Quantity	Coordinate System	Location	Elements
S1≥0	Stresses	Element	Element centers	CQUAD4, CQUAD8, CTRIA3, CTRIA6
S1M≥0	Stresses	Material	Element centers	CQUAD4, CTRIA3
S1G≥0	Stresses	Material	Grid points to which elements connect	CQUAD4, CTRIA3

NUMOUT, in conjunction with BIGER, controls the amount of stress output.

1. NUMOUT = +N requests that N element stresses be printed (or punched) for each element type.
2. NUMOUT=0 outputs all elements in a group when one or more exceeds BIGER. Some of the elements will have stresses small than BIGER. This is conceptually the same as describing an element set in case control, and limiting output in this manner. Stress files obtained with element group filtering may be used for xy plotting and other postprocessor options with DMAP alters. By contrast, the stress file when NUMOUT = -2 is more discontinuous, and may not be used for xy plotting.
3. NUMOUT = 0 does not sort but filters according to BIGER by element group. In static analysis an element group is defined as all case control selected elements for a given load case for SORT1 output. For SORT2 output an element group is defined as the data for a given element type for all load cases. In transient analysis an element group is defined as all case control selected elements at a given time for SORT1 output. For SORT2 output an element group is defined as the data for a given element at all time steps. The element group option applies only to output types described above for PARAM,S1. This option is not available with output types selected by PARAMs S1G and S1M.
4. NUMOUT = -1 requests that stresses be sorted and only those stresses with an absolute value that is greater than BlGER will be output.
5. NUMOUT = -2 (the default) does not sort but filters according to BIGER. Related parameters include BIGER, NOELOF, NOELOP, and NOGPF.

BIGER controls the elements for which stresses will be printed. Elements with stresses that are smaller in absolute value than BIGER will not be output. The quantity tested is element type dependent. Related parameters include CURV, NUMOUT, S1, S1G, and S1M. SRTOPT controls the scanning option to be performed.

SRTOPT Value	Description
0	Filter / sort on maximum magnitude.
1	Filter / sort on minimum magnitude.
2	Filter / sort on maximum algebraic value.
3	Filter / sort on minimum algebraic value.

SRTELTYP controls the element type to be processed, as described in the following table.

SRTELTYP Value	Description
0	All element types will be processed.
> 0	Only element type SRTELTYP will be processed.

NUMOUT1 and BlGER1 serve the same function as NUMOUT and BIGER except that they apply only to composite element stresses and do not require PARAM,S1i,+1.

NUMOUT2 and BIGER2 serve the same function as NUMOUT and BIGER except that they apply only to composite element failure indices and do not require PARAM,S1i,+1.

S1AG Default = -1

S1AM See CURV.

SCALEMAS, Real, no Default, SOL 700 only

DTMIN Option to scale all element masses (generated by density times volume) such that its time step never becomes less than:
dt=STEPFCT*DTMIN
where
DT = timestep originally calculated by Dytran_lsdyna a this particular time
STEPFCT = timestep safety factor (see PARAM,STEPFCT)
DTMIN = value specified on this parameter

Note:	The amount of mass scaling varies with time. Added mass does contribute to total (kinetic) energies in the model and should be used carefully. Generally, a 5-8% mass increase at each component level (not global level) is usually acceptable.

SCRSPEC Default = -1 (SOLs 103 and 115 only)

SCRSPEC=0 requests that structural response be calculated for response spectra input in normal modes analysis. See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide* for a discussion of this capability. The scaled response calculations are made for elements and grid points of the residual structure only. The responses are summed with the ABS, SRSS, NRL, or NRLO convention, depending on the value of PARAM,OPTION. If the SRSS, NRL, or NRLO options are used, close natural frequencies will be summed by the ABS convention, where close natural frequencies meet the inequality $f_{i+1} < \text{CLOSE} \cdot f_i$. Both PARAM,OPTION and PARAM,CLOSE may be set in any subcase, allowing summation by several conventions in a single run.

In Version 70, the NRL option has been modified slightly to correspond to the NAVSEA-0908-LP-000-3010 specification. NRLO provides the V69 NRL.

SDRPOPT Default = SDRP

SDPROPT controls the storage of the principal stresses and strains in the stress and strain tables (OES1 and OSTR1 data blocks) in p-version analysis. By default, the principal stresses and strains are stored in the stress and strain tables to support postprocessing. PARAM,SDRPOPT,OFP requests that the principal stresses and strains are not stored in the tables. This can result in a significant reduction in disk space usage.

SEMAP Default = SEMAP

SEMAPOPT Default = 42

SEMAPPRT Default = 3

The superelement map (SEMAP table) contains several lists useful for determining how the program has partitioned superelement models. It is printed automatically each time this table is generated. It consists of three major parts:

GPM The Grid Point Map contains a list of each grid point, its interior superelement placement, and the SElD of all grid points connected directly to it by elements. Three tables follow that summarize the connectivity between superelements, sorted on grid point sequence, SElD, and the number of connections.

ISM The Individual Superelement Map lists the interior grid points, exterior grid and scalar points, elements, and time and storage space estimates for each superelement.

SDT The Superelement Definition Table contains the SEID of every superelement in the model, the processing order, and a pictorial representation of the superelement tree.

SEMAP, SEMAPOPT, and SEMAPPRT are used to control the amount of output that is printed and other special features. The possible values for SEMAP are shown in the following table.

SEMAP Value	Output and Application
SEMAP (default)	ISM, SDT. The lengthy GPM is suppressed. This is the appropriate value for use after the model is stable and only minor changes are to be made.
SEMAPALL	GPM, ISM, SDT. All tables are printed. This value is useful on the initial debug run of a model and when making extensive modeling changes.
SEMAPCON	Only the summary tables of the GPM and the estimation data is output. This is a useful value when iterating to an economic partitioning scheme for large, complex models.
SEMAPEST	Only the estimation data is printed. This is useful when evaluating several alternative partitioning schemes.

SEMAP Value	Output and Application
SEMAPPUN	No output is printed. The exterior grid points of the superelement with a SEID that is input on SEMAPOPT are placed on a CSUPER entry image on the PUNCH file, allowing the superelement to be used as an external superelement. If SEMAPOPT > 0, the superelement entry is given an SSID of SEMAPOPT. If SEMAPOPT < 0, the exterior points listed are those of the residual structure, but the CSUPER entry is given an SSID of |SEMAPOPT|.

Other special features are available with parameters SEMAPOPT and SEMAPPRT. They are fully described under parameters OPT1 and OPT2 in the description of the TABPRT module in the *MD Nastran 2006 DMAP Programmer's Guide*.

If the default value of SEMAP is used, the other two parameters may be used to further refine this output, as described in *MD Nastran 2006 DMAP Programmer's Guide* under the TABPRT module description. The printing of the SEMAP table can be avoided by the use of PARAM,SEMAPPRT,-1.

SENSUOO Default = NO

By default, in dynamic sensitivity analysis in SOL 200, displacements at the o-set due to pseudo-loads do not include any effect due to inertia and damping. If PARAM,SENSUOO,YES is specified then these effects will be computed in a quasi-static manner and included in the sensitivity analysis.

SEP1XOVR Default = 0

The old and new location of moved shell grid points are printed if SEP1XOVR = 16. When the RSSCON shell-to-solid element connector is used. By default, the moved shell grid points are not printed, SEP1XOVR = 0. See the description of PARAM,TOLRSC for more details.

SEQOUT Default = 0

See OLDSEQ.

SERST Default = AUTO

By default, all restarts are considered automatic (see "**Restart Procedures**" on page 398 of the *MSC.Nastran Reference Guide*). If none of the following Case Control commands are entered, then SEALL=ALL is the default action: SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL.

These commands may be used to partition the analysis into several runs. By default, the restart will proceed in automatic fashion for each command, regenerating only that data that is affected by modifications in the Bulk Data and Case Control or changes in upstream superelements. If the user wishes to overwrite the data, even if it is not affected by modifications to the data, then PARAM,SERST,MANUAL must be entered.

With PARAM,SERST,AUTO or MANUAL, all superelements will be processed through Phase 0 (see "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*). This phase includes execution of the sequencer module (SEQP), initial superelement processing (SEP1), and initial geometry processing (GP1 and GP2) modules, which can result in significant CPU overhead. If this overhead is not desired, then PARAM,SERST,SEMI will limit Phase 0 and Phase 1 to only those superelements specified on the SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL Case Control commands. If none of these commands is entered, then execution will skip Phase 0 and 1.

In the modal solution sequences (SOLs 110, 111, 112, 145, 146, and 200), the modes of the residual structure are automatically computed in Phase 2 if any SE-type command (e.g., SEMG=n) is requested for the residual structure. If PARAM,SERST,SEMI and no SE-type command is specified for the residual structure, then, by default, its modes will not be recomputed. This logic is intended for restarts from SOL 103 into one of the modal solutions. If, however, the modes have not already been computed or need to be recomputed, then PARAM,SERST,RSMDS must be specified to force the calculation of the residual structure modes.

If PARAM,SERST,SEDR is specified, then Phases 0, 1, and 2 will be skipped. This option is intended for data recovery (Phase 3) runs only.

The options of SEMI, RSMDS, and SEDR are intended for models that are defined on more than one database; i.e., superelements are defined on separate databases (multiple MASTER DBsets) and processed in separate runs. Also, with this technique, databases are attached with the DBLOCATE File Management statement rather than the RESTART File Management statement. In general, these options are not recommended because they disable the automatic restart capability, which compromises the database integrity.

SESDAMP Default = NO

Modal damping is calculated for superelements if PARAM,SESDAMP,YES is specified. An SDAMPING Case Control command that selects a TABDMP1 Bulk Data entry must also be specified in the desired superelement's subcase. By default, modal damping is added to viscous damping (B). If you insert PARAM,KDAMP,-1 (or PARAM,KDAMPFL,-1 for fluid superelements) then modal damping will be added to structural damping (K4).

SESEF Default = -1 (SOLs 103 and 115 only)

If SESEF = 0 in superelement normal modes analysis, the fraction of total strain energy for a superelement in each of the system's modes is output in the vector SESEFA for tip superelements and in SESEFG for nontip superelements. If SESEF = 1, strain energy fractions are output, and expansion of the eigenvectors from a-set size to g-set is branched over for tip superelements. If SESEF = -1 (the default value), no strain energy fractions are computed.

Output requests must be present in order for strain energy fractions to be calculated. If SESEF = 1, no other output results for tip superelements.

SHIFT1 Default = -1.234

The negative shift used when computing massless mechanism constraint modes. For very stiff model (1000. hz for the first flexible mode), consider using a larger value.

SHLDAMP Default = SAME

If SAME, then structural damping is obtained from MID1 material of PSHELL. If DIFF or any value not equal to SAME, each MIDi field of the PSHELL will have its own structural damping. See Remark **5.** of the PSHELL.

SIGMA Default = 0.0

The radiant heat flux is proportional to

$$\text{SIGMA} \cdot (T_{grid} + T_{ABS})^4$$

where SIGMA is the Stefan-Boltzmann constant, T_{grid} is the temperature at a grid point, and T_{ABS} is the scale factor for absolute temperature and may be specified by PARAM,TABS. These parameters must be given in units consistent with the rest of the data in the model. The value for SIGMA in SI units is

$$5.67 \times 10^{-8} \text{ watts} / \text{m}^2 K^4$$

The default value causes radiant heat effects to be discarded.

SKINOUT Default = NONE

Request that sets of grid and element lists be output for both the fluid and structure at the fluid-structure interface.
NONE - Requests no output of sets.
PUNCH - Requests set output to .pch only.
PRINT - Requests set output to .f06 only.
ALL - Requests set output to both .pch and ..f06.
See the Case Control command FLSPOUT as an alternative selection.

SKPAMP Default = 0

For SOLs 145, 146, and 200, SKPAMP = -1 suppresses all unsteady aerodynamic calculations. The automatic restart performs a similar function without this parameter. Specifying it ensures suppression of the calculations, regardless of the determination of the automatic restart.

SLOOPID Default = 0 (SOL 129 and 159 only)

In a nonlinear transient analysis (SOLs 129 and 159) restart, SLOOPID identifies the initial conditioning previous nonlinear analysis run (SOLs 106 and 153 respectively). Setting SLOOPlD greater than 0 will cause SOLs 129 and 159 to start from the static deformed position.

SMALLQ Default = 0.0

By default MD Nastran removes unused superelement q-set degrees-of-freedom from the residual structure solution set. Set this parameter to a small value (e.g., 1.0E-10) if you do not want unused superelement q-set degrees-of-freedom removed.

SNORM Default = 20.0

SNORM > 0.0 requests the generation of unique grid point normals for adjacent shell elements (see **Figure 5-2**). Unique grid point normals are generated for the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements. The grid point normal is the average of the local normals from all adjacent shell elements including CQUAD8 and CTRIA6 elements. If grid point normals are present, they are used in all element calculations of the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements.

SNORM	Tolerance in Degrees
> 0.0	Unique grid point normals are generated if each angle between the grid point normal and each local normal of the adjacent shell elements is smaller than SNORM. SNORM Bulk Data entries overwrite a generated normal.
= 0.0	The generation of grid point normals is turned off. The user can define normals with the SNORM Bulk Data entry.
< 0.0	Grid point normals are not generated. SNORM Bulk Data entries are ignored.

Caution: If the grid shown in **Figure 5-2** is located on a symmetric half model boundary and, hence, Shell 2 is not present, you may attain the same result as a full model by specifying the normal direction with the SNORM Bulk Data entry.

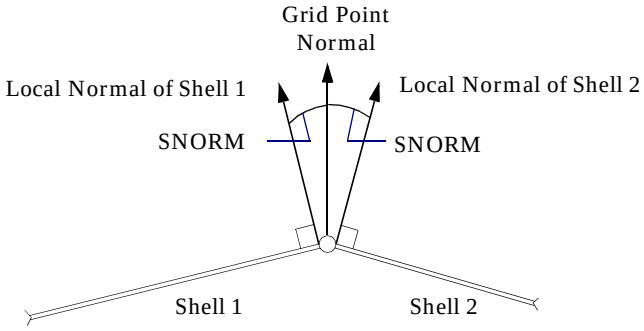


Figure 5-2 Unique Grid Point Normal for Adjacent Shell Elements

SNORMPRT Default = -1

PARAM,SNORMPRT,>0 writes the grid point normals of the model in the basic coordinate system to the .f06 and / or .pch files.

SNORMPRT	Switch to Print Out Normals
≤ 0	No output
1	Print out to the punch file (.pch)
2	Print out to the print file (.f06)
3	Print out to the punch (.pch) and print file (.f06)

SOFTEXIT Default = NO

In SOL 200, if soft convergence is achieved during optimization, before completing the maximum number of design iterations, the user may request an exit with PARAM,SOFTEXIT,YES.

SOLADJC Default = 0

PARAM SOLADJC indicates if adjoint solution vectors are to be calculated during the analysis:

-1 Do not calculate adjoint solution vectors during the analysis. Any required adjoint solution vectors will be computed during sensitivity analysis.

≥0 For ANALYSIS=DFREQ, the adjoint vectors will be computed during the solution if:

1. All frequency response DRESP1 entries are grid responses.
2. Each subcase has the same set of excitation frequencies.
3. The number of degrees-of-freedom referenced on DRESP1 entries < (number of independent design variables + number of type-2 properties + number of spawned nonlinear beam library properties) x (number of frequencies) x (number of subcases.)
4. PARAM AUTOADJ=YES (Default)

SOLID Default = 0

SOLID also allows several models to be stored in the same graphics database created by PARAM,POST,0.

SPARSEDM Default = YES

See SPARSEDR.

SPARSEDR Default = YES

SPARSEDR Default = YES

SPARSEDR=YES limits the data recovery matrix calculations to recover only those grid points specified on SET commands referenced by grid point output requests (DISP, SPCF, etc.) or those points connected to elements specified on element output requests (STRESS, FORCE, etc.). In SOL 200, the design model and design responses are also taken into consideration for determining which grid points are needed for data recovery.

SPARSEDM=YES is for SOL 200 and takes further advantage of small design models / responses during the adjoint or pseudo-load sensitivity calculations.

These methods take advantage of very small output requests, and / or small design models in SOL 200, for large models resulting in significant CPU and disk space savings.

If, however, the output requests and / or the size of the design model in SOL 200 require the calculation of the solution over a large enough percentage of degrees-of-freedom, then it is more efficient to compute the solution at all grid points. The user PARAMeter SPDRRAT (Default75) and SPDMRAT (Default60) specifies this percentage.

The sparse data recovery method is not supported in Aeroelastic (SOL 146), Flutter (SOLs 145 and 200), Complex Eigenvalue (SOLs 107, 110, 145, and 200), Nonlinear (SOLs 106, 129, 153, 159, and 400) and Cyclic Symmetry Analysis (SOLs 114, 115, and 118).

The sparse data recovery method is deactivated when the following Case Control commands are specified: EKE, ESE, EDE, and CMSENERGY.

PARAM,DDRMM is ignored under PARAM,SPARSEDR,YES. To restore the data recovery solution process to pre-V2004 methods insert PARAM,SPARSEDR,NO.

SPDRRAT Default = 75

See SPARSEDR.

SPCGEN Replaced by the PUNCH Keyword on the AUTOSPC Case Control command.

SPDMRAT Default = 60

See SPARSEDR.

SRCOMPS Default = NO

SRCOMPS controls the computation and printout of ply strength ratios. If SRCOMPS=YES, ply strength ratios are output for composite elements that have failure indices requested.

SRTELTYP Default = 0

See S1.

SRTOPT Default = 0

See S1.

START Default = 0

See OLDSEQ.

STIME Default = 0.0 (SOLs 109, 112, 129 and 159 only)

In restarts from previous transient analysis runs, the user provides STIME = t_N where t_N is the last time step of the subcase to be continued with a new or changed subcase in the new run. Thus, the loading and printout will start from t_N as though the original run contained the new subcase data.

In SOLs 109 and 112 restarts from previous SOLs 109 and 112 runs, STIME is used to specify the proper starting time of the restart run. If STIME exceeds the last output time of the previous run, the starting time is assumed to be the last output time. Otherwise, the starting time is assumed to be the output time of the previous run (not necessarily the last output time) that is closest to STIME. In other words, the starting time of the restart run need not be the last output time of the previous run, but may be any time earlier than that. The program informs the user that it is a restart run and indicates the starting time (determined as above) that is used for the restart run.

In SOLs 109 and 112 restarts, the user must ensure that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run. The user may, however, specify different TSTEP and DLOAD requests in Case Control and also different TSTEP and dynamic loading entries in the Bulk Data compared to the previous run. The loading and the results output from the restart run will start from the new starting time.

Note: That the responsibility of ensuring that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run is left to the user; the program does not check for this condition. If this condition is not met, the program may terminate the execution with a fatal error or give erroneous results.

STRUCTMP Replaced by a keyword on the FLSPOUT Case Control command.

SUBCASID Default = 0

PARAM,SUBCASID,n where n is greater than zero, specifies that the restart proceeds from SUBCASE n in nonlinear static analysis, SOL 106. SUBCASID is an alternative to SUBID and is recommended over SUBID which indicates the subcase sequence number.

SUBID Default = 1

In SOL 106 by default, the restart proceeds from the last LOOPID in the last subcase. SUBID may be used to specify an earlier subcase by specifying the sequential number (for SEID = 0) of the subcase. In SOLs 106 and 153, PARAM,LOOPID may also be specified for an earlier LOOPID. SUBCASID is an alternative to SUBID and is recommended over SUBID. See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide* for further discussion.

SUPAERO Default = ZONA

If SUPAERO=ZONA, then the ZONA51 code is used for supersonic aerodynamic calculations. If SUPAERO=CPM, then the CPM method is used. If ZONA51 is not available at a particular installation, PARAM,SUPAERO,CPM must be specified to avoid a fatal error when performing supersonic aerodynamic analyses. Only one supersonic aerodynamics method can be selected in a given run.

SUPER Default = $\begin{cases} 0 \text{ (nonsuperelement sequences)} \\ -1 \text{ (superelement sequences)} \end{cases}$

See OLDSEQ.

TABID Default = 2

TABID controls the punch output for response spectra. See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*. A related parameter is RSPECTRA.

TABS Default = 0.0

TABS is used to convert units of the temperature input (°F or °C) to the absolute temperature (°R or °K). Specify:

PARAM,TABS,273.16	When Celsius is used
PARAM,TABS,459.69	When Fahrenheit is used

Refer to the Bulk Data entry, "**CREEP**" on page 1304 for a creep analysis with SOLs 106 or 153. Refer to PARAM,SlGMA for heat transfer analysis.

TCHECK Default = 1

TCHECK=1, filtering algorithm is on for topology optimization. (Default)
TCHECK=0, no filtering algorithm.

TDMIN Default = 0.0

Topology minimum member diameter in the basic coordinate system. (Real $\geq$ 0.0, Default = 0.0, i.e., no minimum member size control.) This option is applied on 2D and 3D elements only.

TESTNEG Default = $\begin{cases} -2 \text{ for Newton's method} \\ 1 \text{ for Arc-length method} \end{cases}$

In nonlinear static analysis (SOLs 106 and 153), this parameter specifies the action to take when negative terms are encountered on the factor diagonal of matrix decomposition. Negative terms indicate that the differential stiffness has introduced a structural instability. The instability may be real (structural buckling) or mathematical (the current iteration appears unstable, but a stable solution exists).

TESTNEG	Results
-1	Stop if negative terms occur.
1 or 0	Continue if negative terms occur.
-2	If negative terms exist, do not use differential stiffness.
2	Do not use differential stiffness.

TINY Default = 1.E-3

Any elements with strain energy that is less than a TINY percentage of the total strain energy for any superelement will not be printed or made available for postprocessing by MSC.Patran or other programs. TINY may also be used to suppress the printing of small numbers in the constraint check matrix $[E_{mh}]$ described in "**Geometry Processing in SubDMAP PHASE0**" on page 400 of the *MSC.Nastran Reference Guide*.

TOLRSC Default = 0.05

When the RSSCON shell-to-solid element connector is used, the connecting grid points of the shell element are moved on to the solid face if the grid points are close enough. The tolerable distance of the shell grid point to the solid edge or face is $\varepsilon \cdot h$ where h is the height of the solid edge; see the sample figure below. The relative tolerance is user modifiable using the parameter.

PARAM,TOLRSC,ε

The default for the relative tolerance is $\varepsilon = 0.05$. Rigid body invariance is satisfied with double-precision accuracy if the shell grid points are adjusted.

TSTATIC Default = -1 (SOLs 129 and 159 only)

If TSTATlC = 1, a static solution may be obtained while ignoring inertial and damping forces. This option is available only with the adaptive time-stepping method (see METHOD = "ADAPT" on the Bulk Data entry, "**TSTEPNL**" on page 2589).

UNSYMF Default = No

In SOL 106, nonlinear statics, PARAM,UNSYMF,YES is required to include damping effects in the calculation of complex eigenvalues. See PARAM,NMLOOP.

UPDTBSH Default = NO

UPDTBSH controls the update of boundary shapes generated by auxiliary boundary model analysis in SOL 200. By default, the auxiliary boundary models and shapes are generated only once at the initial design cycle and will not be updated in subsequent cycles even if the shape of the primary model is changing. PARAM,UPDTBSH,YES requests that the auxiliary models and shapes are updated and reanalyzed at every cycle.

USETPRT Default = -1

USETSEL Default = 0

USETPRT controls the tabular printout of the degree-of-freedom sets. See "**Degree-of-Freedom Sets**" on page 939.

Sequence	Print	USETPRT
None	None(default)	-1
	Row sort only	0
Internal	Column sort only	1
	Row and Column sort	2
	Row sort only	10
External	Column sort only	11
	Row and Column sort	12

The degrees-of-freedom can be listed in ascending order according to their internal or external sequence number, but not both. The external sequence number is the grid, scalar, or extra point identification number. The internal sequence number is the number assigned after resequencing (see PARAM,OLDSEQ).

The row sort is not recommended in p-version analysis because large integers are generated for hierarchical grid point identification numbers and they will be truncated.

For a given sequence there are two types of tables that may be printed: row sort and column sort. For row sort, a table is printed for each set selected by USETSEL. Here is an example of row sort (USETPRT = 0 or 10):

```
          U S E T   D E F I N I T I O N   T A B L E   ( I N T E R N A L   S E Q U E N C E ,   R O W   S O R T )
                                                A   DISPLACEMENT SET
          -1-       -2-       -3-       -4-       -5-       -6-       -7-       -8-       -9-       -10-

1=        2-1       2-2
```

For column sort, a single table is printed for the following sets: SB, SG, L, A, F, N, G, R, O, S, M, E. Here is an example of column sort (USETPRT=1 or 11):

```
      U S E T   D E F I N I T I O N   T A B L E   ( I N T E R N A L   S E Q U E N C E ,   C O L U M N   S O R T )

          EXT GP. DOF  INT DOF  INT GP.    SB    SG    L     A     F     N     G     R     O     S     M     E
-------------------------------------------------------------------------------------------------------------------
              1 - 1      1-       1 G       1                                  1     1                 1
                - 2      2-                 2                                  2     2                 2
                - 3      3-                       1                            3     3                 3
                - 4      4-                       2                            4     4                 4
                - 5      5-                       3                            5     5                 5
                - 6      6-                       4                            6     6                 6
```

USETSEL specifies the sets which will be printed in the row sort (USETPRT = 0 or 10). In order to select specific sets to be printed, you must sum their corresponding decimal equivalent numbers. For example, sets A, L, and R are selected with USETSEL=128+256+8=392.

USETSEL	Sets Printed
-1	All sets as defined in "**Degree-of-Freedom Sets**" on page 939.
0	Mutually exclusive sets only; i.e., sets M, SB, SG, O, Q, R, C, B, E, and A.

USETSTRi Input-character-Default' '

USETSTR1 through USETSTR4 specifies the sets that will be printed by the specification of parameters USETPRT and USETSEL. Any set in "**Degree-of-Freedom Sets**" on page 939 may be specified. A ":" is used as a separator. In the following example, the m-set (degrees-of-freedom eliminated by multipoint constraints) and s-set (degrees-of-freedom eliminated by single point constraints) are specified.

Example: PARAM,USETSTR1,M:S

VMOPT Default = 0

If VMOPT=1, then the virtual mass will be included in the mass matrix at the same time as all other mass elements. In other words, the component modes will reflect the virtual mass. By default, virtual mass is included after the component modes are computed.

If VMOPT=2 the modes of the structure or component without the fluid are computed first ("dry" modes). The fluid effects are added in the modal basis during the residual flexibility computation to produce the "wet" modes for the component. Both eigenvalue tables are printed, allowing comparison of the dry and wet modes. The wet modes are used in modal dynamic analysis. The cost savings result from the dense Virtual Mass matrix being kept out when computing dry modes in the physical basis. Its presence can increase memory and computation times by an order of magnitude. The VM is added only in the smaller generalized basis used in Residual Flexibility Computations. The approximations introduced by this approach are generally small due to the homogeneous nature of the fluid. This approach was provided in earlier versions with the vma.v* series of SSSALTERS. It is the preferred method when the number of wetted elements exceeds several hundred, for reasons of efficiency.

VREF Default = 1.0

In modal flutter analysis, the velocities are divided by VREF to convert units or to compute flutter indices.

VUELJUMP Default = 1000

VUGJUMP Default = 1000

Specifies the separation in identification numbers for display elements and grid points generated in p-version analysis. The defaults are sufficient for a 9 9 9 display element mesh.

Identification numbers for display elements and grid points start with 10001001 and 201001001, respectively. For example, by default the identification numbers for the display elements of the first p-element will be numbered 100001001 through 100002000 and the second p-element 100002001 through 100003000, etc.

VUBEAM Default = VUBEAM

VUHEXA Default = VUHEXA

VUPENTA Default = VUPENTA

VUQUAD4 Default = VUQUAD4

VUTETRA Default = VUTETRA

VUTRIA3 Default = VUTRIA3

These parameters are used in p-version analysis to specify the names of the display elements in the data recovery output tables; such as those created by the VUGRID Case Control command and PARAM,POST. They should be used if your postprocessor does not recognize display elements. For example, PARAM,VUHEXA,CHEXA renames the display element VUHEXA to “CHEXA” in the output files.

WRBEAMB Integer, Default = 0

0 Write equivalent radius for all beams (see PARAM, BEAMBEA) whether beam-beam contact is anticipated or not. The equivalent radius is the 7th field of Marc’s GEOMETRY values for beam type elements.

-1 Do not write equivalent radius (7th field is blank). This might be necessary for versions of MSC.Marc earlier than 2003.

WTMASS Default = 1.0

The terms of the structural mass matrix are multiplied by the value of WTMASS when they are generated. In coupled fluid-structure analysis WTMASS is applied to the structural portion of the model only. WTMASS applies to MFLUID entries but it is not recommended for use in hydroelastic problems.

W3, W4
W3FL, W4FL Default = 0.0

The damping matrix for transient analysis is assembled from the equation:

$$[B_{dd}] = [B^1_{dd}] + [B^2_{dd}] + \frac{G}{W3}[K^1_{dd}] + \frac{1}{W4}[K^4_{dd}]$$

In coupled fluid-structure analysis, W3 and W4 are applied to structural portion of the model and W3FL and W4FL to the fluid portion of the model. The default values of 0.0 for W3, W4, W3FL, and W4FL cause the

$$[K^1_{dd}] \text{ and } [K^4_{dd}]$$

Iterms to be ignored in the damping matrix, regardless of the presence of the PARAM, G or GFL or

$$[K_{dd}^{4}] \cdot [K_{dd}^{1}]$$

is the stiffness.

$$[K_{dd}^{4}]$$

is the structural damping and is created when GE is specified on the MATi entries.

$$[K_{dd}^{1}]$$

is the stiffness. The units of W3, W4, W3FL, and W4FL are radians per unit time. (See "**Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS**" on page 449 of the *MSC.Nastran Reference Guide* for further discussion.)

In SOLs 129 and 159, W4 may vary between subcases. However, the linear portion of the model uses only the W4 value from the first subcase and the values in the subsequent subcases are applied to the nonlinear portion of the model.

WR3, WR4 Default = 0.0, no rotor damping or circulation terms.

Specifies "average" excitation frequency for calculation of rotor damping and circulation terms. See "Equations Used in Analyses" on page 192 of the *MSC.Nastran 2004 Release Guide* for equations.

XFLAG Default = 0

By default (XFLAG = 0), when temperature loads and element deformations are present, the element strain energy for the linear elements is calculated using the following equation:

$$E = \frac{1}{2}u^{T}K_{e}u - u^{T}P_{et}$$

where u is the deformation, K_e is the element stiffness and P_{et} is the element load vector for temperature differences and element deformations. If XFLAG is set to 2, the element strain energy for linear elements is calculated using the following equation:

$$E = \frac{1}{2}u^T K_e u - \frac{1}{2}u^T P_{et}$$

The latter formula is the same strain energy calculation used for nonlinear elements.

ZROCMAS Default = NO

When performing component modal synthesis with free or mixed boundary conditions, the c-set mass is normally included during the calculation of the component modes. If the component has large masses on the c-set degrees-of-freedom, or if the user requests too many modes for the component, the c-set residual flexibility will become singular. This causes a failure of the component reduction. The singularity may be avoided by setting ZROCMAS to YES, which will exclude the c-set mass when calculating the component modes.

XYUNIT n

XYUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of design optimization results and design sensitivity data in comma separated value format for use in a spreadsheet.

5.2 Parameter Applicability Tables

Table 5-1 lists parameter applicability to the solution sequences (SOLs 101 through 112). Table 5-2 lists parameter applicability to the solution sequences (SOLs 114 through 600).

B	Must be specified in the Bulk Data Section only
E	May be specified in either the Bulk Data and / or Case Control Section
C	Must be specified in the Case Control Section only

Table 5-1 PARAMeter Names in SOLs 101 Through 114

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
ACOUT	B				E	E	E	E	E	E	
ACSYM						B			B		
ADJMETH											
ADPCON				E							
ADSTAT							B			B	
AESDISC											
AESMAXIT											
AESMETH											
AESRNDM											
AESTOL											
ALPHA1					B	B	B	B	B	B	
ALPHA2					B	B	B	B	B	B	
ALTRED	B		B								B
ALTSHAPE	B	B			B	B	B	B	B	B	
ASCOUP	B	B	B	B	B	B	B	B	B	B	B
ASING	E	E	E	E	E	E	E	E	E	E	E
AUNITS											
AUTOADJ											
AUTOQSET	B	B	B	B	B	B	B	B			
AUTOSPCR				E							

Table 5-1 PARAMeter Names in SOLs 101 Through 114 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114) (continued)										
	101	103	105	106	107	108	109	110	111	112	114
AUTOSPRT		E		E	E	E	E	E	E	E	
BAILOUT	E	E	E	E	E	E	E	E	E	E	E
BEIGRED				E	E	E	E	E	E	E	
BETA											
BIGER	E	E	E								E
BIGER1	E	E	E								E
BIGER2	E	E	E								E
BUCKLE				E							
CB1, CB2	E	E	E	E	E	E	E	E	E	E	E
CDIF											
CFDIAGP	E	E	E	E	E	E	E	E	E	E	E
CFRANDEL	E	E	E	E	E	E	E	E	E	E	E
CHECKOUT	E	E	E	E	E	E	E	E	E	E	E
CK1, CK2, CK3	E	E	E	E	E	E	E	E	E	E	E
CLOSE		B									
CM1, CM2	E	E	E	E	E	E	E	E	E	E	E
CONFAC	B	B	B	B	B	B	B	B	B	B	B
COUPMASS	E	E	E	E	E	E	E	E	E	E	E
CP1, CP2	E		E	E		E	E		E	E	E
CURV	E	E	E								E
CURVPLOT	E	E	E	E							E
CWDIAGP	E	E	E	E	E	E	E	E	E	E	E
CWRANDEL	E	E	E	E	E	E	E	E	E	E	E
DBALL	E	E	E	E	E	E	E	E	E	E	E
DBCCONV	E	E	E	E	E	E	E	E	E	E	E
DBCDIAG	E	E	E	E	E	E	E	E	E	E	E
DBCOVWRT	E	E	E	E	E	E	E	E	E	E	E
DBDICT	B	B	B	B	B	B	B	B	B	B	B
DBDN	E	E	E	E	E	E	E	E	E	E	E

Table 5-1 PARAMeter Names in SOLs 101 Through 114 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114) (continued)										
	101	103	105	106	107	108	109	110	111	112	114
DBDRPRJ	B	B	B	B	B	B	B	B	B	B	B
DBDRVER	B	B	B	B	B	B	B	B	B	B	B
DBEXT	E	E	E	E	E	E	E	E	E	E	E
DBRCV	E	E	E	E	E	E	E	E	E	E	E
DBUP	E	E	E	E	E	E	E	E	E	E	E
DDRMM									B	B	
DESPCH											
DESPCH1											
DFREQ						B			B		
DOPT	E	E	E								E
DPEPS											
DSNOKD			B								
DSZERO	B	B	B								
DYNSPCF		E			E	E	E	E	E	E	
ENFMOTN						E	E		E	E	
ERROR	E	E	E	E	E	E	E	E	E	E	E
EST	E	E	E	E	E	E	E	E	E	E	E
EXTDR	B	B			B	B	B	B	B	B	
EXTDROUT	B	B			B	B	B	B	B	B	
EXTDRUNT	B	B			B	B	B	B	B	B	
EXTOUT	B	B	B	B	B	B	B	B	B	B	B
EXTRCV	E	E	E	E	E	E	E	E	E	E	E
EXTUNIT	B	B	B	B	B	B	B	B	B	B	B
FACTOR	B	B	B	B	B	B	B	B	B	B	B
FIXEDB	E	E									
FKSYMFAC				E							
FLUIDSE		B			B	B	B	B	B	B	
FOLLOWK		E	E	E	E	E	E	E	E	E	
FRQDEPO						B			B		

Table 5-1 PARAMeter Names in SOLs 101 Through 114 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114) (continued)										
	101	103	105	106	107	108	109	110	111	112	114
FZERO		E		E	E	E	E	E	E	E	
G					B	B	B	B	B	B	
GEOMU	E	E	E	E	E	E	E	E	E	E	E
GFL					B	B	B	B	B	B	
GPECT	E	E	E	E	E	E	E	E	E	E	E
GRDPNT	E	E	E	E	E	E	E	E	E	E	E
GUSTAERO											
GYROAVG						E			E		
HEATSTAT	B										
HFREQ								B	B	B	
HFREQFL								B	B	B	
IFP	E	E	E	E	E	E	E	E	E	E	E
IFTM									B		
INREL	E	E	E		E	E	E	E	E	E	
IRES	E		E	E		E	E		E	E	E
ITAPE	B	B	B								
IUNIT	B	B	B								
KDAMP								B	B	B	
KDAMPFL								B	B	B	
KDIAG				E							
K6ROT	E	E	E	E	E	E	E	E	E	E	E
LANGLE				B							
LFREQ								B	B	B	
LFREQFL								B	B	B	
LGDISP				E							
LMFACT	B	B	B								
LMODES		E						B	B	B	
LMODESFL								B	B	B	
LOADU	E	E	E	E	E	E	E	E	E	E	E

Table 5-1 PARAMeter Names in SOLs 101 Through 114 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114) (continued)										
	101	103	105	106	107	108	109	110	111	112	114
LOOPID				E							
MACH											
MAXLP											
MAXRATIO	E	E	E	E	E	E	E	E	E	E	E
MESH	B										
METHCMRS		E			E	E	E	E	E	E	
MINIGOA	B	B	B	B	B	B	B	B	B	B	B
MODACC								B	B	B	
MODEL	B	B	B	B	B	B	B	B	B	B	B
MPCX	B	B	B	B	B	B	B	B	B	B	B
NASPRT											
NDAMP											
NEWSET	B										
NINTPTS	E	E	E								E
NLAYERS				E							
NLHTLS				E							
NLMAX		E		E	E	E	E	E	E	E	
NLMIN		E		E	E	E	E	E	E	E	
NMLOOP				E	E	E	E	E	E	E	
NOCOMPS	E	E	E	E							E
NOELOF	E	E	E								E
NOELOP	E	E	E								E
NOGPF	E	E	E								E
NOMSGSTR	E	E	E								E
NONCUP										B	
NQSET		E			E	E	E	E	E	E	
NLTOL				B							
NUMOUT	E	E	E								E
NUMOUT1	E	E	E								E

Table 5-1 PARAMeter Names in SOLs 101 Through 114 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114) (continued)										
	101	103	105	106	107	108	109	110	111	112	114
NUMOUT2	E	E	E								E
OCMP	E	E	E	E							E
OEE	E	E	E	E							E
OEF	E	E	E	E	E	E	E	E	E	E	E
OEFX	E	E	E	E	E	E	E	E	E	E	E
OELMSET	E	E	E	E	E	E	E	E	E	E	E
OES	E	E	E	E	E	E	E	E	E	E	E
OESE	E	E	E	E	E	E	E	E	E	E	E
OESX	E	E	E	E	E	E	E	E	E	E	E
OG	E	E	E	E							E
OGEOM	E	E	E	E	E	E	E	E	E	E	E
OGPF	E	E	E	E							E
OGPS	E	E	E				E			E	E
OGRDOPT	E	E	E	E	E	E	E	E	E	E	E
OGRDSET	E	E	E	E	E	E	E	E	E	E	E
OLDSEQ	B	B	B	B	B	B	B	B	B	B	B
OMAXR	E	E	E	E	E	E	E	E	E	E	E
OMID	E	E	E	E	E	E	E	E	E	E	E
OMSGLVL	E	E	E	E	E	E	E	E	E	E	E
OPCHSET	E	E	E	E	E	E	E	E	E	E	E
OPG	E		E	E		E	E		E	E	E
OPGEOM											
OPGTKG											
OPPHIB											
OPPHIPA											
OPTEXIT											
OPTION		B									
OQG	E	E	E	E	E	E	E	E	E	E	E
OSWELM	B	B	B	B	B	B	B	B	B	B	B

Table 5-1 PARAMeter Names in SOLs 101 Through 114 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114) (continued)										
	101	103	105	106	107	108	109	110	111	112	114
OSWPPT	B	B	B	B	B	B	B	B	B	B	B
OUG	E	E	E	E	E	E	E	E	E	E	E
OUGCORD	E	E	E	E	E	E	E	E	E	E	E
OUMU		E									
OUNIT1	E	E	E	E	E	E	E	E	E	E	E
OUNIT2	E	E	E	E	E	E	E	E	E	E	E
OUTOPT	E	E	E	E							E
PATVER	E	E	E	E	E	E	E	E	E	E	E
PDRMSG	E	E	E	E	E	E	E	E	E	E	E
PEDGEP	E	E			E	E	E	E	E	E	
PENFN	B	B	B								
PLTMSG	E	E	E	E	E	E	E	E	E	E	E
POST	E	E	E	E	E	E	E	E	E	E	E
POSTEXT	E	E	E	E	E	E	E	E	E	E	E
POSTU	E	E	E	E	E	E	E	E	E	E	E
PREFDB							E	E	E	E	
PRINT											
PROUT	E	E	E	E	E	E	E	E	E	E	E
PRPA	E		E								
PRPHIVZ		E			E	E	E	E	E	E	E
PRPJ	E		E			E	E		E	E	E
PRTCSTM	E	E	E	E	E	E	E	E	E	E	E
PRTEQXIN	E	E	E	E	E	E	E	E	E	E	E
PRTGPDT	E	E	E	E	E	E	E	E	E	E	E
PRTGPL	E	E	E	E	E	E	E	E	E	E	E
PRTGPTT	E	E	E	E	E	E	E	E	E	E	E
PRTMAXIM	E	E	E								
PRTMGG	E	E	E	E	E	E	E	E	E	E	E
PRTPG	E	E	E	E	E	E	E	E	E	E	E

Table 5-1 PARAMeter Names in SOLs 101 Through 114 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114) (continued)										
	101	103	105	106	107	108	109	110	111	112	114
PRTRESLT	E	E	E	E	E	E	E	E	E	E	E
PVALINIT	B	B			B	B	B	B	B	B	
Q											
RESLTOPT	E	E	E	E	E	E	E	E	E	E	E
RSPECTRA							E			E	
RSPRINT							E			E	
S1	E	E	E	E			E			E	E
S1A	E	E	E	E							E
S1AG	E	E	E	E							E
S1AM	E	E	E	E							E
S1G	E	E	E	E							E
S1M	E	E	E	E							E
SCRSPEC		B									
SEMAP	B	B	B	B	B	B	B	B	B	B	B
SEMAPOPT	B	B	B	B	B	B	B	B	B	B	B
SEMAPPRT	B	B	B	B	B	B	B	B	B	B	B
SENSUOO											
SEP1XOVR	B	B	B	B	B	B	B	B	B	B	B
SEQOUT	B	B	B	B	B	B	B	B	B	B	B
SERST	B	B	B	B	B	B	B	B	B	B	B
SESDAMP		E			E	E	E	E	E	E	
SESEF		E									
SHLDAMP									E		
SIGMA											
SKINOUT		E			E	E	E	E	E	E	
SKPAMP											
SLOOPID											
SMALLQ	B	B	B	B	B	B	B	B	B	B	B
SNORM	B	B	B	B	B	B	B	B	B	B	B

Table 5-1 PARAMeter Names in SOLs 101 Through 114 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114) (continued)										
	101	103	105	106	107	108	109	110	111	112	114
SNORMPRT	B	B	B	B	B	B	B	B	B	B	B
SOFTEXIT											
SOLADJC											
SOLID	B	B	B	B	B	B	B	B	B	B	B
SPARSEDM	E	E	E			E	E		E	E	
SPARSEDR	E	E	E			E	E		E	E	
SPDDMAT											
SPDRRAT	E	E	E			E	E		E	E	
SRTELTYP	E	E	E								E
SRTOPT	E	E	E								E
START	B	B	B	B	B	B	B	B	B	B	B
STIME							E			E	
SUBCASID				E							
SUBID				E							
SUPAERO											
SUPER	B	B	B	B	B	B	B	B	B	B	B
TABID							E			E	
TABS											
TCHECK											
TDMIN											
TESTNEG				E							
TINY	E	E	E								E
TOLRSC	B	B	B	B	B	B	B	B	B	B	B
TSTATIC											
UNSYMF				B							
UPDTBSH											
USETPRT	E	E	E	E	E	E	E	E	E	E	E
USETSEL	E	E	E	E	E	E	E	E	E	E	E
VMOPT		E	E	E	E	E	E	E	E	E	E

Table 5-1 PARAMeter Names in SOLs 101 Through 114 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114) (continued)										
	101	103	105	106	107	108	109	110	111	112	114
VREF						B				B	
VUBEAM	E	E			E	E	E	E	E	E	
VUELJUMP	E	E			E	E	E	E	E	E	
VUGJUMP	E	E			E	E	E	E	E	E	
VUHEXA	E	E			E	E	E	E	E	E	
VUPENTA	E	E			E	E	E	E	E	E	
VUQUAD4	E	E			E	E	E	E	E	E	
VUTETRA	E	E			E	E	E	E	E	E	
WTMASS	E	E	E	E	E	E	E	E	E	E	E
W3							E			E	
W3FL							E			E	
W4							E			E	
W4FL							E			E	
XFLAG	E		E								E
XYUNIT											
ZROCMAS		E		E	E	E	E	E	E	E	

Table 5-2 PARAMeter Names in SOLs 115 Through 700

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
ACOUT										E			
ACSYM										B			
ADJMETH										B			
ADPCON				E				E	E		E		
ADSTAT													
AESDISC					B					B			
AESMAXIT					B					B			
AESMETH					B					B			
ADJMETH													
AESRNDM					B					B			
AESTOL					B					B			
ALPHA1			B	B		B	B	B	B	B			
ALPHA2			B	B		B	B	B	B	B			
ALTRED													
ALTSHAPE													
ASCOUP	B	B	B	B	B	B	B	B	B	B	B		
ASING	E	E	E	E	E	E	E	E	E	E	E		
AUNITS					B					B			
AUTOQSET			B					B					
AUTOSPC												B	
AUTOSPCR											E		
AUTOSPRT	E		E	E	E	E	E		E		E		
BAILOUT	E	E	E	E	E	E	E	E	E	E	E		
BEIGRED			E	E		E	E			E	E		
BETA				E					E				
BIGER	E	E			E			E		E			
BIGER1	E	E			E			E		E			
BIGER2	E	E			E			E		E			
BUCKLE											E		

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
CB1, CB2	E	E	E	E	E	E	E	E	E	E	E		
CDIF										E			
CFDIAGP	E	E	E	E	E	E	E	E	E	E	E		
CFRANDEL	E	E	E	E	E	E	E	E	E	E	E		
CHECKOUT	E	E	E	E	E	E	E	E	E	E	E	E	E
CK1, CK2, CK3	E	E	E	E	E	E	E	E	E	E	E	E	E
CLOSE	B												
CM1, CM2	E	E	E	E	E	E	E	E	E	E	E		
CONFAC	B	B	B	B	B	B	B	B	B	B	B		
COUPMASS	E	E	E	E	E	E	E	E	E	E	E		
CP1, CP2		E	E	E	E		E	E	E	E	E		
CURV	E	E			E			E		E			
CURVPLOT	E	E			E					E	E		
CWDIAGP	E	E	E	E	E	E	E	E	E	E	E		
CWLDIGNR													B
CWRANDEL	E	E	E	E	E	E	E	E	E	E	E		
DBALL	E		E	E	E	E	E	E	E	E	E		
DBCCONV	E		E	E	E	E	E	E	E	E	E		
DBCDIAG	E		E	E	E	E	E	E	E	E	E		
DBCOVWRT	E		E	E	E	E	E	E	E	E	E		
DBDICT	B		B	B	B	B	B	B	B	B	B		
DBDRPRJ	B		B	B	B	B	B	B	B	B	B		
DBDRVER	B		B	B	B	B	B	B	B	B	B		
DBDN	E		E	E	E	E	E	E	E	E	E		
DBEXT	E	E	E	E	E	E	E	E	E	E	E		
DBRCV	E		E	E	E	E	E	E	E	E	E		
DBUP	E		E	E	E	E	E	E	E	E	E		
DDRMM							B			E			
DESPCH										E			

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
DESPCH1										E			
DFREQ			B				B			B			
DOPT	E				E			E		E			
DPEPS										B			
DPHFLG													
DSNOKD										B			
DSZERO													
DYBEAMIP													B
DYBLDTIM													B
DYBULKL													B
DYBULKLQ1													B
DYCMPFLG													B
DYCONECDT													B
DYCONENMASS													B
DYCONIGNORE													B
DYCONPENOPT													B
DYCONRWPNAL													B
DYCONSKIPRWG													B
DYCONSLSFAC													B
DYCONTHKCHG													B
DYCOWPRD													B
DYCOWPRP													B
DYDCOMP													B
DYDTOUT													B
DYDYLOAD													B
DYELAS1C													B
DYELAS1F													B
DYELAS1R													B
DYELPLET													B

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
DYELPLFL													B
DYELPLSY													B
DYENDTIM													B
DYENERGYHGEN													B
DYENGFLG													B
DYHRGIHQ													B
DYHRGQH													B
DYIEVERP													B
DYINISTEP													B
DYLDKND													B
DYMATS1													B
DYMAXINT													B
DYMAXSTEP													B
DYMINSTEP													B
DYNAMES													B
DYNEIPH													B
DYNEIPS													B
DYNINTSL													B
DYNLOADS													B
DYNRBE23													B
DYN3THDT													B
DYPRMSPC													B
DYRBE3													B
DYRBE2TY													B
DYRLTFLG													B
DYNSPCF	E		E			E	E			E			
DYSHELLFORM													B
DYSHGE													B
DYSHINP													B

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
DYSHTHICK													B
DYSIGFLG													B
DYSTATIC													B
DYSTEPFCT													B
DYSTEPFCTL													B
DYSTRFLG													B
DYSTSSZ													B
DYTERMNENDMAS													B
DYTSTEPDT2MS													B
DYTSTEPERODE													B
ENFMOTN							E			E			
EPPRT													
EPSILONT													
EPZERO													
ERROR	E		E	E	E	E	E	E	E	E	E	E	E
EST	E		E	E	E	E	E	E	E	E	E	E	E
EXTDR													
EXTDROUT													
EXTDRUNT													
EXTOUT	B		B	B	B	B	B	B	B	B	B		
EXTRCV	E		E	E	E	E	E	E	E	E	E		
EXTUNIT	B		B	B	B	B	B	B	B	B	B		
FACTOR	B		B	B	B	B	B	B	B	B	B		
FIXEDB										E			
FKSYMFAC											E		
FLEXINCR													
FLUIDMP													
FLUIDSE										B			
FOLLOWK	E	E								E	E	B	

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
FRQDEPO							B			B			
FZERO											E		
G			B	E		B	B		B	B			
GFL													
GEOMU	E		E	E	E	E	E	E	E	E	E		
GFL							B			B			
GPECT	E		E	E	E	E	E	E	E	E	E		
GRDPNT	E		E	E	E	E	E	E	E	E	E	E	
GUSTAERO						B	B			B			
GYROAVG													
HEATCMD												B	
HEATSTAT													
HFREQ						B	B			B			
HFREQFL							B			B			
IFP	E		E	E	E	E	E	E	E	E			
IFTM							B						
INREL										B			
INRLM													
IRES			E	E	E	E	E	E	E	E	E		
ITAPE										B			
IUNIT										B			
KDAMP						B				B			
KDAMPFL							B			B			
KDIAG								E			E		
K6ROT	E		E	E	E	E	E	E	E	E	E		
LANGLE				B				B	B		B		
LFREQ						B	B			B			
LFREQFL							B			B			
LGDISP				E				E	E		E	B	

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
LMFACT											B		
LMODES						B	B			B			
LMODESFL							B			B			
LOADU	E		E	E				E	E	E	E		
LOOPID				E				E	E		E		
MACH						B	B			B			
MALIAS02												B	
MALIAS03												B	
MARC3901												B	
MARC3D												C	
MARCASUM												B	
MARCAUTD												B	
MARCAUTO												B	
MARCAXEL												B	
MARCBATCH												B	
MARCBEAM												B	
MARCBUG												C	
MARCBUSH												B	
MARCCBAR												B	
MARCCENT												B	
MARCCON2												B	
MARCCON3												B	
MARCCPY												B	
MARCDEF												B	
MARCDILT												B	
MARCDIS2												B	
MARCDIS3												B	
MARCDIS4												B	
MARCDMIG												B	

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARCEKND												B	
MARCEXIT												B	
MARCFIL												B	
MARCFRIC												B	
MARCGAPP												B	
MARCHEAT												B	
MARCHOST												B	
MARCIAMN												B	
MARCITER												B	
MARCLOWE												B	
MARCLUMP												B	
MARCMEM												B	
MARCMNF												B	
MARCND99												B	
MARCNOER												B	
MARCOFFT												B	
MARCONLY												B	
MARCONTF												B	
MARCOOCC												B	
MARCOP2												B	
MARCOPT												B	
MARCOTIM												B	
MARCPARR												B	
MARCPINN												B	
MARCPOS												B	
MARCPOST												B	
MARCPR99												B	
MARCPRES												B	
MARCPRN												B	

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARCPRNG												B	
MARCPRNH												B	
MARCPROG												B	
MARCPTH												B	
MARCRADD												B	
MARCRBAL												B	
MARCRBAR												B	
MARCRBE2												B	
MARCRBE3												B	
MARCREVR												B	
MARCRIGD												B	
MARCRUN												B	
MARCSAME												B	
MARCSETT												B	
MARCSINC												B	
MARCSIZ3												B	
MARCSIZ4												B	
MARCSIZ5												B	
MARCSIZ6												B	
MARCSLHT												B	
MARCSOLV												B	
MARCSPCD												B	
MARCSTIFF												B	
MARCSUMY												B	
MARCT16												B	
MARCT19												B	
MARCTABL												B	
MARCTEDF												B	
MARCTEDN												B	

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARCTEMP												B	
MARCTIEC												B	
MARCTOL												B	
MARCTVL												B	
MARCUSUB												B	
MARCVERS												B	
MARCWDIS												B	
MARCWIND												B	
MARELSTO												B	
MARGPFOR												B	
MARIBOOC												B	
MARMPCHK												B	
MARNOSET												B	
MARNOT16												B	
MARPLANE												B	
MARUPDAT												B	
MAXLP				E					E				
MAXRATIO	E		E	E	E	E	E	E	E	E	E	E	
MCONTACT												B	
MDUMLOAD												B	
MESH								B	B	B			
METHCMRS			E	E		E	E		E	E			
MEXTRNOD												B	
MHEATSHL												B	
MHEATUNT												B	
MHOUBOLT												B	
MINIGOA	B		B	B	B	B	B	B	B	B	B	B	
MINRECCC												B	
MLSTRAIN												B	

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MODACC						B	B						
MODEL	B		B	B	B	B	B	B	B	B	B	B	
MOFFCORE												B	
MOP2TITL												B	
MPCX	B		B	B	B	B	B	B	B	B	B	B	
MRAFFLOR												B	
MRAFFLOT												B	
MRAFFLOW												B	
MRALIAS												B	
MRBEAMB												B	
MRBEPARAM												B	
MRBIGMEM												B	
MRBUKMTH												B	
MRC2DADD												B	
MRCOMPOS												B	
MRCONRES												B	
MRCONVER												B	
MRDISCMB												B	
MREIGMTH												B	
MREL1103												B	
MRELRB												B	
MRENUELE												B	
MRENUMMT												B	
MRESTALL												B	
MRESULTS												B	
MRFINITE												B	
MRFOLLO2												B	
MRFOLLOW												B	
MRFOLOW1												B	

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRFOLOW3												B	
MRFOLOW4												B	
MRGAPUSE												B	
MRHYPMID												B	
MRMAXMEM												B	
MRMEMSUM												B	
MRNOCOR												B	
MRMAT8A3												B	
MRMAT8E3												B	
MRMAT8N1												B	
MRMAT8N3												B	
MRMAXISZ												B	
MRMAXNUM												B	
MRMTXKGG												B	
MRORINTS												B	
MRPLOAD4												B	
MRPSHELL												B	
MRRELNOD												B	
MRRCFILE												B	
MRSETNAM												B	
MSOLMEM,MBYTE												B	
MRSPAWN2												B	
MSPEEDCW												B	
MSPEEDOU												B	
MSPEEDP4												B	
MSPEEDSE												B	
MRSPRING												B	
MRT16OP2												B	
MRT16STP												B	

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRTABLS1												B	
MRTABLS2												B	
MRTIMING												B	
MRTSHEAR												B	
MSOLMEM												B	
MUSBKEEP												B	
NASPRT										B			
NDAMP				E					E				
NEWSET													
NINTPTS	E				E			E		E			
NLAYERS				E							E		
NLHTLS								E			E		
NLMAX											E		
NLMIN										E	E		
NMLOOP										E	E		
NOCOMPS	E				E			E		E	E		
NOELOF	E				E			E		E			
NOELOP	E				E			E		E			
NOGPF	E				E			E		E			
NOMSGSTR	E				E			E		E			
NONCUP										B			
NQSET	E		E	E	E	E	E		E				
NLTOL									E		B		
NUMOUT	E				E			E		E			
NUMOUT1	E				E			E		E			
NUMOUT2	E				E			E		E			
OCMP	E									E	E		
OEE	E									E	E	E	
OEF	E		E	E	E	E	E	E	E	E	E	E	

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
OEFX	E		E	E	E	E	E	E	E	E	E	E	
OELMSET	E	E	E	E	E	E	E	E	E	E	E		
OES	E		E	E	E	E	E	E	E	E	E	E	
OESE	E		E	E	E	E	E	E	E	E	E	E	
OESX	E		E	E	E	E	E	E	E	E	E	E	
OG	E				E			E		E	E	E	
OGEOM	E		E	E	E	E	E	E	E	E	E	E	
OGPF	E				E					E	E	E	
OGPS	E				E					E			
OGRDOPT	E	E	E	E	E	E	E	E	E	E	E		
OGRDSET	E	E	E	E	E	E	E	E	E	E	E		
OLDSEQ	B		B	B	B	B	B	B	B	B	B	B	
OMAXR	E		E	E	E	E	E	E	E	E	E	E	
OMID	E	E	E	E	E	E	E	E	E	E	E	E	
OMSGLVL	E	E	E	E	E	E	E	E	E	E	E		
OPCHSET	E	E	E	E	E	E	E	E	E	E	E		
OPG			E	E		E	E		E	E	E	E	
OPGEOM					B	B	B			B			
OPGTKG					B	B	B			B			
OPPHIB						B	B			B			
OPPHIPA						B	B			B			
OPTEXIT										B			
OPTION	B												
OQG	E		E	E	E	E	E	E	E	E	E	E	
OSWELM	B	B	B	B	B	B	B			B	B	B	
OSWPPT	B	B	B	B	B	B	B			B	B	B	
OUG	E		E	E	E	E	E	E	E	E	E	E	
OUGCORD	E		E	E	E	E	E	E	E	E	E		
OUMU	E									E			

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
OUNIT1	E		E	E	E	E	E	E	E	E	E		
OUNIT2	E		E	E	E	E	E	E	E	E	E		
OUTOPT	E				E			E		E	E		
PARTMEM													
PATVER	E		E	E	E	E	E	E	E	E	E		
PDRMSG	E		E	E	E	E	E	E	E	E	E		
PEDGEP													
PENFN											B		
PH2OUT											B		
PLTMSG	E		E	E	E	E	E	E	E	E	E		
POST	E		E	E	E	E	E	E	E	E	E	E*	
POSTEXT	E	E	E	E	E	E	E	E	E	E	E		
POSTU	E		E	E	E	E	E	E	E	E	E		
PREFDB										E			
PRINT						B				B			
PROUT	E		E	E	E	E	E	E	E	E	E		
PRPA					E			E		E			
PRPHIVZ	E		E	E	E	E	E		E	E			
PRPJ			E		E		E	E	E	E			
PRTCSTM	E		E	E	E	E	E	E	E	E	E		
PRTEQXIN	E		E	E	E	E	E	E	E	E	E		
PRTGPDT	E		E	E	E	E	E	E	E	E	E		
PRTGPL	E		E	E	E	E	E	E	E	E	E		
PRTGPTT	E		E	E	E	E	E	E	E	E	E		
PRTMAXIM	E		E	E	E	E	E	E	E	E			
PRTMGG	E		E	E	E	E	E	E	E	E	E		
PRTPG	E		E	E	E	E	E	E	E	E	E		
PRTRESLT	E		E	E	E	E	E	E	E	E			
PVALINIT										B			

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
Q					B	B	B			B			
RCONTACT												B	
RESLTOPT	E	E	E	E	E	E	E	E	E	E	E		
RMSINT													
RSPECTRA													
RSPRINT													
S1	E				E			E		E	E		
S1A	E				E			E		E	E		
S1AG	E				E			E		E	E		
S1AM	E				E			E		E	E		
S1G	E				E			E		E	E		
S1M	E				E			E		E	E		
SCALEMAS													B
SCRSPEC	B												
SEMAP	B		B	B	B	B	B	B	B	B	B		
SEMAPOPT	B		B	B	B	B	B	B	B	B	B		
SEMAPPRT	B		B	B	B	B	B	B	B	B	B		
SENSUOO										B			
SEP1XOVR	B		B	B	B	B	B	B	B	B	B		
SEQOUT	B		B	B	B	B	B	B	B	B	B		
SERST	B		B	B	B	B	B	B	B	B	B		
SESDAMP	E	E	E	E		E							
SESEF	E												
SHLDAMP										E			
SIGMA									B				
SKINOUT										E			
SKPAMP						B	B			B			
SLOOPID				E									
SMALLQ	B		B	B	B	B	B	B	B	B	B		

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
SNORM	B		B	B	B	B	B	B	B	B	B		
SNORMPRT	B		B	B	B	B	B	B	B	B	B		
SOFTEXIT										B			
SOLADJC										B			
SOLID	B		B	B	B	B	B	B	B	B	B		
SPARSEDM					E					E			
SPARSEDR					E					E			
SPDMRAT										E			
SPDRRAT					E					E			
SRTELTYP	E				E			E		E			
SRTOPT	E				E			E		E			
START	B		B		B	B	B	B	B	B	B		
STIME				E					E				
STRUCTMP										E			
SUBCASID											E		
SUBID								E			E		
SUPAERO					E	E	E			E			
SUPER	B		B	B	B	B	B	B	B	B	B		
TABID													
TABS								E	E	E			
TCHECk													
TDMIN													
TESTNEG								E			E		
TINY	E				E			E		E			
TOLRSC	B		B	B	B	B	B	B	B	B	B		
TSTATIC				E					E				
UNSYMF											B		
UPDTBSH										E			
USETPRT	E		E	E	E	E	E	E	E	E	E		

Table 5-2 PARAMeter Names in SOLs 115 Through 700 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
USETSEL	E		E	E	E	E	E	E	E	E	E		
VMOPT	E		E	E	E	E	E	E	E	E	E		
VREF						B				B			
VUBEAM													
VUELJUMP													
VUGJUMP													
VUHEXA													
VUPENTA													
VUQUAD4													
VUTETRA													
WTMASS	E		E	E	E	E	E	E	E	E	E	B	B
W3				B					E	B		B	
W3FL										B			
W4				B					E	B		B	
W4FL										B			
XFLAG					E					E			
XYUNIT										E			
ZROCMAS											E		

SECTION
6 Item Codes

- Element Stress (or Strain) Item Codes
- Element Force Item Codes
- Fluid Virtual Mass Pressure Item Codes
- 2D Slideline and 3D Surface Contact Item Codes
- Element Strain Energy Item Codes

Item codes are integer numbers assigned to specific output quantities; such as, the third translational component of displacement, the major principal stress at Z1 in a CQUAD4 element, or the torque in a CBAR element. Item codes are specified on the following input statements:

- DRESP1 entry for Design Sensitivity and Optimization (SOL 200).
- X-Y Plotting commands. See "**Plotting**" on page 527 of the *MSC.Nastran Reference Guide*.
- DTI,INDTA entry for stress sorting.

The following tables provide item codes for:

- **Table 6-1**. Element Stress or Strain.
- **Table 6-2**. Element Force.
- **Table 6-3**. Fluid Virtual Mass Pressure.
- **Table 6-4**. Heat Transfer Flux.
- **Table 6-5**. Slideline Contact Output.
- **Table 6-6**. Element Strain Energy Item Codes.

The following superscripts appear in the tables and indicate:

1. Data for components marked with the symbol (1) are included in the data block MES output from module DRMS1. (See ***MD Nastran 2006 DMAP Programmer's Guide***.)
2. Composite Element Stresses and Failure Indices.

6.1 Element Stress (or Strain) Item Codes

All item codes refer to stresses (or strains) unless otherwise denoted.

If output is magnitude/phase, the magnitude replaces the real part, and the phase replaces the imaginary part. Strain item codes are equivalent to stress item codes. However, strain is computed for only some elements. See **Table 3-1** in the *MSC.Nastran Reference Guide*.

Table 6-1 Element Stress-Strain Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CAXIF2 (47)	2[1]	Radial axis	2[1]	Radial axis	RM
	3[1]	Axial axis	3[1]	Axial axis	RM
	4[1]	Tangential edge	4[1]	Tangential edge	RM
	5[1]	Circumferential edge	5[1]	Circumferential edge	RM
			6[1]	Radial axis	IP
			7[1]	Axial axis	IP
			8[1]	Tangential edge	IP
			9[1]	Circumferential edge	IP
CAXIF3 (48)	2[1]	Radial centroid	2[1]	Radial centroid	RM
	3[1]	Circumferential centroid	3[1]	Circumferential centroid	RM
	4[1]	Axial centroid	4[1]	Axial centroid	RM
	5[1]	Tangential edge 1	5[1]	Tangential edge 1	RM
	6[1]	Circumferential edge 1	6[1]	Circumferential edge 1	RM
	7[1]	Tangential edge 2	7[1]	Tangential edge 2	RM
	8[1]	Circumferential edge 2	8[1]	Circumferential edge 2	RM
	9[1]	Tangential edge 3	9[1]	Tangential edge 3	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	10^1	Circumferential edge 3	10^1	Circumferential edge 3	RM
			11^1	Radial centroid	IP
			12^1	Circumferential centroid	IP
			13^1	Axial centroid	IP
			14^1	Tangential edge 1	IP
			15^1	Circumferential edge 1	IP
			16^1	Tangential edge 2	IP
			17^1	Circumferential edge 2	IP
			18^1	Tangential edge 3	IP
			19^1	Circumferential edge 3	IP
CAXIF4 (49)	2^1	Radial centroid	2^1	Radial centroid	RM
	3^1	Circumferential centroid	3^1	Circumferential centroid	RM
	4^1	Axial centroid	4^1	Axial centroid	RM
	5^1	Tangential edge 1	5^1	Tangential edge 1	RM
	6^1	Circumferential edge 1	6^1	Circumferential edge 1	RM
	7^1	Tangential edge 2	7^1	Tangential edge 2	RM
	8^1	Circumferential edge 2	8^1	Circumferential edge 2	RM
	9^1	Tangential edge 3	9^1	Tangential edge 3	RM
	10^1	Circumferential edge 3	10^1	Circumferential edge 3	RM
	11^1	Tangential edge 4	11^1	Tangential edge 4	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	12[1]	Circumferential edge 4	12[1]	Circumferential edge 4	RM
			13	Radial centroid	IP
			14	Circumferential centroid	IP
			15	Axialcentroid	IP
			16	Tangential edge 1	IP
			17	Circumferential edge 1	IP
			18	Tangential edge 2	IP
			19	Circumferential edge 2	IP
			20	Tangential edge 3	IP
			21	Circumferential edge 3	IP
			22	Tangential edge 4	IP
			23	Circumferential edge 4	IP
CBAR (34)	2[1]	End A-Point C	2[1]	End A-Point C	RM
	3[1]	End A-Point D	3[1]	End A-Point D	RM
	4[1]	End A-Point E	4[1]	End A-Point E	RM
	5[1]	End A-Point F	5[1]	End A-Point F	RM
	6[1]	Axial	6[1]	Axial	RM
	7	End A maximum	7[1]	End A-Point C	lP
	8	End A minimum	8[1]	End A-Point D	IP
	9	Safety margin in tension	9[1]	End A-Point E	IP
	10[1]	End B-Point C	10[1]	End A-Point F	IP
	11[1]	End B-Point D	11[1]	Axial	IP

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	12[1]	End B-Point E	12[1]	End B-Point C	RM
	13[1]	End B-Point F	13[1]	End B-Point D	RM
	14	End B maximum	14[1]	End B-Point E	RM
	15	End B minimum	15[1]	End B-Point F	RM
	16	Safety margin in compression	16[1]	End B-Point C	IP
			17[1]	End B-Point D	IP
			18[1]	End B-Point E	IP
			19[1]	End B-Point F	IP
CBAR (100)	2	Station Distance/Length	2	Station Distance/Length	RM
	3	Point C	3	Point C	RM
Intermediate	4	Point D	4	Point D	RM
Stations	5	Point E	5	Point E	RM
	6	Point F	6	Point F	RM
	7	Axial	7	Axial	RM
	8	Maximum	8	Maximum	RM
	9	Minimum	9	Minimum	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	10	Margin of Safety (Item codes above are given for End A. For codes 2 through 10 at intermediate stations add (K-1)*9 and K is the station number, and for codes at End B, K=number of stations plus 1.)	10 11 12 13 14 15 16	Point C Point D Point E Point F Axial Maximum Minimum (Item codes above are given for End A. For codes 2 through 16 at intermediate stations add (K-1)*15 nd K is the station number, and for codes at End B, K=number of stations plus 1.)	IP IP IP IP IP IP
CBAR (238)	2	Station Distance / Length	(Not Supported)		
	3	Max Axial Stress			
	4	Min Axial Stress			
	5	Max Shear Stress in xy			
	6	Min Shear Stress in xy			
	7	Max Shear Stress in zx			
	8	Min Shear Stress in zx			
	9	Max vonMises Stress			
CBEAM (2) Linear	2	External grid point ID	2	External grid point ID	
	3	Station distance / length	3	Station distance / length	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	4[1]	Long. Stress at Point C	4[1]	Long. Stress at Point C	RM
	5[1]	Long. Stress at Point D	5[1]	Long. Stress at Point D	RM
	6[1]	Long. Stress at Point E	6[1]	Long. Stress at Point E	RM
	7[1]	Long. Stress at Point F	7[1]	Long. Stress at Point F	RM
	8	Maximum stress	8[1]	Long. Stress at Point C	IP
	9	Minimum stress	9[1]	Long. Stress at Point D	IP
	10	Safety margin in tension	10[1]	Long. Stress at Point E	IP
	11	Safety margin in compression (Item codes are given for end A. Addition of the quantity (K-1)10 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through10 for intermediate stations.)	11[1]	Long. Stress at Point F (Item codes are given for end A. Addition of the quantity (K-1)10 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)	IP
CBEAM (94) Nonlinear	2	External grid point ID		Not applicable	
	3	C (Character)			
	4	Long. Stress at point C			
	5	Equivalent stress			
	6	Total strain			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	7	Effective plastic strain			
	8	Effective creep strain (Item codes 3 through 8 are repeated for points D, E, and F. Then the entire record (from 2 through N) is repeated for end B of the element.)			
CBEAM (239) Arbitrary Cross Section	2	Station Distance / Length	(Not Supported)		
	3	Max Axial Stress			
	4	Min Axial Stress			
	5	Max Shear Stress in xy			
	6	Min Shear Stress in xy			
	7	Max Shear Stress in xz			
	8	Min Shear Stress in xz			
	9	Max von Mises Stress			
CBEND (69)	2	External grid point ID	2	External grid point ID	
	3	Circumferential angle	3	Circumferential angle	
	4[1]	Long. Stress at Point C	4[1]	Long. Stress at Point C	RM
	5[1]	Long. Stress at Point D	5[1]	Long. Stress at Point D	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	6[1]	Long. Stress at Point E	6[1]	Long. Stress at Point E	RM
	7[1]	Long. Stress at Point F	7[1]	Long. Stress at Point F	RM
	8	Maximum stress	8[1]	Long. Stress at Point C	IP
	9	Minimum stress	9[1]	Long. Stress at Point D	IP
	10	Safety margin in tension	10[1]	Long. Stress at Point E	IP
	11	Safety margin in compression	11[1]	Long. Stress at Point F	IP
		(Item codes are given for end A. Item codes 12 through 21 point to the same information for end B.)		(Item codes are given for end A. Item codes 12 through 21 point to the same information for end B.)	
CBUSH (102)	2	Translation-x	2	Translation-x	R
	3	Translation-y	3	Translation-y	R
	4	Translation-z	4	Translation-z	R
	5	Rotation-x	5	Rotation-x	R
	6	Rotation-y	6	Rotation-y	R
	7	Rotation-z	7	Rotation-z	R
			8	Translation-x	I
			9	Translation-y	I
			10	Translation-z	I
			11	Rotation-x	I
			12	Rotation-y	I
			13	Rotation-z	I

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CBUSH1D (40)	1	Element ID			
	2	Axial force			
	3	Axial displacement			
	4	Axial velocity		Not applicable	
	5	Axial stress			
	6	Axial strain			
	7				
	8				
CCONEAX (35)	2	Harmonic or point angle		Not applicable	
	3	Z1=Fiber Distance 1			
	4[1]	Normal v at Z1			
	5[1]	Normal u at Z1			
	6[1]	Shear uv at Z1			
	7	Shear angle at Z1			
	8	Major principal at Z1			
	9	Minor principal at Z1			
	10	Maximum shear at Z1			
	11	Z2= Fiber Distance 2			
	12[1]	NormaI v at Z2			
	13[1]	Normal u at Z2			
	14[1]	Shear uv at Z2			
	15	Shear angle at Z2			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	16	Major principal at Z2			
	17	Minor principal at Z2			
	18	Maximum shear at Z2			
CDUM3 thru CDUM9 (55-61)	2^1	S1	2^1	S1	RM
	3^1	S2	3^1	S2	RM
	4^1	S3	4^1	S3	RM
	5^1	S4	5^1	S4	RM
	6^1	S5	6^1	S5	RM
	7^1	S6	7^1	S6	RM
	8^1	S7	8^1	S7	RM
CELAS1 (11)	2^1	Stress	2^1	Stress	RM
			3^1	Stress	IP
CELAS2 (12)	2^1	Stress	2^1	Stress	RM
			3^1	Stress	IP
CELAS3 (13)	2^1	Stress	2^1	Stress	RM
			3^1	Stress	IP
CGAP (86)	2	Normal x		Not applicable	
	3	Shear y			
	4	Shear z			
	5	Axial u			
	6	Shear v			
	7	Shear w			
	8	Slip v			
	9	Slip w			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CHEXA (67) Linear	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID (0=center)	5	External grid ID (0=center)	
	6[1]	Normal x	6[1]	Normal x	RM
	7[1]	Shear xy	7[1]	Normal y	RM
	8	First principal	8[1]	Normal z	RM
	9	First principal x cosine	9[1]	Shear xy	RM
	10	Second principal x cosine	10[1]	Shear yz	RM
	11	Third principal x cosine	11[1]	Shear zx	RM
	12	Mean pressure	12[1]	Normal x	lP
	13	von Mises or octahedral shear stress	13[1]	Normal y	IP
	14[1]	Normal y	14[1]	Normal z	IP
	15[1]	Shear yz	15[1]	Shear xy	IP
	16	Second principal	16[1]	Shear yz	IP
	17	First principal y cosine	17[1]	Shear zx	IP
	18	Second principal y cosine	18-121	Items 5 through 17 repeated for 8 corners	
	19	Third principal y cosine			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	20[1]	Normal z			
	21[1]	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-193	Items 5 through 25 repeated for 8 corners			
CHEXA (93) Nonlinear	2	Grid / Gauss			
	3	External grid ID (0=center)			
	4	Stress-X			
	5	Stress-Y			
	6	Stress-Z			
	7	Stress-XY			
	8	Stress-YZ			
	9	Stress-ZX			
	10	Equivalent stress			
	11	Effective plastic strain		Not applicable	
	12	Effective creep strain			
	13	Strain-X			
	14	Strain-Y			
	15	Strain-Z			
	16	Strain-XY			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	17	Strain-YZ			
	18	Strain-ZX			
	19-146	Items 3 through 18 Repeated for 8 corners			
CHEXAFD (202) Nonlinear Finite Deformation with 8 grid points	2	Grid / Gauss			
	3	Grid / Gauss ID (0=center)			
	4	Cauchy stress-X			
	5	Cauchy stress-Y			
	6	Cauchy stress-Z			
	7	Cauchy stress-XY			
	8	Cauchy stress-YZ			
	9	Cauchy stress-ZX			
	10	Pressure $p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$		Not applicable	
	11	Volumetric strain J-1			
	12	Logarithmic strain-X			
	13	Logarithmic strain-Y			
	14	Logarithmic strain-Z			
	15	Logarithmic strain-XY			
	16	Logarithmic strain-YZ			
	17	Logarithmic strain-ZX			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	18-122	Items 3 through 17 repeated for 7 Gauss / grid points			
CHEXAFD (207) Nonlinear Finite Deformation with 20 grid points	2-17	Same as CHEXAFD (202)			
	18-407	Items 3 through 17 repeated for 26 Gauss points		Not applicable	
CONROD (10) Linear	2[1]	Axial stress	2[1]	Axial stress	RM
	3	Axial safety margin	3[1]	Axial stress	IP
	4[1]	Torsional stress	4[1]	Torsional stress	RM
	5	Torsional safety margin	5[1]	Torsional stress	IP
CONROD (92) Nonlinear	2	Axial stress			
	3	Equivalent stress			
	4	Total strain			
	5	Effective plastic strain		Not applicable	
	6	Effective creep strain			
	7	Linear torsional stress			
CPENTA (68) Linear	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID	5	External grid ID	
		(0=center)		(0=center)	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	6[1]	Normal x	6[1]	Normal x	RM
	7[1]	Shear xy	7[1]	Normal y	RM
	8	First principal	8[1]	Normal z	RM
	9	First principal x cosine	91	Shear xy	RM
	10	Second principal x cosine	10[1]	Shear yz	RM
	11	Third principal x cosine	11[1]	Shear zx	RM
	12	Mean pressure	12[1]	Normal x	IP
	13	von Mises or Octahedral shear stress	13[1]	Normal y	IP
	14[1]	Normal y	14[1]	Normal z	IP
	15[1]	Shear yz	15[1]	Shear xy	IP
	16	Second principal	16[1]	Shear yz	IP
	17	First principal y cosine	17[1]	Shear zx	IP
	18	Second principal y cosine	18-95	Items 5 through 17 repeated for 6 corners	
	19	Third principal y cosine			
CPENTA (68) Linear (Cont.)	20[1]	Normal z			
	21[1]	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	25	Third principal z cosine			
	26-151	Items 5 through 25 repeated for 6 corners			
CPENTA (91) Nonlinear	2	Grid / Gauss			
	3	External grid ID (0=center)			
	4	Normal x stress			
	5	Normal y stress			
	6	Normal z stress			
	7	Shear xy stress			
	8	Shear yz stress			
	9	Shear zx stress			
	10	Equivalent stress		Not applicable	
	11	Effective plastic strain			
	12	Effective creep strain			
	13	Normal x strain			
	14	Normal y strain			
	15	Normal z strain			
	16	Shear xy strain			
	17	Shear yz strain			
	18	Shear zx strain			
	19-114	Items 3 through 18 Repeated for 6 corners			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CPENTAFD (204) Nonlinear Finite Deformation with 6 grid points	2-17	Same as CHEXAFD (201)			
	18-92	Items 3 through 17 repeated for 5 Gauss points		Not applicable	
CPENTAFD (209) Nonlinear Finite Deformation with 15 grid points	2-17	Same as CHEXAFD (201)			
	18-317	Items 3 through 17 repeated for 20 Gauss points		Not applicable	
CQUAD4 (33) Linear	2	Z1=Fiber distance 1	2	Z1=Fiber distance 1	
	3[1]	Normal x at Z1	3[1]	Normal x at Z1	RM
	4[1]	Normal y at Z1	4[1]	Normal x at Z1	IP
	5[1]	Shear xy at Z1	5[1]	Normal y at Z1	RM
	6	Shear angle at Z1	6[1]	Normal y at Z1	IP
	7	Major principal at Z1	7[1]	Shear xy at Z1	RM
	8	Minor principal at Z1	8[1]	Shear xy at Z1	IP
	9	von Mises or maximum shear at Z1	9	Z2=Fiber distance 2	
	10	Z2=Fiber distance 2	10[1]	Normal x at Z2	RM
	11[1]	NormaI x at Z2	11[1]	NormaI x at Z2	IP
	12[1]	Normal y at Z2	12[1]	Normal y at Z2	RM
	13[1]	Shear xy at Z2	13[1]	Normal y at Z2	IP
	14	Shear angle at Z2	14	Shear xy at Z2	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	15	Major principal at Z2	15	Shear xy at Z2	IP
	16	Minor principal at Z2			
	17	von Mises or maximum shear at Z2			
CQUAD4 (90) Nonlinear	2	Z1=Fiber distance 1 (Plane stress only)			
	3	Stress-X (at Z1, if plane stress)			
	4	Stress-Y (at Z1, if plane stress)			
	5	Stress-Z (Plane strain only)			
	6	Stress-XY (at Z1, if plane stress)			
	7	Equivalent stress (at Z1, if plane stress)			
	8	Plastic strain (at Z1, if plane stress)			
	9	Creep strain (at Z1, if plane stress)		Not applicable	
	10	Strain-X (at Z1, if plane stress)			
	11	Strain-Y (at Z1, if plane stress)			
	12	Strain-Z (plane strain only)			
	13	Strain-XY (at Z1, if plane stress)			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	14-25	Items 2 through 13 repeated for fiber distance Z2 (plane stress only)			
CQUAD4[2] (95) Composite	2	Lamina Number			
	3	Normal-1			
	4	Normal-2			
	5	Shear-12			
	6	Shear-1Z			
	7	Shear-2Z		Not applicable	
	8	Shear angle			
	9	Major principal			
	10	Minor principal			
	11	Maximum shear			
CQUAD4 (144) CORNER Output	1	EID	1	EID	
	2	CEN /	2	CEN /	
	3	4	3	4	
	4	Z1-Fiber distance	4	Z1-Fiber distance	
	5	Normal x at Z1	5	Normal x at Z1	RM
	6	Normal y at Z1	6	Normal x at Z1	IP
	7	Shear xy at Z1	7	Normal y at Z1	RM
	8	Shear angle at Z1	8	Normal y at Z1	IP
	9	Major principal at Z1	9	Shear xy at Z1	RM
	10	Minor principal at Z1	10	Shear xy at Z1	IP
	11	von Mises or maximum shear at Z1	11	Z2-Fiber distance	
	12	Z2-Fiber distance	12	Normal x at Z2	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	13	Normal x at Z2	13	Normal x at Z2	IP
	14	Normal y at Z2	14	Normal y at Z2	RM
	15	Shear xy at Z2	15	Normal y at Z2	IP
	16	Shear angle at Z2	16	Shear xy at Z2	RM
	17	Major principal at Z2	17	Shear xy at Z2	IP
	18	Minor principal at Z2	18	Grid 1	
	19	von Mises or maximum shear at Z2	19-32	Same as 4 through 17 for corner 1	
	20	Grid 1	33	Grid 2	
	21-36	Same as 4 through19 for corner 1	34-47	Same as 4 through 17 for corner 2	
	37	Grid 2	48	Grid 3	
	38-53	Same as 4 through19 for corner 2	49-62	Same as 4 through 17 for corner 3	
	54	Grid 3	63	Grid 4	
	55-70	Same as 4 through 19 for corner 3	64-77	Same as 4 through 17 for corner 4	
	71	Grid 4			
	72-87	Same as 4 through 19 for corner 4			
CQUAD8 (64)	5[1]	Normal x at Z1	5[1]	Normal x at Z1	RM
	6[1]	Normal y at Z1	6[1]	Normal x at Z1	IP
	7[1]	Shear xy at Z1	7[1]	Normal y at Z1	RM
	8	O Shear angle at Z1	8[1]	Normal y at Z1	IP
	9	Major principal at Z1	9[1]	Shear xy at Z1	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	10	Minor principal at Z1	10[1]	Shear xy at Z1	IP
	11	von Mises or maximum shear at Z1	12[1]	Normal x at Z2	RM
	13[1]	NormaI x at Z2	13[1]	NormaI x at Z2	IP
	14[1]	Normal y at Z2	14[1]	Normal y at Z2	RM
	15[1]	Shear xy at Z2	15[1]	Normal y at Z2	IP
	16	Q Shear angle at Z2	16[1]	Shear xy at Z2	RM
	17	Major principal at Z2	17[1]	Shear xy at Z2	IP
	18	Minor principal at Z2	20-32	Same as items 5 through 17 for corner 1	
	19	von Mises or maximum shear at Z2			
	19	von Mises or maximum shear at Z2	35-47	Same as items 5 through 17 for corner 2	
	22-36	Same as items 5 through 19 for corner 1		Same as items 5 through 17 for corner 2	
			50-62	Same as items 5 through 17 for corner 3	
	39-53	Same as items 5 through 19 for corner 2		Same as items 5 through 17 for corner 3	
	56-70	Same as items 5 through 19 for corner 3	65-77	Same as items 5 through 17 for corner 4	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	73-87	Same as items 5 through 19 for corner 4			
CQUAD8[2] (96) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CQUADFD (201) Nonlinear Finite Deformation with 4 grid points	2	Grid / Gauss			
	3	Grid / Gauss ID (0=center)			
	4	Cauchy stress-X			
	5	Cauchy stress-Y			
	6	Cauchy stress-Z			
	7	Cauchy stress-XY			
	8	Pressure $p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$		Not applicable	
	9	Volumetric strain J-1			
	10	Logarithmic strain-X			
	11	Logarithmic strain-Y			
	13	Logarithmic strain-XY			
	14-46	Items 3 through 13 repeated for 3 Gauss points			
CQUADFD (208) Nonlinear Finite Deformation with 8 or 9 grid points	2-13	Same as CQUADFD (201)			
	14-101	Items 3 through 13 repeated for 8 Gauss points		Not applicable	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CQUADR (82) Linear		Same as CQUAD4(144)		Same as CQUAD4(144)	
		CORNER output			
CQUADR (232) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CQUADXFD (214) Nonlinear Finite Deformation with 4 grid points	2	Grid / Gauss			
	3	Gauss ID			
	4	Cauchy stress-X (radial)			
	5	Cauchy stress-Y (axial)			
	6	Cauchy stress-Z (circumferential)			
	7	Cauchy stress-XY			
	8	Pressure $p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$		Not applicable	
	9	Volumetric strain J-1			
	10	Logarithmic strain-X (radial)			
	11	Logarithmic strain-Y (axial)			
	12	Logarithmic strain-Z (circumferential)			
	13	Logarithmic strain-XY			
	14-46	Items 3 through 13 repeated for remaining 3 Gauss points			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CQUADXFD (215) Nonlinear Finite Deformation with 8 or 9 grid points	2-13 14-101	Same as QUADXFD (214) Items 3 through 13 repeated for remaining 8 Gauss points		Not applicable	
CROD (1) Linear		Same as CONROD(10)		Same as CONROD(10)	
CROD (89) Nonlinear		Same as CONROD(92)		Not applicable	
CSHEAR (4)	2	Maximum shear	2	Maximum shear	RM
	3[1]	Average shear	3	Maximum shear	lP
	4	Safety margin	4[1]	Average shear	RM
			5[1]	Average shear	IP
CSLOT3 (50)	2	Radial centroid	2	Radial centroid	RM
	3	Axial centroid	3	Axial centroid	RM
	4	Tangential edge 1	4	Tangential edge 1	RM
	5	Tangential edge 2	5	Tangential edge 2	RM
	6	Tangential edge 3	6	Tangential edge 3	RM
			7	Radial centroid	IP
			8	Axialcentroid	IP
			9	Tangential edge 1	IP
			10	Tangential edge 2	IP
			11	Tangential edge 3	IP
CSLOT4 (51)	2	Radial centroid	2	Radial centroid	RM
	3	Axial centroid	3	Axial centroid	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	4	Tangential edge 1	4	Tangential edge 1	RM
	5	Tangential edge 2	5	Tangential edge 2	RM
	6	Tangential edge 3	6	Tangential edge 3	RM
	7	Tangential edge 4	7	Tangential edge 4	RM
			8	Radial centroid	IP
			9	Axial centroid	IP
			10	Tangential edge 1	lP
			11	Tangential edge 2	IP
			12	Tangential edge 3	IP
			13	Tangential edge 4	IP
CTETRA (39) Linear	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID (0=center)	5	External grid ID (0=center)	
	6[1]	Normal x	6[1]	Normal x	RM
	7[1]	Shear xy	7[1]	Normal y	RM
	8	First principal	8[1]	Normal z	RM
	9	First principal x cosine	9[1]	Shear xy	RM
	10	Second principal x cosine	10[1]	Shear yz	RM
	11	Third principal x cosine	11[1]	Shear zx	RM
	12	Mean pressure	12[1]	Normal x	IP

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	13	von Mises or octahedral shear stress	13[1]	Normal y	IP
	14[1]	Normal y	14[1]	Normal z	IP
	15[1]	Shear yz	15[1]	Shear xy	IP
	16	Second principal	16[1]	Shear yz	IP
	17	First principal y cosine	17[1]	Shear zx	IP
	18	Second principal y cosine	18-69	Items 5 through 17 repeated for four corners	
	19	Third principal y cosine			
	20[1]	Normal z			
	21[1]	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-109	Items 5 through 25 repeated for four corners			
CTETRA (85) Nonlinear	2	Grid / Gauss			
	3	External grid ID (0=center)			
	4	Stress-X			
	5	Stress-Y			
	6	Stress-Z			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	7	Stress-XY			
	8	Stress-YZ			
	9	Stress-ZX			
	10	Equivalent stress			
	11	Effective plastic strain		Not applicable	
	12	Effective creep strain			
	13	Strain-X			
	14	Strain-Y			
	15	Strain-Z			
	16	Strain-XY			
	17	Strain-YZ			
	18	Strain-ZX			
	19-82	Items 3 through 18 Repeated for four corners			
CTETRAFD (205) Nonlinear Finite Deformation with 4 grid points	2-17	Same as CHEXAFD (202)		Not applicable	
CTETRAFD (210) Nonlinear Finite Deformation witn 10 grid points	2-17	Same as CHEXAFD (202)			
	18-77	Items 3 through 17 repeated for 4 Gauss points		Not applicable	
CTRIA3 (74)		Same as CQUAD4(33)		Same as CQUAD4(33)	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CTRIA3[2] (97) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIA3 (88) Nonlinear		Same as CQUAD4(90)		Not applicable	
CTRIA6 (75) Linear	5[1]	Normal x at Z1	5[1]	Normal x at Z1	RM
	6[1]	Normal y at Z1	6[1]	Normal x at Z1	IP
	7[1]	Shear xy at Z1	7[1]	Normal y at Z1	RM
	8	Q shear angle at Z1	8[1]	Normal y at Z1	lP
	9	Major principal at Z1	9[1]	Shear xy at Z1	RM
	10	Minor principal at Z1	10[1]	Shear xy at Z1	lP
	11	von Mises or maximum shear at Z1	12[1]	Normal x at Z2	RM
	13[1]	NormaI x at Z2	13[1]	Normal x at Z2	IP
	14[1]	Normal y at Z2	14[1]	Normal y at Z2	RM
	15[1]	Shear xy at Z2	15[1]	Normal y at Z2	IP
	16	Q shear angle at Z2	16[1]	Shear xy at Z2	RM
	17	Major principal at Z2	17[1]	Shear xy at Z2	IP
	18	Minor principal at Z2	20-32	Same as items 5 through 17 for corner 1	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	19 22-36 39-53 56-70	von Mises or maximum shear at Z2 Same as items 5 through 19 for corner 1 Same as items 5 through 19 for corner 2 Same as items 5 through 19 for corner 3	35-47 50-62	Same as items 5 through 17 for corner 2 Same as items 5 through 17 for corner 3	
CTRIA6[2] (98)		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIAFD (206) Nonlinear Deformation with 3 grid points	2-13	Same as CQUADFD(201)		Not applicable	
CTRIAFD (211) Nonlinear Finite Deformation with 6 grid points	2-13 14-35	Same as CQUADFD (201) Items 3 through 12 repeated for 2 Gauss points		Not applicable	
CTRlAR (70) Linear		Same as CTRlA6(75)		Same as CTRlA6(75)	
CTRIAR (233) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIAX6 (53)	3[1] 4[1]	Radial Azimuthal	3[1] 4[1]	Radial Radial	RM IP

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	5[1]	Axial	5[1]	Azimuthal	RM
	6[1]	Shear stress	6[1]	Azimuthal	lP
	7	Maximum principal	7	Axial	RM
	8	Maximum shear	8	Axial	lP
	9	von Mises or octahedral	9	Shear	RM
	11-17	Same as items 3 through 9 for corner 1	10	Shear	IP
			12-19	Same as items 3 through 10 for corner 1	
	19-25	Same as Items 3 through 9 for corner 2			
			21-28	Same as items 3 through 10 for corner 2	
	27-33	Same as items 3 through 9 for corner 3			
			30-37	Same as items 3 through 10 for corner 3	
CTRIAXFD (212) Nonlinear Finite Deformation with 3 grid points	2-13	Same as CQUADXFD (214)		Not applicable	
CTRIAXFD (213) Nonlinear Finite Deformatio n with 6 grid points	2-13	Same as CQUADXFD (214)			
	14-35	Items 3 through 13 repeated for 2 Gauss points		Not applicable	
CTUBE (3) Linear		Same as CONROD (10)		Same as CONROD(10)	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CTUBE (87) Nonlinear		Same as CONROD(92)		Not applicable	
VUHEXA (145) VUPENTA (146) VUTETRA (147) for HEXAp, PENTAp, TETRAp if SDRPOPT= SDRP (**with** principals)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	VU grid ID for corner 1			
	4	Normal x	4	Normal x	RM
	5	Normal y	5	Normal y	RM
	6	Normal z	6	Normal z	RM
	7	Shear xy	7	Shear xy	RM
	8	Shear yz	8	Shear yz	RM
	9	Shear zx	9	Shear zx	RM
	10	First principal	10	Normal x	IP
	11	Second principal	11	Normal y	IP
	12	Third principal	12	Normal z	IP
	13	Mean pressure	13	Shear xy	IP
	14	von Mises / Octahedral	14	Shear yz	IP
			15	Shear zx	IP
	15-26	Repeat items 3-14 for corner 2	16-28	Repeat items 3-15 for corner 2	
	27-38	Repeat items 3-14 for corner 3	29-41	Repeat items 3-15 for corner 3	
	39-50	Repeat items 3-14 for corner 4	42-54	Repeat items 3-15 for corner 4	
	51-62	Repeat items 3-14 for corner 5 (VUPENTA, VUHEXA)	55-67	Repeat items 3-15 for corner 5 (VUPENTA, VUHEXA)	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	63-74	Repeat items 3-14 for corner 6 (VUPENTA, VUHEXA)	68-80	Repeat items 3-15 for corner 6 (VUPENTA, VUHEXA)	
	75-86	Repeat items 3-14 for corner 7 (VUHEXA)	81-93	Repeat items 3-15 for corner 7 (VUHEXA)	
	87-98	Repeat items 3-14 for corner 8 (VUHEXA)	94-106	Repeat items 3-15 for corner 8 (VUHEXA)	
VUHEXA (145) VUPENTA (146) VUTETRA (147) for HEXAp, PENTAp, TETRAp if SDRPOPT=OFP (**no** principals)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	VU grid ID for corner 1			
	4	Normal x	4	Normal x	RM
	5	Normal y	5	Normal y	RM
	6	Normal z	6	Normal z	RM
	7	Shear xy	7	Shear xy	RM
	8	Shear yz	8	Shear yz	RM
	9	Shear zx	9	Shear zx	RM
			10	Normal x	IP
			11	Normal y	IP
			12	Normal z	IP
			13	Shear xy	IP
			14	Shear yz	IP
			15	Shear zx	IP
	10-16	Repeat items 3-9 for corner 2	16-28	Repeat items 3-15 for corner 2	
	17-23	Repeat items 3-9 for corner 3	29-41	Repeat items 3-15 for corner 3	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	24-30	Repeat items 3-9 for corner 4	42-54	Repeat items 3-15 for corner 4	
	31-37	Repeat items 3-9 for corner 5 (VUPENTA, VUHEXA)	55-67	Repeat items 3-15 for corner 5 (VUPENTA, VUHEXA)	
	38-44	Repeat items 3-9 for corner 6 (VUPENTA, VUHEXA)	68-80	Repeat items 3-15 for corner 6 (VUPENTA, VUHEXA)	
	45-51	Repeat items 3-9 for corner 7 (VUHEXA)	81-93	Repeat items 3-15 for corner 7 (VUHEXA)	
	52-58	Repeat items 3-9 for corner 8 (VUHEXA)	94-106	Repeat items 3-15 for corner 8 (VUHEXA)	
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN= FIBER; 4th char. of ICORD= X Y, Z (**local** coordinate system); and SDRPORT =SDRP (**with** principals)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			
	7	VU grid ID for corner 1			
	8	Z1 fiber distance			
	9	Z2 fiber distance			
	10	Normal x at Z1	10	Normal x at Z1	RM
	11	Normal y at Z1	11	Normal y at Z1	RM
	12	Shear xy at Z1	12	Shear xy at Z1	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	13	Shear Angle at Z1	13	0.0	RM
	14	Major principal at Z1	14	0.0	RM
	15	Minor principal at Z1	15	0.0	RM
	16	vonMises / Max.Shear at Z1	16	Normal x at Z2	RM
	17	NormaI x at Z2	17	Normal y at Z2	RM
	18	Normal y at Z2	18	Shear xy at Z2	RM
	19	Shear xy at Z2	19	0.0	RM
	20	Shear Angle at Z2	20	0.0	RM
	21	Major principal at Z2	21	0.0	RM
	22	Minor principal at Z2	22	Normal x at Z1	IP
	23	vonMises / Max.Shear at Z2	23	Normal y at Z1	IP
			24	Shear xy at Z1	IP
			25	0.0	IP
			26	0.0	IP
			27	0.0	IP
			28	Normal x at Z2	IP
			29	Normal y at Z2	IP
			30	Shear xy at Z2	IP
			31	0.0	IP
			32	0.0	IP
			33	0.0	IP
	24-40	Repeat items 7-23 for corner 2	34-60	Repeat items 7-33 for corner 2	
	41-57	Repeat items 7-23 for corner 3	61-87	Repeat items 7-33 for corner 3	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	58-74	Repeat items 7-23 for corner 4 (VUQUAD)	88-104	Repeat items 7-33 for corner 4 (VUQUAD)	
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN= FIBER; if 4th char. of ICORD=X Y, Z (**local** coordinate system); and SDRPOPT =OFP (**no** principals)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat / curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			
	7	VU grid ID for corner 1			
	8	Z1 fiber distance			
	9	Z2 fiber distance			
	10	Normal x at Z1	10	Normal x at Z1	RM
	11	Normal y at Z1	11	Normal y at Z1	RM
	12	Shear xy at Z1	12	Shear xy at Z1	RM
	13	0.0	13	0.0	RM
	14	0.0	14	0.0	RM
	15	0.0	15	0.0	RM
	16	0.0	16	Normal x at Z2	RM
	17	Normal x at Z2	17	Normal y at Z2	RM
	18	Normal y at Z2	18	Shear xy at Z2	RM
	19	Shear xy at Z2	19	0.0	RM
	20	0.0	20	0.0	RM
	21	0.0	21	0.0	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	22	0.0	22	Normal x at Z1	IP
	23	0.0	23	Normal y at Z1	IP
			24	Shear xy at Z1	IP
			25	0.0	IP
			26	0.0	IP
			27	0.0	IP
			28	Normal x at Z2	IP
			29	Normal y at Z2	IP
			30	Shear xy at Z2	IP
			31	0.0	IP
			32	0.0	IP
			33	0.0	IP
	24-40	Repeat items 7-23 for corner 2	34-60	Repeat items 7-33 for corner 2	
	41-57	Repeat items 7-23 for corner 3	61-87	Repeat items 7-33 for corner 3	
	58-74	Repeat items 7-23 for corner 4 (VUQUAD)	88-104	Repeat items 7-33 for corner 4 (VUQUAD)	
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN= FIBER and 4th char. of ICORD=F (**fixed** coordinate system)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	7	VU grid ID for corner 1			
	8	Z1 fiber distance			
	9	Z2 fiber distance			
	10	Normal x at Z1	10	Normal x at Z1	RM
	11	Normal y at Z1	11	Normal y at Z1	RM
	12	Shear xy at Z1	12	Shear xy at Z1	RM
	13	Shear yz at Z1	13	Shear yz at Z1	RM
	14	Shear zx at Z1	14	Shear zx at Z1	RM
	15	Normal z at Z1	15	Normal z at Z1	RM
	16	NormaI x at Z2	16	Normal x at Z2	RM
	17	NormaI y at Z2	17	Normal y at Z2	RM
	18	Shear xy at Z2	18	Shear xy at Z2	RM
	19	Shear yz at Z2	19	Shear yz at Z2	RM
	20	Shear zx at Z2	20	Shear zx at Z2	RM
	21	Normal z at Z2	21	Normal z at Z2	RM
	22	0.0	22	Normal x at Z1	IP
	23	0.0	23	Normal y at Z1	IP
			24	Shear xy at Z1	IP
			25	Shear yz at Z1	IP
			26	Shear zx at Z1	IP
			27	Normal z at Z1	IP
			28	Normal x at Z2	IP
			29	Normal y at Z2	IP
			30	Shear xy at Z2	IP
			31	Shear yz at Z2	IP
			32	Shear zx at Z2	IP
			33	Normal z at Z2	IP

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	24-40	Repeat items 7-23 for corner 2	34-60	Repeat items 7-33 for corner 2	
	41-57	Repeat items 7-23 for corner 3	61-87	Repeat items 7-33 for corner 3	
	58-74	Repeat items 7-23 for corner 4 (VUQUAD)	88-104	Repeat items 7-33 for corner 4 (VUQUAD)	
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN= STRCUR and 4th char. of ICORD=X Y, Z (**local** coordinate system)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat / curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			
	7	VU grid ID for corner 1			
	8	0.0			
	9	0.0			
	10	Membrane Strain x	10	Membrane Strain x	RM
	11	Membrane Strain y	11	Membrane Strain y	RM
	12	Membrane Strain xy	12	Membrane Strain xy	RM
	13	0.0	13	0.0	RM
	14	0.0	14	0.0	RM
	15	0.0	15	0.0	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	16	Bending Curvature x	16	Bending Curvature x	RM
	17	Bending Curvature y	17	Bending Curvature y	RM
	18	Bending Curvature xy	18	Bending Curvature xy	RM
	19	Shear yz	19	Shear yz	RM
	20	Shear zx	20	Shear zx	RM
	21	0.0	21	0.0	RM
	22	0.0	22	Membrane Strain x	IP
	23	0.0	23	Membrane Strain y	IP
			24	Membrane Strain xy	lP
			25	0.0	IP
			26	0.0	IP
			27	0.0	IP
			28	Bending Curvature x	IP
			29	Bending Curvature y	IP
			30	Bending Curvature xy	IP
			31	Shear yz	IP
			32	Shear zx	IP
			33	0.0	IP
	24-40	Repeat items 7-23 for corner 2	34-60	Repeat items 7-33 for corner 2	
	41-57	Repeat items 7-23 for corner 3	61-87	Repeat items 7-33 for corner 3	
	58-74	Repeat items 7-23 for corner 4 (VUQUAD)	88-104	Repeat items 7-33 for corner 4 (VUQUAD)	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN=STRCUR and 4th char. of ICORD=F (**fixed** coordinate system)	1	VU element ID *10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			
	7	VU grid ID for corner 1			
	8	0.0			
	9	0.0			
	10	Membrane Strain x	10	Membrane Strain x	RM
	11	Membrane Strain y	11	Membrane Strain y	RM
	12	Membrane Strain xy	12	Membrane Strain xy	RM
	13	Membrane Strain yz	13	Membrane Strain yz	RM
	14	Membrane Strain zx	14	Membrane Strain zx	RM
	15	Membrane Strain z	15	Membrane Strain z	RM
	16	Bending Curvature x	16	Bending Curvature x	RM
	17	Bending Curvature y	17	Bending Curvature y	RM
	18	Bending Curvature xy	18	Bending Curvature xy	RM

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	19	Bending Curvature yz	19	Bending Curvature yz	RM
	20	Bending Curvature zx	20	Bending Curvature zx	RM
	21	Bending Curvature z	21	Bending Curvature z	RM
	22	0.0	22	Membrane Strain x	IP
	23	0.0	23	Membrane Strain y	IP
			24	Membrane Strain xy	IP
			25	Membrane Strain yz	IP
			26	Membrane Strain zx	IP
			27	Membrane Strain z	IP
			28	Bending Curvature x	lP
			29	Bending Curvature y	IP
			30	Bending Curvature xy	IP
			31	Bending Curvature yz	lP
			32	Bending Curvature zx	IP
			33	Bending Curvature z	IP
	24-40	Repeat items 7-23 for corner 2	34-60	Repeat items 7-33 for corner 2	
	41-57	Repeat items 7-23 for corner 3	61-87	Repeat items 7-33 for corner 3	
	58-74	Repeat items 7-23 for corner 4 (VUQUAD)	88-104	Repeat items 7-33 for corner 4 (VUQUAD)	

Table 6-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
VUBEAM (191) for BEAMp	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD (not used)			
	5	VU grid ID for end 1			
	6	x/L position			
	7	Y-coordinate of output point C			
	8	Z-coordinate of output point C			
	9	W-coordinate of output point C			
	10	Normal x at C	10	Normal x at C	RM
	11	Shear xy at C	11	Shear xy at C	RM
	12	Shear zx at C	12	Shear zx at C	RM
			13	Normal x at C	IP
			14	Shear xy at C	IP
			15	Shear zx at C	IP
	13-18	Repeat items 7-12 for output point D	16-24	Repeat items 7-15 for output point D	
	19-24	Repeat items 7-12 for output point E	25-33	Repeat items 7-15 for output point E	
	25-30	Repeat items 7-12 for output point F	34-42	Repeat items 7-15 for output point F	
	31	Max longitudinal			
	32	Min longitudinal			
	33-60	Repeat items 5-32 for end 2	43-80	Repeat items 5-42 for end 2	

6.2 Element Force Item Codes

All items are element forces (or moments) unless otherwise indicated.

Table 6-2 Element Force Item Codes

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
CBAR (34)	2[1]	Bending End A plane 1	2[1]	Bending End A plane 1	RM
	3[1]	Bending End A plane 2	3[1]	Bending End A plane 2	RM
	4[1]	Bending End B plane 1	4[1]	Bending End B plane 1	RM
	5[1]	Bending End B plane 2	5[1]	Bending End B plane 2	RM
	6[1]	Shear plane 1	6[1]	Shear plane 1	RM
	7[1]	Shear plane 2	7[1]	Shear plane 2	RM
	8[1]	Axial force	8[1]	Axial force	RM
	9[1]	Torque	9[1]	Torque	RM
			10[1]	Bending End A plane 1	IP
			11[1]	Bending End A plane 2	IP
			12[1]	Bending End B plane 1	IP
			13[1]	Bending End B plane 2	IP
			14[1]	Shear plane 1	IP
			15[1]	Shear plane 2	IP
			16[1]	Axial force	lP
			17[1]	Torque	IP
CBAR (100)	2	Station Distance/Length	2	Station Distance/Length	
	3	Bending Moment Plane 1	3	Bending Moment Plane 1	RM

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	4	Bending Moment Plane 2	4	Bending Moment Plane 2	RM
	5	Shear Force Plane 1	5	Shear Force Plane 1	RM
	6	Shear Force Plane 2	6	Shear Force Plane 2	RM
	7	Axial	7	Axial	RM
	Item codes are given for end A. Addition of the quantity (K-1) * 7 to the item code points to the same information for other stations, where K is the station number. K=8 for end B and 2 through 7 for intermediate stations.		8	Torque	RM
			9	Bending Moment Plane 1	IP
			10	Bending Moment Plane 2	IP
			11	Shear Force Plane 1	IP
			12	Shear Force Plane 2	IP
			13	Axial	IP
			14	Torque	IP
			(Item codes above are given for End A. For codes 2 through 14 at intermediate stations add (K-1) * 13 and K is the station number, and for codes at End B, K+number of stations plus 1.)		
CBEAM (2)	2	External grid point ID	2	External grid point ID	
	3	Station distance / length	3	Station distance / length	
	4[1]	Bending moment plane 1	4[1]	Bending moment plane 1	RM

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	5[1]	Bending moment plane 2	5[1]	Bending moment plane 2	RM
	6[1]	Web shear plane 1	6[1]	Web shear plane 1	RM
	7[1]	Web shear plane 2	7[1]	Web shear plane 2	RM
	8[1]	Axial force	8[1]	Axial force	RM
	9[1]	Total torque	9[1]	Total torque	RM
	10[1]	Warping torque	10[1]	Warping torque	RM
		(Item codes are given for end A. Addition of the quantity (K-1) 9 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)	11[1]	Bending moment plane 1	IP
			12[1]	Bending moment plane 2	IP
			13[1]	Web shear plane 1	lP
			14[1]	Web shear plane 2	IP
			15[1]	Axial force	IP
			16[1]	Total torque	IP
			17[1]	Warping torque	lP
				(Item codes are given for end A. Addition of the quantity (K-1) 16 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)	

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
CBEND (69)	2	External grid point ID	2	External grid point ID	
	3[1]	Bending moment plane 1	3[1]	Bending moment plane 1	RM
	4[1]	Bending moment plane 2	4[1]	Bending moment plane 2	RM
	5[1]	Shear plane 1	5[1]	Shear plane 1	RM
	6[1]	Shear plane 2	6[1]	Shear plane 2	RM
	7[1]	Axial force	7[1]	Axial force	RM
	8[1]	Torque (Item codes are given for end A. Item codes 9 through 15 point to the same information for end B.)	8[1]	Torque	RM
			9[1]	Bending moment plane 1	IP
			10[1]	Bending moment plane 2	IP
			11[1]	Shear plane 1	IP
			12[1]	Shear plane 2	IP
			13[1]	Axial force	lP
			14[1]	Torque (Item codes are given for end A. Item codes 15 through 27 point to the same information for end B.)	IP
CBUSH (102)	2	Force-x	2	Force-x	RM
	3	Force-y	3	Force-y	RM
	4	Force-z	4	Force-z	RM
	5	Moment-x	5	Moment-x	RM

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	6	Moment-y	6	Moment-y	RM
	7	Moment-z	7	Moment-z	RM
			8	Force-x	IP
			9	Force-y	IP
			10	Force-z	IP
			11	Moment-x	IP
			12	Moment-y	IP
			13	Moment-z	IP
CDAMP1 (20)		Same as CELAS1		Same as CELAS1	
CDAMP2 (21)		Same as CELAS1		Same as CELAS1	
CDAMP3 (22)		Same as CELAS1		Same as CELAS1	
CDAMP4 (23)		Same as CELAS1		Same as CELAS1	
CDUM3 thru CDUM9 (55-61)	2[1]	F1	2[1]	F1	RM
	3[1]	F2	3[1]	F2	RM
	4[1]	F3	4[1]	F3	RM
	5[1]	F4	5[1]	F4	RM
	6[1]	F5	6[1]	F5	RM
	7[1]	F6	7[1]	F6	RM
	8[1]	F7	8[1]	F7	RM
	9[1]	F8	9[1]	F8	RM
	10[1]	F9	10[1]	F9	RM
			11[1]	F1	IP
			12[1]	F2	lP

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
			13[1]	F3	IP
			14[1]	F4	lP
			15[1]	F5	IP
			16[1]	F6	IP
			17[1]	F7	IP
			18[1]	F8	IP
			19[1]	F9	IP
CELAS1 (11)	2[1]	Force	2[1]	Force	RM
CELAS2 (12)		Same as CELAS1		Same as CELAS1	
CELAS3 (13)		Same as CELAS1		Same as CELAS1	
CELAS4 (14)		Same as CELAS1		Same as CELAS1	
CGAP (38)	2	Normal x		Not applicable	
	3	Shear y			
	4	Shear z			
	5	Axial u			
	6	Shear v			
	7	Shear w			
	8	Slip v			
	9	Slip w			
CONROD (10)	2[1]	Axial force	2[1]	Axial force	RM
	3[1]	Torque	3[1]	Axial force	IP
			4[1]	Torque	RM
			5[1]	Torque	lP

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
CQUAD4 (33) Linear	2[1]	Membrane force x	2[1]	Membrane force x	RM
	3[1]	Membrane force y	3[1]	Membrane force y	RM
	4[1]	Membrane force xy	4[1]	Membrane force xy	RM
	5[1]	Bending moment x	5[1]	Bending moment x	RM
	6[1]	Bending moment y	6[1]	Bending moment y	RM
	7[1]	Bending moment xy	7[1]	Bending moment xy	RM
	8[1]	Shear x	8[1]	Shear x	RM
	9[1]	Shear y	9[1]	Shear y	RM
			10[1]	Membrane force x	IP
			11[1]	Membrane force y	IP
			12[1]	Membrane force xy	IP
			13[1]	Bending moment x	IP
			14[1]	Bending moment y	IP
			15[1]	Bending moment xy	IP
			16[1]	Shear x	IP
			17[1]	Shear y	IP
CQUAD4 (95) Composite	2-3	Theory or blank		Not applicable	
	4	Lamina number			
	5	FP (failure index) / SP (strength ratio) for direct stresses			
	6	Failure mode for Maximum strain theory			

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	7	FB (failure index) / SB (strength ratio) or -1 for interlaminar shear-stress			
	8	MAX of FP, FB or -1 or MIN of SP, SB or -1			
	9	Failure flag			
CQUAD4 (144) Corner Output	1	EID	1	EID	
	2	CEN /	2	CEN /	
	3	4	3	4	
	4	Membrane x	4	Membrane x	RM
	5	Membrane y	5	Membrane y	RM
	6	Membrane xy	6	Membrane xy	RM
	7	Bending x	7	Bending x	RM
	8	Bending y	8	Bending y	RM
	9	Bending xy	9	Bending xy	RM
	10	Shear x	10	Shear x	RM
	11	Shear y	11	Shear y	RM
	12	Grid 1	12	Membrane x	IP
	13-20	Same as 4 through 11 for corner 1	13	Membrane y	IP
			14	Membrane xy	IP
	21	Grid 2	15	Bending x	IP
	22-29	Same as 4 through 11 for corner 2	16	Bending y	IP
			17	Bending xy	IP
	30	Grid 3	18	Shear x	IP
	31-38	Same as 4 through 11 for corner 3	19	Shear y	IP
			20	Grid 1	

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	39	Grid 4	21-36	Same as 4 through 19 for corner 1	
	40-47	Same as 4 through 11 for corner 4	37	Grid 2	
			38-53	Same as 4 through 19 for corner 2	
			54	Grid 3	
			55-70	Same as 4 through 19 for corner 3	
			71	Grid 4	
			71-87	Same as 4 through 19 for corner 4	
CQUAD8 (64) Linear	4[1]	Membrane force x	4[1]	Membrane force x	RM
	5[1]	Membrane force y	5[1]	Membrane force y	RM
	6[1]	Membrane force xy	6[1]	Membrane force xy	RM
	7[1]	Bending moment x	7[1]	Bending moment x	RM
	8[1]	Bending moment y	8[1]	Bending moment y	RM
	9[1]	Bending moment xy	9[1]	Bending moment xy	RM
	10[1]	Shear x	10[1]	Shear x	RM
	11[1]	Shear y	11[1]	Shear y	RM
	13-20	Same as items 4 through 11 for corner 1	12[1]	Membrane force x	IP
	22-29	Same as items 4 through 11 for corner 2	13[1]	Membrane force y	IP
	31-38	Same as items 4 through 11 for corner 3	14[1]	Membrane force xy	IP
	40-47	Same as items 4 through 11 for corner 4	15[1]	Bending moment x	lP
			16[1]	Bending moment y	IP

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
			17[1]	Bending moment xy	IP
			18[1]	Shear x	lP
			19[1]	Shear y	IP
			21-36	Same as items 4 through 19 for corner 1	
			38-53	Same as items 4 through 19 for corner 2	
			55-70	Same as items 4 through 19 for corner 3	
			72-87	Same as items 4 through 19 for corner 4	
CQUAD8[2] (96) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CQUADR (82)		Same as CQUAD8(64)		Same as CQUAD8(64)	
CROD (1)		Same as CONROD(10)		Same as CONROD(10)	
CSHEAR (4)	2[1]	Force 4 to 1	2[1]	Force 4 to 1	RM
	3[1]	Force 2 to 1	3[1]	Force 2 to 1	RM
	4[1]	Force 1 to 2	4[1]	Force 1 to 2	RM
	5[1]	Force 3 to 2	5[1]	Force 3 to 2	RM
	6[1]	Force 2 to 3	6[1]	Force 2 to 3	RM
	7[1]	Force 4 to 3	7[1]	Force 4 to 3	RM
	8[1]	Force 3 to 4	8[1]	Force 3 to 4	RM
	9[1]	Force 1 to 4	9[1]	Force 1 to 4	RM

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	10[1]	Kick force on 1	10[1]	Force 4 to 1	IP
	11[1]	Shear 12	11[1]	Force 2 to 1	lP
	12[1]	Kick force on 2	12[1]	Force 1 to 2	IP
	13[1]	Shear 23	13[1]	Force 3 to 2	IP
	14[1]	Kick force on 3	14[1]	Force 2 to 3	lP
	15[1]	Shear 34	15[1]	Force 4 to 3	IP
	16[1]	Kick force on 4	16[1]	Force 3 to 4	IP
	17[1]	Shear 41	17[1]	Force 1 to 4	IP
			18[1]	Kick force on 1	RM
			19[1]	Shear 12	RM
			20[1]	Kick force on 2	RM
			21[1]	Shear 23	RM
			22[1]	Kick force on 3	RM
			23[1]	Shear 34	RM
			24[1]	Kick force on 4	RM
			25[1]	Shear 41	RM
			26[1]	Kick force on 1	IP
			27[1]	Shear 12	lP
			28[1]	Kick force on 2	IP
			29[1]	Shear 23	IP
			30[1]	Kick force on 3	IP
			31[1]	Shear 34	IP
			32[1]	Kick force on 4	IP
			33[1]	Shear 41	IP

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
CTRIA3 (74) Linear		Same as CQUAD4(33)		Same as CQUAD4(33)	
CTRIA3[2] (97) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRlA6 (75) Linear	4[1]	Membrane force x	4[1]	Membrane force x	RM
	5[1]	Membrane force y	5[1]	Membrane force y	RM
	6[1]	Membrane force xy	6[1]	Membrane force xy	RM
	7[1]	Bending moment x	7[1]	Bending moment x	RM
	8[1]	Bending moment y	8[1]	Bending moment y	RM
	9[1]	Bending moment xy	9[1]	Bending moment xy	RM
	10[1]	Shear x	10[1]	Shear x	RM
	11[1]	Shear y	11[1]	Shear y	RM
	13-20	Same as items 4 through 11 for corner 1	12[1]	Membrane force x	IP
			13[1]	Membrane force y	IP
	22-29	Same as items 4 through 11 for corner 2	14[1]	Membrane force xy	IP
			15[1]	Bending moment x	IP
	31-38	Same as items 4 through 11 for corner 3	16[1]	Bending moment y	IP
			17[1]	Bending moment xy	IP
			18[1]	Shear x	IP
			19[1]	Shear y	IP
			21-36	Same as items 4 through 19 for corner 1	

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
			38-53	Same as items 4 through 19 for corner 2	
			55-70	Same as items 4 through 19 for corner 3	
CTRIA6[2] (98) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIAR (70)		Same as CTRIA6(75)		Same as CTRIA6(75)	
CTUBE (3)		Same as CONROD(10)		Same as CONROD(10)	
CVISC (24)		Not applicable		Same as CONROD(10)	
CWELDP (118) if PARTPAT or ELPAT	2	mz bending end A plane 1	2	mz bending end A plane 1	RM
	3	my bending end A plane 2	3	my bending end A plane 2	RM
	4	mz bending end B plane 1	4	mz bending end B plane 1	RM
	5	my bending end B plane 2	5	my bending end B plane 2	RM
CWELDC (117) if MSET = OFF	6	fy shear force plane 1	6	fy shear force plane 1	RM
	7	fz shear force plane 2	7	fz shear force plane 2	RM
	8	fx axial force	8	fx axial force	RM
	9	mx torque	9	mx torque	RM
			10	mz bending end A plane 1	IP
			11	my bending end A plane 2	lP

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
CWELD (200) if MSET = ON			12	mz bending end B plane 1	IP
			13	my bending end B plane 2	IP
			14	fy shear force plane 1	lP
			15	fz shear force plane 2	IP
			16	fx axial force	IP
			17	mx torque	IP
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if 4th char. of ICORD=X Y, Z (**local** coordinate system)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	0.0			
	7	VU grid ID for corner 1			
	8	Membrane Force x	8	Membrane Force x	RM
	9	Membrane Force y	9	Membrane Force y	RM
	10	Membrane Force xy	10	Membrane Force xy	RM
	11	0.0	11	0.0	RM
	12	0.0	12	0.0	RM
	13	0.0	13	0.0	RM
	14	Bending Moment x	14	Bending Moment x	RM

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	15	Bending Moment y	15	Bending Moment y	RM
	16	Bending Moment xy	16	Bending Moment xy	RM
	17	Shear zx	17	Shear zx	RM
	18	Shear yz	18	Shear yz	RM
	19	0.0	19	0.0	RM
			20	Membrane Force x	IP
			21	Membrane Force y	IP
			22	Membrane Force xy	lP
			23	0.0	IP
			24	0.0	IP
			25	0.0	IP
			26	Bending Moment x	IP
			27	Bending Moment y	lP
			28	Bending Moment xy	lP
			29	Shear zx	lP
			30	Shear yz	lP
			31	0.0	IP
	20-32	Repeat items 7-19 for corner 2	32-56	Repeat items 7-31 for corner 2	
	33-45	Repeat items 7-19 for corner 3	57-81	Repeat items 7-31 for corner 3	
	46-58	Repeat items 7-19 for corner 4 (VUQUAD)	82-106	Repeat items 7-31 for corner 4 (VUQUAD)	

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if 4th char. of ICORD=F (**fixed** coordinate system)	1	VU element ID *10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	0.0			
	7	VU grid ID for corner 1			
	8	Membrane Force x	8	Membrane Force x	RM
	9	Membrane Force y	9	Membrane Force y	RM
	10	Membrane Force xy	10	Membrane Force xy	RM
	11	Membrane Force yz	11	Membrane Force yz	RM
	12	Membrane Force zx	12	Membrane Force zx	RM
	13	Membrane Force z	13	Membrane Force z	RM
	14	Bending Moment x	14	Bending Moment x	RM

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	15	Bending Moment y	15	Bending Moment y	RM
	16	Bending Moment xy	16	Bending Moment xy	RM
	17	Bending Moment yz	17	Bending Moment yz	RM
	18	Bending Moment zx	18	Bending Moment zx	RM
	19	Bending Moment z	19	Bending Moment z	RM
			20	Membrane Force x	IP
			21	Membrane Force y	IP
			22	Membrane Force xy	IP
			23	Membrane Force yz	IP
			24	Membrane Force zx	IP
			25	Membrane Force z	IP
			26	Bending Moment x	lP
			27	Bending Moment y	IP
			28	Bending Moment xy	IP
			29	Bending Moment yz	lP
			30	Bending Moment zx	IP
			31	Bending Moment z	IP

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	20-32	Repeat items 7-19 for corner 2	32-56	Repeat items 7-31 for corner 2	
	33-45	Repeat items 7-19 for corner 3	57-81	Repeat items 7-31 for corner 3	
	46-58	Repeat items 7-19 for corner 4 (VUQUAD)	82-106	Repeat items 7-31 for corner 4 (VUQUAD)	

Table 6-2 Element Force Item Codes (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
VUBEAM (191) for BEAMp	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD (not used)			
	5	VU grid ID for end 1			
	6	x/L position			
	7	Force x	7	Force x	RM
	8	Shear Force y	8	Shear Force y	RM
	9	Shear Force z	9	Shear Force z	RM
	10	Torsional Moment x	10	Torsional Moment x	RM
	11	Bending Moment y	11	Bending Moment y	RM
	12	Bending Moment z	12	Bending Moment z	RM
			13	Force x	IP
			14	Shear Force y	IP
			15	Shear Force z	IP
			16	Torsional Moment x	lP
			17	Bending Moment y	IP
			18	Bending Moment z	IP
	13-20	Repeat items 5-12 for end 2	19-32	Repeat items 5-18 for end 2	

6.3 Fluid Virtual Mass Pressure Item Codes

Table 6-3 Fluid Virtual Mass Pressure Item Codes

Element Name	Real Fluid Pressure		Complex Fluid Pressure		
	Code	Item	Code	Item	Real/Mag. or Imag./Phase
Plate	2	Fluid pressure	2	Pressure	RM
Family			3	Pressure	IP

Table 6-4 Heat Transfer Item Codes

(Curve type is FLUX.)

Element Name (Code)	Code	Item
Heat Transfer Elements	2	Element type
	3[1]	
	4[1]	x gradient
	5[1]	y gradient
	6[1]	z gradient
	7	x flux
	8	y flux
	9	z flux
CHBDYE (107)	4	Applied load
	5	Free convection
	6	Forced convection
	7	Radiation
	8	Total
CHBDYG (108)	Same as CHBDYE	Same as CHBDYE
CHBDYP (109)	Same as CHBDYE	Same as CHBDYE

6.4 2D Slideline and 3D Surface Contact Item Codes

Table 6-5 Contact Item Codes

Element Name (Code)	Real Element Data		
	Item Code	Slideline Item	3D Surface Item (SOL 600)
CSLIFID	1	Slave grid point	
(116)	2	Contact region identification number	(39) Contact Touched Body
	3	Master grid 1	
	4	Master grid 2	
	5	Surface coordinate	(38) Contact Status
	6	Normal force	(35) Normal Force
	7	Shear force	(37) Friction Force
	8	Normal stress	(34) Normal Stress
	9	Shear stress	(36) Friction Stress
	10	Normal gap	
	11	Slip	
	12	Slip ratio (Shear force / u*normal force)	
	13-14	Slip code (Character)	

Notes:

1. Numbers in parenthesis refer to MARCOUT nodal post codes.
2. 3D Surface contact is available in SOL 600 only.

6.5 Element Strain Energy Item Codes

Table 6-6 Element Strain Energy Item Codes

Element Name	Real Element Data	
	Item Code	Item
Element groups A and B	2	Element strain energy
Element groups A and B	3	Percent of total energy
Element group A	4	Element strain energy density

Remark:

1. Element group A includes elements of CBAR, CBEAM, CBEND, CONROD, CHEXA, CPENTA, CQUAD4, CQUADR, CROD, CSHEAR, CTETRA, CTRIA3, CTRIA6, CTRIAR, AND CTUBE. Element group B includes elements of CELAS1, CELAS2, CELAS3, AND CGAP.

SECTION
7

Degree-of-Freedom Sets

- Degree-of-Freedom Set Definitions
- Degree-of-Freedom Set Bulk Data Entries

7.1 Degree-of-Freedom Set Definitions

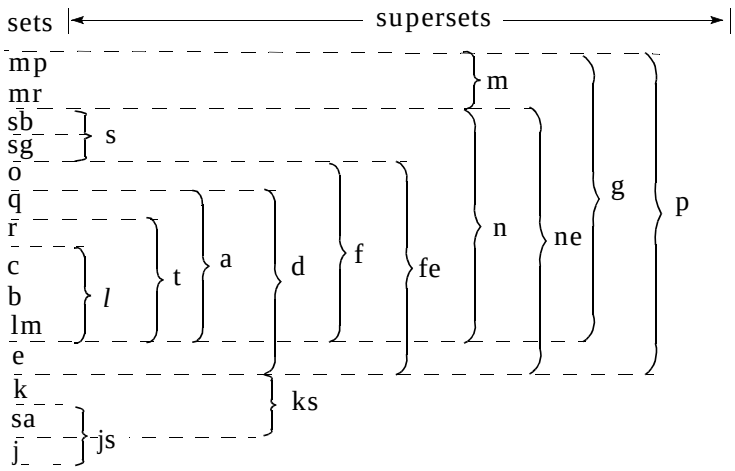
Each degree-of-freedom is a member of one mutually exclusive set. Set names have the following definitions:

Set Name	Definition
mp	Degrees-of-freedom eliminated by multipoint constraints.
mr	Degrees-of-freedom eliminated by multipoint constraints created by the rigid elements using the LGELIM method on the Case Control command RIGID.
sb*	Degrees-of-freedom eliminated by single-point constraints that are included in boundary condition changes and by the AUTOSPC feature.
sg*	Degrees-of-freedom eliminated by single-point constraints that are specified on the PS field on GRID Bulk Data entries.
o	Degrees-of-freedom omitted by structural matrix partitioning.
q	Generalized degrees-of-freedom assigned to component modes and residual vectors.
r	reference degrees-of-freedom used to determine free body motion.
c	Degrees-of-freedom that are free during component mode synthesis or dynamic reduction.
b	Degrees-of-freedom fixed during component mode analysis or dynamic reduction.
lm	Lagrange multiplier degrees-of-freedom created by the rigid elements using the LAGR method on the Case Control command, RIGID.
e	extra degrees-of-freedom introduced in dynamic analysis.
sa	Permanently constrained aerodynamic degrees-of-freedom.
k	Aerodynamic mesh point set for forces and displacements on the aero mesh.
j	Aerodynamic mesh collocation point set (exact physical interpretation is dependent on the aerodynamic theory).

*Strictly speaking, sb and sg are not exclusive with respect to one another. Degrees-of-freedom may exist in both sets simultaneously. Since these sets are not used explicitly in the solution sequences, this need not concern the user. However, those who use these sets in their own DMAPs should avoid redundant specifications

when using these sets for partitioning or merging operations. That is, a degree-of-freedom should not be specified on both a PS field of a GRID entry (sg set) and on a selected SPC entry (sb set). Redundant specifications will cause UFM 2120 in the VEC module and behavior listed in *MD Nastran 2006 DMAP Programmer's Guide* for the UPARTN module. These sets are exclusive, however, from the other mutually exclusive sets.

Each degree-of-freedom is also a member of one or more combined sets called "supersets." Supersets have the following definitions:



Set Name	**Meaning** (+ indicates union of two sets)
s = sb + sg	all degrees-of-freedom eliminated by single point constraints
l = b + c + *l*m	the structural degrees-of-freedom remaining after the reference degrees-of-freedom are removed (degrees-of-freedom left over)
t = *l* + r	the total set of physical boundary degrees-of-freedom for superelements
a = t + q	the set assembled in superelement analysis
d = a + e	the set used in dynamic analysis by the direct method
f = a + o	unconstrained (free) structural degrees-of-freedom
fe = f + e	free structural degrees-of-freedom plus extra degrees-of-freedom
n = f + s	all structural degrees-of-freedom not constrained by multipoint constraints
ne = n + e	all structural degrees-of-freedom not constrained by multipoint constraints plus extra degrees-of-freedom
m = mp + mr	all degrees-of-freedom eliminated by multiple constraints
g = n + m	all structural (grid) degrees-of-freedom including scalar degrees-of-freedom
p = g + e	all physical degrees-of-freedom

Set Name	Meaning (+ indicates union of two sets)
ks = k + sa	the union of k and the re-used s-set (6 dof per grid)
js = j + sa	the union of j and the re-used s-set (6 dof per grid)
fr = o + *l*	statically independent set minus the statically determinate supports (fr = f – q – r)
v = o + c + r	the set free to vibrate in dynamic reduction and component mode synthesis

The a-set and o-set are created in the following ways:

1. If only OMITi entries are present, then the o-set consists of degrees-of-freedom listed explicitly on OMITi entries. The remaining f-set degrees-of-freedom are placed in the b-set, which is a subset of the a-set.
2. If ASETi or QSETi entries are present, then the a-set consists of all degrees-of-freedom listed on ASETi entries and any entries listing its subsets, such as QSETi, SUPORTi, CSETi, and BSETi entries. Any OMITi entries are redundant. The remaining f-set degrees-of-freedom are placed in the o-set.
3. If there are no ASETi, QSETi, or OMITi entries present but there are SUPORTi, BSETi, or CSETi entries present, then the entire f-set is placed in the a-set and the o-set is not created.
4. There must be at least one explicity ASETi, QSETi, or OMITi entry for the o-set to exist, even if the ASETi, QSETi, or OMITi entry is redundant.

In dynamic analysis, additional vector sets are obtained by a modal transformation derived from real eigenvalue analysis of the a-set. These sets are as follows:

ξ_o = rigid body (zero frequency) modal degrees-of-freedom

ξ_f = finite frequency modal degrees-of-freedom

ξ_i = $\xi_o + \xi_f$, the set of all modal degrees-of-freedom

One vector set is defined that combines physical and modal degrees-of-freedom:

u_h = $\xi_i + u_e$, the set of all modal degrees-of-freedom

The membership of each degree-of-freedom can be printed by use of the Bulk Data entries PARAM,USETPRT and PARAM,USETSEL.

7.2 Degree-of-Freedom Set Bulk Data Entries

Degrees-of-freedom are placed in sets as specified by the user on the following Bulk Data entries:

Name	Bulk Data Entry Name
m	MPC, MPCADD, MPCAX, POlNTAX, RBAR, RBE1, RBE2, RBE3, RROD, RSPLINE, RTRPLT, GMBC, GMSPC*
sb	SPC, SPC1, SPCADD, SPCAX, FLSYM, GMSPC*, BNDGRID, (PARAM,AUTOSPC,YES)
sg	GRID, GRIDB, GRDSET (PS field)
o	OMIT, OMIT1, OMlTAX, GRID (SEID field), SESET
q	QSET, QSET1
r	SUPORT, SUPORT1, SUPAX
c	CSET, CSET1
b	BSET, BSET1
e	EPOINT
sa	CAEROi
k	CAEROi
a	ASET, ASET1, Superelement exterior degrees-of-freedom, CSUPEXT

*Placed in set only if constraints are not specified in the basic coordinate system.

In superelement analysis, the appropriate entry names are preceded by the letters SE, and have a field reserved for the superelement identification number. This identification is used because a boundary (exterior) grid point may be in one mutually exclusive set in one superelement and in a different set in the adjoining superelement. The SE-type entries are internally translated to the following types of entry for the referenced superelement:

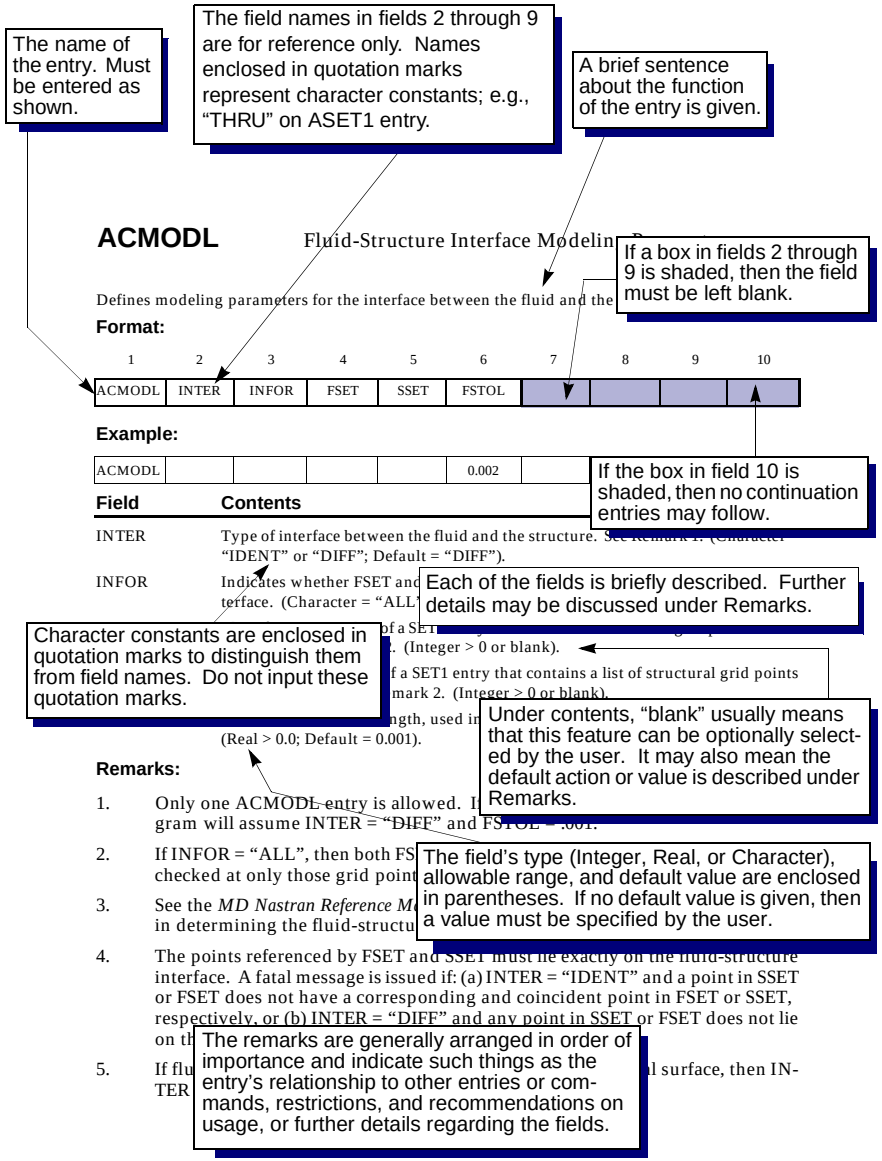
Entry Type	Equivalent Type
SEQSETi	QSETi
SESUP	SUPORT
SECSETi	CSETi
SEBSETi	BSETi

CHAPTER

8 Bulk Data Entries

- Key to Descriptions
- Bulk Data Entry Descriptions

8.1 Key to Descriptions



The Bulk Data Section

The Bulk Data Section contains entries that specify model geometry, element connectivity, element and material properties, constraints (boundary conditions), and loads. Some entries, such as loads and constraints, are selected by an appropriate Case Control command.

Entries are prepared in either fixed or free field format. The descriptions in this section show only the fixed format. Entries that are used by the MSGMESH program are not included in this guide. For a description of the various format options and the MSGMESH entries, see the "**Use of Parameters**" on page 35 of the *MSC.Nastran Reference Guide*.

8.2 Bulk Data Entry Descriptions

Each Bulk Data entry is described as follows:

Description

A brief sentence about the function of the entry is given.

Format

The name of the entry is given in the first field. The subsequent fields are described under the Field and Contents Section. Shaded fields must be left blank. If field 10 is shaded, then no continuation entries are permitted. Character strings enclosed in quotation marks must be specified without the quotation marks as shown in the example.

Example

A typical example is given.

Field and Contents

Each of the fields 2 through 9 that are named in the Format section is briefly described under Contents. The field's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The field must be specified by the user if no default value is given.

Remarks

The remarks in the Remarks Section are generally arranged in order of importance and indicate such things as how the Bulk Data entry is selected in the Case Control Section, its relationship to other entries, restrictions and recommendations on its use, and further descriptions of the fields.

Format of Bulk Data Entries

Real, Integer, and Character Input Data

MD Nastran is quite particular about the input requirements for data entry. The three possible types of data entries are Integer, Real, and Character (sometimes called literal, or BCD-binary coded decimal). The three types of data are described as follows:

Integer	Cannot contain a decimal point.
Real	Must contain a decimal point.
Character	Can be alphanumeric, but must always start with an alpha character and be 8 characters or less in length.

Real numbers may be entered in a variety of ways. For example, the following are all acceptable versions of the real number seven:

7.0 .7E1 0.7+1

.70+1 7.E+0 70.-1

Free, Small, and Large Field Formats

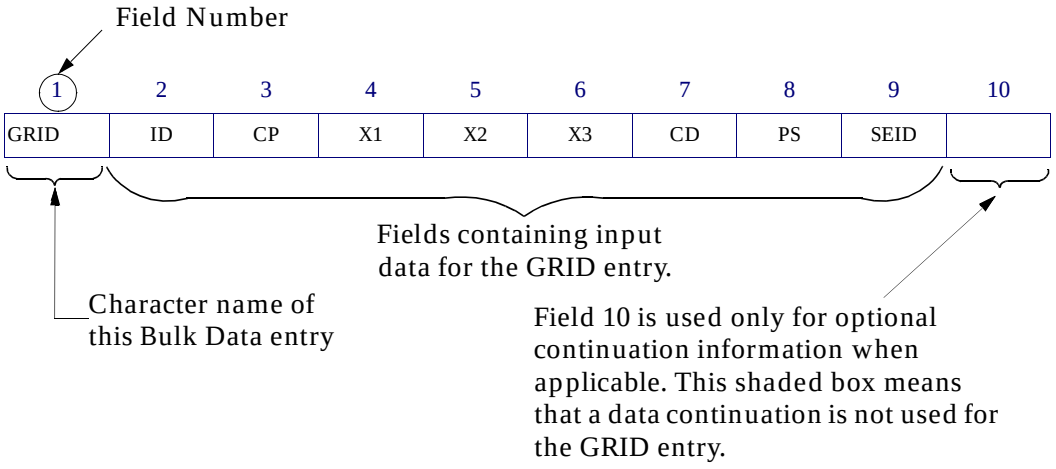
MD Nastran has three different field formats for input data:

Free Field Format	Input data fields are separated by commas.
Small Field Format	Ten fields of eight characters each.
Large Field Format	Ten fields-fields containing actual data are sixteen characters each. Large fields are used when greater numerical accuracy is required.

The NASTRAN statement, File Management Section, Executive Control Section, and Case Control Section use free field format. The Bulk Data Section allows the use of any of the three formats.

MD Nastran Bulk Data contains ten fields per input data entry. The first field contains the character name of the Bulk Data item (e.g., GRID, CBAR, MAT1, etc.). Fields two through nine contain data input information for the Bulk Data entry. The tenth field never contains data-it is reserved for entry continuation information, if applicable.

Consider the format of a typical MD Nastran Bulk Data entry, the GRID entry, which is used in MD Nastran to describe the geometry of the structural model.



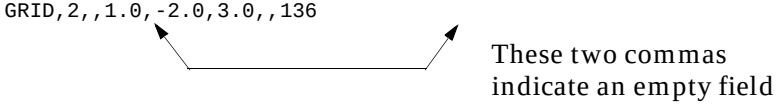
Example:

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

We will now represent this example in free field, small field, and large field formats.

Free Field Format

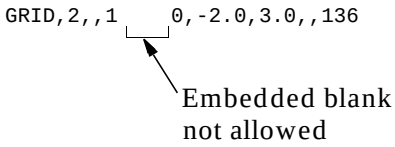
In free field format, data fields are separated by commas or blanks (commas are strongly recommended). The following shows the GRID Bulk Data entry example in free field format:



The rules for free field format are as follows:

- Free field data entries must start in column 1.
- To skip one field, use two commas in succession. To skip two fields, use three commas in succession (and so on).
- Integer or character fields with more than eight characters cause a fatal error.
- Real numbers with more than eight characters are rounded off and lose some precision. For example, an entry of 1.2345678+2 becomes 123.4568. If more significant digits are needed, use the large field format.

- Free field data cannot contain embedded blanks. An example of a free field embedded blank is shown:



- A dollar sign terminates the entry and comments may follow.

The free field data entry capability in MD Nastran have been enhanced to support easy to use data input formats. The following examples illustrate the possible forms of the free field data input and the resulting translation to the fixed-field format.

Entry with or without user continuation mnemonics.

```
MATT9,1101,2 ,3 ,4 ,,,,8 ,+P101
+P101,9 ,,,,13
```

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1101	2	3	4				8	+P101
+P101	9				13				

```
GRID,100,,1.0,0.0,0.0,,456
```

Translates to:

GRID	100		1.0	0.0	0.0		456		

The continuation mnemonics can be included with the data, provided that the data are within 80 characters. For example, the free field entry with continuation mnemonics.

```
SPC1,100,12456,1,2,3,4,5,6+SPC-A,+SPC-A,7,8,9,10
```

Translates to:

1	2	3	4	5	6	7	8	9	10
SPC1	100	12456	1	2	3	4	5	6	+SPC-A
+SPC-A	7	8	9	10					

In the second form, the continuation mnemonics are not included because they are not required. This is illustrated by the entry with automatic continuation:

```
SPC1,100,12456,1,2,3,4,5,6,7,8,9,10
```

Translates to:

1	2	3	4	5	6	7	8	9	10
SPC1	100	12456	1	2	3	4	5	6	
	7	8	9	10					

If more than 80 characters of data are required, the free field may be continued in the next line provided that the next entry starts with a comma in the first column. The next entry will be a logical continuation of the first. For example, the free-field entry:

```
MATT9,1151,2 ,3 ,4 ,,,,8
,9 ,,,,13
```

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1151	2	3	4				8	
	9				13				

Which is equivalent to:

```
MATT9,1151,2 ,3 ,4 ,,,,8 ,+
+,9 ,,,,13
```

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1151	2	3	4				8	+
+	9				13				

The free field data entry can be used to input mixed Small Field, Large Field continuations. Note that the plus (+) and asterisk (*) characters are used to indicate Small Field and Large Field input form respectively when free field dta entry is used. For example, the entries:

```
MATT9*,1302,2 ,,4 ,+
+,,,,,13
```

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9*	1302		2				4		+
+					13				

```
MATT9,1303,2 ,3 ,4 ,,,,8 ,+
*,9 ,,,,+
*,13
```

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1303	2	3	4				8	+
+	9								
*	13								

```
MATT9,1355,2 ,3 ,,5 ,,,8 ,+
*,,10 ,,,+
+,17
```

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1355	2	3		5			8	+
+			10						+
+	17								

System cell 363 must be set to 1 (i.e., system(363)=1, or STRICTUAI=1) if the free-field entry is continued by terminating the parent with a comma. The next entry will be a logical continuation of the first. It is not required to end the first entry at any specific point. This is illustrated by the entry:

CHEXA,200, 200, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19, 20

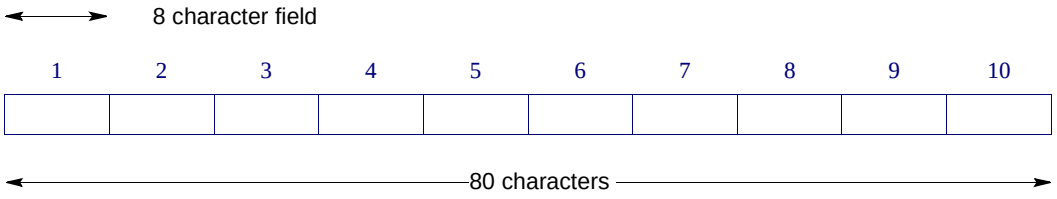
Translates to:

1	2	3	4	5	6	7	8	9	10
CHEXA	200	200	1	2	3	4	5	6	
	7	8	9	10	11	12	13	14	
	15	16	17	18	19	20			

Because of the feature allowing more than 10 fields of data to be entered on one free field entry, IT IS NOT ALLOWED to terminate a single free field entry with a comma.

Small Field Format

Small field format separates a Bulk Data entry into ten equal fields of eight characters each:

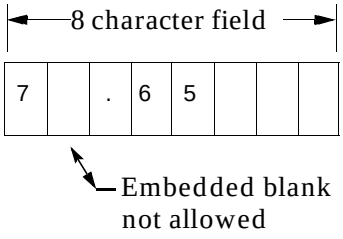


The following is an example of the GRID entry in small field format:

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

The rules for small field format are as follows:

- Fields 1 and 10 must be left justified.
- Fields 2 through 9 do not need to be either right or left justified, although aligning the data fields is good practice.
- Small field input data cannot contain any embedded blanks. An example of a small field embedded blank is shown below:



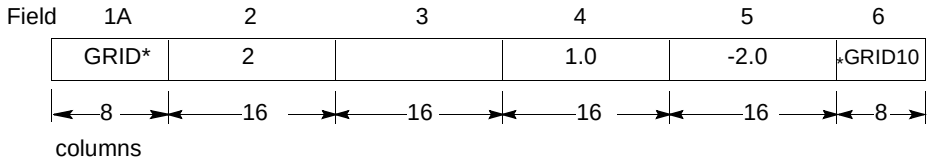
Large Field Format

A high degree of numerical accuracy is required in some MD Nastran applications. Large field format is used when small field format does not provide enough significant digits (recall that a minus sign, decimal point, and the “E” in scientific notation count as characters).

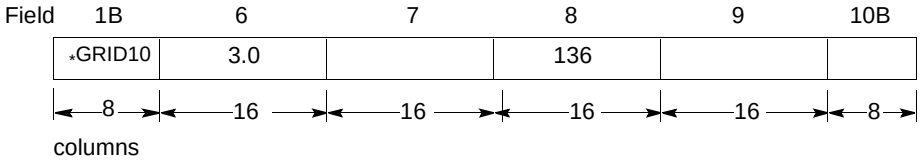
Large field format requires (at least) two lines for each entry: the first and last field of each line contains eight columns, and the fields in between contain 16 columns. Short field becomes two lines. Large field entries are denoted by an asterisk (*) immediately following the character string in field 1A of the first line and immediately preceding the character string in field 1B of the second line.

The following is an example of the GRID Bulk Data entry example in large field format:

First Line: (Left half of single field)



Second Line: (Right half of single field)



Continuations

Some Bulk Data entries require more than eight fields (72 columns) of data. Continuations are required in such cases. To do this, a parent entry (the first line) is followed by one or more continuation entries on subsequent lines. For example, consider the following PBAR simple beam property entry (do not worry about what each field represents-this will be explained later):

Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	El	E2	F1	F2	
	K1	K2	I12						

Continuation Example:

PBAR	39	6	2.9	1.86	2.92	.48			+PB1
+PB1	0.	0.	0.	1.	1.	1.	1.	0.	+PB2
+PB2	.86	.86							

+PB1 in field 10 of the parent entry is an arbitrary (and unique) user-defined pointer to field 1 of the second line. +PB2 in the second line points the third line, and so on.

Continuation fields can also be generated automatically by MD Nastran (this approach is the recommended practice). To automatically generate a continuation, the continuation line (or lines) must immediately follow the parent Bulk Data entry. In addition, fields 1 and 10 of the continuation line (or lines) must be left blank. In the case of double-width generated continuations are not blank in field 1, but have an "*" in column 1. MD Nastran will then generate unique continuations for you. This process is illustrated in the following example:

Input (.DAT) file:

```
CHEXA,       1,        10,        3,       5,       7,        1,       15,      17,
    ,      19,        13,        4,       6,       8,        2,       10,     11,
    ,      12,         9,       16,      18,      20,       14
```

Output (.F06) file:

```
GEAR TOOTH EXAMPLE         SEPTEMBER  29, 1993  MSC.Nastran  9/ 4/91   PAGE     3

        S O R T E D   B U L K   D A T A   E C H O
      CARD
      COUNT  .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
      1-     CHEXA   1   10   3   5       7    1  15       17    +000001
      2-     ++00000119  13   4   6       8    2  10       11    +000002
      3-     ++00000212  9    16  18      20   14                 +000003
```

MD NastranContinuation fields (fields one and ten) are replicated using the following conventions:

1. Only letters of the alphabet and integers may be used. They are coded into a base 36 number. That is, the sequence of numbers is 0,1, 2, ..., 8, 9, A, B, ...
2. The first character in field one or ten is not incremented.
3. The continuation fields are incremented by +1 regardless of the value specified by the user.
4. The number of characters in an incremented field will not be increased. For example, if the first field is "0", the thirty-seventh field will also be "0", resulting in an illegal entry. A method to solve this problem would be to start with a first field of "00". This will provide thirty-six squared unique fields.
5. At least one field in fields 2 through 8 of continuation entries must be non-blank.

Replication

Replication is a limited data generation capability which may be used in a fixed or free-field format.

1. Duplication of fields from the preceding entry is accomplished by coding the symbol =.
2. Duplication of all trailing fields from the preceding entry is accomplished by coding the symbol ==.
3. Incrementing a value from the previous entry is indicated by coding *x or *(x), where x is the value of the increment. “x” should be a real number for real fields or an integer for integer fields.
4. Repeated replication is indicated by coding =n or =(n), where n is the number of images to be generated using the values of the increments on the preceding entry.
5. Data items may be enclosed with parentheses or the parentheses may be deleted.
6. The MSGMESH capability includes the capabilities described here, plus the following capabilities as long as NASTRAN MESH is specified in the File Management Section.
    - Continuation entry fields may be incremented or decremented.
    - Repeated replication is indicated by coding =(n) in field 1, where n is number of entry images to be generated using the values of increments from the current or preceding replication entry.

      Entered entries:

      ```
      GRID,101 ,17,1 .0,10.5,,17,3456
      =(4), *(1),=,*(0.2),==$
      ```

      Generated entries:

GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.5		17	3456		
GRID	103	17	1.4	10.5		17	3456		
GRID	104	17	1.6	10.5		17	3456		
GRID	105	17	1.8	10.5		17	3456		

    - A blank in field 1 indicates immediate continuation entry replication. The default continuation entry increment is 1. Example:

      ```
      BSET1,123,1,2,3,4,5,6,7
      ,,*7,*7,*7,*7,*7,*7,*7
      =(3)
      ```

Generated entries:

BSET1	123	1	2	3	4	5	6	7	+00001
++00001		8	9	10	11	12	13	14	+00002
++00002		15	16	17	18	19	20	21	+00003
++00003		22	23	24	25	26	27	28	+00004
++00004		29	30	31	32	33	34	35	+00005

- A “=(D)” in field 1 indicates delayed continuation entry replication. A maximum of 9 entries may be replicated as a group. The default continuation entry increment is 10. Example:

  Entered entries:

```
CTRIA3,10,1,1,10,11/+C1
=(D),*(1),=,=,*(1),*(1)/*(20)
+C1,,,2.0,1.0,1.0
=(2),==
```

  Generated entries:

CTRIA3	10	1	1	10	11				+C1
+C1			2.0	1.0	1.0				
CTRIA3	11	1	1	11	12				+C21
+C21			2.0	1.0	1.0				
CTRIA3	12	1	1	12	13				+C41
+C41			2.0	1.0	1.0				

- Parentheses are optional on replication entries and an equal sign may replace an asterisk.

  The following is an example of the use of replication, automatic continuation field generation, and the free field format:

```
GRID,101 ,17,1.0,10.5,,17,3456
=,*1,=,*0.2, *(0.1), == $ COMMENTS MAY APPEAR AFTER $
=3
EIGR,13,GIV,,30.
,MASS
CBAR,1 ,1 ,101 ,102,0.,0.,1.,,+0
=,*1,=,*1,*1====*1
+0,56
*1,=$
```

The above free-field entries will generate the following Bulk Data in the 8-column format, as seen in the SORTED BULK DATA ECHO:

Note: A “,” should always be used after the “*1” for the continuation increment even if fixed field format is being used.

CBAR	1	1	101	102	0.	0.	1.		+0
+0	56								
CBAR	2	1	102	103	0.	0.	1.		+1
+1	56								
EIGR	13	GIV		30.					+000001
++000001	MASS								
GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.6		17	3456		
GRID	103	17	1.4	10.7		17	3456		
GRID	104	17	1.6	10.8		17	3456		
GRID	105	17	1.8	10.9		17	3456		

The automatically generated continuation entries start with the number 1, are incremented by 1, and are padded with zeros and plus signs as shown above. If this feature is used, it is the user’s responsibility not to enter continuation entries that also use this convention. In particular, data generated on another run and then written to the PUNCH file with the ECHO=PUNCH, will cause problems when introduced into other data with blank continuation fields.

Bulk Data Entry Summary

This section contains a summary of all Bulk Data entries. The entries are categorized as Geometry, Elements, Material Properties, Constraints, Loads, Solution Control, and Miscellaneous. Entries that are exclusive to MD Nastran Implicit Nonlinear analysis (SOL 600) have been grouped together at the end of the summary.

Geometry

Grid Points

GRID	Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.
GRIDB	Defines the location of a geometric grid point on a fluid point (RINGFL entry) for an axisymmetric fluid model and/or axisymmetric structure. Also defines the boundary of the fluid.
GRDSET	Defines default options for fields 3, 7, 8, and 9 of all GRID entries.
SEQGP	Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

Coordinate Systems

BAROR	Defines default values for field 3 and fields 6 through 8 of the CBAR entry.
BEAMOR	Defines default values for field 3 and fields 6 through 8 of the CBEAM entry.
CORDiC	Cylindrical coordinate system definition.
CORDiR	Rectangular coordinate system definition.
CORDiS	Spherical coordinate system definition.
CORD3G	Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system.

Scalar Points

EPOINT	Defines extra points for use in dynamic problems.
SEQEP	Redefines the sequence of extra points to optimize bandwidth.
SEQGP	Grid and scalar point number resequencing.
SPOINT	Defines scalar points.

Fluid Points

ACMODL	Defines modeling parameters for the interface between the fluid and the structure.
FREEPT	Defines the location of points on the surface of a fluid for recovery of surface displacements in a gravity field.
FSLIST	Defines the fluid points (RINGFL entry) that lie on a free surface boundary.
GRID	Defines fluid points in coupled fluid-structural analysis.
GRIDB	Grid point location on RINGFL.
GRIDF	Defines a scalar degree-of-freedom for harmonic analysis of a fluid.
GRIDS	Defines a scalar degree-of-freedom with a two-dimensional location. Used in defining pressure in slotted acoustic cavities.
PRESPT	Defines the location of pressure points in the fluid for recovery of pressure data.
RINGFL	Defines a circle (fluid point) in an axisymmetric fluid model.
SLBDY	Defines a list of slot points that lie on an interface between an axisymmetric fluid and a set of evenly spaced radial slots.

Axisymmetry

AXIC	Defines the existence of an axisymmetric conical shell problem.
AXIF	Defines basic parameters and the existence of an axisymmetric fluid analysis.
AXSLOT	Defines the harmonic index and the default values for acoustic analysis entries.
FLSYM	Defines the relationship between the axisymmetric fluid and a structural boundary having symmetric constraints. The purpose is to allow fluid boundary matrices to conform to structural symmetry definitions.
POINTAX	Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested. These points are not subject to constraints via MPCAX, SPCAX, or OMITAX entries.

RINGAX	Defines a ring for conical shell problem.
SECTAX	Defines a sector of a conical shell.

Cyclic Symmetry

CYAX	Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis.
CYJOIN	Defines the boundary points of a segment in cyclic symmetry problems.

Superelement Analysis

CSUPER	Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.
CSUPEXT	Assigns exterior points to a superelement.
EXTRN	Defines a boundary connection for an external superelement.
GRID	Defines interior points for a superelement.
RELEASE	Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.
SEBNDRY	Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.
SEBULK	Defines superelement boundary search options and a repeated, mirrored, or collector superelement.
SECONCT	Explicitly defines grid and scalar point connection procedures for a partitioned superelement.
SEELT	Reassigns superelement boundary elements to an upstream superelement.
SEEXCLD	Defines grid points that will be excluded during the attachment of a partitioned superelement.
SELABEL	Defines a label or name to be printed in the superelement output headings.
SELOC	Defines a partitioned superelement relocation by listing three noncolinear points in the superelement and three corresponding points not belonging to the superelement.

SEMPLN	Defines a mirror plane for mirroring a partitioned superelement.
SEQSEP	Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.
SESET	Defines interior grid points for a superelement.
SETREE	Specifies superelement reduction order.

p-element and Adaptivity Analysis

FEEDGE	Defines a finite element edge and associates it with a curve.
FEFACE	Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.
GMBNDC	Defines a geometric boundary consisting of p-element edges along a curve interface. The boundary may consist of edges of shell, beam, or p-solid elements.
GMBNDS	Defines a geometric boundary consisting of p-element faces along a surface interface. The boundary may consist of faces of p-solid or p-shell elements.
GMCORD	Defines a convective / follower coordinate system on an FEEDGE, GMCURV, FEFACE, or GMSURF entry.
GMCURV	Defines geometric curve that will be used in element geometry, load definition, and boundary condition definition.
GMINTC	Defines curve interface elements to connect dissimilar meshes.
GMINTS	Defines an interface element along a surface interface between boundaries of multiple subdomains.
GMSURF	Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.
PINTC	Defines properties for curve interface elements (see GMINTC).
PINTS	Defines the properties for interface elements along surface interfaces between boundaries of multiple subdomains of p-elements.
POINT	Define edge point for FEEDGE entry.

Aeroelastic Control Points

AECOMP	Defines a component for use in aeroelastic monitor point definition.
AECOMPL	Defines a component for use in aeroelastic monitor point definition as a union of other components.
MONPNT1	Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system.
UXVEC	Specification of a vector of aerodynamic control point (extra point) values.

Elements

Line Elements

BAROR	Default for orientation and property for CBAR.
BEAMOR	Default for orientation and property for CBEAM.
CBAR	Defines a simple beam element.
CBEAM	Defines a beam element.
CBEND	Connection definition for curved beam.
CBUSH1D	Defines the connectivity of a one-dimensional spring and viscous damper element.
CFAST	Defines a fastener with material orientation connecting two surface patches.
CONROD	Defines a rod element without reference to a property entry.
CROD	Defines a tension-compression-torsion element.
CTUBE	Defines a tension-compression-torsion tube element.
CWELD	Defines a weld or fastener connecting two surface patches or points.
PBAR	Defines the properties of a simple beam element (CBAR entry).
PBARL	Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.
PBCOMP	Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry.
PBEAM	Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.

PBEAML	Defines the properties of a beam element by cross-sectional dimensions.
PBEND	Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).
PFAST	Defines the CFAST fastener property values.
PBMSECT	Defines the shape of arbitrary cross-sectioon for CBEAM element.
PBRSECT	Defines the shape of arbitrary cross-section for CBAR element.
PBUSH1D	Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
PROD	Defines the properties of a rod element (CROD entry).
PTUBE	Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).
PWELD	Defines the properties of connector (CWELD) elements.

Surface Elements

CQUAD	Defines a plane strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.
CQUAD4	Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.
CQUAD8	Defines a curved quadrilateral shell or plane strain element with eight grid points.
CQUADR	Defines an isoparametric membrane and bending quadrilateral plate element.
CSHEAR	Defines the properties of a shear panel (CSHEAR entry).
CTRIA3	Defines an isoparametric membrane-bending or plane strain triangular plate element.
CTRIA6	Defines a curved triangular shell element or plane strain with six grid points.
CTRIAR	Defines an isoparametric membrane-bending triangular plate element. However, this element does not include membrane-bending coupling. It is a companion to the CQUADR element.

PCOMP	Defines the properties of an n-ply composite material laminate.
PLPLANE	Defines the properties of a fully nonlinear (i.e., large strain and large rotation) hyperelastic plane strain or axisymmetric element.
PSHELL	Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.
PSHEAR	Defines the properties of a shear panel (CSHEAR entry).
SNORM	Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.

Solid Elements

CHEXA	Defines the connections of the six-sided solid element with eight to twenty grid points.
CPENTA	Defines the connections of a five-sided solid element with six to fifteen grid points.
CTETRA	Defines the connections of the four-sided solid element with four to ten grid points.
PLSOLID	Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.
PSOLID	Defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries).
PSOLIDD	Additional (LS-DYNA specific) property specification information may be provided using the entry when materials MATD010 or MATD015 are used.

Scalar and Bushing Elements

CBUSH	Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.
CBUSH1D	Defines the connectivity of a one-dimensional spring and viscous damper element.
CELASi	Connection definition for scalar spring, also property definition for i=2 or 4.
GENEL	Defines a general element.
PBUSH	Defines the nominal property values for a generalized spring-and-damper structural element.

PBUSHT	Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.
PBUSH1D	Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
PELAS	Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).
PELAST	Defines the frequency dependent properties for a PELAS Bulk Data entry.

Axisymmetric Elements

CCONEAX	Defines a conical shell element.
CQUADX	Defines an axisymmetric quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
CTRIAX	Defines an axisymmetric triangular element with up to 6 grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
CTRIAX6	Defines an isoparametric and axisymmetric triangular cross section ring element with midside grid points.
PCONEAX	Defines the properties of a conical shell element described on a CCONEAX entry.

p-element Interface Elements

GMINTC	Defines a p-interface element along a curve.
GMINTS	Defines a p-interface element along a surface.
PINTC	Property definition for GMINTC.
PINTS	Property definition for GMINTS.

Rigid Elements

RBAR	Defines a rigid bar with six degrees-of-freedom at each end.
RBE1	Defines a rigid body connected to an arbitrary number of grid points.

RBE2	Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points.
RBE3	Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.
RJOINT	Defines a rigid joint element connecting two coinciding grid points.
RROD	Defines a pin-ended element that is rigid in translation.
RSPLINE	Defines multipoint constraints for the interpolation of displacements at grid points.
RSSCON	Defines multipoint constraints to model clamped connections of shell-to-solid elements.
RTRPLT	Defines a rigid triangular plate.
RTRPLT1	Defines a rigid triangular plate (alternate).

Mass Elements

CMASSi	Connection definition for scalar mass, also property definition for i=2 or 4.
CONM1	Defines a 6 x 6 symmetric mass matrix at a geometric grid point.
CONM2	Defines concentrated mass at a grid point.
PMASS	Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).
NSM	Non Structural Mass by ID.
NSM1	Non Structural Mass (alternate form).
NSMADD	Non Structural Mass Set Combination.
NSML	Lumped Non Structural Mass by ID.
NSML1	Lumped Non Structural Mass (alternate form).

Damping Elements

CBUSH1D	See line elements.
CDAMPi	Connection definition for a scalar damper, also property definition for i=2 or 4.
CVISC	Defines a viscous damper element.

PBUSH1D	See line elements.
PDAMP	Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.
PDAMP5	Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only.
PDAMPT	Defines the frequency-dependent properties for a PDAMP Bulk Data entry.
PVISC	Defines properties of a one-dimensional viscous damping element (CVISC entry).

Fluid and Acoustic Elements

CAABSF	Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.
CAXIFi	Defines an axisymmetric fluid element that connects i = 2, 3, or 4 fluid points.
CFLUIDi	Defines three types of fluid elements for an axisymmetric fluid model.
CHACAB	Defines the acoustic absorber element in coupled fluid-structural analysis.
CHACBR	Defines the acoustic barrier element.
CHEXA	Connection definition for a pentahedron element in coupled fluid-structural analysis.
CPENTA	Connection definition for a tetrahedron element in coupled fluid-structural analysis.
CSLOTi	Defines slot element for acoustic cavity analysis.
CTETRA	Defines the connections of the four-sided solid element with four to ten grid points.
ELIST	Defines a list of structural elements for virtual fluid mass.
PAABSF	Defines the properties of a frequency-dependent acoustic absorber element.
PACABS	Defines the properties of the acoustic absorber element.
PACBAR	Defines the properties of the acoustic barrier element.

PANEL	Selects the set of structural grid points that define one or more panels.
PSOLID	Defines the fluid properties of solid elements (CHEXA, CPENTA, and CTETRA entries).
SET1	Defines a list of structural grid points for aerodynamic analysis, XY-plots for SORT1 output, and the PANEL entry.

Heat Transfer Elements

BDYOR	Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.
CHBDYi	Connection definition for surface element (CHBDYE, CHBDYG, CHBDYP).
CONTRLT	Thermal control element for heat transfer analysis.
PHBDY	A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements.

The following elastic elements may also be used as heat conduction elements:

Linear:	CBAR, CROD, CONROD, CTUBE, CBEAM, CBEND.
Membrane:	CTRIA3, CTRIA6, CQUAD4, CQUAD8.
Axisymmetric:	CTRIAX6.
Solid:	CTETRA, CHEXA, CPENTA.

Dummy Elements

ADUMi	Defines attributes of the dummy elements ($1 \le i \le 9$)
CDUMi	Defines a dummy element ($1 \le i \le 9$)
PDUMi	Defines the properties of a dummy element ($1 \le i \le 9$). Referenced by the CDUMi entry.
PLOTEL	Defines a one-dimensional dummy element for use in plotting.

Contact or Gap Elements

BCONP	Defines the parameters for a contact region and its properties.
BFRIC	Defines frictional properties between two bodies in contact.

BLSEG	Defines a curve that consists of a number of line segments via grid numbers that may come in contact with another body.
BWIDTH	Defines widths or thicknesses for line segments in 3-D or 2-D slideline contact defined in the corresponding BLSEG Bulk Data entry.
CGAP	Defines a gap or friction element.
PGAP	Defines the properties of the gap element (CGAP entry).

Crack Tip Elements

CRAC2D	Defines a two-dimensional crack tip element.
CRAC3D	Defines a three-dimensional crack tip element.
PRAC2D	Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element.
PRAC3D	Defines the properties of the CRAC3D structural element.

Aerodynamic Elements

AEFACT	Defines real numbers for aeroelastic analysis.
AELINK	Defines relationships between or among AESTAT and AESURF entries.
AELIST	Defines a list of aerodynamic elements to undergo the motion prescribed with the AESURF Bulk Data entry for static aeroelasticity.
AESTAT	Specifies rigid body motions to be used as trim variables in static aeroelasticity.
AESURF	Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points.
AESURFS	Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry.
CAERO1	Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords.
CAERO2	Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.

CAERO3	Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface.
CAERO4	Defines an aerodynamic macro element for Strip theory.
CAERO5	Defines an aerodynamic macro element for Piston theory.
CSSCHD	Defines a scheduled control surface deflection as a function of Mach number and angle of attack.
PAERO1	Defines associated bodies for the panels in the Doublet-Lattice method.
PAERO2	Defines the cross-sectional properties of aerodynamic bodies.
PAERO3	Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.
PAERO4	Defines properties of each strip element for Strip theory.
PAERO5	Defines properties of each strip element for Piston theory.

Aerodynamic to Structure Interconnection

SET1	Defines a list of structural grid points.
SET2	Defines a list of structural grid points in terms of aerodynamic macro elements.
SPLINE1	Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
SPLINE2	Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
SPLINE3	Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.
SPLINE4	Defines a curved surface spline for interpolating motion and/or forces for aeroelastic problems on general aerodynamic geometries using either the Infinite Plate, Thin Plate or Finite Plate splining method.
SPLINE5	Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by irregular arrays of aerodynamic points.

Connector Elements

CFAST	Defines a fastener with material orientation connecting two surface patches.
CWELD	Defines a weld or fastener connecting two surface patches or points.
CWSEAM	Defines a seam element connecting two surfaces patches.
PFAST	Defines the CFAST fastener property values.
PWELD	Defines the property of connector (CWELD) elements.
PWSEAM	Defines the CWSEAM-PWSEAM fastener property values.
SWLDPRM	Overrides default values of parameters for connector search.

Materials

Isotropic

MAT1	Defines the material properties for linear isotropic materials.
MAT4	Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.
MATHP	Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).
RADM	Defines the radiation properties of a boundary element for heat transfer analysis.

Anisotropic

MAT2	Defines the material properties for linear anisotropic materials for two-dimensional elements.
MAT3	Defines the material properties for linear orthotropic materials used by the CTRIAX6 element entry.
MAT5	Defines the thermal material properties for anisotropic materials.

MAT8	Defines the material property for an orthotropic material for isoparametric shell elements.
MAT9	Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see PSOLID entry description).

Temperature Dependent

MATTi	Table references for temperature-dependent MATi materials.
RADMT	Specifies table references for temperature dependent RADM entry radiation boundary properties.
TABLEMi	Tabular functions for generating temperature-dependent material properties.
TABLEST	Table references for temperature dependent MATS1 materials.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPAX	Defines temperature sets for conical shell problems.
TEMPD	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.
TEMPPi	Defines temperature field for surface elements.
TEMPRB	Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Stress Dependent

CREEP	Defines creep characteristics based on experimental data or known empirical creep law.
MATS1	Specifies stress-dependent material properties for use in applications involving nonlinear materials.
TABLES1	Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

Fluid

AXIF	Includes default values for mass density and bulk modulus.
AXSLOT	Includes default values for mass density and bulk modulus.
BDYLIST	Defines the boundary between a fluid and a structure.
CFLUIDi	Includes mass density and bulk modulus.
CSLOTi	Includes mass density and bulk modulus.
FSLIST	Includes mass density at free surface.
MAT10	Defines material properties for fluid elements in coupled fluid-structural analysis.
MFLUID	Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.
SLBDY	Includes mass density at interface between fluid and radial slots.

Constraints and Partitioning

Single Point Constraints

FLSYM	Symmetry control for boundary in axisymmetric fluid problem.
GRID	Includes single point constraint definition.
GRIDB	Includes single point constraint definition.
GRDSET	Includes default for single point constraints.
SPC	Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis).
SPC1	Defines a set of single point constraints.
SPCADD	Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.
SPCAX	Defines a set of single-point constraints or enforced displacements for conical shell coordinates.
SPCOFFi	Defines degrees-of-freedom to be excluded from the AUTOSPC operation.

Multipoint Constraints

MPC	Defines a linear relationship for two or more degrees-of-freedom.
MPCADD	Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.
MPCAX	Defines multipoint constraints for conical shell problems.
POINTAX	Defines multipoint constraints for point on conical shell.
RBAR	Defines multipoint constraints for rigid bar.
RBEi	Defines multipoint constraints for RBE1, RBE2, RBE3.
RROD	Defines multipoint constraints for rigid rod.
RSPLINE	Defines multipoint constraints for spline element.
RTRPLT	Defines multipoint constraints for rigid triangular plate.

Partitioning

ASET	Defines degrees-of-freedom in the analysis set (a-set)
ASET1	Defines degrees-of-freedom in the analysis set (a-set)
CSUPEXT	Assigns exterior points to a superelement.
GRID	Defines interior points for a superelement.
OMIT	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
OMIT1	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
OMITAX	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
RELEASE	Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.
SEELT	Reassigns superelement boundary elements to an upstream superelement.
SESET	Defines interior grid points for a superelement.

Free Body Supports

CYSUP	Defines fictitious supports for cyclic symmetry analysis.
SUPAX	Defines determinate reaction degrees-of-freedom in free bodies for conical shell analysis.
SUPORTi	Defines degrees-of-freedom for determinate reactions.

p-element Geometry Constraints

GMBC	Defines enforced displacements for GRID, FEEDGE, FEFACE, GMCURV, and GMSURF entries.
GMSPC	Defines constraints for entries.

Component Mode Boundary Conditions

BSET	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
BSET1	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
CSET	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
CSET1	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
QSET	Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.
QSET1	Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.
SEBSET	Defines boundary degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
SEBSET1	Defines fixed boundary points for superelement.
SECSET	Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.
SECSET1	Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.
SENQSET	Defines number of internally generated scalar points for superelement dynamic reduction.

SEQSET	Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.
SEQSET1	Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.
SESUP	Defines determinate reaction superelement degrees-of-freedom in a free-body analysis.

User Sets

DEFUSET	Defines new names for degree-of-freedom sets.
SEUSET	Defines a degree-of-freedom set for a superelement.
SEUSET1	Defines a degree-of-freedom set for a superelement.
USET	Defines a degree-of-freedom set.
USET1	Defines a degrees-of-freedom set.

Loads

Static Loads

ACCEL	Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based upon the tabular input defined on this Bulk Data entry.
ACCEL1	Defines static acceleration loads at individual GRID points.
CLOAD	Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOL 106 or 153).
DEFORM	Defines enforced axial deformation for one-dimensional elements for use in statics problems.
FORCE	Defines a static concentrated force at a grid point by specifying a vector.
FORCEAX	Defines a concentrated force on a conical shell ring.
FORCEi	Defines concentrated load at grid point.
GRAV	Defines acceleration vectors for gravity or other acceleration loading.

LOAD	Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, SLOAD, RFORCE, and GRAV entries.
LOADCYH	Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.
LOADCYN	Defines a physical static or dynamic load for use in cyclic symmetry analysis.
LOADCYT	Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = “AXI” on the CYSYM entry.
MOMAX	Defines a static concentrated moment load on a ring of a conical shell.
MOMENT	Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.
MOMENTi	Defines moment at grid point.
PLOAD	Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.
PLOAD1	Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined.
PLOAD2	Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements.
PLOAD4	Defines a pressure load on a face of a CHEXA, CPENTA, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element.
PLOADX1	Defines surface traction to be used with the CQUADX, CTRIAX, and CTRIAX6 axisymmetric element.
PRESAX	Defines the static pressure loading on a conical shell element.
RFORCE	Defines a static loading condition due to an angular velocity and/or acceleration.

SPCD	Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.
SLOAD	Defines concentrated static loads on scalar or grid points.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPD	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.
TEMPPi	Defines temperature field for surface elements.
TEMPRB	Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPAX	Defines temperature sets for conical shell problems.

Dynamic Loads

ACSRCE	Defines the power versus frequency curve for a simple acoustic source.
DAREA	Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with RLOADi and TLOADi entries.
DELAY	Defines the time delay term τ in the equations of the dynamic loading function.
DLOAD	Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.
DPHASE	Defines the phase lead term θ in the equation of the dynamic loading function.
LOADCYH	Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.
LOADCYN	Defines a physical static or dynamic load for use in cyclic symmetry analysis.

LSEQ	Defines a sequence of static load sets.
NOLINi	Nonlinear transient load definition.
NLRGAP	Defines a nonlinear transient radial (circular) gap.
RLOADi	Frequency dependent excitation definition.
TABLEDi	Tabular functions for generating dynamic loads.
TLOADi	Time dependent excitation definition.

Heat Transfer Loads

CONV	Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).
CONVM	Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).
PCONV	Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.
PCONVM	Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.
QBDY1	Defines a uniform heat flux into CHBDYj elements.
QBDY2	Defines grid point heat flux into CHBDYj elements.
QBDY3	Defines a uniform heat flux load for a boundary surface.
QHBDY	Defines a uniform heat flux into a set of grid points.
QVECT	Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.
QVOL	Defines a rate of volumetric heat addition in a conduction element.
RADBC	Specifies an CHBDYi element face for application of radiation boundary conditions.
RADBND	Specifies Planck’s second radiation constant and the wavelength breakpoints used for radiation exchange problems.
RADCAV	Identifies the characteristics of each radiant enclosure.

RADLST	Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.
RADMTX	Provides the $F_{ji} = A_j f_{ji}$ exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry.
RADSET	Specifies which radiation cavities are to be included for radiation enclosure analysis.
SLOAD	Defines concentrated static loads on scalar or grid points.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPBC	Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions.
TEMPD	Specifies default initial temperature at grid points.
VIEW	Defines radiation cavity and shadowing for radiation view factor calculations.
VIEW3D	Defines parameters to control and / or request the Gaussian Integration method of view factor calculation for a specified cavity.

p-element Loads

GMBC	Defines enforced displacements for GRID, FEEDGE, GMCURV, FEFACE, and GMSURF entries.
GMCONV	Defines convection boundary conditions.
GMLOAD	Defines the forces and moments to be applied to a FEEDGE, GMCURV, FEFACE, or GMSURF entry.
GMQVOL	Defines volumetric heat loads.
TEMPF	Defines the thermal loading to be applied to a group of elements.

Solution Control

Buckling Analysis

EIGB	Defines data needed to perform buckling analysis.
EIGRL	Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

Eigenvalue Analysis

EIGC	Defines data needed to perform complex eigenvalue analysis.
EIGR	Defines data needed to perform real eigenvalue analysis.
EIGRL	Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.
EIGP	Defines poles that are used in complex eigenvalue extraction by the Determinant method.
RVDOF, RVDOF1	Degrees-of-freedom specificatin for residual vector calculation.

Cyclic Symmetry

CYSYM	Defines parameters for cyclic symmetry analysis.

Frequency Response

FREQ	Defines a set of frequencies to be used in the solution of frequency response problems.
FREQi	Defines a set of frequencies for problem solution.
TABDMP1	Defines modal damping as a tabular function of natural frequency.

Random Response

RANDPS	Defines load set power spectral density factors for use in random analysis.
RANDT1	Defines time lag constants for use in random analysis autocorrelation function calculation.
RCROSS	Cross-power spectral density and cross-correlation function output.
TABRND1	Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

Rotordynamics

RGYRO	Specifies synchronous or asynchronous analysis, reference rotor, and rotation speed of the reference rotor.
ROTORG	Specifies grids that compose the rotor line model.

RSPINR	Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis.
RSPINT	Specifies rotor spin rates for nonlinear transient analysis.
UNBALNC	Specifies an unbalanced load for transient analysis in terms of a cylindrical system with the rotor rotation axis as the z-axis.

Transient Response

TIC	Defines values for the initial conditions of variables used in structural transient analysis.
TSTEP	Defines time step intervals at which a solution will be generated and output in transient analysis.
TSTEPNL	Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. TSTEPNL is intended for SOLs 129, 159, and 99.

Nonlinear Static Analysis

ITER	Defines options for the iterative solver in SOLs 101, 106, 108, 111 and 153.
NLPARM	Defines a set of parameters for nonlinear static analysis iteration strategy.
NLPCI	Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106).

Original Design Sensitivity Analysis (DSA)

DSCONS	Defines a design constraint in design sensitivity analysis (original DSA). See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*.
DVAR	Defines a design variable for design sensitivity analysis (original DSA) described in "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*.
DVSET	Defines a set of element properties that vary in a fixed relation to a design variable for design sensitivity analysis (original DSA). See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*.

Optimization (SOL 200 Only)

BNDGRID	Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200).
DCONADD	Defines the design constraints for a subcase as a union of DCONSTR entries.
DCONSTR	Defines design constraints.
DDVAL	Define real, discrete design variable values for discrete variable optimization.
DEQATN	Defines a design variable for design optimization.
DESVAR	Defines a design variable for design optimization.
DLINK	Relates one design variable to one or more other design variables.
DOPTPRM	Overrides default values of parameters used in design optimization.
DRESP1	Defines a set of structural responses that is used in the design either as constraints or as an objective.
DRESP2	Defines equation responses that are used in the design, either as constraints or as an objective.
DRESP3	Defines an external response using user-supplied routine.
DSCREEN	Defines screening data for constraint deletion.
DTABLE	Defines a table of real constants that are used in equations (see DEQATN entry).
DVBSHAP	Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.
DVCREL1	Defines the relation between a connectivity property and design variables.
DVCREL2	Defines the relation between a connectivity property and design variables with a user-supplied equation.
DVGRID	Defines the relationship between design variables and grid point locations.
DVMREL1	Defines the relation between a material property and design variables.
DVMREL2	Defines the relation between a material property and design variables with a user-supplied equation.

DVPREL1	Defines the relation between an analysis model property and design variables.
DVPREL2	Defines the relation between an analysis model property and design variables with a user-supplied equation.
DVSHAP	Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.
MODTRAK	Specifies parameters for mode tracking in design optimization (SOL 200).
TOPVAR	

Aerodynamic Matrix Generation

MKAERO1	Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.
MKAERO2	Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

Aeroelastic Stability Analysis

FLFACT	Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.
FLUTTER	Defines data needed to perform flutter analysis.
DIVERG	Defines Mach numbers (m) for a divergence analysis in SOLs 144 and 200.

Aeroelastic Response Analysis

AEDW	Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG).
AEFORCE	Defines a vector of absolute forces (it will ***not*** be scaled by dynamic pressure) associated with a particular control vector.
AEPARM	Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point).
AEPRESS	Defines a vector of pressure / unit dynamic pressure associated with a particular control vector.
GUST	Defines a stationary vertical gust for use in aeroelastic response analysis.

TABRNDG	Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.
TRIM	Specifies constraints for aeroelastic trim variables. The SPLINE1 and SPLINE4 entries need to be here for the finite plate spline.

Aerodynamic Parameters

AERO	Gives basic aerodynamic parameters for unsteady aerodynamics.
AEROS	Defines basic parameters for static aeroelasticity.

p-element and Adaptivity Analysis

ADAPT	Defines controls for p-version adaptive analysis.
PSET	Describes polynomial order distribution and is selected by the ADAPT Case Control command.
PVAL	Describes polynomial order distribution and is selected by the ADAPT Bulk Data entry.

Miscellaneous

Comments

$	Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Delete

/	Removes entries on restart.

Parameters

PARAM	Specifies values for parameters used in solution sequences or user-written DMAP programs.
MDLPRM	Specifies parameters which affect the solution of the structural model.

Direct Matrix Input

CONM1	Defines a 6x6 mass matrix at a geometric grid point.
DMI	Defines matrix data blocks.

DMIG	Defines direct input matrices related to grid, extra, and/or scalar points.
DMIG,UACCEL	Defines rigid body accelerations in the basic coordinate system.
DMIAX	Defines axisymmetric (fluid or structure) related direct input matrix terms.
TF	Defines a dynamic transfer function.

Direct Matrix Input for Aeroelasticity

DMIJ	Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries.
DMIJI	Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2.
DMIK	Defines direct input matrices related to physical (displacement) degrees-of-freedom (ks-set) of aerodynamic grid points.

Tabular Input

DTI	Defines table data blocks.
DTI,ESTDATA	Provides override data for time and space estimation for superelement processing operations.
DTI,INDTA	Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.
DTI,SETREE	Defines a superelement tree that determines the superelement processing order.
DTI,SPECSEL	Correlates spectra lines specified on TABLED1 entries with damping values.
DTI,SPSEL	Correlates output requests with frequency and damping ranges.
TABDMP1	Defines modal damping as a tabular function of natural frequency.
TABLEDi	Tabular functions for generating dynamic loads.
TABLEMi	Tabular functions for generating temperature-dependent material properties.

TABLES1	Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).
TABRND1	Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.
TABLE3D	Specify a function of three variables for the GMBC, GMLOAD, and TEMPF entries only.

Non Structural Mass Distribution Selection

NSM	Non Structural Mass entry by id,value.
NSM1	Non Structural Mass entry by value,id.
NSMADD	Non Structural Mass as sum of listed sets.
NSML	Lumped non structural mass entry by id,value.
NSML1	Lumped non structural mass entry by value,id.

Output Control

BOUTPUT	Defines slave nodes at which output is requested.
CBARAO	Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output.
ECHOOFF	Marks the point or points in the input file to deactivate printed echo of the Bulk Data.
ECHOON	Marks the point or points in the input file to activate printed echo of the Bulk Data.
FREEPT	Surface point location for data recovery in hydroelastic problems.
PLOTEL	Defines a one-dimensional dummy element for use in plotting.
POINTAX	Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested.
PRESPT	Defines the location of pressure points in the fluid for recovery of pressure data in hydroelastic problems.
SET1	Defines a set of grid points.

TSTEP	Specifies time step intervals for data recovery in transient response.
TSTEPNL	Specifies time step intervals for data recovery in nonlinear transient response.

p-element Output Control

OUTPUT	Output control for p-adaptive analysis.
OUTRCV	Defines options for the output of displacements, stresses, and strains of p-elements.

Solution Control

ITER	Defines options for the iterative solver in SOLs 101, 106, 108, 111 and 153.
RVDOF	Degrees-of-freedom specification for residual vector computations.
RVDOF1	Degrees-of-freedom specification for residual vector computations (alternate form).

End of Input

ENDDATA	Designates the end of the Bulk Data Section.

Include File

INCLUDE	Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

MD Nastran Implicit Nonlinear (SOL 600)

3D Contact Region

BCBODY	Defines a flexible rigid contact body in 2D or 3D.
BCBOX	Defines a 3D contact region.
BCHANGE	Changes definitions of contact bodies.
BCMATL	Defines a 3D contact region by element material.
BCMOVE	Defines movement of bodies in contact.
BCPARA	Defines contact parameters.
BCPROP	Defines a 3d contact region by element properties.
BCTABLE	Defines a contact table.

BSURF	Defines a contact body or surface by element IDs.
GMNURB	3D contact region made up of NURBS.

Initial Conditions

IPSTRAIN	Defines initial plastic strain values.
ISTRESS	Defines initial stress values.
MARCIN	Insert a text string in MSC.Marc.
MARCOUT	Selects data recovery output.

Materials

MATEP	Elasto-plastic material properties.
MATF	Specifies material failure model.
MATG	Gasket material properties.
MATHE	Hyperelastic material propeties.
MATHED	Damage model properties for hyperelastic materials.
MATORT	Elastic 3D orthotropic material properties.
MATTEP	Thermoelastic-Plastic material properties.
MATTG	Temperature variation of interlaminar materials .
MATTHE	Thermo hyperelastic material.
MATTORT	Thermoelastic orthotropic material
MATTVE	Thermo-visco-elastic material properties
MATVE	Viscoelastic material properties
MATVP	Viscoplastic or creep material properties

Solution Control

MPROCS	Provides additional control for parallel processing
NLAUTO	Parameters for automatic load / time stepping.
NLDAMP	Defines damping constants.
NLSTRAT	Strategy parameters for nonlinear structural analysis.

PARAMARC	Parallel domain decomposition.
RESTART	Restart data.

Element Properties

NTHICK	Defines nodal thickness values for beams, plates, and/or shells.

Bolts

MBOLT	Defines a bolt for use in countries outside the USA.
MBOLTUS	Defines a bolt for use in the USA and all other countries.

Solid Composites

MSTACK	Defines the direction that 3D solid comosites are stocked.

Brake Squeal

BRKSQL	Specifies data for brake squeal calculations using SOL 600

MD Nastran Explicit Nonlinear (SOL 700)

Air Bags (SOL 700)

AIRBAG	Single Surface Contact
GBAG	Defines the pressure within an enclosed volume.

Contact (SOL 700)

BCTABLE	Defines a contact table.
BCGRID	Grids to be included in SOL 700 contact analyses.
BCSEG	Grids which are part of an element to be used in SOL 700 contact analyses. (Specified in the Ls-Dyna style)

Coordinate Systems (SOL 700)

CORD1RX	Alternate rectangular coordinate system specification for SOL 700.
CORD2RX	Alternate rectangular coordinate system specification for SOL 700.

CORD3RX	Alternate rectangular coordinate system specification for SOL 700.
CORD3R	Defines a moving rectangular coordinate system using three points (SOL 700 only).

Dampers/Springs (SOL 700)

CDAMP1D	Defines a scalar damper connection for use in SOL 700 only.
CDAMP2D	Defines a scalar damper connection for use in SOL 700 only.
CELAS1D	Defines a scalar spring connection for use in SOL 700 only.
CELAS2D	Defines a scalar spring connection for use in SOL 700 only.
CSPR	Springs with offsets for use in SOL 700.

Direct Text Input (SOL 700)

ENDDYNA	Defines the end of direct text to Dytran-lsdyna.
TODYNA	Defines the start of direct text to Dytran-lsdyna.

Dynamic Relaxation (SOL 700)

DAMPGBL	Defines parameters to be used for static analysis simulation using Dynamic Relaxation for use in SOL 700 only.
DYRELAX	Define controls for dynamic relaxation for restart runs.

Element Properties (SOL 700)

PBEAM71	Defines complex beam properties that cannot be defined using the PBAR or PBEAM entries.
PBEAMD	Defines cross-sectional properties for beam, truss, discrete beam, and cable elements.
PCOMPA	Defines additional properties of a multi-ply laminate composite material.
PELAS1	Defines a spring property designated by a force-deflection curve for SOL 700.
PSHELL1	Defines the properties of SOL 700 shell elements that are much more complicated than the shell elements defined using the PSHELL entry.

PSPHELLD	Defines properties for shell elements.
PSOLIDD	Additional (MSC.Dytran-lsdyna specific) property specification information may be provided using this entry when materials MATD010 or MATD015 are used.

Hourglass Control (SOL 700)

HGSUPPR	Defines the hourglass suppression method, the corresponding hourglass damping coefficients and sets for the bulk viscosity method and coefficients.

Inertial Properties (SOL 700)

D2RINER	Inertial properties can be defined for the new rigid bodies that are created when the deformable parts are switched.

Initial Conditions (SOL 700)

TICD	Defines values for the initial conditions of variables used in structural transient analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom. This entry may not be used for heat transfer analysis. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
TIC3	Allows for the definition of a velocity field of grid points consisting of a rotation and a translation specification. Used in MD Nastran Implicit Nonlinear (SOL 700) only.

Materials (SOL 700)

EOSPOL	Defines the properties of a polynomial equation of state.
MATD001	LS-DYNA Material #1 -- Isotropic elastic material available for beam, shell and solid elements. A specialization of this material allows the modeling for fluids. The fluid option is valid for solid elements only. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD2AN	LS-DYNA style Material # 2 for modeling the elastic-anisotropic behavior of solids, shells, and thick shells. Defines material properties for anisotropic materials in the LS-DYNA style. Used in MD Nastran Implicit Nonlinear (SOL 700) only.

MATD2OR	LS-DYNA Material #2 -- Used to model the elastic-orthotropic behavior of solids, shells, and thick shells. For orthotropic solids and isotropic frictional damping is available. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD003	LS-DYNA material #3 -- Used to model isotropic and kinematic hardening plasticity with the option of including rate effects. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD005	LS-DYNA Material #5 -- Used to model soil and foam. This is a very simple model and works in some ways like a fluid. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD006	LS-DYNA Material #6 -- Used to model the viscoelastic behavior of beams, shells, and solids. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD007	LS-DYNA Material #7 -- Used to model nearly incompressible continuum rubber. The Poisson's ratio is fixed to 0.463. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD009	This material allows equations of state to be considered without computing deviatoric stresses.
MATD012	LS-DYNA Material #12 --Low cost isotropic plasticity model for three-dimensional solids. This is the only model in LS-DYNA for plane stress that does not default to an iterative approach. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD013	LS-DYNA Material #13 -- This is a non-iterative plasticity with simple plastic strain failure model. Used in MD Nastran Implicit Nonlinear (SO 700) only.
MATD014	LS-DYNA Material #14 -- Input for this model is the same as for MATD005; however, when the pressure reaches the failure pressure, the element loses its ability to carry tension. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD015	LS-DYNA Material #15 -- Johnson / Cook strain and temperature sensitive plasticity sometimes used for problems where the strain rates vary over a large range and adiabatic temperature increases due to plastic heating cause material softening. Used in MD Nastran Implicit Nonlinear (SOL 700) only.

MATD018	LS-DYNA Material #18 -- An isotropic plasticity model with rate effects that uses a power law hardening rule. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD019	LS-DYNA Material #19 -- Used to model strain rate dependent material. For an alternative, see MATD024. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD020	LS-DYNA Material #20 -- Used to model rigid materials. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD20M	Merges two or more rigid materials defined using MATD020. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD022	LS-DYNA Material #22 -- Used to model an orthotropic material with optional brittle failure for composites. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD024	LS-DYNA Material #24 -- Used to model an elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD026	LS-DYNA Material #26 -- Used to model honeycomb and foam materials with real anisotropic behavior. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD027	LS-DYNA Material #27 -- Used to model rubber using two variables. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD028	LS-DYNA Material #28 -- A resultant formulation for beam and shell elements including elasto-plastic behavior can be defined. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD030	LS-DYNA Material #30 -- This material model describes the superelastic response present in shape-memory alloys (SMA). Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD031	LS-DYNA Material #31 -- Used to model rubber using the Frazer-Nash formulation. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD032	LS-DYNA Material #32 -- Used for Automotive Glass
MATD054	LS-DYNA Material #54 -- This material is an enhanced version of the composite model material type 22. Arbitrary orthothropic materials, e.g., unidirectional layers in composite shell structures can be defined. Used in MD Nastran Implicit Nonlinear (SOL 700) only.

MATD057	LS-DYNA Material #57 -- Material used to model highly compressible low density foams. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD058	LS-DYNA Material #58 -- Composite and Fabrics
MATD059	LS-DYNA Material #59 -- Material used to model shells or solid composite structures. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD062	LS-DYNA Material #62 --Used to model viscous foams. It was written to represent the Confor Foam on the ribs of EuroSID side impact dummy. It is only valid for solid elements, mainly under compressive loading. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD063	LS-DYNA Material #63 --Used to model crushable foams. It is dedicated to modeling crushable foam with optional damping and tension cutoff. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD064	LS-DYNA Material #64 -- Used to model strain rate sensitive elasto-plastic material with a power law hardening. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD066	This material model is defined for simulating the effects of a linear elastic beam by using six springs each acting about one of the six local degrees-of-freedom.
MATD067	This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom.
MATD068	This material model is defined for simulating the effects of nonlinear elastoplastic, linear viscous behavior of beams by using six springs, each acting about one of the six local degrees-of-freedom.
MATD069	The side impact dummy uses a damper that is not adequately treated by the nonlinear force versus relative velocity curves since the force characteristics are dependent on the displacement of the piston.
MATD070	This special purpose element represents a combined hydraulic and gas-filled damper which has a variable orifice coefficient
MATD071	This model permits elastic cables to be realistically modeled; thus, no force will develop in compression.

MATD073	This material is for Modeling Low Density Urethane Foam with high compressibility and with rate sensitivity which can be characterized by a relaxation curve.
MATD074	This model permits elastic springs with damping to be combined and represented with a discrete beam element type 6.
MATD076	This material model provides a general viscoelastic Maxwell model having up to 6 terms in the prony series expansion and is useful for modeling dense continuum rubbers and solid explosives.
MATD077	LS-DYNA Material #77 -- Used to model a general hyperelastic rubber model combined optionally with linear viscoelasticity as outlined by Christensen. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD080	LS-DYNA Material #80 -- Used to model Ramberg-Osgood plasticity. This model is intended as a simple model of shear behavior and can be used in seismic analysis. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD081	LS-DYNA Material #81 -- Used to model elasto-visco-plastic materials with arbitrary stress versus strain curves and arbitrary strain rate dependency. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD083	LS-DYNA Material #80 -- Rate effects can be modeled in low and medium density foams.
MATD087	This material model provides a cellular rubber model with confined air pressure combined with linear viscoelasticity as outlined by Christensen [1980].
MATD093	This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom.
MATD094	This model permits elastoplastic springs with damping to be represented with a discrete beam element type 6.
MATD095	This material model is defined for simulating the effects of nonlinear inelastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom.
MATD097	This model is used to define a general joint constraining any combination of degrees of freedom between two nodes.

MATD100	LS-DYNA Material #100 -- The material model applies to beam elements to solid elements with hourglass control. The failure models apply to both beam and solid elements. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD119	This is a very general spring and damper model. This beam is based on the MATDS06 material model.
MATD121	This is Material Type 121 -- This is a very general spring and damper model. This beam is based on the MATDS06 material model and is a one-dimensional version of MATD119.
MATD126	The major use of this material model is for aluminum honeycomb crushable foam materials with anisotropic behavior.
MATD127	LS-DYNA Material #127 -- Used to model rubber using the Arruda-Boyce formulation. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATD181	LS-DYNA Material #181 -- Used to model rubber or foam using a simplified formulation. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
MATDB01	Defines a seat belt material.
MATDS01	Defines a translational or rotational elastic spring located between two nodes.
MATDS02	Defines a translational or rotational linear damperlocated between two nodes.
MATDS03	Defines a translational or rotational elastoplastic spring with isotropic hardening located between two nodes.
MATDS04	Defines a translational or rotational nonlinear elastic spring with arbitrary force versus displacement or moment versus rotation, located between two nodes.
MATDS05	Defines a translational or rotational viscous damper with arbitrary force versus displacement or moment versus rotation, located between two nodes.
MATDS06	Defines a translational or rotational nonlinear spring with arbitrary loading and unloading definitions, located between two nodes.
MATDS07	Defines a translational or rotational three Parameter Maxwell Viscoelastic spring located between two nodes.

MATDS08	Defines a translational or rotational inelastic tension or compression only spring located between two nodes.
MATDS13	Defines a translational spring located between two nodes.
MATDS14	Define a translational spring located between two nodes.
MATDS15	Defines a translational spring located between two nodes.
MATRIG	Defines the properties of a rigid body.

Multi-Variable Parameters (SOL 700)

DYPARAM	Bulk data parameters for Dytran-LsDyna with extra fields (SOL 700 only).

Prescribed Boundary Motion (SOL 700)

SPCD2	Defines an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes.

Response Measurement (SOL 700)

ACC	Creates time history output of an accelerometer in local coordinate system.
ACCMETR	Accelerometer
SBSENSR	Defines a seat belt sensor.

Restarts (SOL 700)

DYCHANG	For a SOL 700 restart analysis, change certain solution options.
DYDELEM	Deletes properties or element using a list for SOL 700 restarts.
DYRIGSW	Defines materials to be switched from rigid to deformable and deformable to rigid in a restart.
RESTART	Specifies writing or reading of restart data for Nonlinear Analysis when MSC.Marc or Dytran-lsdyna is executed from MD Nastran.

Rigid Elements (SOL 700)

BJOIN	Defines (multiple) pairs of grid points of one-dimensional and/or shell elements to be joined during the analysis.
RBE2A	Defines extra nodes for rigid body.

RBE2D	Defines a nodal rigid body.
RBE2F	Defines nodal constraint sets for translational motion in global coordinates.
RBE2GS	Defines an RBE2 connecting the two closest grids to GS.
RBE3D	Defines rigid interpolation constraints in the MSC.Dytran style. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
RBJOINT	Defines a joint between two rigid bodies.
RCONN	Defines a rigid connection between the different parts of Lagrangian meshes (tied surfaces).
WALL	Defines a rigid plane through which specified Lagrangian grid points cannot penetrate. Used in MD Nastran Implicit Nonlinear (SOL 700) only.
WALLGEO	Defines a geometric rigid wall with an analytically described form. Four forms are possible.

Seat Belts (SOL 700)

CBELT	Defines a seat belt element.
PBELTD	Defines section properties for the seat belt elements.
PBDISCR	Defines properties for 6 DOF discrete beam elements.
SBPRET	Defines a seat belt pretensioner.
SBRETR	Defines a seat belt retractor.
SBSLPR	Defines seat belt slipring.

SOL 700 Output Control

DYTIMHS	Specifies various types of time history output and form of the output for SOL 700.

Switch Rigid/Deformable (SOL 700)

D2RAUTO	Defines a set of parts to be switched to rigid or to deformable at some stage in the calculation.
D2R0000	Defines materials to be switched to rigid at the start of the calculation.

Tables (SOL 700)

TABLEDR	Defines a table to reference other tables.

Time Step Control (SOL 700)

DYTERMT	Stop a SOL 700 analysis depending on specified displacement conditions.
TSTEPNL	Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. TSTEPNL is intended for SOLs 129, 159, 400, 600 and 700.

Welds (SOL 700)

CBUTT	Defines a butt weld in the LS-DYNA style. Replaces CWELD for SOL 700.
CCRSFIL	Defines a cross-fillet weld in the LS-DYNA style. Replaces CWELD for SOL 700.
CFILLET	Defines a fillet weld in the LS-DYNA style. Replaces CWELD for SOL 700.
COMBWLD	Defines a complex or combined weld in the LS-DYNA style. Replaces CWELD for SOL 700.
CSPOT	Defines a complex or combined weld in the LS-DYNA style. Replaces CWELD for SOL 700.

$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Format:

$ followed by any characters out to column 80.

Example:

```
$ TEST FIXTURE-THIRD MODE
```

Remarks:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

/ Delete

Removes entries on restart.

Format:

1	2	3	4	5	6	7	8	9	10
/	K1	K2							

Example:

/	4								

Field	Contents
K1	Sorted sequence number of first entry in sequence to be removed. (Integer > 0)
K2	Sorted sequence nummber of last entry in sequence to be removed. (Integer > 0; Default = K1)

Remarks:

1. This entry causes Bulk Data entries having sort sequence numbers K1 through K2 to be removed from the Bulk Data. The sort sequence numbers appear in the output of the previous run under the sorted Bulk Data echo.
2. If K2 is blank, only entry K1 is removed from the Bulk Data.
3. If the current execution is not a restart, the delete entries are ignored.
4. K2 may be specified as larger than the actual sequence number of the last entry. This is convenient when deleting entries to the end of the Bulk Data Section.

ACC (SOL 700) Accelerometer Output

Creates time history output of an accelerometer in local coordinate system.

Format:

1	2	3	4	5	6	7	8	9	10
ACC	AID	CID	SID	SAMPLE					

Example(s):

ACC	22	3	10	1.E-4					

Field	Contents	Type	Default
NAME	Unique ACC name	I > 0	Required
CID	Local coordinate system defined in the BULK DATA Section.	I > 0	Required
SID	Unique SET number	I > 0	Required
SAMPLE	Sampling rate. This value determines the time step when the measurements will be taken and stored in the time history file.	R > 0	Required

Remark:

1. The set SID referenced must contain ONLY 1 grid point ID, which must be the same node that defines the origin of the (moving) coordinate system referenced with CID.

ACCEL Acceleration Load

Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based upon the tabular input defined on this Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
ACCEL	SID	CID	N1	N2	N3	DIR			
	LOC1	VAL1	LOC2	VAL2	Continues in Groups of 2				

Example(s):

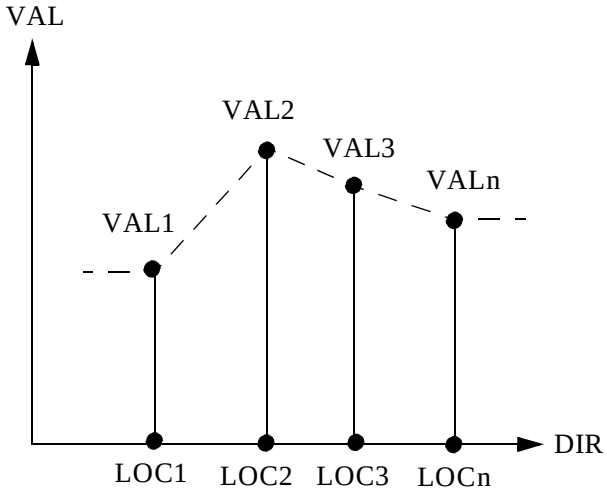
ACCEL	100	2	0.0	1.0	2.0	X			
	0.0	1.0	1000.0	3.0					

Field	Contents
SID	Load set identification number (Integer>0)
CID	Coordinate system identification number. (Integer>0: Default=0)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one $Ni \neq 0.0$)
DIR	Component direction of acceleration variation. (Character; one of X, Y, and Z)
LOCi	Location along direction DIR in coordinate system CID for specification of a load scale factor. (Real)
VALi	The load scale factor associated with location LOCi. (Real)

Remarks:

1. For all grids of the model, the acceleration vector is defined by $\vec{a} = VAL \cdot \vec{N}$, where $\vec{N}$ is the vector defined by (N1, N2, N3). The magnitude of $\vec{a}$ is equal to *VAL* times the magnitude of $\vec{N}$. The scale factor *VAL* for each grid is found linearly interpolating the DIR coordinate of the grid between table values *LOCi/VALi*. If the GRID point coordinate in coordinate system CID is outside the range of the table, VAL is determined either from *VAL1* or *VALn* (the last value, see the following figure).

2. This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL1 loads, by specification on a LOAD entry. The SID on an ACCEL entry may not be the same as that of any other load entry.
3. This acceleration load does not include effects due to mass on scalar points.
4. A CID of zero references the basic coordinate system.
5. The DIR field must contain one of the characters X, Y, or Z. The DIR direction defines the direction of acceleration load variation along direction 1, 2, or 3 respectively of coordinate system CID.
6. A minimum of two pairs of {LOCi, VALi} data must be specified.



Definition of Load Scale Factor vs Location

ACCEL1 Acceleration Load

Defines static acceleration loads at individual GRID points.

Format:

1	2	3	4	5	6	7	8	9	10
ACCEL1	SID	CID	A	N1	N2	N3			
	GRIDID1	GRIDID2	-etc.-						

Example(s):

ACCEL1	100	2	10.0	1.0	2.0	0.0			
	1	2	3	4	THRU	10	BY	2	
	20	21	THRU	30	40	52	69	70	
	82	90	100						

Field	Contents
SID	Load set identification number (Integer>0)
CID	Coordinate system identification number. (Integer>0: Default=0)
A	Acceleration vector scale factor. (Real)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one $Ni \neq 0.0$)
GRIDIDi LIST	List of one or more GRID point identification numbers. Key words “THRU” and “BY” can be used to assist the listing. (Integer>0)

Remarks:

1. The acceleration vector is defined by $\vec{a} = A \cdot \vec{N}$, where $\vec{N}$ is the vector defined by (N1, N2, N3). The magnitude of $\vec{a}$ is equal to *A* times the magnitude of $\vec{N}$.
2. This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL loads, by specification on a LOAD entry. The SID on an ACCEL1 entry may not be the same as that of any other load entry.
3. This acceleration load does not include effects due to mass on scalar points.
4. A CID of zero references the basic coordinate system.

ACCMETR (SOL 700) Accelerometer

The accelerometer is fixed to a rigid body containing the three nodes defined below. Whenever computed accelerations are compared to experimental results or whenever computed accelerations are compared between different runs, accelerometers are essential. Raw nodal accelerations contain considerable numerical noise and their comparisons are generally meaningless and, therefore, misleading.

Format:

1	2	3	4	5	6	7	8	9	10
ACCMETR	SBACID	NID1	NID2	NID3	IGRAV	INTOPT			

Example:

ACCMETR	12	64	54	53	0	0			

Field	Contents
SBACID	Accelerometer ID. A unique number must be used. (Integer $\geq$ 0, Required)
NID1	Node 1 ID. (Integer > 0, Required)
NID2	Node 2 ID. (Integer > 0, Required)
NID3	Node 3 ID. (Integer > 0, Required)
IGRAV	Gravitational accelerations due to body force loads. (Integer $\geq$ 0, Default = 0) = 0: included in acceleration output = 1: removed from acceleration output
INTOPT	Integration option. If the accelerometer undergoes rigid body translation without rotation this option has no effect; however, if rotation occurs, option 1 may provide better agreement with the output of the accelerometer. (Integer $\geq$ 0, Default = 0) = 0: velocities are integrated from the global accelerations and transformed into the local system of the accelerometer = 1: velocities are integrated directly from the local accelerations of the accelerometer.

Remarks:

1. Corresponds to Ls-Dyna entry * ELEMENT_SEATBELT_ACCELEROMETER.
2. The presence of the accelerometer means that the accelerations and velocities of node 1 will be output to all output files in local instead of global coordinates.

   The local coordinate system is defined by the three nodes as follows:

   - local **x** from node 1 to node 2,
   - local **z** perpendicular to the plane containing nodes, 1, 2, and 3 (**z** = **x** × **a)**, where **a** is from node 1 to node 3),
   - local **y** = **z** × **x.**

   The three nodes should all be part of the same rigid body. The local axis then rotates with the body.

ACMODL Fluid-Structure Interface Parameters

Defines modeling parameters for Fluid-Structure Interface.

Format:

(METHOD="BW")

1	2	3	4	5	6	7	8	9	10
ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD	SKNEPS	DSKNEPS	
	INTOL	ALLSSET	SRCHUNIT						

(METHOD="CP")

ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD			

Example(s):

(No entry recommended)

ACMODL	IDENT				NORMAL				

Field	Contents
INTER	Type of structure-fluid interface. (Character = "IDENT" or "DIFF"; Default = "DIFF")
INFOR	For METHOD="BW" and INTER = "DIFF", indicates if FSET and SSET are used to define the fluid-structure interface, "NONE" if not used, and whether they contain grids or elements. (Character = "GRIDS", "ELEMENTS", "ALL", or "NONE", Default = "NONE") For METHOD="CP" and INTER="DIFF", indicates if FSET and SSET are used to define the fluid-structure interface, "NONE" if not used. See Remark **10.** (Character = "ALL", or "NONE", Default = "NONE")
FSET	Optional identification of a SET1 entry that contains a list of fluid elements or grids on the fluid "skin". See Remark **2.** (Integer > 0 or blank)
SSET	Optional identification of a SET1 entry that contains a list of structural elements or grids on the structure-fluid interface. See Remark **2.** (Integer > 0 or blank)
NORMAL	Fluid normal tolerance. See Remark **5.** (Real; Default = 1.0 (Real; .001 for IDENT)

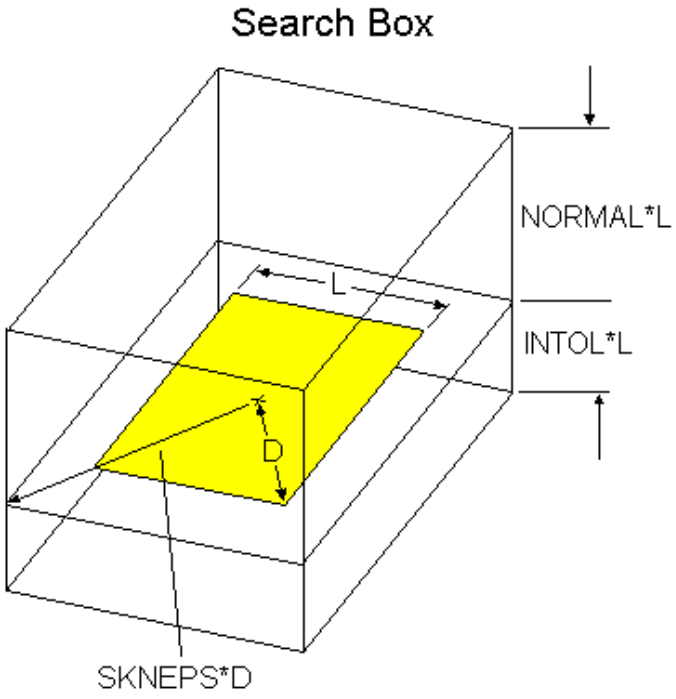
Field	Contents
METHOD	Default = “BW” “BW” = Body in White method “CP” = Closed Pressure Vessel See Remark **10.**
SKNEPS	Fluid skin growth tolerance. (Real; Default 0.5)
DSKNEPS	Secondary fluid skin growth tolerance (Real; Default .75)
INTOL	Tolerance of inward normal. (Real; Default .5)
ALLSSET	If “NO” then SSET structure is searched and coupled if found. If ‘YES’ then all the structure given by SSET is coupled. (Character = ’YES’, or ‘NO’; Default = ’NO’)
SRCHUNIT	Search units. (Character=‘ABS’ for absolute model units or ‘REL’ for relative model units based on element size; Default=’REL’)

Remarks:

1. Only one ACMODL entry is allowed. In general, for large irregular models, it is recommended that, initially, this entry not be used, so the defaults will be applied.
2. For METHOD=”BW” and INTER = “DIFF” (default), FSET and SSET refer to either grids or elements as selected below. For INTER = “IDENT”, FSET and SSET refer to grids.
   a. For INTER = “DIFF”, INFOR = “ELEMENTS”, for FSET, the search algorithm is restricted to elements referenced by FSET. For SSET, the search algorithm is restricted to elements referenced by SSET. This allows the user to de-select specific structural faces of a solid structural element. Both these sets are optional and the user can have one without the other.
   b. For INTER = “DIFF”, INFOR = “GRIDS”, for FSET, the search algorithm is restricted to grids referenced by FSET. This allows the user to deselect fluid grids. Fluid grid selection is the only way to deselect specific fluid faces. For SSET, the search algorithm is restricted to grids referenced by SSET. This allows the user to de-select structural grids. Both these sets are optional and the user can have one without the other.
   c. For INTER = “IDENT”, INFOR = “ALL”, the points referenced by FSET and SSET must lie exactly on the fluid-structure interface. These sets are optional, but if used, both must be present or no fluid interface is calculated.

3. For INTER = “DIFF”, a .PCH file is created with a SET1 representing the fluid “skin” and a SET1 representing the structure interface. This file is useful for graphic post-processing for viewing the interface. It also produces the sets that can be used as FSET and SSET.
4. For ALLSSET = ‘NO’ (default) the elements and grids determined by the couplings algorithm are written to the .PCH file. The user can then deselect elements or grids as defined by the .PCH file by editing them out of the SET1 entries defined in the file and referencing the edited SET1 with the SSET. To add structural elements that the coupling algorithm did not include in the .PCH file, it is not sufficient to just include them on the SET1 entry referenced by SSET. In addition, ALLSSET = ‘YES’ must be specified.
5. NORMAL determines the height of the fluid box in the outward normal direction to the fluid surface. The fluid box is used to locate the structural elements used in defining the fluid-structure coupling matrix. If L is the smallest edge of the fluid element surface, then the height of the box is L x NORMAL.

   For INTER = “IDENT”, NORMAL = .001 is the default and represents a tolerance, in units of length, used in determining the fluid-structure interface.
6. SKNEPS represents the enlargement of the plane of the fluid surface used to define the search box. The diagonal distance from the center of the fluid surface to each surface grid is pushed out (diagonal x (1. + SKNEPS)).



7. DSKNEPS represents a secondary enlargement of the plane of the fluid surface used to define the search box if SKNEPS fails to find ANY structural elements. The diagonal distance from the center of the fluid surface to each surface grid is pushed out (diagonal x (1. + DSKNEPS)).
8. INTOL represents a normal direction into the fluid for the case when the fluid protrudes past the structural interface. It is defined as L x INTOL where L is the smallest edge of the fluid element surface.
9. The area of each structural element projected normal to the fluid element will be used as a weighting function. The expression is of the form

$$\{F_j\} = [W][R]([R]^T[W][R])^{-1}\begin{Bmatrix} R_i \\ 0 \\ 0 \end{Bmatrix}$$

where $\{F_j\}$ is the vector of resulting load distribution at the grids of each of the j structural elements. $[W]$ is a weighting function. $[R]$ is the MD Nastran rigid body distribution matrix. R_i is the resultant pressure force for a unit grid pressure of the fluid element.

10. The default METHOD is the new “BW” searching algorithm that requires a special license.

    The pre-Version 2004 method is selected with METHOD = “CP” in which:

    - the search box is not used so the SKNEPS, DSKNEPS, INTOL, ALLSET, SRCHUNIT fields are ignored.
    - if INFOR = ‘ALL’ (METH = ‘CP’ only), then both FSET and SSET must be specified and matching is checked at only those grid points referenced by FSET and SSET.
    - FSET and SSET refer to grids.
    - NORMAL = blank is the default (recommended), $1. \leq$ NORMAL $\leq 10.$ gets acceptable results. In this case, NORMAL represents a maximum cutoff value measured in physical units. When NORMAL = ‘blank’, MD Nastran will compute the cutoff value.

      This field replaces the pre-Version 2004 FSTOL field. Different units are also used so pre-Version 2004 FSTOL values may need to be changed to obtain the same results.

ACSRCE Acoustic Source Specification

Defines acoustic source as a function of power vs. frequency.

$$\text{Source Strength} = \{A\} \cdot \left[\frac{1}{2\pi f}\sqrt{\frac{8\pi CP(f)}{\rho}}\right] e^{i(\theta + 2\pi f\tau)}$$

$$C = \sqrt{B/\rho}$$

Format:

1	2	3	4	5	6	7	8	9	10
ACSRCE	SID	EXCITEID	DELAY	DPHASE	TP	RHO	B		

Example:

ACSRCE	103	11			12	1.0	15.0		

Field	Contents
SID	Load set identification number. (Integer > 0)
EXCITEID	Identification number of DAREA or SLOAD entry set the defines $\{A\}$. (Integer > 0)
DELAY	Defines time delay τ. (Integer $\geq$ 0, real or blank). If it is a non-zero integer, it represents the identification number of DELAY Bulk Data entry that defines τ. If it is real, then it directly defines the value of τ that will be used for all fluid degrees-of-freedom that are excited by this dynamic load entry. See also Remark 4.
DPHASE	Defines phase angle θ. (Integer $\geq$ 0, real or blank). If it is a non-zero integer, it represents the identification number of DPHASE Bulk Data entry that defines θ (in degrees). If it is real, then it directly defines the value of θ (in degrees) that will be used for all fluid degrees-of-freedom that are excited by this dynamic load entry. See also Remark 4.
TP	Identification number of a TABLEDi entry that defines power versus frequency, $P(f)$. (Integer $\geq$ 0 or blank)
RHO	Density of the fluid. (Real > 0.0)
B	Bulk modulus of the fluid. (Real > 0.0)

Remarks:

1. Acoustic sources must be selected in the Case Control Section with DLOAD = SID.
2. For additional remarks, see the RLOAD1 entry description.
3. SID must be unique for all ACSRSE, RLOADi, and TLOADi entries.
4. The referenced EXCITEID, DELAY, and DPHASE entries must specify fluid points only.

ADAPT Version Adaptivity Control

Defines controls for p-version adaptive analysis.

Format:

1	2	3	4	5	6	7	8	9	10
ADAPT	SID	ADGEN	MAXITER	PSTRTID	PMINID	PMAXID			
	PART=name1, option1=value1, option2=value2, etc., PART=name2								
	option1=value1, option2=value2, etc., PART=name3, etc.								

Example:

ADAPT	127		3	23		45			
	PART=LOWSTR, ELSET=11, TYPE=UNIP, SIGTOL=22								
	PART=HISTR, ELSET=111, ERREST=2, EPSTOL=.002								

Field	Contents	Type	Default
SID	Adapt entry ID selected in Case Control by ADAPT command.	Integer > 0	None
ADGEN	ID of the first PVAL entry generated in the adaptive process. See Remark **14.**	Integer > PSTRTID, PMINID, PMAXID	1000
MAXITER	Number of analyses performed before adaptive process is stopped.	Integer > 0	3
PSTRTID	ID of PVAL entry describing the starting p-order distribution.	Integer > 0	None
PMINID	ID of PVAL entry describing the minimum p-order distribution. See Remark **10.**	Integer > 0	PSTRTID
PMAXID	ID of PVAL entry describing the maximum p-order distribution. See Remark **10.**	Integer > 0	PSTRTID
optioni = valuei	Assigns a value to an option described later. See Remark **16.**		

Field	Contents	Type	Default
PART	Part name of the elements defined in ELSET and controlled by TYPE, ERREST, ERRTOL, SIGTOL, and EPSTOL.	Character	PART = MODEL
ELSET	ID of the SET command under the SETS DEFINITION command. See Remark 7.	Integer > 0	ELSET = 999999
TYPE	p-order adjustment. See Remark 3.	Character or Integer > 0	TYPE = EBEP
ERREST	Error estimator activation flag. See Remark 2.	Integer $\geq$ 0	ERREST = 1
ERRTOL	Error tolerance. Required if MAXITER is not specified.	0.0<Real<1.0	ERRTOL=0.01
SIGTOL	Stress tolerance. If the von Mises stress at the center of the element is below this value, the element will not participate in the error analysis.	Real $\geq$ 0.0	SIGTOL = 0.0
EPSTOL	Strain tolerance. If the von Mises strain at the center of the element is below this value, the element will not participate in the error analysis.	Real $\geq$ 0.0	EPSTOL = 1.0E-8

Remarks:

1. Only one ADAPT entry may be specified. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
2. The error estimator is activated by ERREST = 1 and is based on strain energy density sensitivity and stress discontinuity in neighboring elements. ERREST = 0 means no error estimation will be performed on the PART.

3. The types of p-order adjustment are:

Type	Description
EBEP	The p-order will increase only in the elements that are required by the error analysis.
UNIP	If any element in the group has an error larger than the tolerance, all elements will be increased by one order in each direction.
NOCH	The p-order of the group does not change during the iterations.
LIST	The PVAL distribution specified as PSTRTID is used for the first iteration. The user is required to provide PVAL entries with IDs starting with ADGEN, and these p-distributions will be used in the following iterations

4. If a PVAL ID is not specified for PSTRTID, PMINID, or PMAXID, then this is equivalent to no change at the last PVAL ID found for the element.
5. The elements specified in the SET could overlap. In this case, the highest p_1, highest p_2, highest p_3 (the polynomial order of the elements in three directions) determined by the error estimator will be used.
6. n restart, PMINID and PMAXID must not refer to any PVAL identification number that was generated in the previous run(s). Also, PARAM,PVALINIT must specify the desired PVAL identification number from which to restart.
7. If an element in the SET does not have a PVAL for PSTRTID or PMINID or PMAXID, it will be excluded from the adaptivity process.
8. SET = 999999 is a reserved set that includes all elements.
9. The user can specify as many PARTs as needed.
10. Each finite element has to have a unique PVAL for PSTRTID, PMINID, PMAXID. Any overlap of the PVAL specification will result in a warning message and the use of the PVAL with the highest pi field (highest p_2 if same p_1 and highest p_3 if same p_1 and p_2) and the lowest CID value.

11. The p-distribution for an element specified by the PVAL entry referenced by PMAXID must be larger than the distribution specified by the PSTRTID, which must be larger than the distribution specified by the PMINID. A warning message will be issued if these conditions are not met, and the data is reset.
12. The solution vector of all the elements listed in the SET entries for all loads and boundary conditions will be used in the error estimation. New p values are generated for all the elements.
13. When ERREST = 0, no error analysis is performed. The p-value of the elements in the set are increased uniformly starting from p-values specified on the PVAL entry referenced by PSTRTID up to values specified on the PVAL entry referenced by PMAXID.
14. The intermediate PVAL entries generated will have an ID starting with ADGEN; thus, ADGEN must be larger than PSTRTID, PMINID, and PMAXID.
15. The displacement and stress output can be requested by a DATAREC Case Control command.
16. Each optioni = valuei must be specified on the same entry. In other words, optioni and valuei may not be specified on two separate continuation entries.

ADUMi Dummy Element Attributes

Defines attributes of the dummy elements ($1 \leq i \leq 9$).

Format:

1	2	3	4	5	6	7	8	9	10
ADUMi	NG	NC	NP	ND	ELNM				

Example:

ADUM2	8	2	1	3	CTRIM6				

Field	Contents
NG	Number of grid points connected by DUMi dummy element. (Integer > 0)
NC	Number of additional fields (Ai) on the CDUMi connection entry. (Integer $\geq$ 0)
NP	Number of additional fields (Ai) on the PDUMi property entry. ($24 \geq$ Integer ≥ 0)
ND	Number of displacement components at each grid point used in generation of the differential stiffness matrix. Zero implies no differential stiffness. (Integer 3 or 6)
ELNM	The name of the element connection and property entry. In the example above, the connection entry is named “CTRIM6" and the property entry is named “PTRIM6".

AECOMP Component for an Integrated Load Monitor Point

Defines a component for use in aeroelastic monitor point definition.

Format:

1	2	3	4	5	6	7	8	9	10
AECOMP	NAME	LISTTYPE	LISTID1	LISTID2	LISTID3	LISTID4	LISTID5	LISTID6	
	LISTID7	-etc.-							

Example:

AECOMP	WING	AELIST	1001	1002					

Field	Contents
NAME	A character string of up to eight characters identifying the component. (Character)
LISTTYPE	One of CAERO or AELIST for aerodynamic components and SET1 for structural components. Aerodynamic components are defined on the aerodynamic ks-set mesh while the structural components are defined on the g-set mesh. See Remarks **2.** and **3.**
LISTIDi	The identification number of either SET1, AELIST or CAEROi entries that define the set of grid points that comprise the component. See Remarks **2.** and **3.**

Remarks:

1. The Identification name must be unique among all AECOMP and AECOMPL entries.
2. If the component is defined on the structure, a LISTIDs must refer to SET1 entry(ies) that define the list of associated GRID points. For the AELIST or CAERO option, the LISTIDs must refer to AELIST or CAERO i entries, respectively. Note that, for DLM models (CAERO1/2), the set of points defined by the AELIST are the box identification numbers. For example, if the control surface's grids are desired, the same AELIST used for the AESURF can be referred to here. An AECOMP component must be defined as either an aerodynamic mesh component or a structural component. The two mesh classes cannot be combined into a single component.

3. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.

AECOMPL Component for an Integrated Load Monitor Point

Defines a component for use in aeroelastic monitor point definition as a union of other components.

Format:

1	2	3	4	5	6	7	8	9	10
AECOMPL	NAME	LABEL1	LABEL2	LABEL3	LABEL4	LABEL5	LABEL6	LABEL7	
	LABEL8	-etc.-							

Example:

AECOMPL	HORIZ	STAB	ELEV	BALANCE					

Field	Contents
NAME	A character string of up to eight characters Identifying the component. (Character)
LABELi	A string of 8 characters referring to the names of other components defined by either AECOMP or other AECOMPL entries.

Remarks:

1. The Identification name must be unique among all AECOMP and AECOMPL entries.
2. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.

AEDW Parametric Normal Wash Loading for Aerodynamics

Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG). From this downwash vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

Format:

1	2	3	4	5	6	7	8	9	10
AEDW	MACH	SYMXZ	SYMXY	UXID	DMIJ	DMIJI			

Example:

AEDW	0.90	SYMM	ASYMM	101	ALP1				

Field	Contents
MACH	The Mach number for this force, see Remark 2. (Real ≥ 0.0, ≠ 1.0)
SYMXZ,SYMXY	The symmetry of this force vector. One of SYMM, ASYMM or ANTI (Character).
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this downwash vector.
DMIJ	The name of a DMI or DMIJ entry that defines the downwash.
DMIJI	The name of a DMIJI entry that defines the CAERO2 interference element “downwashes”.

Remarks:

1. The AEDW, AEFORCE and AEPRESS are associated with the current AECONFIG using either Case Control (if in the main Bulk Data Section) or using the BEGIN AECONFIG=<config> if in a partition of the Bulk Data.
2. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero “downwash” (j-set) input is needed on the interference body elements.
3. Mach numbers > 1.0 require that the supersonic aerodynamic option be available.

AEFACT Aerodynamic Lists

Defines real numbers for aeroelastic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
AEFACT	SID	D1	D2	D3	D4	D5	D6	D7	
	D8	D9	-etc.-						

Example:

AEFACT	97	.3	.7	1.0					

Field	Contents
SID	Set identification number. (Unique Integer > 0)
Di	Number. (Real)

Remarks:

1. AEFACT entries must be selected by a CAEROi or PAEROi entry.
2. Embedded blank fields are not allowed.
3. To specify division points, there must be one more division point than the number of divisions.
4. When referenced by the CAERO3 entry, AEFACT defines the aerodynamic grid points. The ID number of the first point defined by each AEFACT entry is the value of the CAERO3 ID that selected the AEFACT entry. The ID of each following point defined on the AEFACT is incremented by 1.

AEFORCE Parametric Force for Aerodynamics

Defines a vector of absolute or “per unit dynamic pressure” forces associated with a particular control vector. This force vector may be defined on either the aerodynamic mesh (ks-set) or the structural mesh (g-set). The force vector will be used in static aeroelastic trim.

Format:

1	2	3	4	5	6	7	8	9	10
AEFORCE	MACH	SYMXZ	SYMXY	UXID	MESH	LSET	DMIK	PERQ	

Example:

AEFORCE	0.90	SYMM	ASYMM	101	AERO		BETA		

Field	Contents
MACH	The Mach number for this force. (Real ≥ 0.0, ≠ 1.0)
SYMXZ,SYMXY	The symmetry conditions for this force vector. One of SYMM, ASYMM or ANTI. (Character)
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this force vector.
MESH	One of AERO or STRUCT that declares whether the force vector is defined on the aerodynamic ks-set mesh or the structural g-set mesh.
LSET	SID of a load set that defines the vector. See Remark 2. (Integer > 0 if MESH=STRUCT)
DMIK	The name of a DMIK entry that defines the aerodynamic force vector. See Remark 3. (Character; required if MESH=AERO)
PERQ	The string PERQ or blank. If PERQ, the corresponding FORCE set is scaled by dynamic pressure. (Default = blank)

Remarks:

1. The AEFORCE is associated with the current AECONFIG and must be entered for the appropriate Mach numbers and aerodynamic symmetries.
2. For the STRUCT mesh option, the LSET can refer to any existing load type (e.g., FORCE1, PLOAD4 or LOAD) that is available to define static loads.

3. For the AERO mesh option, the DMIK Bulk Data are used. Any forces associated with the aerodynamic model's permanently SPC'd degrees-of-freedom (which are dependent on the type of aerodynamic model being used) will be ignored.
4. For the STRUCT mesh option, setting the PERQ field to "PERQ" will cause the LSET data to be scaled by dynamic pressure.

AELINK Links Aeroelastic Variables

Defines relationships between or among AESTAT and AESURF entries, such that:

$$u^D + \sum_{i=1}^{n} \text{Ci}\ u_i^I = 0.0$$

where:

u^D = dependent variable

u_i^I = independent variable

Format:

1	2	3	4	5	6	7	8	9	10
AELINK	ID	LABLD	LABL1	C1	LABL2	C2	LABL3	C3	
	LABL4	C4	-etc.-						

Example:

AELINK	10	INBDA	OTBDA	-2.0					

Field	Contents
ID	If an integer > 0 is specified, this is the TRIM set ID selected in Case Control and the AELINK only applies to that subcase. If an integer value of 0 or the character string “ALWAYS” is specified, this AELINK is applicable to all subcases. (Integer $\geq$ 0 or the “ALWAYS” character string.)
LABLD	Character string to identify the dependent aerodynamic variable. (Character)
LABLi	Character string to identify the i-th independent aerodynamic variable. (Character)
Ci	Linking coefficient for the i-th variable. (Real)

Remarks:

1. If the ID is a positive integer, the AELINK entry (or entries) is selected by the TRIM=ID in Case Control.

2. If the ID is 0 or the character string ALWAYs, the linking relationship applies to all subcases
3. The entry constrains the dependent variable to be a linear combination of the independent variables.
4. LABLD data must be unique for a given ID or if ID=0 or AWAYS is being used (i.e., the variable cannot be constrained more than once).
5. LABLD and LABLi refer to AEPARM, AESTAT or AESURF Bulk Data entries.

AELIST Aerodynamic Elements List

Defines a list of aerodynamic elements to undergo the motion prescribed with the AESURF Bulk Data entry for static aeroelasticity.

Format:

1	2	3	4	5	6	7	8	9	10
AELIST	SID	E1	E2	E3	E4	E5	E6	E7	
	E8	-etc.-							

Example:

AELIST	75	1001	THRU	1075	1101	THRU	1109	1201	
	1202								

Field	Contents
SID	Set identification number. (Integer > 0)
Ei	List of aerodynamic boxes generated by CAERO1 entries to define a surface. (Integer > 0 or “THRU”)

Remarks:

1. These entries are referenced by the AESURF entry.
2. When the “THRU” option is used, all intermediate grid points must exist. The word “THRU” may not appear in field 3 or 9 (2 or 9 for continuations).
3. Intervening blank fields are not allowed.

AEPARM General Controller for Use in Trim

Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point). The forces associated with this controller will be derived from AEDW, AEFORCE and AEPRESS input data.

Format:

1	2	3	4	5	6	7	8	9	10
AEPARM	ID	LABEL	UNITS						

Example:

AEPARM	5	THRUST	LBS						

Field	Contents
ID	Controller identification number. (Integer > 0)
LABEL	Controller name. See Remark 1. (Character)
UNITS	Label used to describe the units of the controller values. (Character)

Remarks:

1. Controller LABELs that comprise the unique set relative to all the AESURF, AESTAT and AEPARM entries will define the set of trim variable degrees-of-freedom for the aeroelastic model.
2. Unit labels are optional and are only used to label outputs. No units will be associated with the controller if left blank.

AEPRESS Parametric Pressure Loading for Aerodynamics

Defines a vector of pressure / unit dynamic pressure associated with a particular control vector. From this pressure vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

Format:

1	2	3	4	5	6	7	8	9	10
AEPRESS	MACH	SYMXZ	SYMXY	UXID	DMIJ	DMIJI			

Example:

AEPRESS	0.90	SYMM	ASYMM	101	ALP1				

Field	Contents
MACH	The Mach number for this force, see Remark 2. (Real ≥ 0.0, ≠ 1.0)
SYMXZ,SYMXY	The symmetry of this force vector. One of SYMM, ASYMM or ANTI. (Character)
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this pressure vector.
DMIJ	The name of a DMI or DMIJ entry that defines the pressure per unit dynamic pressure.
DMIJI	The name of a DMIJI entry that defines the CAERO2 interference element “downwashes”.

Remarks:

1. The AEDW, AEFORCE, and AEPRESS are associated with the current AECONFIG using Case Control.
2. Mach numbers > 1.0 require that the supersonic aerodynamic option be available.
3. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero “downwash” (j-set) input is needed on the interference body elements.

AERO Aerodynamic Physical Data

Gives basic aerodynamic parameters for unsteady aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
AERO	ACSID	VELOCITY	REFC	RHOREF	SYMXZ	SYMXY			

Example:

AERO	3	1.3+4	100.	1.-5	1	-1			

Field	Contents
ACSID	Aerodynamic coordinate system identification. See Remark **2.** (Integer $\geq$ 0; Default is the basic coordinate system)
VELOCITY	Velocity for aerodynamic force data recovery and to calculate the BOV parameter. See Remark **5.** (Real)
REFC	Reference length for reduced frequency. (Real > 0.0)
RHOREF	Reference density. (Real > 0.0)
SYMXZ	Symmetry key for the aero coordinate x-z plane. See Remark **6.** (Integer = +1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default = 0)
SYMXY	The symmetry key for the aero coordinate x-y plane can be used to simulate ground effect. (Integer = -1 for symmetry, 0 for no symmetry, and +1 for antisymmetry; Default = 0)

Remarks:

1. This entry is required for aerodynamic problems. Only one AERO entry is allowed.
2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction.
3. Set SYMXY = -1 to simulate ground effect.
4. PARAM,WTMASS does not affect aerodynamic matrices. RHOREF must be input in mass units.
5. VELOCITY is used only in aeroelastic response analysis, and it must be equal to V on the GUST Bulk Data entry.

6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMXY, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

AEROS Static Aeroelasticity Physical Data

Defines basic parameters for static aeroelasticity.

Format:

1	2	3	4	5	6	7	8	9	10
AEROS	ACSID	RCSID	REFC	REFB	REFS	SYMXZ	SYMXY		

Example:

AEROS	10	20	10.	100.	1000.	1			

Field	Contents
ACSID	Aerodynamic coordinate system identification. See Remark **2.** (Integer ≥ 0; Default is the basic coordinate system)
RCSID	Reference coordinate system identification for rigid body motions. (Integer ≥ 0; Default is the basic coordinate system)
REFC	Reference chord length. (Real > 0.0)
REFB	Reference span. (Real > 0.0)
REFS	Reference wing area. (Real > 0.0)
SYMXZ	Symmetry key for the aero coordinate x-z plane. See Remark **6.** (Integer = +1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default = 0)
SYMXY	The symmetry key for the aero coordinate x-y plane can be used to simulate ground effects. (Integer = +1 for antisymmetry, 0 for no symmetry, and -1 for symmetry; Default = 0)

Remarks:

1. This entry is required for static aeroelasticity problems. Only one AEROS entry is allowed.
2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction (T1).
3. The RCSID must be a rectangular coordinate system. All AESTAT degrees-of-freedom defining trim variables will be defined in this coordinate system.
4. REFB should be full span, even on half-span models.
5. REFS should be half area on half-span models.

6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMXY, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

AESTAT Static Aeroelasticity Trim Variables

Specifies rigid body motions to be used as trim variables in static aeroelasticity.

Format:

1	2	3	4	5	6	7	8	9	10
AESTAT	ID	LABEL							

Example:

AESTAT	5001	ANGLEA							

Field	Contents
ID	Identification number of an aerodynamic trim variable degree-of-freedom. See Remark 1. (Integer > 0)
LABEL	An alphanumeric string of up to eight characters used to identify the degree-of-freedom. See Remark 1. (Character)

Remarks:

1. The degrees-of-freedom defined with this entry represent rigid body motion in the reference coordinate system defined on the AEROS entry. The standard labels that define the various rigid body motions are as follows:

Table 8-1 Standard Labels Defining Rigid Body Motions

LABEL	Degree-of-Freedom Motion	Description
ANGLEA	u_r (R2)	Angle of Attack
SIDES	u_r (R3)	Angle of Sideslip
ROLL	$\dot{u}_r$ (R1)	Roll Rate = pb/2V
PITCH	$\dot{u}_r$ (R2)	Pitch Rate = qc/2V
YAW	$\dot{u}_r$ (R3)	Yaw Rate = rb/2V
URDD1	$\ddot{u}_r$ (T1)	Longitudinal (See Remark 3.)
URDD2	$\ddot{u}_r$ (T2)	Lateral

Table 8-1 Standard Labels Defining Rigid Body Motions (continued)

LABEL	Degree-of-Freedom Motion	Description
URDD3	$\ddot{u}_r$ (T3)	Vertical
URDD4	$\ddot{u}_r$ (R1)	Roll
URDD5	$\ddot{u}_r$ (R2)	Pitch
URDD6	$\ddot{u}_r$ (R3)	Yaw

These reserved names may be defined on the AEPARM entry instead, in which case the incremental load due to the unit perturbation of the rigid body degree-of-freedom (as it will with AESTAT). See the AEPARM, AEPRESS, and AEFORCE entries.

2. The degrees-of-freedom defined with this entry are variables in the static aeroelastic trim solution, unless they are constrained by referencing them with a TRIM Bulk Data entry.
3. If a label other than those above is specified, then the user must either generate the corresponding forces with an AELINK or via a DMI Bulk Data entry along with a DMAP alter that includes the DMIIN module and additional statements to merge into the appropriate matrices. Or, using AEPARM and the AEDW, AEPRESS, and/or AEFORCE, you can accomplish this purpose without the need for any alters.

AESURF Aerodynamic Control Surface

Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points. The forces associated with this controller will be derived from rigid rotation of the aerodynamic model about the hinge line(s) and from AEDW, AEFORCE and AEPRESS input data. The mass properties of the control surface can be specified using an AESURFS entry.

Format:

1	2	3	4	5	6	7	8	9	10
AESURF	ID	LABEL	CID1	ALID1	CID2	ALID2	EFF	LDW	
	CREFC	CREFS	PLLIM	PULIM	HMLLIM	HMULIM	TQLLIM	TQULIM	

Example:

AESURF	6001	ELEV	100	100	200	200			
	10.0	180.0			-1.4E4	1.2E4	20	30	

Field	Contents
ID	Controller identification number. (Integer > 0)
LABEL	Controller name. (Character)
CIDi	Identification number of a rectangular coordinate system with a y-axis that defines the hinge line of the control surface component. (Integer > 0)
ALIDi	Identification of an AELIST Bulk Data entry that identifies all aerodynamic elements that make up the control surface component. (Integer > 0)
EFF	Control surface effectiveness. See Remark 4. (Real ≠ 0.0; Default = 1.0)
LDW	Linear downwash flag. See Remark 2. (Character, one of LDW or NOLDW; Default = LDW).
CREFC	Reference chord length for the control surface. (Real > 0.0; Default = 1.0)
CREFS	Reference surface area for the control surface. (Real > 0.0; Default = 1.0)

Field	Contents
PLLIM,PULIM	Lower and upper deflection limits for the control surface in radians. (Real, Default = $\pm \pi/2$)
HMLLIM,HMULIM	Lower and upper hinge moment limits for the control surface in force-length units. (Real, Default = no limit)
TQLLIM,TQULIM	Set identification numbers of TABLEDi entries that provide the lower and upper deflection limits for the control surface as a function of the dynamic pressure. (Integer > 0, Default = no limit)

Remarks:

1. The ID on AESURF, AESTAT, and AEPARM entries are ignored. AESURFS can be used to define mass properties of the control surface.
2. The degrees-of-freedom defined on this entry represent a rigid body rotation of the control surface components about their hinge lines. In the default LDW (Linear DownWash) case, the downwash due to a unit perturbation of the control surface will be computed as part of the database. In the NOLDW case, the user must prescribe the controller's effects by direct definition of the induced forces using the AEPRESS, AEDW and/or AEFORCE entries.
3. Either one or two control surface components may be defined.
4. If EFF is specified, then the forces produced by this surface are modified by EFF (e.g., to achieve a 40% reduction, specify EFF=0.60).
5. The continuation is not required.
6. The CREFC and CREFS values are only used in computing the nondimensional hinge moment coefficients.
7. Position limits may be specified using either PiLIM or TQiLIM, but not both.
8. Position and hinge moment limits are not required.

AESURFS Structural Grids on an Aerodynamic Control Surface

Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry. The mass associated with these structural nodes define the control surface moment(s) of inertia about the hinge line(s).

Format:

1	2	3	4	5	6	7	8	9	10
AESURFS	ID	LABEL		LIST1		LIST2			

Example:

AESURFS	6001	ELEV		6002		6003			

Field	Contents
ID	Controller identification number, see Remark **1.** (Integer > 0)
LABEL	Controller name, see Remark **1.** (Character)
LISTi	Identification number of a SET1 entry that lists the structural grid points that are associated with this component of this control surface. (Integer > 0)

Remarks:

1. The LABEL on the AESURFS entry must match one on an AESURF entry. The ID is ignored.
2. The mass of the GRID points listed on the SETi entries is used to compute the mass moment of inertia of the control surface about its i-th hinge line. The presence of these data will allow the hinge moments to include the inertial forces in the computations. These data are optional, and, if omitted, result in hinge moments which include only the applied, aeroelastically corrected, forces.
3. These data will be associated to a structural superelement by grid list or partitioned SUPER=<seid> if the AESURFS is defined in the main bulk data section.

ASET Degrees-of-freedom for the a-set

Defines degrees-of-freedom in the analysis set (a-set)

Format:

1	2	3	4	5	6	7	8	9	10
ASET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

ASET	16	2	23	3516	1	4			

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive a-set. They may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
2. When ASET, ASET1, QSET, and / or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
3. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry is specified then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.

ASET1 Degrees-of-freedom for the a-set, Alternate Form of ASET Entry

Defines degrees-of-freedom in the analysis set (a-set)

Format:

1	2	3	4	5	6	7	8	9	10
ASET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

ASET1	345	2	1	3	10	9	6	5	
	7	8							

Alternate Format and Example:

ASET1	C	ID1	“THRU”	ID2					
ASET1	123456	7	THRU	109					

Field	Contents
C	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; for THRU option, ID1 < ID2)

Remarks:

1. Degrees-of-freedom specified on this entry form members of the a-set that is exclusive from other sets defined by Bulk Data entries. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.
2. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
3. If the alternate format is used, all points in the sequence ID1 through ID2 are not required to exist, but there must be at least one degree-of-freedom in the a-set for the model, or a fatal error will result. Any points implied in the THRU that do not exist will collectively produce a warning message but will otherwise be ignored.

4. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry is specified then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.

AXIC Conical Shell Problem Flag

Defines the existence of an axisymmetric conical shell problem.

Format:

1	2	3	4	5	6	7	8	9	10
AXIC	H								

Example:

AXIC	15								

Field	Contents
H	Highest harmonic defined for the problem. (0 < Integer < 998)

Remarks:

1. Only one AXIC entry is allowed. When the AXIC entry is present, most other entries are not allowed. The types that are allowed with the AXIC entry are listed below:

CCONEAX	MATT1	SPCADD
DAREA	MOMAX	SPCAX
DELAY	MOMENT	SUPAX
DLOAD	MPCADD	TABDMP1
DMI	MPCAX	TABLED1
DMIG	NOLIN1	TABLED2
DPHASE	NOLIN2	TABLED3
EIGB	NOLIN3	TABLED4
EIGC	NOLIN4	TABLE11
EIGP	NSM	TABLEM2
EIGR	NSM1	TABLEM3
EIGRL	NSMADD	TABLEM4
EPOINT	OMITAX	TEMPAX
FORCE	PARAM	TF
FORCEAX	PCONEAX	TIC

FREQ	POINTAX	TLOAD1
FREQ1	PRESAX	TLOAD2
FREQ2	RINGAX	TSTEP
GRAV	RFORCE	
LOAD	RLOAD1	
MAT1	RLOAD2	
MAT2	SECTAX	

2. For a discussion of the conical shell element, see the "**Conical Shell Element (RINGAX)**" on page 155 of the *MSC.Nastran Reference Guide.*

AXIF Fluid Related Axisymmetric Parameters

Defines basic parameters and the existence of an axisymmetric fluid analysis.

Format:

1	2	3	4	5	6	7	8	9	10
AXIF	CID	G	DRHO	DB	NOSYM	F			
	N1	N2	N3	N4	N5	-etc.-			

Example:

AXIF	2	32.2	0.12	2.4+5	YES				
	1	2	3		4		7	10	

Alternate Formats and Examples of Continuation Data:

	N1	“THRU”	Ni						
	0	THRU	10						

	N1	“THRU”	Ni	“STEP”	NS				
	0	THRU	9	STEP	3				

AXIF	100	-386.0		0.0	NO				
	0	THRU	50	STEP	5				
	52								
	54	THRU	57						
	61	THRU	65						
	68		71		72	75			
	81	92							

Field	Contents
CID	Fluid coordinate system identification number. (Integer > 0)
G	Value of gravity for fluid elements in the axial direction. (Real)
DRHO	Default mass density for fluid elements. (Real > 0.0 or blank)
DB	Default bulk modulus for fluid elements. (Real)

Field	Contents
NOSYM	Request for nonsymmetric (sine) terms of series. (Character: “YES” or “NO”)
F	Flag specifying harmonics. (Blank if harmonic is specified, or Character: “NONE”)
Ni	Harmonic numbers for the solution, represented by an increasing sequence of integers. On continuation entries, without the “THRU” option, blank fields are ignored. “THRU” implies all numbers including upper and lower harmonics. ($0 \leq$ Integer < 100, or Character: “THRU”, “STEP” or blank)
NS	Every NSth step of the harmonic numbers specified in the “THRU” range is used for solution. If field 5 is “STEP”, Ni = i*NS+N1 where i is the number of harmonics. (Integer)

Remarks:

1. Only one AXIF entry is allowed.
2. CID must reference a cylindrical or spherical coordinate system.
3. Positive gravity (+G) implies that the direction of free fall is in the -Z direction of the fluid coordinate system.
4. The DRHO value replaces blank values of RHO on the FSLIST, BDYLIST and CFLUIDi entries.
5. The DB value replaces blank values of B on the CFLUIDi entries. If the CFLUIDi entry is blank and DB is zero or blank, the fluid is incompressible.
6. If NOSYM = “YES”, both sine and cosine terms are specified. If NOSYM = “NO”, only cosine terms are specified.
7. If F = “NONE”, no harmonics are specified, no fluid elements are necessary, and no continuations may be present. In this case, AXIS = “FLUID” should not be specified in the Case Control Section.
8. Superelements cannot be used.

AXSLOT Axisymmetric Slot Analysis Parameters

Defines the harmonic index and the default values for acoustic analysis entries.

Format:

1	2	3	4	5	6	7	8	9	10
AXSLOT	RHOD	BD	N	WD	MD				

Example:

AXSLOT	0.003	1.5+2	3	0.75	6				

Field	Contents
RHOD	Default density of fluid in units of mass/volume. (Real = 0.0 or blank)
BD	Default bulk modulus of fluid in units of force/volume ratio change. (Real > 0.0 or blank)
N	Harmonic index number. (Integer $\geq$ 0)
WD	Default slot width. (Real $\geq$ 0.0 or blank)
MD	Default number of slots. (Integer $\geq$ 0 or blank)

Remarks:

1. Only one AXSLOT entry is allowed.
2. If any of the RHO, B, and M fields on the GRID, SLBDY, CAXIFi, and CSLOTi entries are blank, then values must be specified for the RHOD, BD and MD fields.
3. If the number of slots (M) is different in different regions of the cavity, this fact may be indicated on the CSLOTi and SLBDY entries. If the number of slots is zero, no matrices for CSLOTi elements are generated.
4. BD=0.0 implies the fluid is incompressible.

BAROR CBAR Entry Default Values

Defines default values for field 3 and fields 6 through 8 of the CBAR entry.

Format:

1	2	3	4	5	6	7	8	9	10
BAROR		PID			X1	X2	X3	OFFT	

Example:

BAROR		39			0.6	2.9	-5.87	OOG	

Alternate Format and Example:

BAROR		PID			G0			OFFT	
BAROR		39			18			OOG	

Field	Contents
PID	Property identification number of the PBAR entry. (Integer > 0 or blank)
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark **5.** (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$, using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB on CBAR entry)
OFFT	Offset vector interpretation flag. See Remark **5.** (Character or blank)

Remarks:

1. The contents of fields on this entry will be assumed for any CBAR entry whose corresponding fields are blank.
2. Only one BAROR entry is allowed.
3. For an explanation of bar element geometry, see the “**Bar Element (CBAR)**” on page 68 of the *MSC.Nastran Reference Guide*.
4. If field 6 is an integer, then G0 is used to define the orientation vector and X2 and X3 must be blank. If field 6 is real or blank, then X1, X2, and X3 are used.

5. OFFT is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset coordinate system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector $\vec{v}$ and the offset system x-axis are then used to define the z and y axes of the offset system.

BCBODY (SOLs 600/700) Flexible Rigid Contact Body in 2D and 3D

Defines a flexible or rigid contact body in 2D or 3D used in SOLs 600 and 700 only.

Use only as many forms (i.e. HEAT, PATCH3D, BEZIER, POLY, CYLIND, SPHERE, NURBS2, or NURBS) as necessary to describe the body (if rigid). Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM (only the first line should be entered for deformable bodies). The "RIGID" header may be used with any of the other rigid entries but only once per body. See Remark **5.** for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

Format:

1	2	3	4	5	6	7	8	9	10
BCBODY	BID	DIM	BEHAV	BSID	ISTYP	FRIC	IDSPL	CONTROL	
	NLOAD	ANGVEL	DCOS1	DCOS2	DCOS3	VELRB1	VELRB2	VELRB3	
	"RIGID"	CGID	NENT	--- Rigid Body Name ---					
	"HEAT"	CFILM	TSINK	CHEAT	TBODY	HCV	HNC	ITYPE	
		BNC	EMISS	HBL					
	"PATCH3D"	NPATCH							
		IDP	G1	G2	G3	G4			
		IDP	G1	G2	G3	G4			
		etc.	(npatch entries)						
	"BEZIER"	NP1	NP2	NSUB1	NSUB2				
		G1	G2	G3	G4	etc		(np1*np2 values)	
	"POLY"	NP1	NP2						
		G1	G2	G3	G4	etc		(np1*np2 values)	
	"CYLIND"	NSUB							
		Gtop	Rtop	Gbottom	Rbottom				
	"SPHERE"	NSUB							
		Gcenter	Radius						
	"LINE"	NPTS						(2D Contact)	

		G1	G2	G3	G4	etc.		(npts values)	
	“ARC”	NPTS	MethArc					(2D Contact)	
		G1	G2	G3	G4	etc.		(npts values)	
	“SPLINE”	NPTS						(2D Contact)	
		G1	G2	G3	G4	etc.		(npts values)	
	“NURBS2D”	NPTU	NORU	NSUB				(2D Contact)	
		G1 or X1	G2 or Y1	G3	G4 or X2	G5 or Y2	G6	[abs(nptu) grids or x, y,z, values]	See Remark 8
		Homo1	Homo2	Homo3	Homo4	etc.		(nptu values)	
		Knot1	Knot2	Knot3	Knot4	Knot5	etc.	(nptu+noru values)	
	“NURBS2”	IDN							
	“NURBS”	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV	NTRIM	
		G1 or X1	G2 or Y1	G3 or Z1	G4 or X2	G5 or Y2	G6 or Z2	G7	See Remark 8
		G8 or X3	G9 or Y3	G10 or Z3	etc.		[abs(nptu)*nptv values]	See Remark 8	
		Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7	
		Homo8	Homo9	etc.			(nptu*nptvvales)		
		Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	`
		Knot8	Knot9	etc.			(nptu+noru+ntpv+norv values)		
		IDtrim	NPTUtrim	NORUtrim	NSUBtrim		(repeat this and all following lines NTRIM times)		
			Xisoparam	Yisoparam			(NPTUtrim entries)		
			Homo1	Homo2	Homo3	etc	(NPTUtrim entries)		

			Knot1	Knot2	Knot3	etc.	(NPTUtrim + NORUtrim entries)		

Examples (of Deformable and Rigid Contact):

Example 1 -- Typical deformable body

BCBODY	1		DEFORM	101	0	.05			

Example 2 -- Simple 4-node rigid patch (see Remark 5 for rigid bodies)

BCBODY	2		RIGID	102	0	.08			
	RIGID				301	302	303	304	

Field	Contents
BID (4,1)	Contact body identification number referenced by BCTABLE, BCHANGE, or BCMOVE. (Integer > 0; required)
DIM	Dimension of body. (Character; Default= 3D) DIM=2D planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves. DIM=3D any 3D body composed of rigid surfaces, shell elements or solid elements. ***Only 3D is supported for MSC.Nastran 2004.***
BEHAV (4,8)	Behavior of curve or surface (Character; Default = DEFORM) DEFORM body is deformable, RIGID body is rigid, SYMM body is a symmetry body, ACOUS indicates an acoustic body, WORK indicates body is a workpiece, HEAT indicates body is a heat-rigid body. ***Only DEFORM and RIGID are supported for MSC.Nastran 2004. DEFORM, RIGID and SYMM and HEAT are also supported in MSC.Nastran 2005 r2 and subsequent versions. See Remark 5. for Rigid Bodies.***
BSID	Identification number of a BSURF, BCBOX, BCPROP or BCMATL entry if BEHAV=DEFORM. (Integer > 0)

Field	Contents
ISTYP (4,3)	Check of contact conditions. (Integer ≥ 0; Default = 2 for deformable and 0 for rigid) For a deformable body: = 0 symmetric penetration, double sided contact. =1 unsymmetric penetration, single sided contact. (Integer > 0) =2 double-sided contact with automatic optimization of contact constraint equations (this option is known as “optimized contact”) For a rigid body: =0 no symmetry condition on rigid body. =1 rigid body is a symmetry plane.
FRIC (6,7)	Friction coefficient. (Real ≥ 0; Default = 0)
IDSPL (4,5)	Activates the SPLINE (analytic) option for a deformable body. (Integer; Default = 0) =0 or blank, SPLINE option is turned off. = 1, The body is smoothed out with splines (2D) or Coons surfaces (3D). >1 Identification number of a BLSEG entry that lists nodes on edges of the body which are excluded from the SPLINE option. These nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. for example, a CQUAD4 with a grid numbering 1,2,4,3 would need pairs of nodes 1,2 2,3 4,3 3,1. The nodal pairs may be entered in any order. See MSC.Marc Volume C SPLINE option documentation for more details.
CONTROL (4,6)	Indicates the type of control for the body. Integer: =-1 for position control, =0 for velocity control. = positive number for load control (the positive number is the grid number which has displacement degrees-of-freedom controlling the body. The position of this grid is at the center of rotation given in the CGID field. The velocity / displacement of the body must be specified by the VELRBi fields rather than SPCD.
NLOAD (4,7)	Enter a positive number if load controlled and rotations are allowed (Integer). The positive number is the grid number which as the rotation(s) of the body as degrees-of-freedom. The position of this grid is at the center of rotation given in the CGID field.

Field	Contents
ANGVEL (6,1)	Angular velocity or angular position about local axis through center of rotation. (Real; Default = 0.0)
DCOS1 (6,4)	3D - First component direction cosine of local axis if ANGVEL is nonzero. (Real)
	2D - First coordinate of initial position of rotation of rigid body.
DCOS2 (6,5)	3D - Second component direction cosine of local axis if ANGVEL is nonzero. (Real)
	2D - Second coordinate of intial position of rotation of rigid body
DCOS3 (6,6)	3D - Third component direction cosine of local axis if ANGVEL is nonzero. (Real)
	2D - Not used.
VELRB1 (5,4)	2D & 3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 1st direction. (Real)
VELRB2 (5,5)	2D & 3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 2nd direction. (Real)
VELRB3 (5,6)	3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 3rd direction. (Real)
	2D - Not used.
"RIGID"	The entries of this continuation line are for the rigid body description. See Remark 5.
CGID (5,i) i=1,2,3 (4,6)	Grid point identification number defining the initial position of the center of rotation for the rigid body or the point where a concentrated force or moment is applied.
NENT (4,2)	Number of geometric entities to describe this rigid surface. A rigid surface can be described by multiple sets of patches, nurbs, etc. For example, if it takes 3 sets of PATCH3D entries to describe a rigid surface, then set NENT=3. (Integer > 0; Default = 1)
Rigid Body Name (4,9)	Name associated with the rigid body. (Default is blank, 24-characters maximum)
“HEAT”	**The entries of this continuation line(s) are for contact in heat transfer. Do not enter these line(s) for structural analyses.**

Field	Contents
CFILM (9,1) / (10,1)	Heat transfer coefficient (film) to environment. (Real or Integer, Default = 0.0 for a heat transfer problem, omit for a structural problem) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the heat transfer coefficient vs temperature. This is usually callled HCVE in the MSC.Marc documentation.
TSINK (9,2) / (10,2)	Environment sink temperature. (Real or Integer, Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value netered is the sink temperature. If Integer, the value entered is the ID of a TABLEM1 entry specifying temperature vs time. At present, this variable should not be a function of temperature.
CHEAT (9,3) / (10,3)	Contact heat transfer coefficient. (Real or Integer, Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the contact heat transfer coefficient vs temperature.
TBODY (9,4) / (10,4)	Body temperature. (Real or Integer, Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLEM1 entry specifying the body temperature vs time. At present, this variable should not be a function of temperature.
HCV (9,5) / (10,5)	Convection coefficient for near field behavior (Real or Integer, Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the near field convection coefficient vs temperature.
HNC (9,6) / (10,6)	Natural convection coefficient for near field behavior (Real or Integer, Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the near field natural convection coefficient vs temperature.
ITYPE [4,8]	An option entry for heat transfer only (Integer, no Default) 1 - Heat sink 4 - Heat conduction body

Field	Contents
BNC (9,7) / (10,7)	Exponent associated with the natural convection coefficient for near field behavior (Real or Integer, Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the exponent associated with the near field natural convection coefficient vs temperature. At present, this variable should not be a function of temperature.
EMISS (9,8) / (10,8)	Emissivity for radiation to the environment or near thermal radiation (Real or Integer, Default = 1.0 for a heat transfer problem, omit for a structural problem). If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 entry specifying the emissivity vs temperature.
HBL (7,6) / (8,6)	Separation distance dependent thermal convection coefficient (Real or Integer, Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is a convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the convection coefficient vs temperature.
PATCH3D	**Entries for this continuation line describe a rigid body made up of as many 4-node patches as desired.**
IDP	ID of the patch (Integer number 1 through highest value).
G1, G2, G3, G4	Grid numbers for each of the 4 nodes of the patch (see Note 5).
BEZIER	**Entries for this continuation line describe a rigid body made up of Bezier Surfaces.**
NP1	Number of points in 1st direction. (Integer > 0)
NP2	Number of points in 2nd direction. (Integer > 0)
NSUB1	Number of subdivisions in 1st direction. (Integer > 0)
NSUB2	Number of subdivisions in 2nd direction. (Integer > 0)
G1, G2, G3, etc	Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)

Field	Contents
"POLY"	**Entries for this continuation line describe a rigid body made up of Poly Surfaces.**
NP1	Number of points in the 1st direction. (Integer > 0)
NP2	Number of points in the 2nd direction. (Integer > 0)
G1, G2, G3, etc	Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)
"CYLIND"	**Entries for this continuation line describe a cylindrical rigid body.**
NSUB	Number of subdivisions. (Integer > 0)
Gtop	Grid point ID of a grid in the center of the top of the cylinder. (Integer > 0)
Rtop	Radius of the top of the cylinder. (Real > 0.0)
Gbottom	Grid point ID of a grid in the center of the bottom of the cylinder. (Integer > 0)
Rbottom	Radius of the bottom of the cylinder. (Real > 0.0)
"SPHERE"	**Entries for this continuation line describe a spherical rigid body.**
NSUB	Number of subdivisions. (Integer > 0)
Gcenter	Grid point ID of a grid in the center of the sphere. (Integer > 0)
Radius	Radius of the sphere. (Real > 0.0)
“LINE”	**Entries for this continuation line describe a 2D rigid body made up of as many line segments as desired.**
NPTS	Number of points in the line segment.
G1, G2, G3, G4	Grid numbers for each of the NPTS points on the line segment.
“ARC”	**Entries for this continuation line describe a 2D rigid body made up of as many segments as desired describing an arc.**
NPTS	Number of points in the arc. NPTS must be 4 for an ARC.
MethArc	Method to generate arc (Integer 0 to 4) (see MSC.Marc volume C Contact description Figures 3-3 and 3-4).

Field	Contents
G1, G2, G3, G4	Grid numbers for each of the 4 points as described by method.
“SPLINE”	**Entries for this continuation line describe a 2D rigid body made up of as many spline segments as desired.**
NPTS	Number of points for the spline.
G1, G2, G3, G4	Grid numbers for each of the NPTS points on the spline.
“NURBS2D”	**Entries for this continuation line describe a 2D rigid body made up of nurmbs.**
NPTU	Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer, no Default)
NORU	Order
NSUB	Number of subdivisions
G1, G2, G3, G4	Grid numbers for each of the NPTU control points
X1, Y1, X2, Y2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.
Homo1, Homo2, Homo3, etc.	Homogeneous coordinates (0.0 to 1.0) (Real). There must be NPTU entries.
Knot1, Knot2, Knot3, etc.	Knot vectors (0.0 to 1.0) (Real). There must be (NPTU+NORU) entries.
"NURBS2"	**Entries for this continuation line describe a rigid body made up of nurbs.**
IDN	ID of a matching GMNURB entry. The GMNURB is an entry that contains the same information as at shown for the NURBS option. (Integer > 0)
"NURBS"	**Entries for this continuation line describe a rigid body made up of nurbs.**

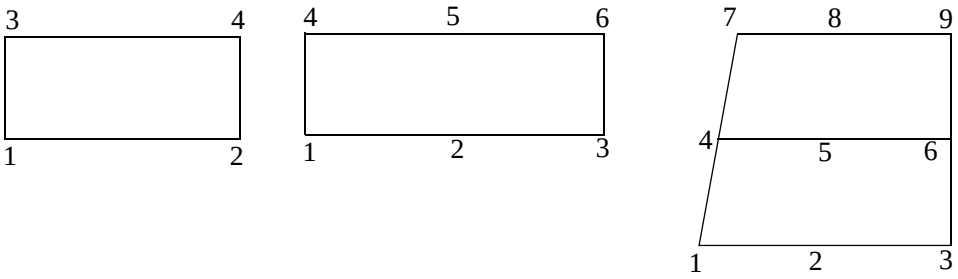
Field	Contents
NPTU	Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer, No Default)
NPTV	Number of control points in V direction. (Integer > 0; required)
NORU	Order along U direction. (Integer > 0; required)
NORV	Order along V direction (Integer > 0; required)
NSUBU	Number of subdivisions in U direction (Integer > 0; required)
NSUBV	Number of subdivisions in V direction (Integer > 0; required)
NTRIM	Number of trimming curves (Integer, $\geq$ 0 or blank)
G1, G2, G3, etc.	Grid point IDs defining control points (Integer > 0). There must be NPTU*NPTV entries.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0). There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)
IDtrim	ID of trimming vector. There must NTRIM of these entries and those entries that follow. (Integer > 0)
NPUTtrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
Xisoparam	First coordinate of point in isoparametric space. (Real)
Ysoparam	Second coordinate of point in isoparametric space. (Real)

Field	Contents
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)

Remarks:

1. Named continuation entries are ignored for a deformable curve or surface (BEHAV=DEFO), except for “HEAT”.
2. The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID.
3. All continuation lines may be omitted if not required.
4. BCBODY is recognized only in SOLs 600 and 700.
5. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
6. (i,j) refers to data block i and field j of MSC.Marc’s CONTACT model definition entry. IDSPL covers the SPLINE history definition in MSC.Marc. For structural analysis (i,j) refers to contact without tables. For heat transfer (i,j) refers to contact with tables.
7. For BEZIER surfaces, enter np1*np2 points in the order shown below:

Mesh	Normal Order	Reversed Order
2x2	1,2,3,4	2,1,4,3
3x2	1,2,3,4,5,6	3,2,1,6,5,4
3x3	1,2,3,4,5,6,7,8,9	3,2,1,6,5,4,9,8,7



8. For NURBS, enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for abs(NPTU) points and set NPTU to a negative value.
9. The heat transfer options are available starting with MSC.Nastran 2005 r2 and must use MSC.Marc 2005 or later.
10. For heat transfer items described using a TABLEM1 ID, the smallest value in the table will be entered into MSC.Marc's 9th contact (with tables) datablock. The table ID will be translated directly to MSC.Marc's 10th contact (with tables) datablock.

BCBOX (SOLs 600/700) 3D Contact Region

Defines a 3D contact region -- all elements within the region define a contact body used in SOLs 600 and 700 only.

Format (Form 1):

1	2	3	4	5	6	7	8	9	10
BCBOX	ID		HOW						
	N1	N2	N3	N4	N5	N6	N7	N8	

Form 2:

BCBOX	ID	COORD	HOW						
			X1	Y1	Z1	X2	Y2	Z2	
			X3	Y3	Z3	X4	Y4	Z4	
			X5	Y5	Z5	X6	Y6	Z6	
			X7	Y7	Z7	X8	Y8	Z8	

Example (for Form 1):

BCBOX	101		0						
	1001	1002	1003	1004	1005	1006	1007	1008	

Field	Contents
ID	Identification of a deformable surface corresponding to a BSID value on the BCBODY entry if the Case Control BCONTACT=BCBOX is specified. All elements corresponding designated box may potentially come into contact. See Remark 2. (Integer > 0)
COORD	Enter COORD in field 3 if x,y,z coordinates of the box are to be specified rather than grid IDs. (Character)
HOW	A flag indicating whether an element is in the defined box or not. (Integer; Default = 0) 0=If only one grid point of an element is in the box, the entire element is considered to be in the box. 1=All grid points comprising the element must be within the box, otherwise the element is considered outside of the box.

Field	Contents
N1-N8	Enter 8 Grid IDs defining a box (hexa-like region) if the third field is blank. (Integer; required if COORD is blank)
Xi, Yi, Zi	Enter eight x,y,z values in the basic coordinate system if the third field is COORD. (Real; required if “COORD” is entered in field 3 of line 1)

Remarks:

1. BCBOX is only recognized in SOLs 600 and 700.
2. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
3. The deformable surface may alternately be defined using BSURF, BCPROP, or BCMATL entries.
4. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
5. All elements corresponding to the IDs entered will be used to define the deformable surface.
6. The model is searched to determine whether each element lies within the specified box region as specified by the HOW criteria option.

BCHANGE (SOL 600) Changes Definitions of Contact Bodies

Changes definitions of contact bodies used in MD Nastran Implicit Nonlinear (SOL 600 only).

Format:

1	2	3	4	5	6	7	8	9	10
BCHANGE	ID	TYPE	NBOD		IDBOD1	N1	N2	INC	
	IDBOD2	N1	N2	INC	IDBOD3	etc.			

Example:

BCHANGE	201	NODE	2		1	1001	1010	1	
	2	2001	2021	2					

Field	Contents
ID	Identification number referenced by a SUBCASE Case Control command. To place an entry in MSC.Marc's phase 0 (main body of the data, set ID = 0). To activate the entry for the 1st subcase, set ID=1, for the 2nd, set ID=2. (Integer $\geq$ 0; required)
Type	Type of modification (Character; required) = NODE defines nodes of a contact body which may come into contact = EXCLUDE excludes node segments in 2D or 4 node patches in 3D
NBOD	Number of bodies to be modified -- must match number of bodies actually entered. More than one N1-N2-INC range may be entered for each body. See N1 below. (Integer > 0; Default = 1)
IDBODi	Identification number of a contact body, BCBODY entry. (Integer > 0)
N1	Starting grid ID.

Note: If more than one N1-N2-INC range is required for a body, enter N1 as a negative value for all ranges except for the last range for which N1 is entered as a positive value.

N2	Ending grid ID.
INC	Grid ID increment.

Remarks:

1. The BCHANGE entry does not apply to rigid bodies. Multiple BCHANGE entries are allowed. A body may be entered more than once with different grid IDs.
2. The BCHANGE entry covers MSC.Marc's history definitions CONTACT NODE and EXCLUDE.
3. BCHANGE is recognized only in MD Nastran Implicit Nonlinear (SOL 600).
4. BCHANGE is useful only for saving computer time and is not recommended for general usage.
5. Warning -- For the NODE option, if some nodes in a body are inadvertently omitted, they may penetrate other bodies.

BCGRID (SOL 700) Contact Node Region

Grids to be included in SOL 700 contact analyses.

Format:

1	2	3	4	5	6	7	8	9	10
BCGRID	CID	GID1	GID2	GID3	GID4	GID5	GID6	GID7	
	GID8	GID9	GID10	GID11	GID12	GID13	-etc.-		

Example:

BCGRID	100	12	14	17	121	234	235	270	
	309	1001	THRU	2000	BY	2			

Field	Contents
ID	Unique identification number of a “cloud” of gridpoints which can be used for SOL 700 contact (Integer > 0, Required)
GID1, GID2,…	Gridpoint ID. THRU indicates a range to be used. BY is the increment to be used within this range (Integer > 0, Required)

Remarks:

1. Corresponds to Ls-Dyna entry *CONTACT_NODES_TO_SURFACE
2. ID must be unique with respect to all other BSURF, BCBOX, BCPROP or BCMATL entries.
3. BCGRID may only be used for SLAVE body definitions on the BCTABLE entry.
4. As many continuation lines as necessary may be used to define all GRID points used in the contact definition.

BCMATL (SOLs 600/700) 3D Contact Region by Element Materials

Defines a 3D contact region by element material. All elements with the specified materials define a contact body used in SOLs 600 and 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCMATL	ID	IM1	IM2	IM3	IM4	IM5	IM6	IM7	
	IM8	IM9	etc.						

Example:

BCMATL	1001	101	201	301					

Field	Contents
ID	Identification of a deformable surface corresponding to a BSID value on the BCBODY entry if the Case Control command, BCONTACT=BCMATL is specified. All elements corresponding to the material IDs specified may potentially come into contact. See Remark 2. (Integer > 0)
IMi	Material ID. A minimum of one entry is required. (Integer)

Remarks:

1. BCMATL is only recognized in SOLs 600 and 700.
2. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
3. The deformable surface may alternately be defined using BSURF, BCBOX, or BCPROP entries.
4. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
5. All elements corresponding to the IDs entered will be used to define the deformable surface.
6. As many continuation lines as necessary may be used to define all material IDs associated with a particular deformable body.
7. BCMATL may not be used to define contact regions made up of composite elements.

BCMOVE (SOL 600) Movement of Bodies in Contact

Defines movement of bodies in contact used in MD Nastran Implicit Nonlinear (SOL 600 only).

Format:

1	2	3	4	5	6	7	8	9	10
BCMOVE	ID	MTYPE	IREL						
	IDRBOD1	IDRBOD2	IDRBOD3	etc.					

Examples:

BCMOVE	33	RELEASE	20						
	1	3	5	7					
BCMOVE	1	approach							

Field	Contents
ID	Identification number referenced by a SUBCASE Case Control command. To place an entry in MSC.Marc's phase 0 (main body of the data, set ID=0). To activate the entry for the 1st subcase, set ID=1, for the 2nd, set ID=2. (Integer $\geq$ 0; required)
MTYPE	Movement type. (Character; Default = APPROACH) = APPROACH all rigid bodies are moved so that they all make contact with deformable bodies. = RELEASE the contact condition is released for selected bodies. = SYNCHRON all rigid bodies are moved until the first rigid body makes contact with a deformable body.
IREL	Flag to indicate how contact forces are removed, for option RELEASE only. (Integer) = 0 contact forces are immediately removed. (Default) > 0 contact forces are reduced to zero over the number of increments specified in this load period. See NLPARM and TSTEP1 for the number of increments.
IDRBODi	Identification numbers of rigid bodies to be released, for option RELEASE only. Points to BCBODY Bulk Data entries.

Remarks:

1. This entry matches MSC.Marc's history definitions RELEASE, APPROACH, and SYNCHRONIZED. Note that MSC.Marc's history definition MOTION CHANGE is done in MD Nastran by describing the enforced motion for the grid which is defined to be the center of rotation of the rigid body, see CGID of the BCBODY entry.
2. For MTYPE=APPROACH and MTYPE=SYNCHRON leave all following fields blank.
3. BCMOVE is recognized only in MD Nastran Implicit Nonlinear (SOL 600).
4. The APPROACH and SYNCHRON options apply to rigid contact surfaces only.
5. You may release a deformable body from contact with either a deformable or rigid body.

BCONP Contact Parameters

Defines the parameters for a contact region and its properties.

Format:

1	2	3	4	5	6	7	8	9	10
BCONP	ID	SLAVE	MASTER		SFAC	FRICID	PTYPE	CID	

Example:

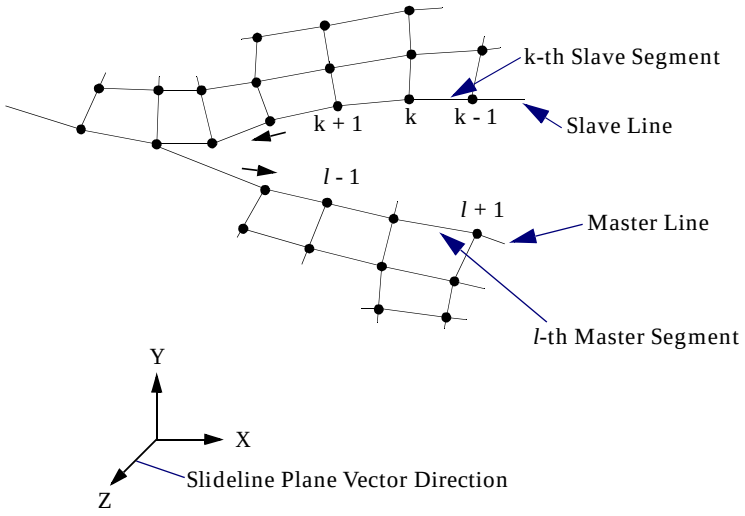
BCONP	95	10	15		1.0	33	1		

Field	Contents
ID	Contact region identification number. See Remark **1.** (Integer > 0)
SLAVE	Slave region identification number. See Remark **2.** (Integer > 0)
MASTER	Master region identification number. See Remark **3.** (Integer > 0)
SFAC	Stiffness scaling factor. SFAC is used to scale the penalty values automatically calculated by the program. See Remark **4.** (Real > 0.0; Default = 1.0)
FRICID	Contact friction identification number. See Remark **5.** (Integer > 0 or blank)
PTYPE	Penetration type. See Remark **6.** (Integer 1 or 2; Default = 1) 1: unsymmetrical (slave penetration only--Default) 2: symmetrical
CID	Coordinate system identification number to define the slideline plane vector and the slideline plane of contact. See Remark **7.** (Integer > 0; Default = 0, which means the basic coordinate system)

Remarks:

1. ID field must be unique with respect to all other BCONP identification numbers.
2. The referenced SLAVE is the identification number in the BLSEG Bulk Data entry. This is the slave line. The width of each slave segment must also be defined to get proper contact stresses. See the Bulk Data entry, “**BWIDTH**” on page 1132 for the details of specifying widths.

3. The referenced MASTER is the identification number in the BLSEG Bulk Data entry. This is the master line. For symmetrical penetration, the width of each master segment must also be defined. See the Bulk Data entry, "**BWIDTH**" on page 1132 for the details of specifying widths.
4. SFAC may be used to scale the penalty values automatically calculated by the program. The program calculates the penalty value as a function of the diagonal stiffness matrix coefficients that are in the contact region. In addition to SFAC, penalty values calculated by the program may be further scaled by the ADPCON parameter (see description of ADPCON parameter for more details). The penalty value is then equal to k*SFAC*|ADPCON|, where k is a function of the local stiffness. It should be noted that the value in SFAC applies to only one contact region, whereas the ADPCON parameter applies to all the contact regions in the model.
5. The referenced FRICID is the identification number of the BFRIC Bulk Data entry. The BFRLC defines the frictional properties for the contact region.
6. In an unsymmetrical contact algorithm only slave nodes are checked for penetration into master segments. This may result in master nodes penetrating the slave line. However, the error depends only on the mesh discretization. In symmetric penetration both slave and master nodes are checked for penetration. Thus, no distinction is made between slave and master. Symmetric penetration may be up to thirty percent more expensive than the unsymmetric penetration.
7. In **Figure 8-1**, the unit vector in the Z-axis of the coordinate system defines the slideline plane vector. The slideline plane vector is normal to the slideline plane. Relative motions outside the slideline plane are ignored, and therefore must be small compared to a typical master segment. For a master segment the direction from master node 1 to master node 2 gives the tangential direction (t). The normal direction for a master segment is obtained by the cross product of the slideline plane vector with the unit tangent vector (i.e., n=z x t). The definition of the coordinate system should be such that the normal direction must point toward the slave region. For symmetric, penetration, the normals of master segments and slave segments must face each other. This is generally accomplished by traversing from master line to slave line in a counterclockwise or clockwise fashion depending on whether the slideline plane vector forms a right-hand or left-hand coordinate system with the slideline plane.



- X-Y plane is the slideline plane. Unit normal in the Z-direction is the slideline plane vector.
- Arrows show positive direction for ordering nodes. Counterclockwise from master line to slave line.
- Slave and master segment normals must face each other.

Figure 8-1 A Typical Finite Element Slideline Contact Region

BCPARA (SOLs 600/700) Contact Parameters

Defines contact parameters used in SOLs 600 and 700.

Format:

1	2	3	4	5	6	7	8	9	10
BCPARA	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	etc				

Example:

BCPARA		NBODIES	4	BIAS	0.5				

Field	Contents
ID	Subcase to which the defined parameters belong. If ID is zero or blank, the parameters belong to all subcases. (Integer)
Param(i)	Name of a parameter. Allowable names are given in **Table 8-2**. (Character)
Value(i)	Value of the parameter. See **Table 8-2**. (Real or Integer)

Table 8-2 Contact Parameters

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
NBODIES (2,1)	Number of contact bodies defined in the analysis. (Integer $\geq$ 0 or blank)
MAXENT* (2,2)	Maximum number of entities created for any contact body. (Integer > 0 or blank; default is max element number or 1.5 times the number of nodes whichever is smaller)
MAXNOD* (2,3)	Maximum number of nodes that lie on the periphery of any deformable contact body. (Integer > 0 or blank; default is the number of nodes)

*No longer required for MSC.Nastran 2005 r2 or any subsequent versions.

Table 8-2 Contact Parameters (continued)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
ERROR (3,2)	Distance below which a node is considered touching a body. Automatically calculated if left blank. (Real; default = blank)
BIAS (3,6)	Contact tolerance bias factor. (Real, $0 \leq$ BIAS ≤ 1, default = 0.9 if field left blank or not entered in the file. To obtain a near zero value, enter 1.0E-16)
ISPLIT (2,7)	Flag for increment splitting procedure. (Integer ≥ 0; Default = 3 for statics and 0 for dynamics) = 0 Uses increment splitting procedures for the fixed time step procedures. = 1 Suppresses splitting for the fixed time step procedures. = 2 Suppresses splitting for adaptive time step procedures. = 3 Uses contact procedure which does not require increment splitting (3 is not available for dynamics).
FNTOL (3,5)	Separation force above which a node separates from a body. Automatically calculated if left blank. (Real; Default = blank)
MAXSEP (2,6)	Maximum number of separations allowed in each increment. **After MAXSEP separations have occurred, if the standard convergence tolerance conditions are achieved, the step will converge.** (Integer > 0; Default = 9999)
ICHECK (2,8)	Flag for interference kinematic check and bounding box check. (Integer > 0) = 1 activates interference kinematic check. = 2 Suppress bounding box checking. = 3 no reset of NCYCLE to zero. = 4 check for separation only when solution has converged, for analytical surfaces only.

Table 8-2 Contact Parameters (continued)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
ICSEP (2,9)	Flag to control separation. (Integer $\geq$ 0) = 0 The node separates and an iteration occurs if the force on the node is greater than the separation force. = 1 If a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does NOT separate in this increment, but separates at the beginning of the next increment. = 2 If a new node comes into contact during this increment, it is not allowed to separate during this increment, prevents chattering. = 3 both 1 and 2 are in effect.
IBSEP (2,12)	Flag for separation based on stresses or forces. (Integer $\geq$ 0; Default = 0) = 0 separation based on forces. = 1 separation based on absolute stresses (force / area) = 2 separation based on absolute stress (extrapolating integration point stresses) = 3 relative nodal stress (force / area) = 4 separation based on relative stress (extrapolating integration point stresses) Only option 2 and 4 can be used with mid-side node elements where the mid-side nodes contact (LINQUAD=-1). Options 2 - 4 are only available with MSC.Marc 2003 and subsequent releases.
ISHELL (2,10)	Parameter governing normal direction and thickness contribution of shells. (Integer $\geq$ 0; Default = 0) = 0 check node contact with top and bottom surface. = 1 nodes only come into contact with bottom layer. = 2 nodes only come into contact with bottom layer and shell thickness is ignored. = -1 nodes only come into contact with top layer. = -2 nodes only come into contact with top layer and shell thickness is ignored.
IPRINT (2,11)	Flag to reduce print out of surface definition. (Integer $\geq$ 0; Default = 0) = 0 full print out. = 1 reduced print out.

Table 8-2 Contact Parameters (continued)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
RVCNST (3,1)	Relative sliding velocity between bodies below which sticking is simulated. If FTYPE=5, then the value of RVCNST is the stick-slip transition region.
FTYPE (2,4)	Friction type. (Integer 0 to 5) = 0 No friction. = 1 Shear friction. = 2 Coulomb Friction. (Default) = 3 Shear friction for rolling. = 4 Coulomb friction for rolling. = 5 Stick-slip Coulomb friction. = 6 Bilinear Coulomb friction. =7 Bilinear Shear friction. (Types 6 and 7 are available only in MSC.Marc 2005 and subsequent releases.)
FKIND (2,5)	Friction kind. (Integer 0 or 1) = 0 friction based on nodal stress. = 1 (default if friction is present and EKIND is not entered) friction based on nodal force.
BEAMB (2,13)	Beam-Beam contact flag. (Integer 0 or 1) = 0 (Default) no beam-beam contact. = 1 activate beam-beam contact options.
FSSMULT (3,7)	Stick-slip friction coefficient multiplier. Applicable only to stick-slip friction. (The friction coefficient is multiplied by this value for the sticking condition.) (Real $\geq$ 0; required, Default = 1.05)
FSSTOL (3,8)	Stick-slip friction force tolerance. Applicable only to stick-slip friction. (Real; Default = 0.05)

Table 8-2 Contact Parameters (continued)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
LINQUAD (2,14)	Higher order element contact flag (Integer, Default=1). =1 the outer boundary of a contact body is described by the corner nodes only and mid-side nodes can't come into contact. =-1 the other boundary is described by a quadratic field and both corner and mid-side nodes are considered in contact. If this flag is set to -1 and IBSEP is blank, IBSEP will be re-set to 2. This option is only available with MSC.Marc 2003 and subsequent releases.
INITCON (2,16)	If INITCON is set to 1, tying relations (MPC's) for surfaces initially in contact will be saved on a file named jid.marc.t01. This option maybe used to model dissimilar meshes. See CONTINUE=101+ on the SOL 600 entry to use these items in the same MD Nastran execution. If CONTINUE is between 101 and 400 on the SOL 600 entry, INITCON=1 will be set automatically. This option is available in the MSC.Nastran 2005 r2 and subsequent releases only. This option is also controlled by PARAM,MINITCON,1.

Remarks:

1. (i,j) refers to data block i and field j of the CONTACT model definition option in MSC.Marc.
2. BCPARA is recognized only in MD Nastran Implicit Nonlinear (SOLs 600 and 700).

BCPROP (SOLs 600/700) 3D Contact Region by Element Properties

Defines a 3D contact region by element properties. All elements with the specified properties define a contact body used in SOLs 600 and 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCPROP	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	etc.						

Example:

BCPROP	1	101	201	301					

Alternate Format:

BCPROP	ID	IP1	THRU	IP2	FS	FK	EXP	VC	
	OPTT	SFT	SSF						

Example for Alternate Format:

BCPROP	25	101	THRU	102	.03	.01	1.0	.033	
	.125	1.0	1.0						

Field	Contents
ID	Identification of a deformable surface corresponding to a BSID value on the BCBODY entry or if the Case Control BCONTACT=BCPROP is specified. All elements corresponding to the property IDs specified that may potentially come into contact. Do not specify mixed property types (use all shell, all solid or all beam properties only). See Remark 2. (Integer > 0)
IPi	Property ID. A minimum of one entry is required. (Integer, no Default)
FS	Static coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot \left\|v_{rel}\right\|}$. SOL 700 only. (Real, Default = 0.0)

FK Dynamic coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $\mu_c = FD + (FS - FD)e^{-DC \cdot |v_{rel}|}$. SOL 700 only. (Real, Default = 0.0)

EXP Exponential decay coefficient. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces on contact $\mu_c = FD + (FS - FD)e^{-DC \cdot |v_{rel}|}$. SOL 700 only. (Real, Default = 1.0)

VC Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{lim} = VC \cdot A_{cont} \cdot A_{cont}$ being the area of the segment contacted by the node in contact. The suggested value for VC is to use the yield stress in shear $VC = \sigma_o / \sqrt{3}$ where σ_o is the yield stress of the contacted material. SOL 700 only. (Real, no Default)

OPTT Optional contact thickness (applies to shells only). SOLs 600 or 700. (Real, no Default)

SFT Optional thickness scale factor. This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements. SOL 700 only. (Real, Default = 1.0)

SSF Scale factor on default slave penalty stiffness. If zero, SSF is taken as unity. SOL 700 only. (Real, Default = 1.0)

Remarks:

1. BCPROP is only recognized in SOLs 600 and 700.
2. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
3. The deformable surface may alternately be defined using BSURF, BCBOX, or BCMATL entries.
4. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
5. All elements corresponding to the IDs entered will be used to define the deformable surface.
6. As many continuation lines as necessary may be used to define all property IDs associated with a particular deformable body.

7. The alternate format is triggered if field 4 contains THRU or field 6 contains a value with a decimal point. THRU and an entry in field 5 are not necessary if the values starting in field 6 apply only to one property.
8. FK, EXP, VC, SSF apply only to SOL 700 and should be left blank for SOL 600. If entered for SOL 600 these values will be ignored.

BCSEG (SOL 700) Contact Segment Defined Using Grids

Grids which are part of an element to be used in SOL 700 contact analyses. (Specified in the Ls-Dyna style)

Format:

1	2	3	4	5	6	7	8	9	10
BCSEG	ID	IBODY	G1	G2	G3	G4			

Example:

1	2	3	4	5	6	7	8	9	10
BCSEG	100	1005	11	12	13	14			

Field	Contents
ID	Unique identification number for this BCSEG entry. (Integer > 0, Required)
IBODY	Identification number of a surface that is called out on the 5th field of a BCBODY entry. (Integer > 0, Required)
G1,G2,G3,G4	GRID point identification numbers of an element on this surface. For quad plates and quad surfaces of solids, enter four grid id's. For triangular plates or triangular surfaces of solids, set G4 = G3. (Integer > 0, Required)

Remarks:

1. This entry is used as shown in the example below:

   BCBODY, 201,,,1005

   BCSEG,1,1005,11,12,13,14

   BCSEG,2,1005,21,22,23,24

   BCSEG,3,1005,31,32,33,34

   (In the above 11-14, 21-24 and 31-34 are GRID ID's)

2. This entry is used only in SOL 700 and is not available in SOL 600.

BCTABLE (SOLs 600/700) Defines a Contact Table

Defines a contact table used in SOL 600 and 700 only.

For SOL 600 or SOL 700 structural analysis, use the Primary Format. For SOL 600 heat transfer or thermal contact, use the Alternate Format and enter parameter PARAM,MRCONVER,11 in the bulk data.

Primary Format:

SOL 600 & SOL 700 Structural Analysis -- not to be used with heat transfer or thermal contact; do not enter PARAM,MRCONVER.

1	2	3	4	5	6	7	8	9	10
BCTABLE	ID	IDSLAVE	IDMAST	NGROUP					
	“SLAVE”	IDSLA1	ERROR	FNTOL	FRIC	CINTERF	IGLUE		
		ISEARCH	ICOORD	JGLUE	TOLID	DQNEAR	DISTID		
		FK	EXP	METHOD	ADAPT	THICK	THICKOF	PENV	
		FACT	TSTART	TEND	MAXPAR	PENCHK	FSB	VSF	
		EROSOP	IADJ	SOFT	DEPTH	BSORT	FRCFRQ	SNLOG	
		ISYM	I2D3D	IGNORE	SPR	MRP	VDC	SSBOPT	
		SFS	SFM	SST	MST	SFST	SFMT	AUTO	
		LCID	FCM	US					
	“MASTERS”	IDMA1	IDMA2	IDMA3	IDMA4	IDMA5	IDMA6	IDMA7	
		IDMA8	IDMA9	...					

Alternate Format:

For SOL 600 heat transfer / SOL 600 thermal contact and certain contact problems only; enter PARAM,MRCONVER,11).

1	2	3	4	5	6	7	8	9	10
BCTABLE	ID	IDSLAVE	IDMAST	NGROUP					
	“SLAVE”	IDSLA1	ERROR	FNTOL	FRIC	CINTERF	IGLUE		
		ISEARCH	ICOORD	JGLUE	TOLID	DQNEAR	DISTID		
		FRLIM	BIAS	SLIDE	HARDS				
		HTC	HCV	HNC	BNC	EMISS	HBL		
		FK	EXP	METHOD	ADAPT	THICK	THICKOF	PENV	

1	2	3	4	5	6	7	8	9	10
		FACT	TSTART	TEND	MAXPAR	PENCHK	FSF	VSP	
		EROSOP	IADJ	SOFT	DEPTH	BSORT	FRCFRQ	SNLOG	
		ISYM	I2D3D	IGNORE	SPR	MPR	VDC	SBOPT	
		SFS	SSFM	SST	MST	SFST	SFMT	AUTO	
		LCID	FCM	US					
	"MASTERS"	IDMA1	IDMA2	IDMA3	IDMA4	IDMA5	IDMA6	IDMA7	
		IDMA8	IDMA9	...					

Examples:

BCTABLE	2			3					
	SLAVE	10			0.2				
	MASTERS	20	30						
	SLAVE	20			0.3				
	MASTERS	10							
	SLAVE	30			0.2				
	MASTERS	10							

BCTABLE	0	1	2	0					

Field	Contents
ID	Identification number of a BCONTACT Case Control command. See Remark **6.** (Integer; required)
IDSLAVE (3,1) [3,1]	Identification number of a BCBODY entry defining the touching body. (Integer > 0 or blank)
IDMAST (5,1) [15,1]	Identification number of a BCBODY Bulk Data entry defining the touched body. (Integer > 0 or blank)
NGROUP (2,1) [2,1]	Number of touching bodies. The continuation entries "SLAVE" and "MASTERS" must be repeated NGROUP times. (Integer > 0 or blank).
"SLAVE"	Indicates that this line defines the touching body and its parameters.

Field	Contents
IDSLA1 (4,1) [3,1]	Identification number of a BCBODY Bulk Data entry defining the touching body. (Integer > 0) For SOL 700, leaving IDSLA1 blank will result in contact for all elements in the model. In this case, you are allowed to use ADAPT=YES.
ERROR (3,2) [3,2]	Distance below which a node is considered touching a body. Default= blank automatic calculation. (Real)
FNTOL (3,5) [5,1]	Separation force above which a node separates from a body. Default is maximum residual force. (Real)
FRIC (3,4) [5,2]	Friction coefficient. (Real > 0; Default = 0)
CINTERF (3,6) [5,3]	Interference closure amount, normal to the contact surface. Default = 0. For CINTERF > 0, overlap between bodies. For CINTERF < 0., gap between bodies. (Real)
IGLUE (3,7) [3,7]	Flag to activate glue option (Integer $\geq$ 0). Default is 0, no glue option, 1. Activates the glue option. In the glue option, all degrees-of- freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. 2. Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.
ISEARCH (3,8) [3,8]	Enter a value of 1 to indicate that the searching order for deformable contact bodies is from the touching body to the touched bodies. Enter 2 to Let the program decide which searching order is optimal for deformable bodies. (Integer; Default = 2)
ICOORD (3,9) [3,9]	Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. (Integer; Default = 0)

Field	Contents
JGLUE (3,10) [3,10]	This option is only relevant if the glue option is invoked (IGLUE > 0). Enter 0 if a node should not separate (default). Enter 1 to invoke the standard separation behavior based on the maximum residual force. (Integer; Default = 0)
TOLID (4,2) [4,2]	Contact tolerance table ID. Used in heat transfer analysis only. (Integer, Default = 0 which means no table ID)
DQNEAR (3,3) [3,3]	Distance below which near themal contact behavior occurs. Used n heat transfer analysis only. (Real, Default = 0, which means near contact does not occur)
DISTID (4,3) [4,3]	Contact near distance table ID. Used in heat transfer only. (Integer, Default = 0 which means near contact does not occur) (Integer, Default = 0 which means no table ID)
FRLIM [5,4]	Friction stress limit. This entry is only used for friction type 6 (Coulomb friction using the bilinear model). If the shear stress due to friction reaches this limit value, then the applied friction force will be reduced so that the maximum friction stress is given by $min(\mu\sigma_n, \sigma_{limit})$, with μ the friction coefficient and σ_n the contact normal stress. (Real, Default = 1.0E20)
BIAS [5,5]	Contact tolerance bias (value between 0.0 and 1.0) A nonblank entry will override the BIAS entered on the BCBODY entry. (Real, no Default)
SLIDE [5,6]	Delayed slide off distance. This entry should not be made unless ICOORD=2 (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance large rthan the delayed slide off distance. By default, the delayed slide off distance is realted to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain.

Field	Contents
HARDS [5,7]	Hard-soft ratio. This entry is only used if double-sided contact with automatic constraint optimization is used, (ISTYP=2 on the BCBODY entry). The hard-soft ratio can be used by the program if there is a significant difference in the (average) stiffness of the contact bodies (expressed by the trace of the initial stress-strain law). If the ratio of the stiffnesses is larger then the hard-soft ratio, the nodes o the softest body are the preferred slave nodes. (Real, Default = 2.0)
HTC [7,1] / [8,1]	Contact heat transfer coefficient. If real, the value entered is the contact heat transfer coefficient. If integer, the value entered is the ID of a TABLEM1 entry specifying the contact heat transfer coefficient vs temperature. (Real, or Integer, Default = 0.0 for a heat transfer problem, omit for a structural problem.)
HCV [7,2] / [8,2]	Convection coefficient for near field behavior. If real, the value entered is the near field convection coefficient. If integer, the value entered is the ID of a TABLEM1 entry specifying the near field convection coefficient vs temperature. (Real, or Integer, Default = 0.0 for a heat transfer problem, omit for a structural problem).
HNC [7,3] / [8,3]	Natural convection coefficient for near field behavior. If real, the value entered is the near field natural convection coefficient. If integer, the value entered is the ID of a TABLEM1 entry specifying the near field natural convection coefficient vs temperature. (Real, or Integer, Default = 0.0 for a heat transfer problem, omit for a structural problem).
BNC [7,4] / [8,4]	Exponent associated with the natural convection coefficient for near field behavior. If real, the value entered is the exponent associated with near field natural convection coefficient. If integer, the value entered is the ID of a TABLEM1 entry specifying the exponent associated with the near field natural convection coefficient vs temperature. (Real, or Integer, Default = 1.0 for a heat transfer problem, omit for a structural problem.)
EMISS [7,5] / [8,5]	Emissivity for radiation to the environment or near thermal radiation. If real, the value entered is the emissivity. If integer, the value entered is the ID of a TABLEM1 entry specifying the emissivity vs temperature. (Real, or Integer, Default = 0.0 for a heat transfer problem, omit for a structural problem.)

Field	Contents
HBL [7,6] / [8,6]	Separation distance dependent thermal convection coefficient. If real, the value entered is the separation distance dependent thermal convection coefficient. If integer, the value entered is the ID of a TABLEM1 entry specifying the separation distance dependent thermal convection coefficient. (Real, or Integer, Default = 0.0 for a heat transfer problem, omit for a structural problem).
FK (SOL 700 only)	Real $\geq$ 0.0, kinetic coefficient of friction. Default = 0.0
EXP (SOL 700 only)	Real $\geq$ 0.0, exponential decay coefficient. Default = 0.0
METHOD (SOL 700 only)	Character, Influences the contact type used. Options are: FULL: Regular Contact (Default) AIRBAG: Single Surface Contact SS1WAY: Surface To Surface Two Way SS2WAY: Surface To Surface One Way RB1WAY: Rigid Body One Way To Rigid Body RB2WAY: Rigid Body Two Way To Rigid Body RNRB: Rigid Nodes To Rigid Body TIEDNS: Tied Nodes to Surface TIEDES: Tied Shell Edge to Surface TIEDSS: Tied Surface to Surface TIEDNSO: Tied Nodes to Surface with Offset TIEDESO: Tied Shell Edge to Surface with Offset TIEDSSO: Tied Surface to Surface with Offset Default = FULL
ADAPT (SOL 700 only)	Character, influences the contact type used Options are NO or YES. Default = NO When ADAPT=YES, the BCBODY entries IDMAi must be defined as: behav=DEFORM bsid references a BCPROP

Field	Contents
THICK (SOL 700 only)	Real $\geq$ 0.0, shell thickness scale factor. Default = 1.0
THICKOF (SOL 700 only)	Real $\geq$ 0.0 artificial contact thickness offset. Default = 0.0
PENV (SOL 700 only)	Real $\geq$ 0.0, overwrites the default maximum penetration distance factor. Default = 1.E20
FACT (SOL 700 only)	Real $\geq$ 0.0, scale factor for the contact forces. Default = 0.1
TSTART (SOL 700 only)	Real $\geq$ 0.0, time at which the contact is activated. Default = 0.0
TEND (SOL 700 only)	Real $\geq$ 0.0, time at which the contact is deactivated. Default = 1.e20
MAXPAR (SOL 700 only)	Real $\geq$ 0.0 Maximum parametric coordinate in segment. Default = 1.025, search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025. This factor allows an increase in the size of the segments. May be useful at sharp corners.
PENCHK (SOL 700 only)	(Integer) Small penetration in contact search option. If the slave node penetrates more than the segment thickness times the factor XPENE (specified by PARAM,DYXPENE), the penetration is ignored and the slave node is set free. The thickness is taken as the shell thickness if the segment belongs to a shell element or it is taken as 1/20 of its shortest diagonal if the segment belongs to a solid element. This option applies to the surface-to-surface contact algorithms. Options are: 0: check is turned off, (Default) 1: check is turned on, 2: check is on but shortest diagonal is used.
FSF (SOL 700 only)	Real, Coulomb friction scale factor. The Coulomb friction value is scaled as $\mu_{sv} = FSF \cdot \mu_c$
VSF (SOL 700 only)	Real, Viscous friction scale factor. If this factor is defined then the limiting force becomes: $F_{lim} = VSF \cdot VC \cdot A_{cont}$

Field	Contents
EROSOP (SOL 700 only)	Integer ≥ 0, erosion / Interior node option. Default = 1 0 = only exterior boundary information is saved 1 = storage is allocated so that eroding contact can occur Otherwise, no contact is assumed after erosion of the corresponding element.
IADJ (SOL 700 only)	Integer ≥ 0, adjacent material treatment option for solid elements. Default = 1 0 = solid element faces are included only for free boundaries 1 = solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact
SOFT (SOL 700 only)	Integer ≥ 0, Soft contraint option: Default=1 0 = penalty formulation 1 = soft constraint formulation (same as original MSC.Dytran method) 2 = pinball segment-based contact 4 = constraint approach for FORMING contact option In the penalty formulation, the interface stiffness is based on the elastic bulk modulii of the materials. In the soft constraint option, the interface stiffness is based on the nodal mass and the global time step size. This method is more suited for contact between two materials where the elastic modulii vary greatly.
DEPTH (SOL 700 only)	Integer > 0, search depth. Default = 2 A value of 1 is sufficiently accurate for most crash applications, and is much less expensive. For improved accuracy, the default = 2
BSORT (SOL 700 only)	Integer ≥ 0, Number of time-steps between bucket sorts. Values of 25-100 are recommended for single-surface contact. Values of 10-15 are recommended for master-slave contact. Default = 0 (automatic determination by the algorithm)
FRCFRQ (SOL 700 only)	Integer ≥ 0, Number of time-steps between contact fore updates for penalty contact formulations. This option can provide significant speed-up of the contact treatment. If used, values exceeding 3 or 4 are dangerous. Considerable care must be exercised when using this option, as this option assumes that contact does not change for FRCFRG time-steps. Default = 1 (force calculations are performed each cycle)

Field	Contents
SNLOG (SOL 700 only)	Integer ≥ 0, shooting node logic. Default = 1 0 = on (adviced for crash simulation) 1 = off
ISYM (SOL 700 only)	Integer ≥ 0, symmetry plane option. Default = 0 0 = off 1 = do not include faces with normal boundary conditions (e.g., segments of brick elements on a symmetry plane). This option is important to retain the correct boundary conditions in the model with symmetry.
I2D3D (SOL 700 only)	Integer ≥ 0, segment searching option. Default = 0 0 = search 2-D elements (shells) before 3D elements (solids) 1 = search 3-D elements (solids) before 2D elements (shells)
IGNORE (SOL 700 only)	Integer ≥ 0, ignore initial penetrations. "Initial" in this context refers to the first timestep that a penetration is encountered. 0 = Take default from PARAM,DYCONIGNORE*****,<value 1 = Allow initial penetrations to exist by tracking it 2 = Move nodes to eliminate initial penetrations Default = 0 (move nodes to eliminate initial penetrations)
SPR (SOL 700 only)	Integer, Option to include the slave side in the NCFORC and the INTFOR interface force files. Options are: 0: slave side forces not included. (Default) 1: slave side forces included.
MPR (SOL 700 only)	Integer, Include the master side in the NCFORC and the INTFOR interface force files. Options are: 0: master side forces not included (Default) 1: master side forces included

Field	Contents
VDC (SOL 700 only)	Real, Viscous damping coefficient in percent of critical. In order to avoid undesirable oscillation in contact, e.g., for sheet forming simulation, a contact damping perpendicular to the contacting surfaces is applied. Damping coefficient $\xi = VDC/100\xi_{wd}$ e.g., VDC = 20. ξ_{crit} is determined in the following fashion by LS-DYNA. $$\xi_{crit} = 2mw; \quad m = min\left(m_{slave}, m_{master}\right) \quad \begin{matrix} mass\,of\,master \\ resp.\,slave\,node \end{matrix}$$ $$w = \sqrt{k \cdot \frac{m_{slave} + m_{master}}{m_{slave} \cdot m_{master}}} \quad k \text{ interface stiffness}$$
SBOPT (SOL 700 only)	Integer, Segment-based contact options (SOFT=2). Options are: 0: defaults to 2. 1: pinball edge-edge contact (not recommended) 2: assume planer segments (default) 3: warped segment checking 4: sliding option 5: do options 3 and 4
SFS (SOL 700 only)	Real, Scale factor on default slave penalty stiffness.
SFM (SOL 700 only)	Real, Scale factor on default master penatly stiffness.
SST (SOL 700 only)	Real, Optional thickness for slave surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
MST (SOL 700 only)	Real, Optional thickness for master surface (overrides true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SFST (SOL 700 only)	(Real) Scale factor for slave surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SFMT (SOL 700 only)	Real, Scale factor for master surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.

Field	Contents
AUTO (SOL 700 only)	Character, Options are: YES: Automatic Contacts Activated (Default) NO: Non-Automatic Contact Activated. This option is not recommended when Distributed Memory Parallel is activated.
LCID (SOL 700 only)	Integer, TABLED1 ID giving force versus penetration behavior for RIGID contact. See also the definition of FCM below. Only allowed and required input when METHOD is RB1WAY, RB2WAY or RNRB.
FCM (SOL 700 only)	Integer, Force calculation method for rigid contact . Only allowed and required input when METHOD is RB1WAY, RB2WAY or RNRB. Options are: 1: Load curve gives total normal force on surface versus maximum penetration of any node (RB1WAY only). 2: Load curve gives normal force on each node versus penetration of node through the surface (all rigid body contact types). 3: Load curve gives normal pressure versus penetration of node through the surface (RB1WAY and RB2WAY only). 4: Load curve gives total normal force versus maximum soft penetration. In this case the force will be followed based on the original penetration point. (RB1WAY only).
US (SOL 700 only)	Real, Unloading stiffness for rigid contact. The default is to unload along the loading curve. This should be equal to or greater than the maximum slope used in the loading curve. Only allowed input when METHOD is RB1WAY, RB2WAY or RNRB.
“MASTERS”	Indicates the start of the list of bodies touched by touching body IDSLA1.
IDMAi (4,i) [15,1]	Identification numbers of BCBODY Bulk Data entries defining touched bodies. (Integer > 0)

Remarks:

1. BCTABLE defines surface contact. It is only recognized in SOLs 600 and 700.

2. If BCTABLE is not given, the default for contact analysis is assumed, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If BCTABLE is given, the default for every body is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. BCTABLE is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.
3. A short input to define two contact bodies exits if the user provides IDSLAVE and IDMAST. Then it is assumed that there are only two contact bodies, NGROUP is ignored and continuation entries are not allowed. Default values are set for the parameters on the continuation entry.
4. If the user leaves IDSLAVE and IDMAST blank, then NGROUP is required and 2*NGROUP continuation entries are expected.
5. The maximum number of touching bodies is 99 for versions prior to MSC.Nastran 2005 r3. For MSC.Nastran 2005 r3 and subsequent versions, the limit is 999.
6. When MSC.Marc is executed from MD Nastran, the BCTABLE with ID=0 will be used in “phase 0”. This phase of MSC.Marc is purely elastic and is sometimes used to move contact bodies together so that they are just touching. If no BCTABLE entries are desired in “phase 0”, then do not enter any BCTABLEs with ID=0. In the examples shown, the first BCTABLE is used in subcase 2 and the second BCTABLE is used in phase 0.
7. The lines starting with FRLIM and HTC are used for SOL 600 heat transfer or SOL 600 thermal contact analyses only and require that PARAM,MRCONVER,11 be entered in the Bulk Data. Standard SOL 600 and SOL 700 structural analysis must omit these lines.
8. (i,j) indicates the ith datablock jth field of MSC.Marc’s Contact Table (without tables) history definition. [i,j] indicates MSC.Marc’s Contact Table (with tables). Heat transfer options require MSC.Nastran 2005 r2 and MSC.Marc 2005 r2 or subsequent versions. SOL 700 options require MSC.Nastran 2005 r3 and dytran-lsdyna 2005 r3 or subsequent versions.
9. The variables for slave FK through IGNORE are for SOL 700 (dytran-lsdyna) only.

   (3,2) indicates the 3rd data block and the 2nd field of MSC.Marc’s CONTACT TABLE history definition.

10. It is not necessary to enter all continuation lines between SLAVE and MASTERS. For example, if an entry is required on the 4th SLAVE line, the first three must be entered with some being blank (and those after the 4th may be omitted.)

BDYLIST Fluid Boundary List

Defines the boundary between a fluid and a structure.

Format:

1	2	3	4	5	6	7	8	9	10
BDYLIST	RHO	IDF1	lDF2	IDF3	IDF4	IDF5	IDF6	IDF7	
	IDF8	-etc.-							

Example:

BDYLIST	.037	432	325	416	203	256	175	153	
	101	105	AXIS						

Field	Contents
RHO	Fluid mass density at boundary. (Real $\geq$ 0.0; Default is DRHO on the AXIF entry)
IDFi	Identification number of a RINGFL entry. (Integer > 0 or Character = “AXIS” may be specified in the first and / or last field on the entry)

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. Each entry defines a boundary if RHO $\neq$ 0.0. The order of the points must be sequential with the fluid on the right with respect to the direction of travel.
3. The word “AXIS” defines an intersection with the polar axis of the fluid coordinate system.
4. There is no limit to the number of BDYLIST entries specified. If the fluid density varies along the boundary, there must be one BDYLIST entry for each interval between fluid points.
5. The BDYLIST entry is not required and should not be used to specify a rigid boundary where structural points are not defined. Such a boundary is automatically implied by the omission of a BDYLIST.

6. If RHO=0.0, no boundary matrix terms will be generated to connect the GRIDB points to the fluid. See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*. This option is a convenience for structural plotting purposes. GRIDB points may be located on a fluid ring (RINGFL entry) only if the rings are included in a BDYLIST.

BDYOR CHBDYi Entry Default Values

Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.

Format:

1	2	3	4	5	6	7	8	9	10
BDYOR	TYPE	IVIEWF	IVIEWB	RADMINF	RADMIDB		PID	GO	
	CE	E1	E2	E3					

Example:

BDYOR	AREA4	2	2	3	3		10		

Field	Contents
TYPE	Default surface type. See Remark 2. (Character)
IVIEWF	Default identification number of front VIEW entry. (Integer > 0 or blank)
IVIEWB	Default identification number of back VIEW entry. (Integer > 0 or blank)
RADMIDF	Default identification number of a RADM entry for front face. (Integer $\geq$ 0 or blank)
RADMIDB	Default identification number of a RADM entry for back face. (Integer $\geq$ 0 or blank)
PID	Default PHBDY property entry identification number. (Integer > 0 or blank)
GO	Default orientation grid point. (Integer $\geq$ 0; Default = 0)
CE	Default coordinate system for defining the orientation vector. (Integer $\geq$ 0 or blank)
E1, E2, E3	Default components of the orientation vector in coordinate system CE. The origin of this vector is grid point G1 on a CHBDYP entry. (Real or blank)

Remarks:

1. Only one BDYOR entry may be specified in the Bulk Data Section.

2. TYPE specifies the type of CHBDYi element surface; allowable values are: POINT, LINE, REV, AREA3, AREA4, ELCYL, FTUBE, AREA6, AREA8, and TUBE.
3. IVIEWF and IVIEWB are specified for view factor calculations only (see VIEW entry).
4. GO is only used from BDYOR if neither GO nor the orientation vector is defined on the CHBDYP entry and GO is > 0.
5. E1, E2, E3 is not used if GO is defined on either the BDYOR entry or the CHBDYP entry.

BEAMOR CBEAM Entry Default Values

Defines default values for field 3 and fields 6 through 8 of the CBEAM entry.

Format:

1	2	3	4	5	6	7	8	9	10
BEAMOR		PID			X1	X2	X3	OFFT	

Example:

BEAMOR		39			0.6	2.9	-5.87	OOG	

Alternate Format and Example:

BEAMOR		PID			G0			OFFT	
BEAMOR		39			86			OOG	

Field	Contents
PID	Property identification number of the PBEAM or PBCOMP entry. (Integer > 0 or blank)
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$, using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 $\neq$ GA or GB on CBEAM entry)
OFFT	Offset vector interpretation flag. See Remark (Character or blank)

Remarks:

1. The contents of fields on this entry will be assumed for any CBEAM entry with corresponding fields that are blank.
2. Only one BEAMOR entry is allowed.
3. For an explanation of beam element geometry, see the CBEAM entry description.
4. If X1 or G0 is integer, G0 is used. If X1 or G0 is blank or real, then X1, X2, X3 is used.

5. For p-version CBEAM elements, field 9 contains the value of the built-in twist measured in radians. The OFFT option cannot be used for p-elements. Otherwise, OFFT in field 9 is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector $\vec{v}$ and the offset system x-axis are then used to define the z and y axes of the offset system.

BFRIC Contact Friction

Defines frictional properties between two bodies in contact.

Format:

1	2	3	4	5	6	7	8	9	10
BFRIC	FID		FSTIF	MU1					

Example:

BFRIC	33			0.3					

Field	Contents
FID	Friction identification number. See Remark **1.** (Integer > 0)
FSTlF	Frictional stiffness in stick. See Remarks **2.** and **3.** (Real > 0.0; Default = automatically selected by the program)
MU1	Coefficient of static friction. (Real > 0.0)

Remarks:

1. This identification number must be unique with respect to all other friction identification numbers. This is used in the FRICID field of BCONP Bulk Data entry.
2. The value of frictional stiffness requires care. A method of choosing its value is to divide the expected frictional strength (MU1*expected normal force) by a reasonable value of the relative displacement that may be allowed before slip occurs. The relative value of displacement before slip occurs must be small compared to expected relative displacements during slip. A large stiffness value may cause poor convergence, while too small a value may cause poor accuracy.

   Frictional stiffness specified by the user is selected as the initial value. If convergence difficulties are encountered during the analysis, the frictional stiffness may be reduced automatically to improve convergence.
3. The stiffness matrix for frictional slip is unsymmetric. However, the program does not use the true unsymmetric matrix. Instead the program uses only the symmetric terms. This is to avoid using the unsymmetric solver to reduce CPU time.

BJOIN (SOL 700)

Defines (multiple) pairs of grid points of one-dimensional and / or shell elements to be joined during the analysis. When the failure criterion for a grid-point pair is satisfied, the grid-point pair is removed from the join and the grid-point motion is computed for the separate grid points. The join ceases to exist when all pairs of the join have failed, after which all of the grid points of the join are treated as separate grid points

Format:

1	2	3	4	5	6	7	8	9	10
BJOIN	BID	SID		TYPE		SN		SS	
		TF							

Example:

BJOIN	1	2		SPOTWELD		1.E3		1.E3	

Field	Contents	Type	Default
BID	BJOIN number.	Integer > 0	Required.
SID	SET number of a Case Control set	Integer > 0	Required.
TOL	Tolerance used in matching grid-point pairs.	Real ≥ 0.0	1.E-4
TYPE	Type of failure criterion.	C	FOMO
	SPOTWELD Spotweld-type failure.		
	SN Failure force in tension.	Real ≥ 0.0	No failure.
	SS Failure force in shear.	Real ≥ 0.0	No failure.
	TF Failure total moment.	Real ≥ 0.0	No failure.

Remarks:

1. Nodes connected by a spot weld cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also, care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.
2. Failure of the spot welds occurs when:

$$\frac{|f_n|^n}{S_n} + \frac{|f_s|^m}{S_s} \geq 1$$

   where f_n and f_s are the normal and shear interface force. Component f_n is nonzero for tensile values only.
3. When the failure time, *TF*, is reached the spot weld becomes inactive and the constrained nodes may move freely.

BLSEG Boundary Line Segments

Defines a curve that consists of a number of line segments via grid numbers that may come in contact with another body.

Format:

1	2	3	4	5	6	7	8	9	10
BLSEG	ID	G1	G2	G3	G4	G5	G6	G7	

Alternate Format:

BLSEG	ID	G1	“THRU”	G2	“BY”	INC			

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	G8	G9	G10	G11	-etc.-				

Continuation Entry Format 2:

	G8	“THRU”	G9	“BY”	INC				

Example:

BLSEG	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Field	Contents
ID	Line segments identification number. See Remark **2.** (Integer > 0)
Gi	Grid point identification numbers on a curve in a continuous topological order so that the normal to the segment points toward the other curve. See Remark **3.** (Integer > 0)
INC	Grid point identification number increment. See Remark **3.** (Integer or blank)

Remarks:

1. A line segment is defined between every two consecutive grid points. Thus, the number of line segments defined is equal to the number of grid points specified minus 1. A corresponding BWlDTH Bulk Data entry may be required to define the width / thickness of each line segment. If the corresponding BWlDTH is not present, the width / thickness for each line segment is assumed to be unity.
2. ID must be unique with respect to all other BLSEG entries. Each line segment has a width in 3-D sideline and a thickness in a 2-D slideline contact to calculate contact stresses. The width / thickness of each line segment is defined via BWIDTH Bulk Data entry. The ID in BLSEG must be same as the ID specified in the BWlDTH. That is, there must be a one to one correspondence between BLSEG and BWlDTH. BWlDTH Bulk Data entry may be omitted only if the width / thickness of each segment is unity.
3. For automatic generation of grid numbers, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing (i.e., the user need not specify BY and the increment value).
4. The normal to the segment is determined by the cross product of the slideline plane vector (i.e., the Z direction of the coordinate system defined in the 'ClD' field of BCONP Bulk Data entry) and the tangential direction of the segment. The tangential direction is the direction from node 1 to node 2 of the line segment.
5. A curve may be closed or open. A closed curve is specified by having the last grid point identification number the same as the first grid number.

BNDFIX Fixed Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFIX	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BNDFIX	2	135	14	6					

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. BNDFIX and BSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BNDFREEi or BNDFIXi entries present, all a-set points are considered fixed during component mode analysis. If there are only BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BNDFIXi and BNDFREEi entries present, the c-set degrees-of-freedom are defined by the BNDFREEi entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

- If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDFIX1 Fixed Boundary Degrees-of-Freedom, Alternate Form of BNDFIX Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFIX1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

BNDFIX1	2	135	14	6	23	24	25	26	
	122	127							

Alternate Format and Example:

BNDFIX1	C	ID1	“THRU”	ID2					
BNDFIX1	3	6	THRU	32					

Field	Contents
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; For “THRU” option, ID1< ID2)

Remarks:

1. BNDFIX and BSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BNDFREEi or BNDFIXi entries present, all a-set points are considered fixed during component mode analysis. If there are only BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BNDFIXi and BNDFREEi entries present, the c-set degrees-of-freedom are defined by the BNDFREEi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
    - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
    - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BNDFREE Free Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFREE	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BNDFREE	124	1	5	23	6	16			

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. BNDFIX and BSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BNDFREEi or BNDFIXi entries present, all a-set degrees-of-freedom are considered fixed during component modes analysis. If there are only BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BNDFIXi and BNDFREEi entries present, the c-set degrees-of-freedom are defined by the BNDFREEi entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

- If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

5. The BNDFREE entry replaces the old CSET entry.

BNDFREE1 Free Boundary Degrees-of-Freedom, Alternate Form of BNDFREE Entry

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFREE1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Example:

BNDFREE1	124	1	5	7	6	9	12	122	
	127								

Alternate Formats and Examples:

BNDFREE1	C	ID1	“THRU”	ID2					
BNDFREE1	3	6	THRU	32					

BNDFREE1		“ALL”							
BNDFREE1		ALL							

Field	Contents
C	Component number. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks)
IDi	Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2)

Remarks:

1. BNDFIX and BSET entries are equivalent to each other. Either one of them or any combination of them may be employed.

2. If there are no BNDFREEi or BNDFIXi entries present, all a-set degrees-of-freedom are considered fixed during component modes analysis. If there are only BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both BNDFIXi and BNDFREEi entries present, the c-set degrees-of-freedom are defined by the BNDFREEi entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
    - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
    - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.
5. The BNDFREE1 entry replaces the old CSET1 entry.

BNDGRID Boundary Grid Points

Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200).

Format:

1	2	3	4	5	6	7	8	9	10
BNDGRID	C	GP1	GP2	GP3	GP4	GP5	GP6	GP7	
	GP8	-etc.-							

Example:

BNDGRID	123	41	42	43	44	45	46	47	
	49								

Alternate Format and Example:

BNDGRID	C	GP1	“THRU”	GP2					
BNDGRID	123	41	THRU	49					

Field	Contents
C	Component number (any unique combination of integers 1 through 6 with no embedded blanks). See Remark 1.
GPi	Shape boundary grid point identification number. (0 < Integer < 1000000; For THRU option, GP1 < GP2)

Remarks:

1. C specifies the components for the listed grid points for which boundary motion is prescribed.
2. Multiple BNDGRID entries may be used to specify the shape boundary grid point identification numbers.
3. Both fixed and free shape boundary grid point identification numbers are listed on this entry.
4. The degrees-of-freedom specified on BNDGRID entries must be sufficient to statically constrain the model.

5. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.

BOUTPUT Output for Slideline Contact

Defines slave nodes at which output is requested.

Format:

1	2	3	4	5	6	7	8	9	10
BOUTPUT	ID	G1	G2	G3	G4	G5	G6	G7	

Alternate Format:

BOUTPUT	ID	G1	“THRU”	G2	“BY”	INC			

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	G8	G9	G10	G11	-etc.-				

Continuation Entry Format 2:

	G8	“THRU”	G9	“BY”	INC				

Example:

BOUTPUT	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Format and Example Using “ALL” (No continuation entry is allowed):

BOUTPUT	ID	ALL							
BOUTPUT	15	ALL							

Field	Contents
ID	Contact region identification number of a BCONP entry for which output is desired. (Integer > 0)

Field	Contents
Gi	Slave node numbers for which output is desired. (Integer > 0)
INC	Grid point identification number increment. See Remark 1. (Integer or blank)

Remark:

1. For automatic generation of grid numbers, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing (i.e., the user need not specify BY and the increment value).

BRKSQL (SOL 600) Specifies Data for Brake Squeal Calculations using SOL 600

Specifies data for brake squeal calculations using MD Nastran Implicit Nonlinear (SOL 600).

Format:

1	2	3	4	5	6	7	8	9	10
BRKSQL	METH	AVSTIF	FACT1			GLUE	ICORD		
	R1	R2	R3	X	Y	Z			
	NASCMD								
	RCFILE								

Example:

BRKSQL	1	5.34E6	1.0						
	0.0	0.0	1.0	2.0	3.0	4.0			
	tran								
	nastb								

Field	Contents
METH	Method flag corresponding to the type of brake squeal calculations to be performed. (Integer, Default = 1) 0 = Performs brake squeal calculations before any nonlinear analysis has taken place (corresponds to MSC.Marc feature, 4302). 1 = Performs brake squeal calculations after all nonlinear load cases (corresponds to MSC.Marc feature, 4304). -1 = Same as ID=0 except it corresponds to MSC.Marc feature, 4301 (not recommended).
AVSTIF	Approximate average stiffness per unit area between the pads and disk. Corresponds to Marc's PARAMETERS fifth datablock, field 1. This value is also known as the initial friction stiffness in MSC.Marc Volume C documentation. AVSTIF is used a penalty contact stiffness for brake squeal, it needs to be a large value but not so large that numerical instabilities result. If AVSTIF is large enough, increasing it by a few orders of magnitude will not appreciably affect the squeal modes. (Real; no Default. Required field)

Field	Contents
FACT1	Factor to scale friction stiffness values calculated by MSC.Marc; see Remark 3. (Real; Default =1.0)
GLUE	Flag specifying whether MPC for non-pad / disk surfaces with glued contact are used or ignored. A value of 0 means ignore the MPC; a value of 1 means include the MPCs (see Remark 6). (Integer, Default = 1)
ICORD	Flag indicating whether coordinates are updated or not. A value of 0 means coordinates are not updated. A value of 1 means coordinates are updated using the formula Cnew=Corig+Defl where Cnew are unpdated coordinates, Corig are original coordinates, and Defl are the final displacements from last MSC.Marc increment. (Integer; Default = 0)
R1	X direction cosine (basic coord system) of axis of rotation; corresponds to MSC.Marc ROTATION A second datablock. (Real; no Default. Required field)
R2	Y direction cosine (basic coord system) of axis of rotation; corresponds to MSC.Marc ROTATION A second datablock
R3	Z direction cosine (basic coord system); corresponds to MSC.Marc ROTATION A second datablock. (Real; no Default. Required field)
X	X coordinate in basic coord system of a point on the axis of rotation; corresponds to MSC.Marc ROTATION A third datablock. (Real; no Default. Required field)
Y	Y coordinate in basic coord system of a point on the axis of rotation; corresponds to MSC.Marc ROTATION A third datablock. (Real; no Default. Required field)
Z	Z coordinate in basic coord system of a point on the axis of rotation; corresponds to MSC.Marc ROTATION A third datablock. (Real; no Default. Required field)

Field	Contents
NASCMD	Name of a command to run Nastran (limited to 64 characters) -- used in conjunction with the CONTINUE options on the SOL 600 entry. The full path of the command to execute Nastran should be entered. The string will be converted to lower case. See Remark 2. (Character; Default=nastran)
RCFILE	Name of an RC file to be used with a secondary Nastran job (limited to 8 characters) -- used in conjunction with the CONTINUE options on the SOL 600 entry. An extension of ".rc" will automatically be added. See Remark 2. (Character; Default=nastb.rc)

Remarks:

1. This entry is used to calculate complex eigenvalues for brake squeal using unsymmetric stiffness friction matrices calculated by MSC.Marc. Options exist to obtain the unsymmetric stiffness matrices using the undeformed geometry (initial contact) or after all specified nonlinear subcases.
2. SOL 600 performs brake squeal calculations, using the following approach. The main (original) Nastran job with input file jid.dat or jid.bdf spawns MSC.Marc just as it does for any other SOL 600 job. MSC.Marc calculates unsymmetric friction stiffness matrices that are saved on a file (jid.marc.bde with associated file jid.marc.ccc). The primary Nastran job then creates input data for a second Nastran job (jid.nast.dat) to use the unsymmetric stiffness matrices in an complex eigenvalue extraction. The primary Nastran job spawns a second Nastran job to calculate the complex eigenvalues. The complex eigenvalues and eigenvectors are found in jid.nast.f06, jid.nast.op2, etc.

   NASCMD is the name of the command to execute the secondary Nastran job. NASCMD can be up to 64 characters long and must be left justified in field 2. The sting as entered will be used as is - except that it will be converted to lower case regardless of whether it is entered in upper or lower case.

   RCFILE is the name of an RC file to be used for the secondary Nastran job. Normally it should be similar to the RC file used for the primary run except that additional memory will normally be necessary to calculate the complex eigenvalues and batch=no should also be specified for UNIX and Linux systems. RCFILE is limited to 8 characters and an extension of ".rc" will be added automatically. This entry will be converted to upper case in MD

Nastran but will be converted to lower case before spawning the complex eigenvalue run. This RC file must be located in the same directory as the MD Nastran input file. This entry is the same as specifying PARAM,MRRCFILE. Only one or the other should be used.

3. MPCs are produced for contact surfaces with glued contact. DMIGs are produced for contact surfaces without glued contact. The brakes and drums should not use glued contact; other regions of the structure can used glued contact.
4. The continuation lines may be omitted if defaults are appropriate.
5. When a BRKSQL entry is used, PARAM,MRMTXNAM and PARAM,MARCFIL1 should not be entered.
6. When brake squeal matrices are output by MSC.Marc, unsymmetric friction stiffness matrices are output for non-glued contact surfaces. For surfaces with glued contact, MPCs are output. The GLUE flag signals SOL 600 to look for these MPCs and combine them with other MPCs that might be in the model using MPCADD, or if no MPCs were originally used to add the MCPs due to glued contact. Glued contact surfaces may not be used for the disk-rotor interface. If GLUE is zero or blank, the MPCs for glued contact in the MSC.Marc brake squeal bde file (if any) will be ignored. Sometimes, MSC.Marc puts out MPCs with only one degree-of-freedom defined. Such MPCs will be ignored; otherwise MD Nastran will generate a fatal error.
7. If ICORD=1, an MSC.Marc t19 file will automatically be produced.

BSET Fixed Analysis Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BSET	2	135	14	6					

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. BSET and BNDFIX entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREEi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREEi entries present or both BSETi/BNDFIXi and CSETi/BNDFREEi entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREEi entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

- If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
- If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BSET1 Fixed Analysis Degrees-of-Freedom, Alternate Form of BSET Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

BSET1	2	135	14	6	23	24	25	26	
	122	127							

Alternate Format and Example:

BSET1	C	ID1	“THRU”	ID2					
BSET1	3	6	THRU	32					

Field	Contents
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; For “THRU” option, ID1< ID2)

Remarks:

1. BNDFIX and BSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREEi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREEi entries present or both BSETi/BNDFIXi and CSETi/BNDFREEi entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREEi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
    - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
    - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

BSURF (SOLs 600/700) Contact Body or Surface

Defines a contact body or surface defined by Element IDs used in SOLs 600 and 700 only.

Format 1:

1	2	3	4	5	6	7	8	9	10
BSURF	ID	ELID1	ELID2	ELID3	ELID4	ELID5	ELID6	ELID7	

Continuation Entry Format 1:

	ELID8	ELID9	etc.						

Alternate Format:

BSURF	ID	ELID1	THRU	ELID2	BY	INC			

Example:

BSURF	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Field	Contents
ID	Identification of a deformable surface corresponding to a BSID value on the BCBODY entry. See Remark **2.** (Integer > 0)
ELIDi	Element identification numbers. If the curve or surface is defined with element ids only, the direction of the normal depends on the grid point numbering. See Remarks.
INC	Identification number increment. See Remark **3.** (Integer or blank)

Remarks:

1. BSURF is recognized in SOLs 600 and 700 only.
2. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.

3. For automatic generation of element ids, the default increment value is 1 if element numbers are increasing or -1 if element numbers are decreasing (i.e., the user need not specify BY and the increment value).
4. The deformable surface may alternately be defined using BCBOX, BCPROP, or BCMATL entries.
5. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.

BWIDTH Boundary Line Segment Width or Thickness

Defines widths or thicknesses for line segments in 3-D or 2-D slideline contact defined in the corresponding BLSEG Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
BWIDTH	ID	W1	W2	W3	W4	W5	W6	W7	

Alternate Format:

BWIDTH	ID	W1	“THRU”	W2	“BY”	INC			

The continuation entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	W8	W9	W10	W11	-etc.-				

Continuation Entry Format 2:

	W8	“THRU”	W9	“BY”	INC				

Example:

BWIDTH	15	2.0	THRU	5.0	BY	1.0			
	2.0	2.0	2.0	2.0					
	35.	THRU	44.						
	1.5	3.4	7.6	0.4	0.7				

Field	Contents
ID	BLSEG entry identification number. (Integer > 0)
Wi	Width values for the corresponding line segments defined in the BLSEG entry. See Remark **1.** (Real > 0.0)
INC	Width value increment. See Remark **2.** (Real or blank)

Remarks:

1. BWIDTH may be omitted if the width of each segment defined in the BLSEG entry is unity. The number of widths to be specified is equal to the number of segments defined in the corresponding BLSEG entry.
2. The default value for INC is 1.0 if the width is increasing or -1.0 if the width is decreasing. That is, the user need not specify BY and the increment value. If the number of widths specified is less than the number of segments defined in the corresponding BLSEG entry, the width for the remaining segments is assumed to be equal to the last width specified.
3. If there is only one grid point in the corresponding BLSEG entry, there is no contributory area associated with the grid point. To compute correct contact stresses an area may be associated with the single grid point by specifying the area in field W1.

CAABSF Frequency-Dependent Acoustic Absorber Element

Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CAABSF	EID	PID	G1	G2	G3	G4			

Example:

CAABSF	44	38	1	10	20				

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number that matches a PAABSF entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of fluid connection points. (Integer $\geq$ 0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. If only G1 is specified then a point impedance is assumed. If G1 and G2 are specified then a line impedance is assumed. If G1, G2, and G3 are specified, then an impedance is associated with the area of the triangular face. If G1 through G4 are specified, then an impedance is associated with the quadrilateral face. See **Figure 8-2**.
3. The CAABSF element must connect entirely to fluid points on the fluid-structure boundary.

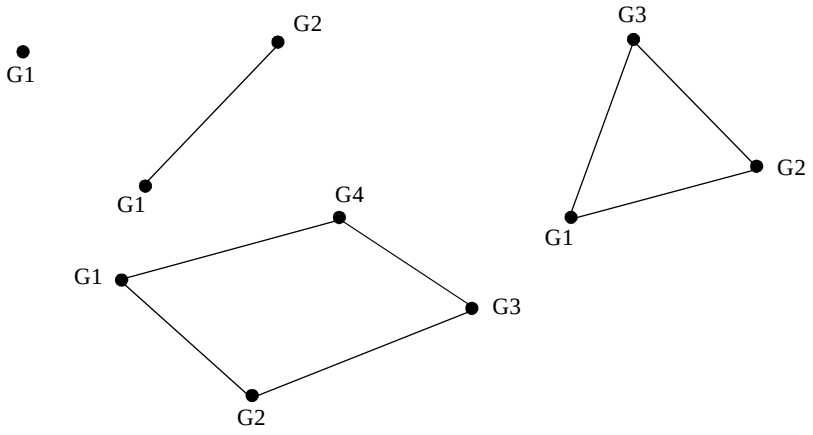


Figure 8-2 Four Types of CAABSF Elements

CACINF3 Acoustic Conjugate Infinite Element Base Connection

Defines an acoustic conjugate infinite element with triangular base

Format:

1	2	3	4	5	6	7	8	9	10
CACINF3	EID	PID	G1	G2	G3				

Field	Contents
EID	Element Identification Number. (Integer > 0)
PID	Property Identification Number of a PACINF entry. (Integer > 0)
Gi	Grid Point Identification Numbers of Element Base Connection Points. (Integer > 0)

CACINF4 Acoustic Conjugate Infinite Element Base Connection

Defines an acoustic conjugate infinite element with quadrilateral base

Format:

1	2	3	4	5	6	7	8	9	10
CACINF4	EID	PID	G1	G2	G3	G4			

Field	Contents
EID	Element Identification Number. (Integer > 0)
PID	Property Identification Number of a PACINF entry. (Integer > 0)
Gi	Grid Point Identification Numbers of Element Base Connection Points. (Integer > 0)

CAERO1 Aerodynamic Panel Element Connection

Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords. This is used for Doublet-Lattice theory for subsonic aerodynamics and the ZONA51 theory for supersonic aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO1	EID	PID	CP	NSPAN	NCHORD	LSPAN	LCHORD	IGID	
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO1	1000	1		3			2	1	
	0.0	0.0	0.0	1.0	0.2	1.0	0.0	0.8	

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PAERO1 entry; used to specify associated bodies. Required even if there are no associated bodies. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer $\geq$ 0; Default = 0)
NSPAN	Number of spanwise boxes; if a positive value is given NSPAN, equal divisions are assumed; if zero or blank, a list of division points is given at LSPAN, field 7. (Integer $\geq$ 0)
NCHORD	Number of chordwise boxes; if a positive value is given NCHORD, equal divisions are assumed; if zero or blank, a list of division points is given at LCHORD, field 8. (Integer $\geq$ 0)
LSPAN	ID of an AEFACT entry containing a list of division points for spanwise boxes. Used only if NSPAN, field 5 is zero or blank. (Integer > 0)
LCHORD	ID of an AEFACT data entry containing a list of division points for chordwise boxes. Used only if NCHORD, field 6 is zero or blank. (Integer > 0)
IGID	Interference group identification; aerodynamic elements with different IGIDs are uncoupled. (Integer > 0)

Field	Contents
X1, Y1, Z1 X4, Y4, Z4	Location of points 1 and 4, in coordinate system CP. (Real)
X12, X43	Edge chord lengths in aerodynamic coordinate system. (Real $\geq$ 0.0, but not both zero.)

Remarks:

1. The boxes and corner point nodes are numbered sequentially, beginning with EID. The user should be careful to ensure that all box and corner point node numbers are unique. There can be overlapping IDs between the structural and aerodynamic model, but MSC.Patran will not then be able to display any results. Also, non-unique corner IDs are allowed, but results cannot be visualized in MSC.Patran.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0. and 1. (In which case the corners of the panel would not be at the reference points.)
3. A triangular element is formed if X12 or X43=0.0
4. The element coordinate system is right-handed as shown in **Figure 8-3**.
5. The continuation is required.
6. It is recommended that NCHORD or LCHORD be chosen so that the typical box chord length Δx satisfies the condition $\Delta x < 0.08 \ V/f$ (recent studies indicate that .02 V/f is needed to get converged stability derivatives) where V is the minimum velocity and f, in hertz, is the maximum frequency to be analyzed (see the *MSC.Nastran Aeroelastic Analysis User's Guide).*

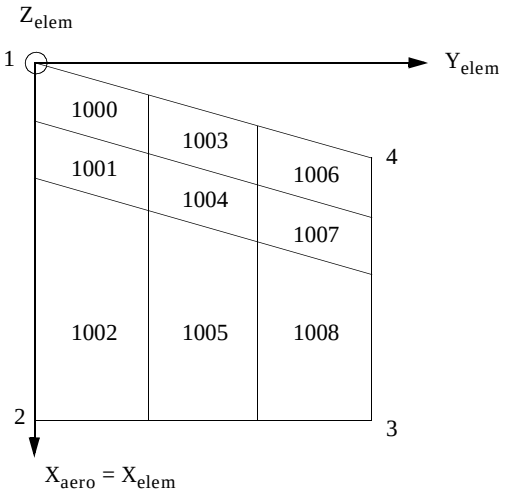


Figure 8-3 Element Coordinate System for Aerodynamic Panel

7. This entry can be used for two different aerodynamic theories: Doublet-Lattice for subsonic and ZONA51 for supersonic. The proper theory is selected based on the specification of Mach number on the MKAEROi or TRIM entry.

CAERO2 Aerodynamic Body Connection

Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO2	EID	PID	CP	NSB	NINT	LSB	LINT	IGID	
	X1	Y1	Z1	X12					

Example:

CAERO2	1500	2	100		4	99		1	
	-1.0	100.	-30.	175.					

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PAERO2 entry. (Integer > 0)
CP	Coordinate system for locating point 1. (Integer ≥ 0; Default = 0)
NSB	Number of slender body elements. If NSB > 0, then NSB equal divisions are assumed; if zero or blank, specify a list of divisions in LSB. (Integer ≥ 0)
NINT	Number of interference elements. If NINT > 0, then NINT equal divisions are assumed; if zero or blank, specify a list of divisions in LINT. (Integer ≥ 0)
LSB	ID of an AEFACT Bulk Data entry for slender body division points; used only if NSB is zero or blank. (Integer ≥ 0)
LINT	ID of an AEFACT data entry containing a list of division points for interference elements; used only if NINT is zero or blank. (Integer ≥ 0)
IGID	Interference group identification. Aerodynamic elements with different IGIDs are uncoupled. (Integer ≥ 0)
X1, Y1, Z1	Location of point 1 in coordinate system CP. (Real)
X12	Length of body in the x-direction of the aerodynamic coordinate system. (Real ≥ 0)

Remarks:

1. Point 1 is the leading point of the body.
2. All CAERO1 (panels) and CAERO2 (bodies) in the same group (IGID) will have aerodynamic interaction.
3. At least one interference element is required for the aerodynamic body specified by this entry.
4. The beams and connection points are numbered sequentially beginning with EID. The user should be careful to ensure that all aero elements and connection point IDs are unique. Overlapping IDs between structure and aerodynamic models are allowed, but will prevent results visualization in MSC.Patran.

   Old rules regarding numbering among Z, ZY, Y bodies and CAERO1 no longer apply: arbitrary ordering is allowed.
5. At least two slender body elements are required for each aerodynamic body.
6. Interference elements are only intended for use with panels.
7. Determining the size of the j-set (i.e., the number of aerodynamic elements) is essential to input D1JE and D2JE matrices. Use the following expressions for locating the proper row in the two matrices:

   J=Number of boxes = + Number of I-elements, z
   = + 2*(Number of I-elements, zy)
   = + Number of I-elements, y
   = + Number of S-elements, z
   = + 2*(Number of S-elements, zy)
   = + Number of S-elements, y

   where I-elements denote interference and S-elements denote slender body.

CAERO3 Aerodynamic Panel Element Configuration

Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO3	EID	PID	CP	LISTW	LISTC1	LISTC2			
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO3	2000	2001	0	22	33				
	1.0	0.0	0.0	100.	17.	130.	0.	100.	

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PAERO3 entry. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer $\geq$ 0; Default = 0)
LISTW	Identification number of an AEFACT entry that lists (x,y) pairs for structural interpolation of the wing. (Integer > 0)
LISTC1, LISTC2	Identification number of AEFACT entries that list (x,y) pairs for control surfaces, if they exist. (Integer $\geq$ 0)
X1, Y1, Z1 X4, Y4, Z4	Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43	Edge chord lengths in the aerodynamic coordinate system. (Real $\geq$ 0, X12 $\neq$ 0)

Remarks:

1. EID must be unique with respect to all other element identification numbers.
2. The (x,y) pairs on LISTW, LISTC1 and LISTC2 AEFACT entries are in the aero element coordinate system (see **Figure 8-4**). The (x,y) pairs define a set of aerodynamic grid points that are independent of Mach number and are selected by the user to be representative of the planform and motions of interest. The (x,y) pairs must be sufficient in number and distribution such

that: the surface spline provides an accurate interpolation between them and the Mach Box centers that are variously located on the planform as a function of Mach number (a complete description of the Mach Box Method is given in the *MSC.Nastran Aeroelastic Analysis User's Guide*).

3. The (x,y) pairs are numbered sequentially, beginning with EID for LISTW, then LISTC1, and finally for LISTC2. On SPLINEi entries, the box numbers (BOX1 and BOX2 on SPLINE1, ID1 and ID2 on SPLINE2, and UKID on SPLINE3) refer to the (x,y) pair sequence number appropriate for the surface (primary, or first or second control) being splined.
4. If cranks and / or control surfaces exist, their locations are given on the PAERO3 entry.
5. The numbering system and coordinate system are shown below:

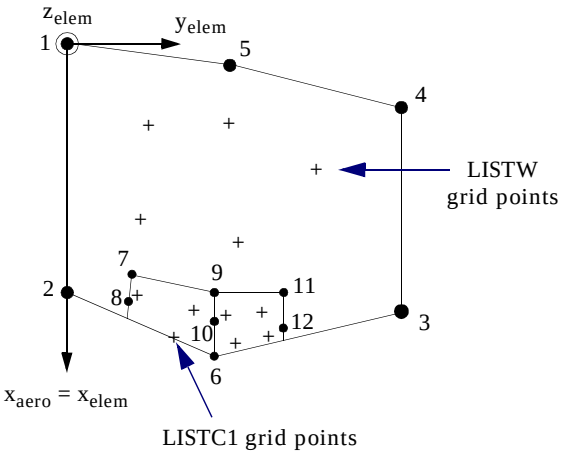


Figure 8-4 CAERO3 Element Configuration

Planform Corners		Control	
1	Leading edge, inboard	7	Hinge line, inboard
2	Trailing edge, inboard	8	On inboard edge (usually at trailing edge)
3	Trailing edge, outboard	9	Hinge line, outboard
4	Leading edge, outboard	10	On outboard edge (usually at trailing edge)
Cranks		Control (if two)	
5	Leading edge	9	Hinge line, inboard
6	Trailing edge	10	On inboard edge (usually at trailing edge)
		11	Hinge line, outboard
		12	On outboard edge (usually at trailing edge)

CAERO4 Aerodynamic Macro-Strip Element Connection

Defines an aerodynamic macro element for Strip theory.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO4	EID	PID	CP	NSPAN	LSPAN				
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO4	6000	6001	100		315				
	0.0	0.0	0.0	1.0	0.2	1.0	0.0	0.8	

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PAERO4 entry. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer $\geq$ 0; Default = 0)
NSPAN	Number of strips; if a positive value is given, NSPAN equal strips are assumed. If zero or blank, LSPAN must be specified. (Integer $\geq$ 0)
LSPAN	ID of an AEFACT entry containing a list of division points for strips. Used only if NSPAN is zero or blank. (Integer > 0)
X1, Y1, Z1 X4, Y4, Z4	Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43	Edge chord lengths in aerodynamic coordinate system. (Real $\geq$ 0.0, and not both zero.)

Remarks:

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and greater than structural grid, scalar, and extra point IDs.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN = 3, the division points are 0.0, 0.333, 0.667, and 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).
3. A triangular element is formed if X12 or X43 = 0.

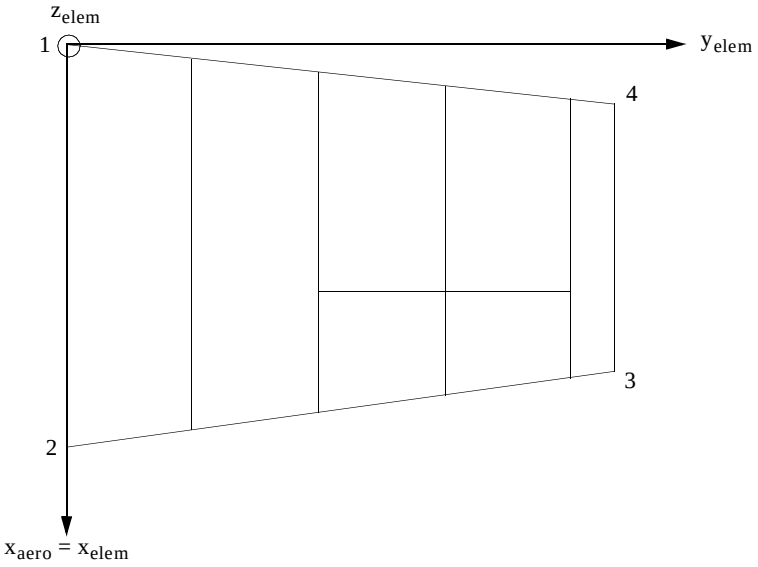


Figure 8-5 CAERO4 Element Connection

CAERO5 Aerodynamic Panel Element Configuration

Defines an aerodynamic macro element for Piston theory.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO5	EID	PID	CP	NSPAN	LSPAN	NTHRY	NTHICK		
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO5	6000	6001	100		315	0	0		
	0.0	0.0	0.0	1.0	0.2	1.0	0.	0.8	

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PAERO5 entry. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer $\geq$ 0; Default = 0)
NSPAN	Number of strips. (Integer. If a positive value is given, equal strips are assumed. If zero or blank, then LSPAN must be specified.)
LSPAN	ID of an AEFACT entry containing a list of division points for strips. Used only if NSPAN is zero or blank. (Integer > 0)
NTHRY	Parameter to select Piston or van Dyke's theory. (Integer = 0, 1, or 2; Default = 0) Blank or 0 = Piston theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ 1 = van Dyke's theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ with no sweep correction (secΛ = 1.0). 2 = van Dyke's theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ with a sweep correction based on the actual Λ.
NTHICK	Parameter to select thickness integrals input. (Integer $\geq$ 0; Default = 0) Blank or 0 = Thickness integrals are computed internally. >0 = Thickness integrals are input directly and are the ID number of an AEFACT entry that lists the *I* and / or *J* integrals.

Field	Contents
X1,Y1, Z1 X4,Y4, Z4	Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43	Edge chord lengths in aerodynamic coordinate system. (Real $\geq$ 0; X12 and X43 cannot both be zero.)

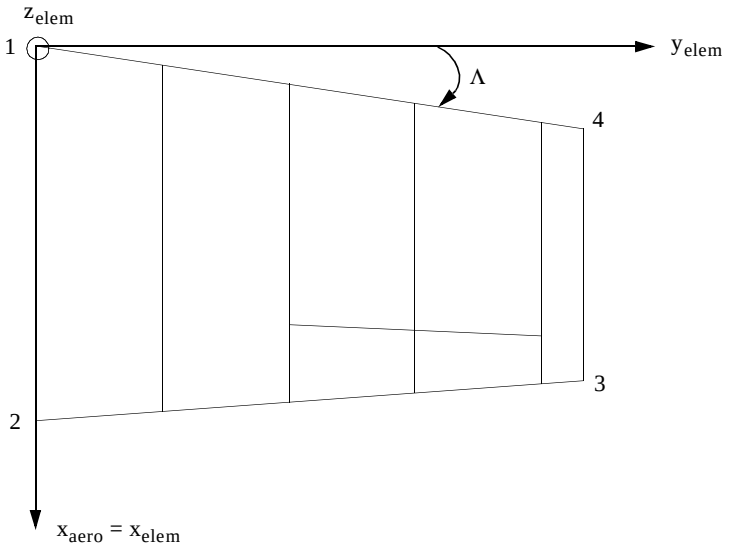


Figure 8-6 CAERO5 Element Configuration

Remarks:

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and different from structural grid IDs.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).
3. A triangular element is formed if X12 or X43 = 0.0.

4. $\bar{C}_1 = m/(m^2 - \sec^2\Lambda)^{1/2}$

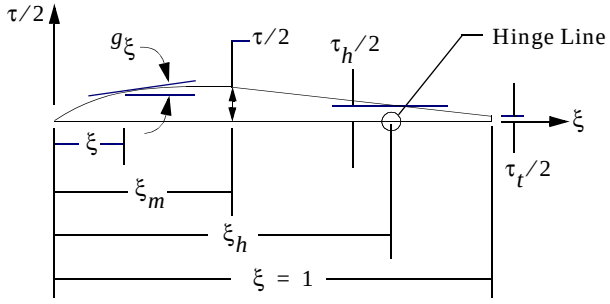
   $\bar{C}_2 = [m^4(\gamma + 1) - 4\sec^2\Lambda(m^2 - \sec^2\Lambda)]/[4(m^2 - \sec^2\Lambda)^2]$

   where:

   m = Mach number
   γ = Specific heat ratio
   Λ = Leading edge sweep angle

   When secΛ = 0.0, Piston theory coefficients are obtained (NTHRY = 1)
   When secΛ = 1.0, van Dyke's coefficients are obtained (NTHRY = blank or 0)
   When sec$\Lambda \neq 0.0$ or $\neq$ 1.0, sweep corrections are included (NTHRY = 2)

5. *I* and *J* thickness integral definitions:



$g_\xi \equiv \frac{dg}{d\xi}$ = slope of airfoil semithickness

$$I_1 = \int_0^1 g_\xi d\xi \qquad J_1 = \int_{\xi_h}^1 g_\xi d\xi$$

$$I_2 = \int_0^1 \xi g_\xi d\xi \qquad J_2 = \int_{\xi_h}^1 \xi g_\xi d\xi$$

$$I_3 = \int_0^1 \xi^2 g_\xi d\xi \qquad J_3 = \int_{\xi_h}^1 \xi^2 g_\xi d\xi$$

$$I_4 = \int_0^1 g_\xi{}^2 d\xi \qquad J_4 = \int_{\xi_h}^1 g^2{}_\xi d\xi$$

$$I_5 = \int_0^1 \xi g_\xi{}^2 d\xi \qquad {}_5 = \int_{\xi_h}^1 \xi g^2{}_\xi d\xi$$

$$I_6 = \int_0^1 \xi^2 g_\xi{}^2 d\xi \qquad J_6 = \int_{\xi_h}^1 \xi^2 g_\xi{}^2 d\xi$$

Figure 8-7 CAERO5 *I* and *J* Thickness Integral Definitions

CAMPBLL Campbell Diagram Parameters

Specifies the parameters for Campbell diagram generation.

Formats:

1	2	3	4	5	6	7	8	9	10
CAMPBLL	CID	VPARM	DDVALID	TYPE	ID	NAME / FID	B		

Examples:

CAMPBLL	10	SPEED	100	FREQ		0.25E-3			

Field	Contents
CID	Identification number of entry (Integer > 0, Required).
VPARM	Variable parameter, allowable entries are: ‘SPEED’, ‘PROP’, ‘MAT’.
	‘SPEED’, reference rotor speed will be varied (rotordynamic option only).
	‘PROP’, element property values will be varied (currently not implemented)
	‘MAT’, element material properties will be varied (currently not implemented).
DDVALID	Identification number of DDVAL entry that specifies the values for the variable parameter (Integer > 0, Required).
TYPE	For VPARM set to ‘SPEED’ allowable entries are: ‘FREQ’ and ‘RPM’.
	For VPARM set to ‘PROP’, allowable entries are the names of property entries, such as ‘PBAR’, ‘PELAS’, etc.
	For VPARM set to ‘MAT’, allowable entries are the names of material entries, such as ‘MAT1’ or ‘MAT2’, etc.
ID	Property or material entry identification number (Integer > 0, not required for VPARM set to ‘SPEED’, required for VPARM set to ‘PROP’ or ‘MAT’).
NAME / FID	For VPARM set to ‘SPEED’, no entry required.

Field	Contents
	For VPARM set to ‘PROP’, property name, such as ‘T’, ‘A’, or field position of the property entry, or word position in the element property table of the analysis model. For VPARM set to ‘MAT’, material property name, such as ‘E’ or ‘RHO’, or field position of the material entry. Names that begin with an integer, such as 12I/T**2, may only be referenced by a field position (character or integer < > 0, required).

Remark:

The ‘PROP’ and ‘MAT’ options are not currently available.

CAXIFi Fluid Element Connections

Defines an axisymmetric fluid element that connects i = 2, 3, or 4 fluid points.

Formats:

1	2	3	4	5	6	7	8	9	10
CAXIF2	EID	IDF1	IDF2			RHO	B		
CAXIF3	EID	IDF1	IDF2	IDF3		RHO	B		
CAXIF4	EID	IDF1	IDF2	IDF3	IDF4	RHO	B		

Examples:

CAXIF2	11	23	25			0.25E-3			
CAXIF3	105	31	32	33		6.47E-3			
CAXIF4	524	421	425	424	422	0.5E-3	2.5+3		

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
IDFi	Identification numbers of connected GRIDF points. (Integer > 0)
RHO	Fluid density in mass units. (Real > 0.0 or blank)
B	Fluid bulk modulus. (Real $\geq$ 0.0 or blank)

Remarks:

1. CAXIFi is allowed only if an AXSLOT entry is also present.
2. The element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO or B is blank, then the corresponding RHOD and BD fields must be specified on the AXSLOT entry.
4. Plot elements are generated for these elements. Because each plot element connects two points, one is generated for the CAXIF2 element, three are generated for the CAXIF3 element, and four plot elements are generated for the CAXIF4 element. In the last case the elements connect the pairs of points (1-2), (2-3), (3-4), and (4-1).
5. If B = 0.0, the fluid is incompressible.

CBAR Simple Beam Element Connection

Defines a simple beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3	OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Example:

CBAR	2	39	7	3	0.6	18.	26.	OOG	
		513							

Alternate Format and Example:

CBAR	EID	PID	GA	GB	G0			OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

CBAR	2	39	7	6	105				
		513							

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PBAR or PBARL entry. (Integer > 0 or blank*; Default = EID unless BAROR entry has nonzero entry in field 3.)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark **8.** (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB)

Field	Contents
OFFT	Offset vector interpretation flag. (character or blank) See Remark **8.**
PA, PB	Pin flags for bar ends A and B, respectively. Used to remove connections between the grid point and selected degrees-of-freedom of the bar. The degrees-of-freedom are defined in the element's coordinate system (see **Figure 8-8**). The bar must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4 is specified, the PBAR entry must have a value for J, the torsional stiffness. (Up to 5 of the unique Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0.) Pin flags combined with offsets are not allowed for SOL 600. Pin flags are not allowed in SOL 700.
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors $\vec{w}_a$ and $\vec{w}_b$, respectively (see **Figure 8-8**) in displacement coordinate systems (or in element system depending upon the content of the OFFT field), at points GA and GB, respectively. See Remark **7.** and **8.** (Real; Default = 0.0) Offsets are not allowed in SOL 700.

* See the BAROR entry for default options for field 3 and fields 6 through 9.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. **Figure 8-8** and **Figure 8-9** define bar element geometry with and without offsets:

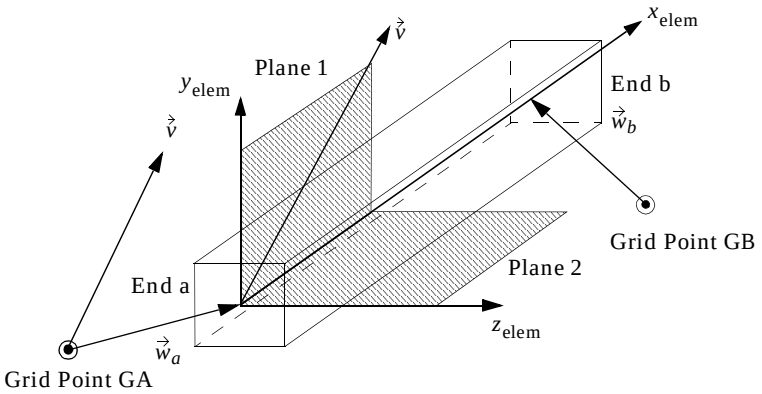


Figure 8-8 CBAR Element Geometry with Offsets

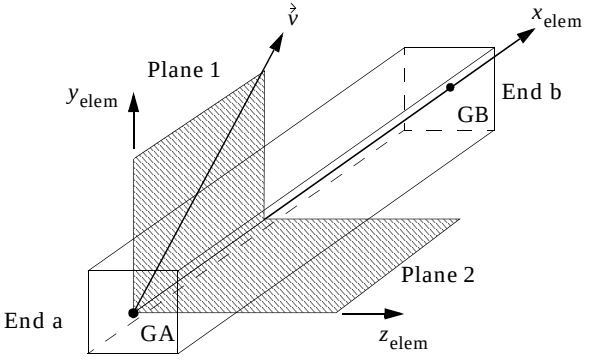


Figure 8-9 CBAR Element Geometry without Offsets

3. **Figure 8-10** and **Figure 8-11** define the elemental force and moment sign convention.

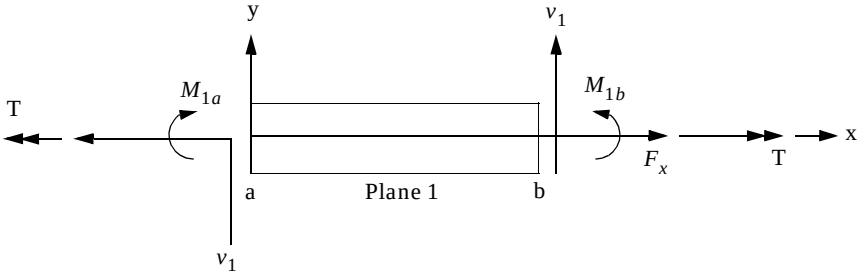


Figure 8-10 CBAR Element Internal Forces and Moments (x-y Plane)

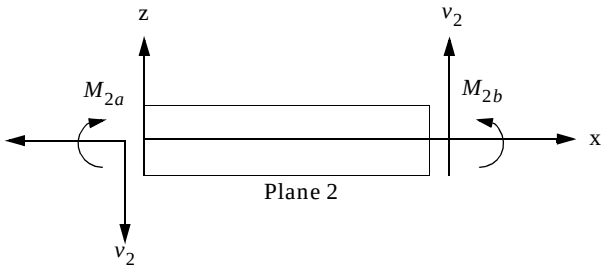


Figure 8-11 CBAR Element Internal Forces and Moments (x-z Plane)

4. The continuation may be omitted if there are no pin flags or offsets.
5. For the case where field 9 is blank and not provided by the BAROR entry, if an integer is specified in field 6, then G0 is used; if field 6 is blank or real, then X1, X2, X3 is used.
6. See "**Grid Point and Coordinate System Definition**" on page 41 of the *MSC.Nastran Reference Guide* for a definition of coordinate system terminology.
7. Offset vectors are treated like rigid elements and are therefore subject to the same limitations.
    - Offset vectors are not affected by thermal loads.
    - The specification of offset vectors is not recommended in solution sequences that compute differential stiffness because the offset vector remains parallel to its original orientation. (Differential stiffness is computed in buckling analysis provided in SOLs 103 and 107 through 112 with the STATSUB command; and also in nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1.)
    - BAR elements with offsets will give wrong buckling results.
    - Masses are not offset for shells.
    - The nonlinear solution in SOL 106 uses differential stiffness due for the iterations to reduce equilibrium errors. An error in the differential stiffness due to offsets may cause the iterations to converge slowly or to diverge. If the solution converges the answer is correct, even though there may be an error in the differential stiffness. However, the special capabilities in SOL 106 to get vibration and buckling modes will produce wrong answers if the differential stiffness is bad.

- The internal “rigid elements” for offset BAR/BEAM elements are rotated in the nonlinear force calculations. Thus, if convergence is achieved, BAR/BEAM elements may be used in SOL 106 with LGDISP,1.

8. OFFT is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset coordinate system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector $\vec{v}$ and the offset system x-axis are then used to define the z and y axes of the offset system.

9. For SOL 600, the BIT field is ignored unless param,MAROFSET is 1 or 2. An extra flag
10. For SOL 700, the BIT field is ignored and a warning is issued.

CBARAO Auxiliary Output Points Along Bar Element Axis (CBAR Entry)

Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output.

Format:

1	2	3	4	5	6	7	8	9	10
CBARAO	EID	SCALE	X1	X2	X3	X4	X5	X6	

Example:

CBARAO	1065	FR	0.2	0.4	0.6	0.8			

Alternate Format and Example:

CBARAO	EID	SCALE	NPTS	X1	DELTAX				
CBARAO	1065	FR	4	0.2	0.2				

Field	Contents
EID	Element identification of a CBAR entry. (0 < Integer < 100,000,000)
SCALE	Defines scale of Xi values. (Character = “LE” or “FR”)
Xi	Series of locations along element axis for stress and force data recovery. (Real > 0.0)
DELTAX	Incremental distance along element axis. (Real)
NPTS	Number of stress recovery points, not including the end points. (Integer > 0)

Remarks:

1. This entry defines intermediate locations on the axis of selected CBAR elements for additional data recovery. The values of Xi are actual distances along the length if SCALE = “LE”. If SCALE = “FR”, the values of Xi are ratios of actual distances to the bar length. A PLOAD1 Bulk Data entry for the CBAR element in question must be present to obtain intermediate data recovery.
2. When the alternate format is used, a series of locations Xi = X[i-1]+DELTAX, i = 1, 2, ..., NPTS is generated.

3. If a CBARAO or PLOAD1 entry is specified and stress and/or force output is requested, then the stresses and/or forces will be calculated at each location Xi and output as a separate line. The force and stress values at the end points of the beam will always be output. This output format will be used for all beam and bar elements.
4. Intermediate loads on the element defined by the PLOAD1 entry will be accounted for in the calculation of element stresses and forces. If no PLOAD1 entry is defined for the element, the shear forces are constant, the moments are linear, and it is not necessary that the user define additional points.
5. For each bar element, either the basic format or the alternate format, but not both, may be used. A maximum of six internal points can be specified with the basic form. The end points must not be listed because data will be generated for them, as explained in Remark **3.** If more than six unequally spaced internal points are desired, it is advisable to subdivide the bar into two or more elements.

CBEAM Beam Element Connection

Defines a beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	X1	X2	X3	OFFT / BIT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

Example:

CBEAM	2	39	7	13	8.2	6.1	-5.6	OOG	
		513		3.0					
	8	5							

Alternate Format and Example:

CBEAM	EID	PID	GA	GB	G0			OFFT / BIT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

CBEAM	2	39	7	13	105				
		513							

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PBEAM, PBCOMP or PBEAML entry. (Integer > 0; Default = EID)*
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark **9.** (Real)

Field	Contents
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then transferred to End A. (Integer > 0; G0 ≠ GA or GB)
OFFT	Offset vector interpretation flag. See Remark **9.** (Character or blank)
BIT	Built-in twist of the cross-sectional axes about the beam axis at end B relative to end A. For beam p-elements only. (Real; Default = 0.0)
PA, PB	Pin flags for beam ends A and B, respectively; used to remove connections between the grid point and selected degrees-of-freedom of the beam. The degrees-of-freedom are defined in the element's coordinate system and the pin flags are applied at the offset ends of the beam (see **Figure 8-12**). The beam must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4, the PBEAM entry must have a nonzero value for J, the torsional stiffness. (Up to five of the unique Integers 1 through 6 with no embedded blanks.) Pin flags are not allowed for beam p-elements. Pin flags combined with offsets are not allowed for SOL 600. Pin flags are not presently allowed in SOL 700.
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors from the grid points to the end points of the axis of the shear center. See Remarks **7.**, **8.** and **9.** (Real; Default = 0.0)
SA, SB	Scalar or grid point identification numbers for the ends A and B, respectively. The degrees-of-freedom at these points are the warping variables $d\theta/dx$. SA and SB cannot be specified for beam p-elements. (Integers > 0 or blank)

* See the BEAMOR entry for default options for field 3 and fields 6 through 9.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.

2. For an additional explanation of the beam element, see the "**Beam Element (CBEAM)**" on page 58 of the *MSC.Nastran Reference Guide*. **Figure 8-12** defines beam element geometry:

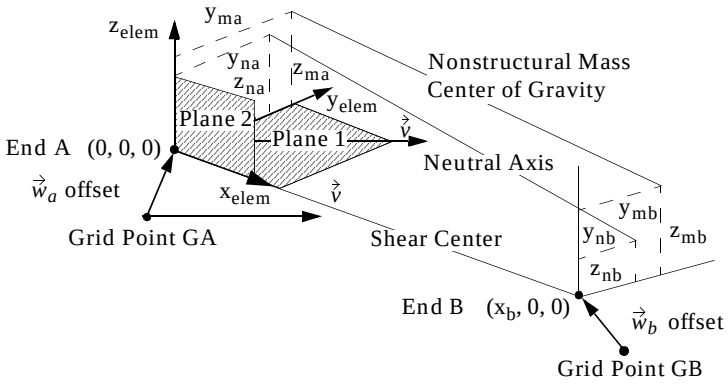


Figure 8-12 CBEAM Element Geometry System (Non p-adaptive)

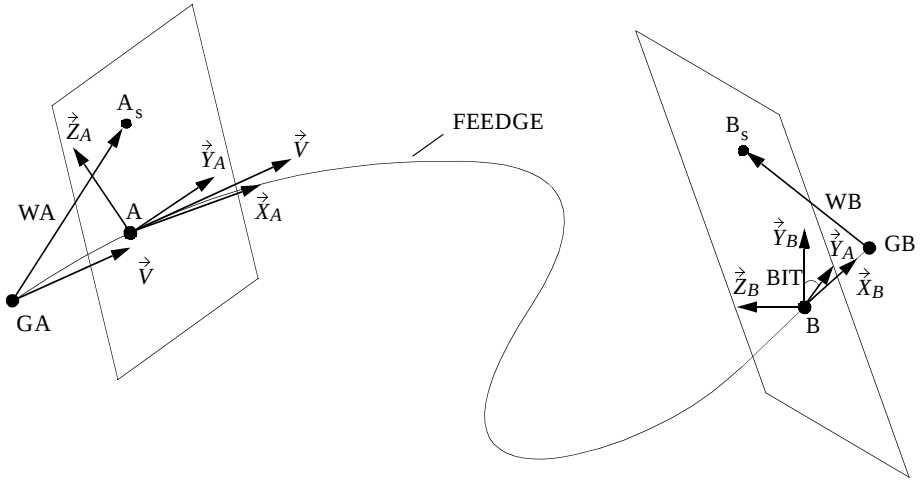


Figure 8-13 CBEAM Element Geometry System (p-adaptive)

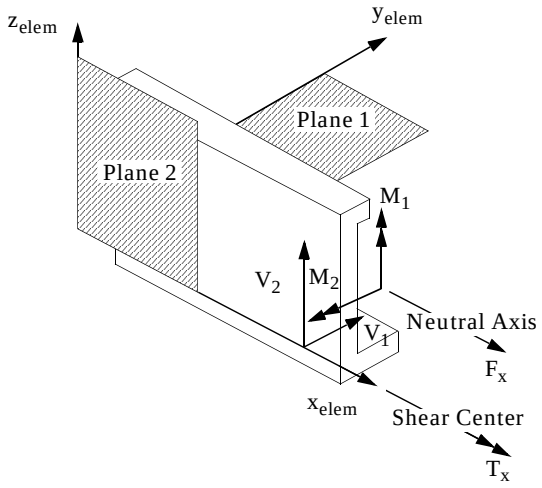


Figure 8-14 CBEAM Internal Element Forces and Moments

3. If field 6 is an integer, then G0 is used. If field 6 is blank or real, then X1, X2, X3 is used.
4. G0 cannot be located at GA or GB.
5. The rules for the continuations entries are:
   - Both continuations may be omitted if there are no pin flags, offsets, or warping variables.
   - If the second continuation is used, then the first continuation must be included, even if all fields are blank.
   - If the second continuation is omitted, torsional stiffness due to warping of the cross section will not be considered.
6. If warping is allowed (SA and SB > 0), then SA and SB must be defined with SPOINT or GRID entries. If GRID entries are used, the warping degree-of-freedom is attached to the first (T1) component.
7. Offset vectors are treated like rigid elements and are therefore subject to the same limitations.
   - Offset vectors are not affected by thermal loads.

- The specification of offset vectors is not recommended in solution sequences that compute differential stiffness because the offset vector remains parallel to its original orientation. (Differential stiffness is computed in buckling analysis provided in SOLs 105 and 200; SOLs 103 and 107 through 112 with the STATSUB command; and also in nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1).

8. If the CBEAM element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
    - By default, the edge of the element is considered straight unless the element is a p-element and the edge is associated to curved geometry with a FEEDGE entry.
    - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and considered to be straight. Edges with midside nodes cannot be shared by p-elements.
    - For the beam p-element, components of the offset vectors parallel to the beam axis (FEEDGE) will be ignored.
    - For the beam p-element, offset vectors can only be specified in the displacement coordinate systems at the grid points.
9. If the element is a p-version element, BIT in field 9 contains the value of the built-in-twist measured in radians. Otherwise, OFFT in field 9 is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset

String	Orientation Vector	End A Offset	End B Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector $\vec{v}$ and the offset system x-axis are then used to define the z and y axes of the offset system.

CBEAM3 Three-node Beam Element Connection

Defines a three-node beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM3	EID	PID	GA	GB	GC	X1	X2	X3	
	W1A	W2A	W3A	W1B	W2B	W3B	W1C	W2C	
	W3C	TWA	TWB	TWC	SA	SB	SC		

Example:

CBEAM3	101	2	201	332	1000	1.0	3.5	-2.0	
		3.0		3.0	2.2	-1.0			
	2.5	10.	15.	20.0	206	301	312		

Alternate Format and Example:

CBEAM3	EID	PID	GA	GB	GC	G0			
	W1A	W2A	W3A	W1B	W2B	W3B	W1C	W2C	
	W3C	TWA	TWB	TWC	SA	SB	SC		

CBEAM3	101	2	201	332	1000	105			
		3.0			2.2	1.0			
	2.5	10.	15.	20.0	206	301	312		

Field	Contents
EID	Unique element identification number. (Integer > 100,000,000)
PID	Property identification number of PBEAM3 or PBEAML entry. (Integer > 0; Required)
GA, GB, GC	Grid point identification numbers of connection points. GA and GB are grid point identification numbers at the two ends of the beam element while GC is the one at the grid point in between. (Integer > 0 or blank; GA, GB and GC must be distinct from each other. See Remark 6.)

Field	Contents
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then transferred to End A. (Integer > 0; G0 ≠ GA or GB or GC)
WiA, WiB, WiC	Components of offsets vectors, measured in the displacement coordinate systems at grid points A, B, and C, from the grid points to the points on the axis of shear center. (Real; Default = 0.0)
TWA, TWB, TWC	Pretwist angles in degrees at A, B, and C, respectively. (Real; Default = 0.0)
SA, SB, SC	Scalar or grid point identification numbers for A, B, and C, respectively. The degrees of freedom at these points are warping variables. (Integer > 0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. If field 7 is an integer, then G0 is used. If field 7 is blank or real, then X1, X2, X3 are used.
3. G0 cannot be located at GA or GB or GC.
4. If warping effect is included in the analysis (SA, SB and SC > 0), then SA, SB, and SC must be defined with either SPOINT or GRID entries. If GRID entries are used, the warping degree of freedom is attached to the first (T1) component.
5. BEAMOR cannot be used to set up default options for field 3 and fields 6 through 8 for CBEAM3 entries.
6. If GC is left blank, then the element degenerates to a two-node straight beam element.

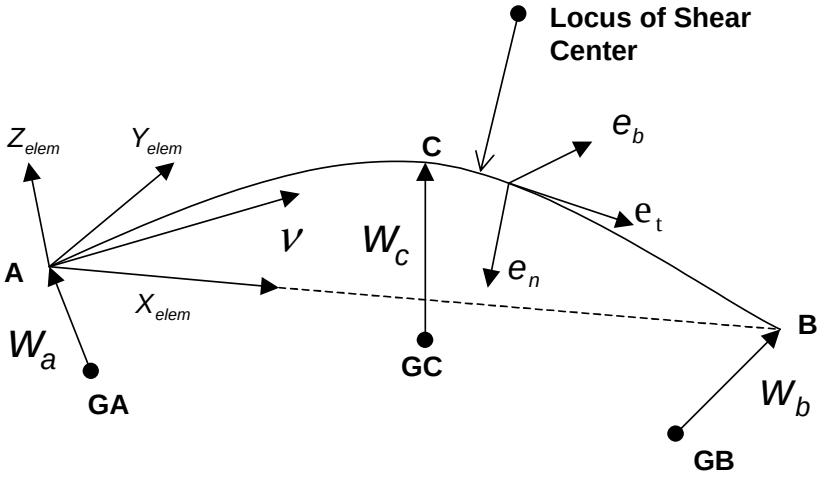


Figure 8-15 CBEAM3 Element Geometry System

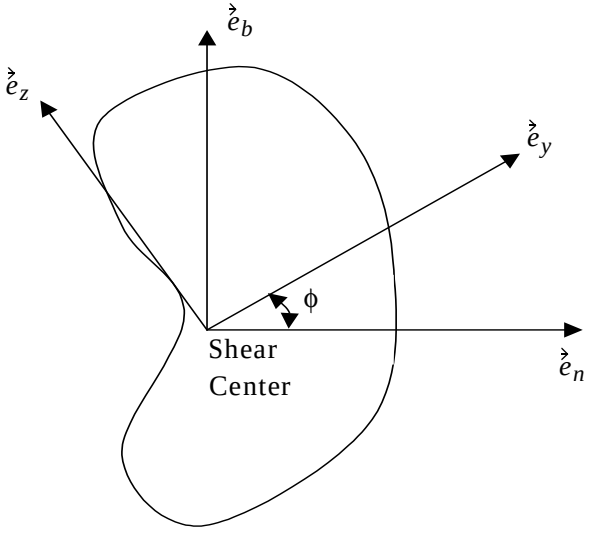


Figure 8-16 Local Coordinate System on Beam Cross-Section

CBELT (SOL 700) Seat Belt Element

Defines a seat belt element.

Format:

1	2	3	4	5	6	7	8	9	10
CBELT	EID	PID	N1	N2	SBRID	SLEN			

Example:

CBELT	12	21	1001	1002	11	.04			

Field	Contents
EID	Element ID. Unique number relative to all elements. (Integer, Required)
PID	Property ID (Integer, Required)
N1	Node 1 ID (Integer, Required)
N2	Node 2 ID (Integer, Required)
SBRID	Retractor ID of a SBRETR entry. (Integer, Required)
SLEN	Initial slack length. (Real, Default = 0.0)

Remarks:

1. The retractor ID should be defined only if the element is initially inside a retractor, see the SBRETR entry.
2. Belt elements are single degree of freedom elements connecting two nodes. When the strain in an element is positive (i.e. the current length is greater then the unstretched length), a tension force is calculated from the material characteristics and is applied along the current axis of the element to oppose further stretching. The unstretched length of the belt is taken as the initial distance between the two nodes defining the position of the element plus the initial slack length.
3. Corresponds to Ls-Dyna entry *ELEMENT_SEATBELT.

CBEND Curved Beam or Pipe Element Connection

Defines a curved beam, curved pipe, or elbow element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEND	EID	PID	GA	GB	X1	X2	X3	GEOM	

Example:

CBEND	32	39	17	19	6.2	5.1	-1.2		

Alternate Format and Example:

CBEND	EID	PID	GA	GB	G0			GEOM	
CBEND	32	39	17	19	106				

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PBEND entry. (Integer > 0; Default = EID)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB)
GEOM	Flag to select specification of the bend element. See Remark 3. (1 < Integer < 4)

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.

2. For an additional explanation of the CBEND element, see the PBEND entry description. **Figure 8-17** and **Figure 8-18** define the element coordinate system and internal forces and moments.

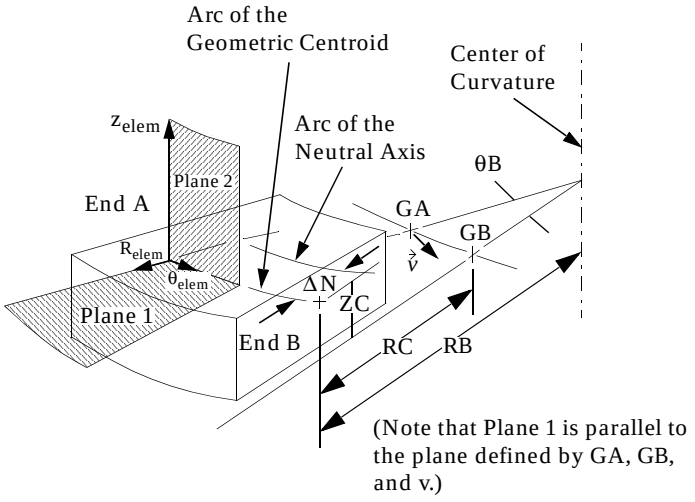


Figure 8-17 CBEND Element Coordinate System

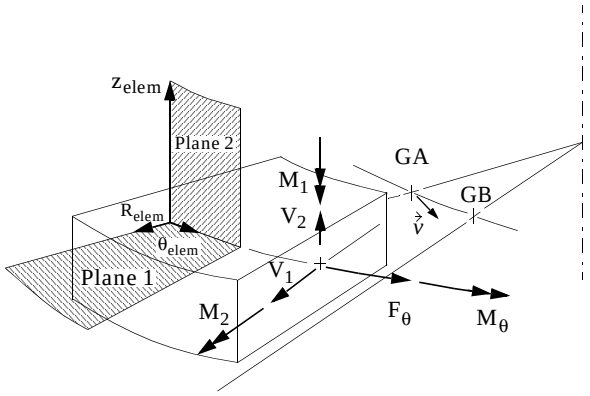
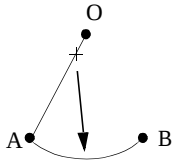
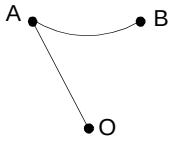
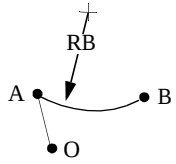
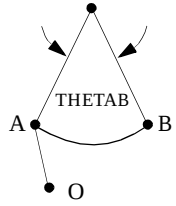


Figure 8-18 CBEND Element Internal Forces and Moments

3. The options for element connection to GA, GB using GEOM are the following.

Table 8-3 GEOM Options

Configuration	GEOM	Description
	1	The center of curvature lies on the line AO (or its extension) or vector $\vec{v}$.
	2	The tangent of centroid arc at end A is parallel to line AO or vector $\vec{v}$. Point O (or vector $\vec{v}$) and the arc $\overline{AB}$ must be on the same side of the chord $\overline{AB}$.
	3	The bend radius (RB) is specified on the PBEND entry: Points A, B, and O (or vector $\vec{v}$) define a plane parallel or coincident with the plane of the element arc. Point O (or vector $\vec{v}$) lies on the opposite side of line AB from the center of the curvature.
	4	THETAB is specified on the PBEND entry. Points A, B, and O (or vector $\vec{v}$) define a plane parallel or coincident with the plane of the element arc. Point O (or vector $\vec{v}$) lies on the opposite side of line AB from the center of curvature.

CBUSH Generalized Spring-and-Damper Connection

Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	GO / X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

Example 1: Noncoincident grid points.

CBUSH	39	6	1	100	75				

Example 2: GB not specified.

CBUSH	39	6	1					0	

Example 3: Coincident grid points (GA=GB).

CBUSH	39	6	1	100				6	

Example 4: Noncoincident grid points with fields 6 through 9 blank and a spring-damper offset.

CBUSH	39	6	1	600					
	0.25	10	0.	10.	10.				

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PBUSH entry. (Integer > 0; Default = EID)
GA, GB	Grid point identification number of connection points. See Remark **6.** (Integer > 0)
Xi	Components of orientation vector $\vec{v}$, from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply vector $\vec{v}$ using grid point GO. Direction of $\vec{v}$ is from GA to GO. $\vec{v}$ is then transferred to End A. See Remark **3.** (Integer > 0)

Field	Contents
CID	Element coordinate system identification. A 0 means the basic coordinate system. If CID is blank, then the element coordinate system is determined from GO or Xi. See **Figure 8-19** and Remark **3.** (Integer ≥ 0 or blank)
S	Location of spring damper. See **Figure 8-19**. (0.0 ≤ Real ≤ 1.0; Default = 0.5)
OCID	Coordinate system identification of spring-damper offset. See Remark **9.** (Integer ≥ -1; Default = -1, which means the offset point lies on the line between GA and GB according to **Figure 8-19**)
S1, S2, S3	Components of spring-damper offset in the OCID coordinate system if OCID ≥ 0. See **Figure 8-20** and Remark **9.** (Real)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. **Figure 8-19** shows the bush element geometry.
3. CID ≥ 0 overrides GO and Xi. Then the element x-axis is along T1, the element y-axis is along T2, and the element z-axis is along T3 of the CID coordinate system. If the CID refers to a cylindrical coordinate system or a spherical coordinate system, then grid GA is used to locate the system. If for cylindrical or spherical coordinate, GA falls on the z-axis used to define them, it is recommended that another CID be selectfced to define the element x-axis.
4. For noncoincident grids (GA ≠ GB), when GO or (X1, X2, X3) is given and no CID is specified, the line AB is the element x-axis and the orientation vector $\vec{v}$ lies in the x-y plane (similar to the CBEAM element).
5. For noncoincident grids (GA ≠ GB), if neither GO or (X1, X2, X3) is specified and no CID is specified, then the line AB is the element x-axis. This option is valid only when K1 (or B1) or K4 (or B4) or both on the PBUSH entry are specified (but K2, K3, K5, K6 or B2, B3, B5, B6 are not specified). If K2, K3, K5, or K6 (or B2, B3, B5, or B6) are specified, a fatal message will be issued.
6. If the distance between GA and GB is less than .0001, or if GB is blank, then CID must be specified. GB blank implies that B is grounded.
7. If PID references a PBUSHT entry, then the CBUSH element may only be defined in the residual structure and cannot be attached to any omitted degrees-of-freedom.

8. Element impedance output is computed in the CID coordinate system. The impedances in this system are uncoupled.
9. If OCID = -1 or blank (default) then S is used and S1, S2, S3 are ignored. If OCID $\geq$ 0, then S is ignored and S1, S2, S3 are used.

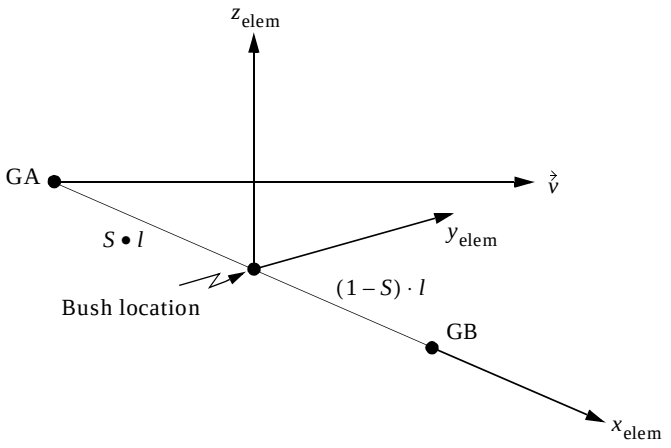
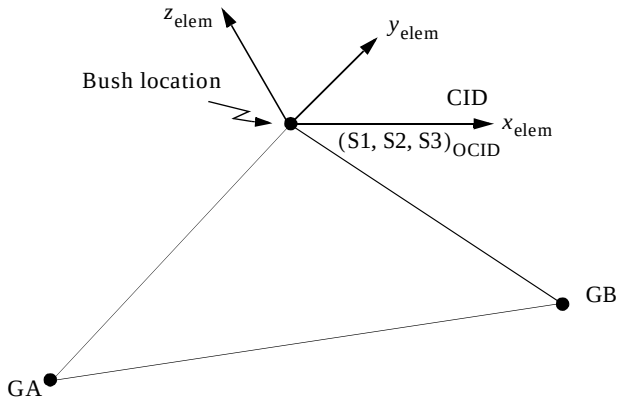


Figure 8-19 CBUSH Element



The material stiffness and damping properties of the elastomer are located at (S1, S2, S3).

Figure 8-20 Definition of Offset S1, S2, S3

10. When CID $\geq$ 0, the element x-axis is set as in Remark **3.** This means that the element force is always computed as Ke $\cdot$ (UB – UA); if UA > UB, a compressive force will result. This is unlike the GO or Xi options, where relative positive elongation in tension and relative negative elongation is compression.

11. The CBUSH element is designed to satisfy rigid body equilibrium requirements. For noncoincident grids, internal rigid links connect the bush location to the grid locations. This results in coupling between translational and rotational degrees-of-freedom at the grids even when no rotational springs or dampers are specified on the PBUSH.
12. For SOL 600, if G0, X1, X2, X3, CID or OCID are entered, a Severe Warning will be issued and Marc will not run. To override this message, enter param,marcbush,1 in the Bulk Data or rc file. SOL 600 translates the spring and damping terms in global coordinates of GA and GB and will use K1 to K6 and B1 to B6 whether or not GA and GB are coincident or not. S, S1, S2, S3 are ignored. If param,marcbush,1 is entered, G0, X1, X2, X3 and OCID are ignored for all CBUSH element in the model. CID is also ignored unless it is zero, in which case K1 and B1 are along the axis of GA to GB and K2-K6 and B2-B6 are ignored if entered.

CBUSH1D Rod Type Spring-and-Damper Connection

Defines the connectivity of a one-dimensional spring and viscous damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH1D	EID	PID	GA	GB	CID				

Example:

CBUSH1D	35	102	108	112					

Field	Contents	Default Values
EID	Element identification number. (0 < Integer < 100,000,000)	Required
PID	Property identification number of a PBUSH1D entry. (Integer > 0).	EID
GA	Grid point id of first grid.	Required
GB	Grid point id of second grid	blank
CID	Coordinate system id. (Integer $\geq$ 0)	blank

Remarks:

1. For noncoincident grids GA $\neq$ GB and if CID is blank, the line GA to GB is the element axis. In geometric nonlinear analysis, the element axis (line GA to GB) follows the deformation of grids GA and GB. See **Figure 8-21**.
2. If CID $\geq$ 0 is specified, the x-axis of the CID coordinate system is the element axis. In geometric nonlinear analysis, the element axis (x-axis of CID) remains fixed.
3. If GA and GB are coincident or if GB is blank, then CID $\geq$ 0 must be specified and the element axis is the x-axis of CID.

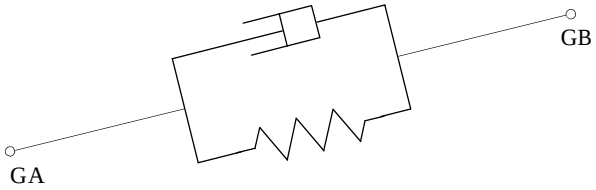


Figure 8-21 Spring and Damper Element

CBUSH2D 2-D Linear-Nonlinear Connection

Defines the connectivity of a two-dimensional Linear-Nonlinear element.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH2D	EID	PID	GA	GB	CID	PLANE	SPTID		

Example:

CBUSH2D	100	101	1001	2001	0	XY			

Field	Contents
EID	Element identification number (Integer > 0, Required).
PID	Property identification number of a PBUSH2D (Integer > 0, Required).
GA	Inner grid (Integer > 0, Required).
GB	Outer grid (Integer > 0, Required).
CID	Coordinate system used to define 2-D plane (Integer $\geq$ 0, Default = 0).
PLANE	Orientation plane in CID: XY, YZ, ZX, see Remark **1.** (Character, Default = 'XY').
SPTID	Optional rotor speed input for use with table lookup or DEQTN generation of element properties (Integer > 0 or blank).

Remarks:

1. The XY, YZ, and ZX planes are relative to the displacements coordinates of GA and GB. The planes correspond to directions 1 and 2. GA and GB should be coincident grids with parallel displacement coordinate systems. The coordinate systems are not checked. Wrong answers will be produced if this rule is not followed.
2. The SPTID input is currently not used.

CBUTT (SOL 700) Butt Weld in the LS-DYNA Style for SOL 700 Only

Defines a butt weld in the LS-DYNA style for use in SOL 700 only. Replaces CWELD for SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
CBUTT	EID	NSID	CID	WINDOW		NPRT			
	TFAIL	EPSF	SIG	BEAT	L	D	LT		

Example:

CBUTT	1002	11	5	0	0	3	1		
	0.005	.07		2.5	4.0	2.0	1.5		

Field	Contents
EID	Unique element identification number. (Integer > 0, required no default)
NSIDs	ID of a set number containing the grid points comprising this weld (Integer > 0, required, no default)
CID	ID of a CORDi entry providing the local output coordinate system for this weld (Integer $\geq$ 0 or blank, blank is the same as zero indicating the basic coordinate system)
FILTER	Number of force vectors saved for filtering. This option can eliminate spurious failures due to numerical force spikes buy memory will be larger if this option is envoked. Enter 0 for no filtering and N for a simple average of force coponents divided by N or the maximum number of force vectors that are stored for the time window option WINDOW (Integer > 0, Default = 0)
WINDOW	Time window for filtering (Real, Default = 0.0 for no filtering)
NPRT	Control of weld force output in file RBDOUT (Integer > 0, Default = 1) NPRT=1 data is output NPRT=2 data is not output
TFAIL	Failure time for this weld (Real > 0 or blank, Default = blank which means not used) Used for ductile failures.
EPSF	Effective plastic strain at failure (Real > 0 or blank, Default = blank which means not used). Used for ductile failures.

Field	Contents
SIG	Stress at failure (Real > 0 or blank, Default = blank which means not used) Used for brittle failures.
BETA	Failure parameter for brittle failure (Real > 0 or blank, default = blank which means not used) Used for brittle failures.
L	Length of butt weld (Real > 0, Required, no Default)
D	Thickness of butt weld (Real > 0, Required, no Default)
LT	Transverse length of butt weld (Real > 0, Required, no Default)

Remarks:

1. No property entry is needed for the CFILLET entry.
2. Ductile butt weld failure, due to plastic straining, is treated identically to spotweld failure. Brittle failure of the butt welds occurs when:

$$\beta\sqrt{\sigma_n^2 + 3(\tau_n^2 + \tau_t^2)} \geq \sigma_f$$

where

σ = normal stress

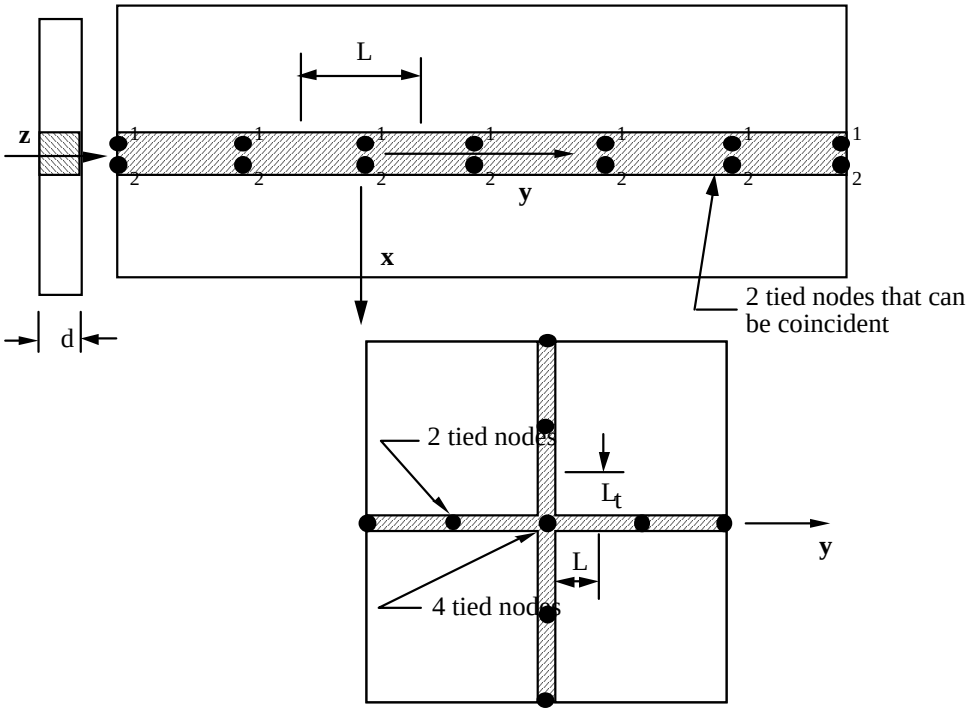
τ_n = shear stress in direction of weld (local y)

τ_t = shear stress normal to weld (local z)

σ_f = failure stress

β = failure parameter

Component σ_n is nonzero for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. The nodes in the butt weld may coincide.



Orientation of the local coordinate system and nodal ordering is shown for butt weld failure.

CCONEAX Axisymmetric Shell Element Connection

Defines a conical shell element.

Format:

1	2	3	4	5	6	7	8	9	10
CCONEAX	EID	PID	RA	RB					

Example:

CCONEAX	1	2	3	4					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PCONEAX entry. (Integer > 0; Default = EID)
RA	Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB)
RB	Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB)

Remarks:

1. This element has limited capabilities. See the *MSC.Nastran Reference Manual*, Section 5.3.3.
2. This entry is allowed only if an AXIC entry is also present.
3. In order to reference this entry on a SET Case Control command, the ID must be modified by

$$IDn = ID \cdot 1000 + n$$

where n is the harmonic number plus one and IDn is the value specified on the SET entry.

CCRSFIL (SOL 700) Cross-Fillet Weld in the LS-DYNA Style for SOL 700 Only

Defines a cross-fillet weld in the LS-DYNA style for use in SOL 700 only. Replaces CWELD for SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
CCRSFIL	EID	NSID	CID	FILTER	WINDOW	NPR	NPRT		
	TFAIL	EPSF	SIG	BETA	L	W	A	ALPHA	
	GA	GB	NCID						

Example:

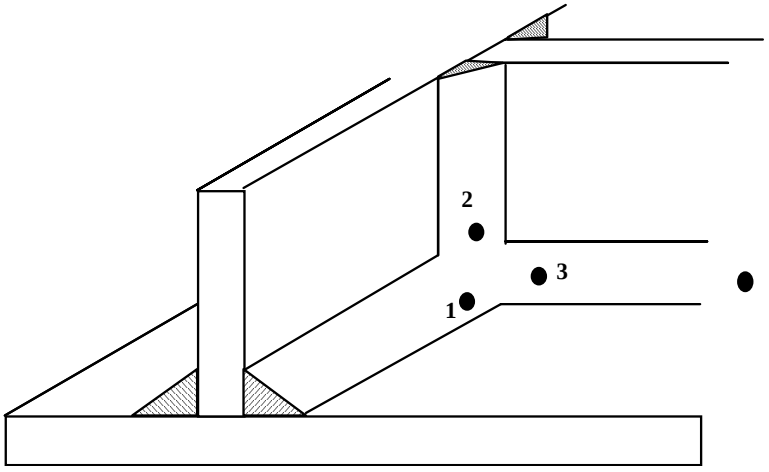
CCRSFIL	1002	11	5	0	0	2	1		
	0.005		40000.	2.5	4.	3.	2.	25.0	
	101	201	11						
	103	203	15						

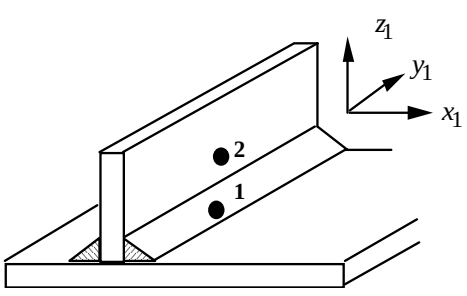
Field	Contents
EID	Unique element identification number. (Integer > 0, required no default)
NSID	ID of a set number containing the grid points comprising this weld (Integer > 0, Required, no Default)
CID	ID of a CORDi entry providing the local output coordinate system for this weld. (Integer $\geq$ 0 or blank, blank is the same as zero indicating the basic coordinate system)
FILTER	Number of force vectors saved for filtering. This option can eliminate spurious failures due to numerical force spikes but memory will be larger if this option is envoked. Enter 0 for no filtering and N for a simple average of force components divided by N or the maximum number of of force vectors that are stored for the time window option WINDOW (Integer > 0, Default =0)
WINDOW	Time window for filtering (Real, Default = 0.0 for no filtering)
NPR	Number of individual nodal pairs in this weld. The third line is repeated NPR times. (Integer > 0, no Default)

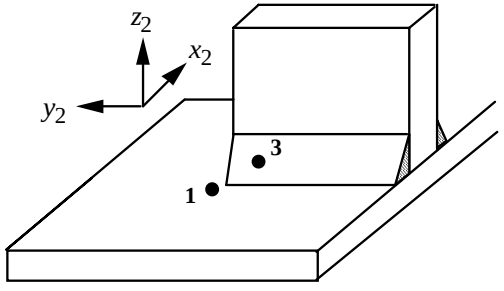
Field	Contents
NPRT	Control of weld force output in file RBOUT (Integer > 0, Default = 1) NPRT=1 data is output NPRT=2 data is not output
TFAIL	Failure time for this weld (Real > 0 or blank, Default = 1.0E20)
EPSF	Effective plastic strain at failure (Real > 0 or blank, default which means not used) Used for ductile failures.
SIG	Stress at failure (Real > 0 or blank, Default = blank which means not used) Used for brittle failures.
BETA	Failure parameter for brittle failure (Real > 0 or blank, Default = blank which means not used) Used for brittle failure. Which means not used. Used for brittle failures.
L	Length of weld (Real > 0, Required, no Default)
W	Width, W, of flange, see figure (Real > 0, Required, no Default)
A	Width, A, of flange, see figure (Real > 0, Required, no Default)
ALPHA	Weld angle, see figure (Real > 0, Required, no Default)
GA	Grid A ID, see figure (Integer > 0, Required, no Default)
GB	Grid B ID, see figure (Integer > 0, Required, no Default)
NCID	Local coordinate system of weld (Integer > 0 or blank, blank is the same as zero indicating the basic coordinate system)

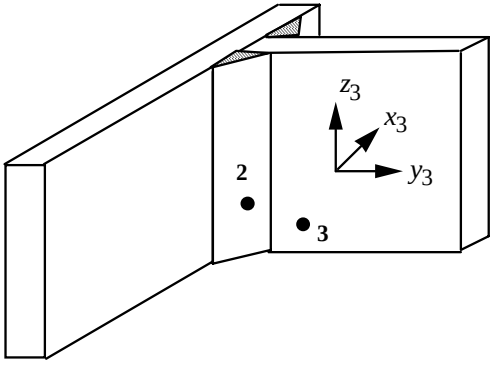
Remarks:

1. No property entry is needed for the CFILLET entry.
2. A simple cross fillet weld illustrates the required input. Here NFW=3 with nodal pairs (A=2, B=1), (A=3, B=1), and (A=3, B=2). The local coordinate axes are shown. These axes are fixed in the rigid body and are referenced to the local rigid body coordinate system which tracks the rigid body rotation.


2
3
1


z_1
y_1
x_1
2
1


z_2
x_2
y_2
3
1


z_3
x_3
y_3
2
3

CDAMP1 Scalar Damper Connection

Defines a scalar damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP1	EID	PID	G1	C1	G2	C2			

Example:

CDAMP1	19	6	0		23	2			

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PDAMP property entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer $\geq$ 0)
C1, C2	Component number. (0 $\leq$ Integer $\leq$ 6; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.

6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

CDAMP1D (SOL 700) Scalar Damper Connection for SOL 700 Only

Defines a scalar damper connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP1D	EID	PID	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CDAMP1D	1001	101	55	1					

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PDAMPn entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer $\geq$ 0)
C1, C2	Component number. (0 $\leq$ Integer $\leq$ 6; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. (Integer $\geq$ 0)
FOLLOW	Method to update the direction vector in which the damper acts: FOLLOW=CORD: direction vector follows the motion of hte coordinate system as specified under CORD.

Remark:

1. Scalar points may be used for G1 and / or G2, in which case the corresponding C1 and / or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.

4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.Available in SOL 700 only.

CDAMP2 Scalar Damper Property and Connection

Defines a scalar damper element without reference to a material or property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2	EID	B	G1	C1	G2	C2			

Example:

CDAMP2	16	2.98	32	1					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
B	Value of the scalar damper. (Real)
G1, G2	Geometric grid point identification number. (Integer $\geq$ 0)
C1, C2	Component number. (0 $\leq$ Integer $\leq$ 6; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.

6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

CDAMP2D (SOL 700) Scalar Damper Connection for SOL 700 Only

Defines a scalar damper connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2D	EID	B	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CDAMP2D	1001	101	55	1					

Field	Contents
EID	Unique element identification number. (Integer > 0)
B	Value of the scalar damper. (Real)
G1, G2	Geometric grid point identification number. (Integer $\geq$ 0)
C1, C2	Component number. (0 $\leq$ Integer $\leq$ 6; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. (Integer $\geq$ 0)
FOLLOW	Method to update the direction vector in which the damper acts: FOLLOW=CORD: direction vector follows the motion of the coordinate system as specified under CORD.

Remark:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.

4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry. Available in SOL 700 only.

CDAMP3 Scalar Damper Connection to Scalar Points Only

Defines a scalar damper element that is connected only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP3	EID	PID	S1	S2					

Example:

CDAMP3	16	978	24	36					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PDAMP entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer $\geq$ 0; S1 $\neq$ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. When CDAMP3 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.

CDAMP4 Scalar Damper Property and Connection to Scalar Points Only

Defines a scalar damper element that connected only to scalar points and without reference to a material or property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP4	EID	B	S1	S2					

Example:

CDAMP4	16	-2.6	4	9					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
B	Scalar damper value. (Real)
S1, S2	Scalar point identification numbers. (Integer $\geq$ 0; S1 $\neq$ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. If this entry is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.

CDAMP5 Scalar Damper with Material Property

Defines a damping element that refers to a material property entry and connection to grid or scalar points. This element is intended for heat transfer analysis only.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP5	EID	PID	G1	G2					

Example:

CDAMP5	1	4	10	20					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Identification number of a PDAMP5 property entry. (Integer > 0; Default = EID)
G1, G2	Grid or scalar point identification numbers. (Integer $\geq$ 0 and G1 $\neq$ G2)

Remarks:

1. G1 or G2 may be blank or zero indicating a constraint.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. CDAMP5 generates a lumped heat capacity in heat transfer analysis.
4. A scalar point specified on CDAMP5 need not be defined on an SPOINT entry.

CDUMi Dummy Element Connection

Defines a dummy element ($1 \le i \le 9$)

Format:

1	2	3	4	5	6	7	8	9	10
CDUMi	EID	PID	G1	G2	G3	G4	-etc.-		
	A1	A2	-etc.-						

Example:

CDUM2	114	108	2	5	6	8	11		
	2.4		3.E4	2		50			

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PDUMi entry. See Remark 2. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0, G1 ≠ G2 … ≠ GN)
Ai	Additional fields. (Real or Integer)

Remarks:

1. The user must write the associated element subroutines for matrix generation, stress recovery, etc., and perform a link edit to replace the dummy routines. See the *MSC.Nastran Programmer's Manual.*
2. If no property entry is required, PID may contain the material identification number.
3. Additional entries are defined in the user-written element routines.
4. CDUM1 is replaced by the CTRIAX6 element. If CDUM1 is used, User Fatal Message 307 will be issued.

CELAS1 Scalar Spring Connection

Defines a scalar spring element.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS1	EID	PID	G1	C1	G2	C2			

Example:

CELAS1	2	6			8	1			

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer $\geq$ 0)
C1, C2	Component number. (0 $\leq$ Integer $\leq$ 6; blank or zero if scalar point.)

Remarks:

1. Scalar points may be used for G1 and / or G2, in which case the corresponding C1 and / or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and / or ground are involved, it is more efficient to use the CELAS3 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

CELAS1D (SOL 700) Scalar Spring Connection for SOL 700 Only

Defines a scalar spring connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS1D	EID	PID	G1	C1	G2	C2			
	CORD	FOLLOW	ITID						

Example:

CELAS1D	1001	101	55	1	8	1			

Field	Contents
EID	Unique element identification number. (Integer > 0)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; 0 or up to six unique, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. (Integer ≥ 0) (SOL 700 only)
FOLLOW	Method to update the direction vector in which the spring acts: For SOL 700: FOLLOW=CORD: Direction vector follows the motion of the coordinate system as specified under CORD.
ITID	ID of TABLEDi providing nonlinear force-deflection curve (SOL 600 only)

Remarks:

1. Scalar points may be used for G1 and / or G2, in which case the corresponding C1 and / or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and / or ground are involved, it is more efficient to use the CELAS3 entry.

2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. Available in SOL 700 only.

CELAS2 Scalar Spring Property and Connection

Defines a scalar spring element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE	S	

Example:

CELAS2	28	6.2+3	32		19	4			

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
K	Stiffness of the scalar spring. (Real)
G1, G2	Geometric grid point or scalar identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; 0 blank or zero if scalar point.)
GE	Damping coefficient. See Remarks **6.** and **8.** (Real)
S	Stress coefficient. (Real)

Remarks:

1. Scalar points may be used for G1 and / or G2, in which case the corresponding C1 and / or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and / or ground are involved, it is more efficient to use the CELAS4 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide.*
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If PARAM,W4 is not specified, GE is ignored in transient analysis. See "**Parameters**" on page 659.

7. If Gi refers to a grid point then Ci refers to degrees-of-freedom in the displacement coordinate system specified by CD on the GRID entry.
8. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.

CELAS2D (SOL 700) Scalar Spring Connection for SOL 700 Only

Defines a scalar spring connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS2D	EID	K	G1	C1	G2	C2			
	CORD	FOLLOW	ITID						

Example:

CELAS1D	1001	101	55	1	8	1			

Field	Contents
EID	Unique element identification number. (Integer > 0)
K	Stiffness of the scalar spring (Real)
G1, G2	Geometric grid point identification number. (Integer $\geq$ 0)
C1, C2	Component number. (0 $\leq$ Integer $\leq$ 6; 0 or up to six unique, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
GE	Damping coefficient. See Remarks **6.** and **8.** (Real)
S	Stress coefficient. (Real)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. (Integer $\geq$ 0) (SOL 700 only)
FOLLOW	Method to update the direction vector in which the spring acts: For SOL 700: FOLLOW=CORD: Direction vector follows the motion of the coordinate system as specified under CORD.
ITID	ID of TABLEDi providing nonlinear force-deflection curve (SOL 600 only)

Remarks:

1. Scalar points may be used for G1 and / or G2, in which case the corresponding C1 and / or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and / or ground are involved, it is more efficient to use the CELAS4 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide.*
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If PARAM,W4 is not specified, GE is ignored in transient analysis. See "**Parameters**" on page 659.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom in the displacement coordinate system specified by CD on the GRID entry.
8. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
9. Available in SOL 700 only.

CELAS3 Scalar Spring Connection to Scalar Points Only

Defines a scalar spring element that connects only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS3	EID	PID	S1	S2					

Example:

CELAS3	19	2	14	15					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer $\geq$ 0; S1 $\neq$ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar spring element may be defined on a single entry.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

CELAS4 Scalar Spring Property and Connection to Scalar Points Only

Defines a scalar spring element that is connected only to scalar points, without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS4	EID	K	S1	S2					

Example:

CELAS4	42	6.2-3	2						

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
K	Stiffness of the scalar spring. (Real)
S1, S2	Scalar point identification numbers. (Integer $\geq$ 0; S1 $\neq$ S2)

Remarks:

1. S1 or S2, but not both, may be blank or zero indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. A structural damping coefficient is not available with CELAS4. The value of g is assumed to be 0.0.
4. No stress coefficient is available with CELAS4.
5. Only one scalar spring element may be defined on a single entry.
6. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
7. A scalar point specified on this entry need not be defined on an SPOINT entry.

CFAST A Shell Patch Fastener Connection

Defines a fastener with material orientation connecting two surface patches.

Format:

1	2	3	4	5	6	7	8	9	10
CFAST	EID	PID	TYPE	IDA	IDB	GS	GA	GB	
	XS	YS	ZS						

Example using PROP:

CFAST	3	20	PROP	21	24	206			

Example using ELEM:

CFAST	7	70	ELEM	27	74	707			

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PFAST entry. (Integer > 0; Default = EID)
TYPE	Specifies the surface patch definition: (Character) If TYPE = 'PROP', the surface patch connectivity between patch A and patch B is defined with two PSHELL (or PCOMP) properties with property ids given by IDA and IDB. See Remark **1.** and **Figure 8-22**. If TYPE = 'ELEM', the surface patch connectivity between patch A and patch B is defined with two shell element ids given by IDA and IDB. See Remark **1.** and **Figure 8-22**.
IDA,IDB	Property id (for PROP option) or Element id (for ELEM option) defining patches A and B. IDA ≠ IDB (Integer > 0)
GS	Grid point defining the location of the fastener. See Remark **2.** (Integer > 0 or blank)
GA,GB	Grid ids of piecing points on patches A and B. See Remark **2.** (Integer > 0 or blank)
XS,YS,ZS	Location of the fastener in basic. Required if neither GS nor GA is defined. See Remark **2.** (Real or blank)

Remarks:

1. The CFAST defines a flexible connection between two surface patches. Depending on the location for the piercing points GA and GB, and the size of the diameter D (see PFAST), the number of unique physical grids per patch ranges from a possibility of 3 to 16 grids. (Currently there is a limitation that there can be only a total of 16 unique grids in the upper patch and only a total of 16 unique grids in the lower patch. Thus, for example, a patch can not hook up to four CQUAD8 elements with midside nodes and no nodes in common between each CQUAD8 as that would total to 32 unique grids for the patch.)

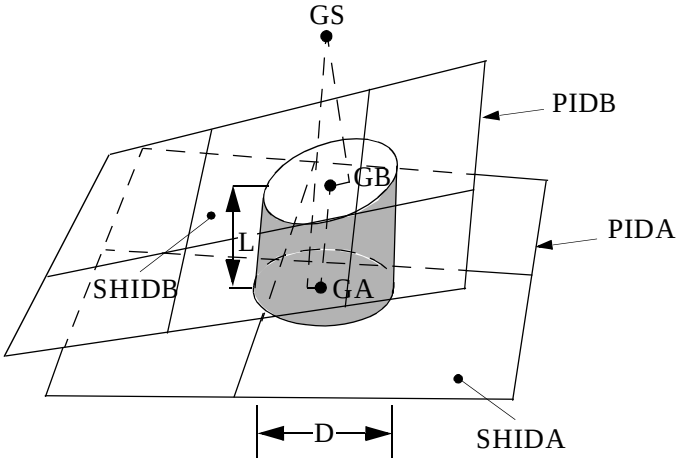


Figure 8-22 Patches Defined with TYPEj= 'PROP' or TYPE = 'ELEM'

2. GS defines the approximate location of the fastener in space. GS is projected onto the surface patches A and B. The resulting piercing points GA and GB define the axis of the fastener. GS does not have to lie on the surfaces of the patches. GS must be able to project normals to the two patches. GA can be specified in lieu of GS, in which case GS will be ignored. If neither GS nor GA is specified, then (XS, YS, ZS) in basic must be specified.

   If both GA and GB are specified, they must lie on or at least have projections onto surface patches A and B respectively. The locations will then be corrected so that they lie on the surface patches A and B within machine precision. The length of the fastener is the final distance between GA and GB. If the length is zero, the normal to patch A is used to define the axis of the fastener.

Diagnostic printouts, checkout runs and control of search and projection parameters are requested on the SWLDPRM Bulk Data entry.

3. The use of param,cfdiagp,yes and param,cfrandel,real_fraction_value allows for the random removal of a percentage of CFAST elements for failure studies.

CFILLET (SOL 700) Fillet Weld in the LS-DYNA Style for SOL 700 Only

Defines a fillet weld in the LS-DYNA style for use in SOL 700. Replaces CWELD for SOL 700.

Formats:

1	2	3	4	5	6	7	8	9	10
CFILLET	EID	NSID	CID	FILTER	WINDOW		NPRT		
	TFAIL	EPSF	SIG	BETA	L	W	A	ALPHA	

Examples:

CFILLET	1002	11	5	0	0	3	1		
	0.005	.07		2.5	4.0	3.0	2.0	25.0	

Field	Contents
EID	Unique element identification number. (Integer > 0, required no Default)
NSID	ID of a set number containing the grid points comprising this weld (Integer > 0, required, no Default)
CID	ID of a CORDi entry providing the local output coordinate system for this weld. (Integer $\geq$ 0 or blank - blank is the same as zero indicating the basic coordinate system)
FILTER	Number of force vectors saved for filtering. This option can eliminate spurious failures due to numerical force spikes but memory will be larger if this option is envoked. Enter 0 for no filtering and N for a simple average of force components divided by N or the maximum number of force vectors that are stored for the time window option WINDOW. (Integer > 0, Default = 0)
WINDOW	Time window for filtering (Real, Default = 0.0 for no filtering)
NPRT	Control of weld force output in file RBDOUT. (Integer > 0, Default = 1) NPRT = 1 data is output NPRT = 2 data is not output
TFAIL	Failure time for this weld (Real > 0 or blank, Default = 1.0E20)
EPSF	Effective plastic strain at failure (Real > 0 or blank, Default = blank which means not used) Used for ductile failures.

Field	Contents
SIG	Stress at failure (Real > 0 or blank, Default = Blank which means not used) Used for brittle failures.
BETA	Failure parameter for brittle failure (Real > 0 or blank, Default = blank which means not used) Used for brittle failures.
L	Length of fillet weld (Real > 0, Required, no Default)
W	Width, A of fillet weld - See **Figure 8-23**. (Real > 0, Required, No Default)
A	Width, A of fillet weld - See **Figure 8-23**. (Real > 0, Required, No Default)
ALPHA	Weld angle in degrees (Real > 0, Required, No Default)

Remarks:

1. No property entry is needed for the CFILLET entry.
2. Ductile fillet weld failure, due to plastic straining, is treated identically to spotweld failure. Brittle failure of the fillet welds occurs when:

$$\beta\sqrt{\sigma_n^2 + 3(\tau_n^2 + \tau_t^2)} \geq \sigma_f$$

where:

σ_n = normal stress

ι_n = shear stress in direction of weld (local y)

ι_t = shear stress normal to weld (local x)

σ_f = failure stress

β = failure parameter

Component σ_n is nonzero for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. In **Figure 8-23** the ordering of the nodes is shown for the 2 node and 3 node fillet welds. This order is with respect to the local coordinate system where the local z axis determines the tensile direction. The nodes in the fillet weld may coincide. The failure of the 3 node fillet weld may occur gradually with first one node failing and later the second node may fail.

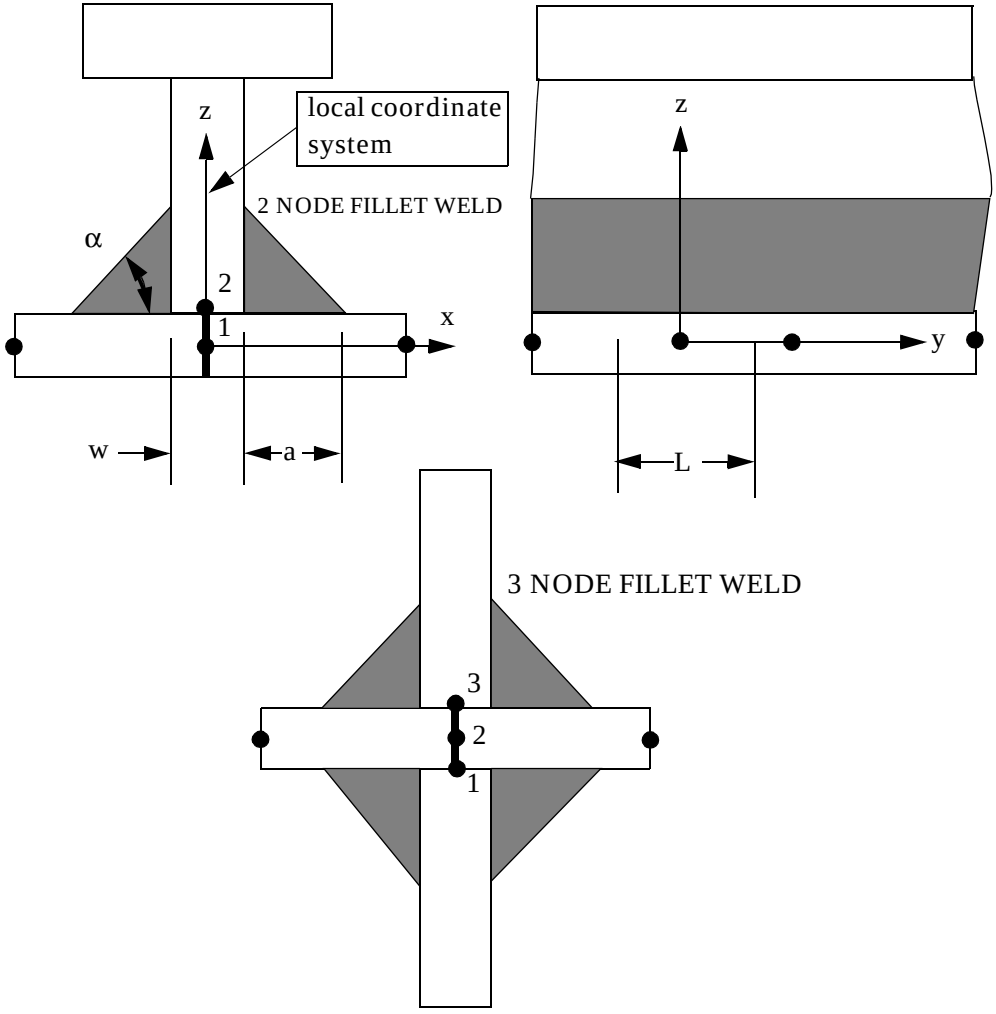


Figure 8-23

Nodal ordering and orientation of the local coordinate system is shown for fillet weld failure. The angle is defined in degrees.

CFLUIDi Fluid Element Connections

Defines three types of fluid elements for an axisymmetric fluid model.

Formats:

1	2	3	4	5	6	7	8	9	10
CFLUID2	EID	IDF1	IDF2			RHO	B		
CFLUID3	EID	IDF1	IDF2	IDF3		RHO	B		
CFLUID4	EID	IDF1	IDF2	IDF3	IDF4	RHO	B		

Examples:

CFLUID2	100	11	14			.025	0.0		
CFLUID3	110	15	13	12		1.2			
CFLUID4	120	11	15	12	14				

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
IDFi	Identification number of a RINGFL entry. (Integer > 0; DF1 ≠ IDF2 ≠ IDF3 ≠ IDF4; all IDFi < 500000)
RHO	Mass density. (Real > 0.0; Default is the value of DRHO on the AXIF entry)
B	Bulk modulus, pressure per volume ratio. (Real; Default is the value of DB on the AXIF entry)

Remarks:

1. CFLUIDi is allowed only if an AXIF entry is also present.
2. Element identification number must be unique with respect to all other fluid, scalar, and structural elements.

3. The volume defined by IDFi is a body of revolution about the polar axis of the fluid coordinate system defined by AXIF. CFLUID2 defines a thick disk with IDF1 and IDF2 defining the outer corners as shown in **Figure 8-24**:

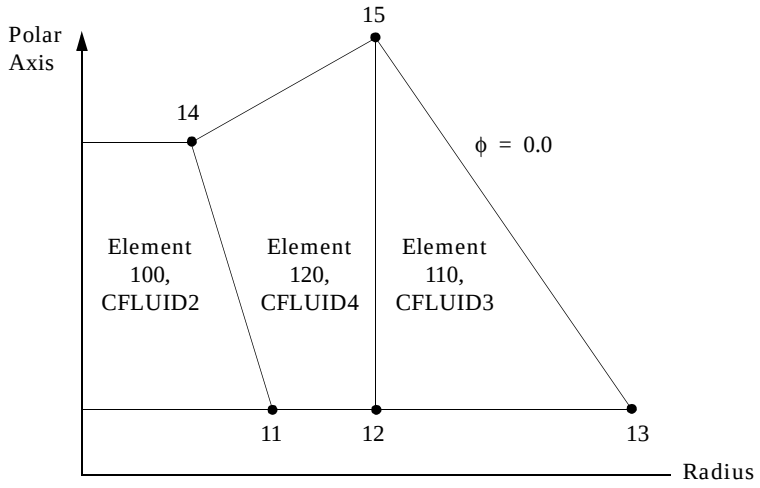


Figure 8-24 CFLUIDi Examples

4. All interior angles must be less than 180°.
5. The order of connected RINGFL points is arbitrary.
6. If B = 0.0, the fluid is incompressible.

CGAP Gap Element Connection

Defines a gap or friction element.

Format:

1	2	3	4	5	6	7	8	9	10
CGAP	EID	PID	GA	GB	X1	X2	X3	CID	

Example:

CGAP	17	2	110	112	5.2	0.3	-6.1		

Alternate Format and Example:

CGAP	EID	PID	GA	GB	GO			CID	
CGAP	17	2	110	112	13				

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PGAP entry. (Integer > 0; Default = EID)
GA, GB	Connected grid points at ends A and B. (Integers > 0; GA ≠ GB)
X1, X2, X3	Components of the orientation vector $\bar{v}$, from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply the orientation vector $\bar{v}$ using grid point GO. Direction of $\bar{v}$ is from GA to GO. (Integer > 0)
CID	Element coordinate system identification number. CID must be specified if GA and GB are coincident (distance from GA to GB $< 10^{-4}$). See Remark **6.** (Integer $\geq$ 0 or blank)

Remarks:

1. The CGAP element is intended for the nonlinear solution sequences 106, 129, 153, and 159. However, it will produce a linear stiffness matrix for the other solutions, but remains linear with the initial stiffness. The stiffness used depends on the value for the initial gap opening (U0 field in the PGAP entry).

2. The gap element coordinate system is defined by one of two following methods:
   - If the coordinate system (CID field) is specified, the element coordinate system is established using that coordinate system, in which the element x-axis is in the T1 direction and the y-axis in the T2 direction. The orientation vector $\bar{v}$ will be ignored in this case.
   - If the CID field is blank and the grid points GA and GB are not coincident (distance from A to $B \geq 10^{-4}$), then the line AB is the element x-axis and the orientation vector $\bar{v}$ lies in the x-y plane (like the CBEAM element).
3. The element coordinate system does not rotate as a result of deflections.
4. Initial gap openings are specified on the PGAP entry and not derived from the separation distance between GA and GB.
5. Forces, which are requested with the STRESS Case Control command, are output in the element coordinate system. F_x is positive for compression.
6. If CID is being used to define the element coordinate system and the CID refers to either a cylindrical or spherical coordinate system then grid GA will be used to locate the system. If grid GA lies on the z-axis of the cylindrical or spherical coordinate system it is recommended that a different coordinate system be used for this element.
7. See PARAM,CDITER for an alternative approach.

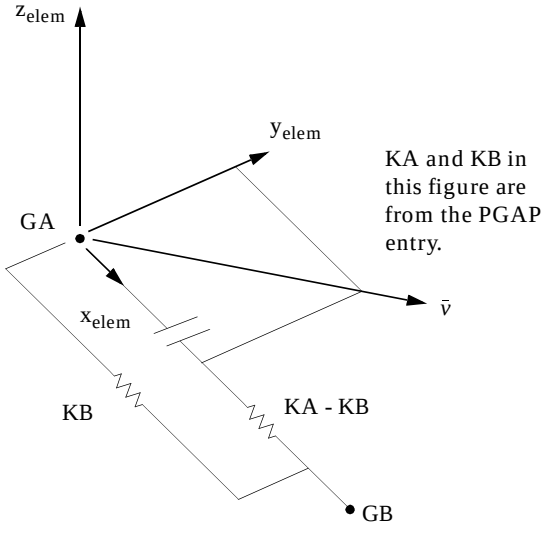


Figure 8-25 CGAP Element Coordinate System

8. Since a large stiffness is used for KA (the closed GAP stiffness), param,g damping should be avoided. Instead damping should be specified on the MATi entries and PARAM,W4 set.

CHACAB Acoustic Absorber Element Connection

Defines the acoustic absorber element in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CHACAB	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12			
			G17	G18	G19	G20			

Example:

CHACAB	95	12	1	2	5	7	8	9	
	24	23							

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PACABS entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer $\geq$ 0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. The second continuation is optional.
5. It is recommended that the edge points be located within the middle third of the edge.

6. The face consisting of grid points G1 through G4 and G9 through G12 is assumed to be in contact with the structure.

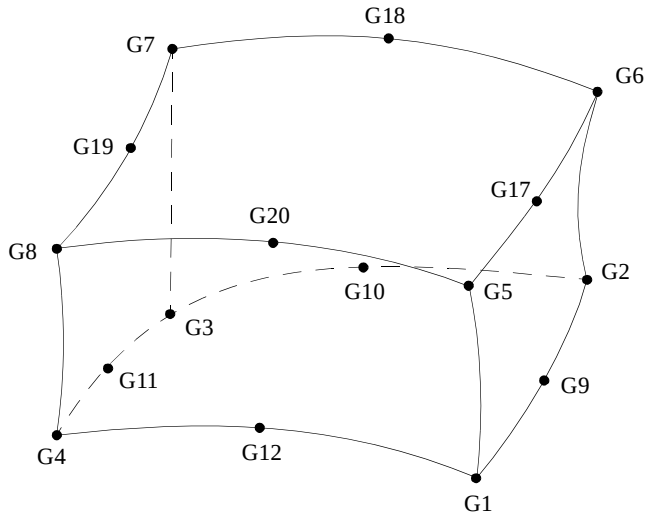


Figure 8-26 CHACAB Element Connection

7. The mass is lumped to the face formed by grid points G5 through G8 and G17 through G20 and defined to be in contact with the fluid. The opposite face has no mass contribution due to the absorber element. Also, the face in contact with the fluid has only translational stiffness in the direction normal to the face.

CHACBR Acoustic Barrier Element Connection

Defines the acoustic barrier element.

Format:

1	2	3	4	5	6	7	8	9	10
CHACBR	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12			
			G17	G18	G19	G20			

Example:

CHACBR	95	12	1	2	5	7	8	9	
	24	23							

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PACBAR entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. The second continuation is optional.
5. It is recommended that the edge points be located within the middle third of the edge.
6. The face consisting of grids G1 through G4 and G9 through G12 is assumed to be the backing that corresponds to MBACK on the PACBAR entry.

7. The face consisting of grid points G5 through G8 and G17 through G20 is assumed to be the septum that corresponds to MSEPTM on the PACBAR entry.

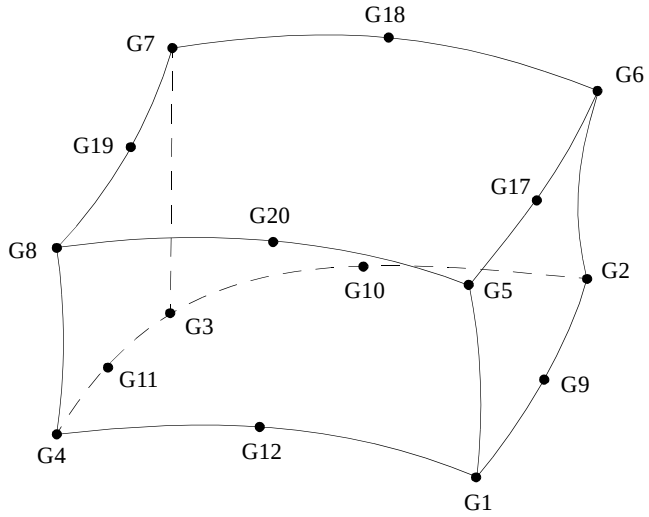


Figure 8-27 CHACBR Element Connection

8. The face in contact with the fluid is defined to be the face formed by grid points G5 through G8 and G17 through G20 and has only translational stiffness in the direction normal to the face.

CHBDYE Geometric Surface Element Definition (Element Form)

Defines a boundary condition surface element with reference to a heat conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYE	EID	EID2	SIDE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		

Example:

CHBDYE	2	10	1	3	3	2	2		

Field	Contents
EID	Surface element identification number for a specific side of a particular element. See Remarks **1.** and **9.** (Unique (0 < Integer < 100,000,000) among all elements.)
EID2	A heat conduction element identification number. (Integer > 0)
SIDE	A consistent element side identification number. See Remark **6.** (1 ≤ Integer ≤ 6)
IVIEWF	A VIEW entry identification number for the front face of surface element. See Remark **2.** for default. (Integer > 0)
IVIEWB	A VIEW entry identification number for the back face of surface element. See Remark **2.** for default. (Integer > 0)
RADMIDF	RADM identification number for front face of surface element. See Remark **2.** for default. (Integer ≥ 0)
RADMIDB	RADM identification number for back face of surface element. See Remark **2.** for default. (Integer ≥ 0)

Remarks:

1. EID is a unique elemental ID associated with a particular surface element. EID2 identifies the general heat conduction element being considered for this surface element.
2. The defaults for IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank both on the CHBDYE entry and the BDYOR entry, then the default is zero.

3. For the front face of shell elements, the right-hand rule is used as one progresses around the element surface from G1 to G2 to ... Gn. For the edges of shell elements or the ends of line elements, an outward normal is used to define the front surface.
4. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
5. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.
6. Side conventions for solid elements.

   The sides of the solid elements are numbered consecutive+ly according to the order of the grid point numbers on the solid element entry. The sides of solid elements are either quadrilaterals or triangles. For each element type, tabulate the grid points (gp) at the corners of each side.

8-node or 20-node CHEXA

side	gp	gp	gp	gp
1	4	3	2	1
2	1	2	6	5
3	2	3	7	6
4	3	4	8	7
5	4	1	5	8
6	5	6	7	8

CPENTA

side	gp	gp	gp	gp
1	3	2	1	
2	1	2	5	4
3	2	3	6	5
4	3	1	4	6
5	4	5	6	

CTETRA

side	gp	gp	gp
1	3	2	1
2	1	2	4
3	2	3	4
4	3	1	4

7. Side conventions for shell elements.

   Side 1 of shell elements (top) are of an AREA type, and additional sides (2 through a maximum of 5 for a QUAD) are of LINE type. (See "**CHBDYG**" on page 1227 for surface type definition.)

   Area Type Sides -- The first side is that given by the right-hand rule on the shell elements grid points.

   Line Type Sides -- The second side (first line) proceeds from grid point 1 to grid point 2 of the shell element, and the remaining lines are numbered consecutively. The thickness of the line is that of the shell element, and the normal to the line is outward from the shell element in the plane of the shell. Note that any midside nodes are ignored in this specification.

8. Side conventions for line elements.

   LINE elements have one linear side (side 1) with geometry that is the same as that of the element and two POINT-type sides corresponding to the two points bounding the linear element (first grid point-side 2; second grid point-side 3).

   The TUBE-type element has two linear sides of type TUBE. The first side represents the outside with diameters equal to that of the outside of the tube. The second side represents the inside with diameters equal to that of the inside of the tube.

   Point Sides -- Point sides may be used with any linear element. The direction of the outward normals of these points is in line with the element axis, but pointing away from the element. The area assigned to these POINT-type sides is consistent with the element geometry.

   Rev Sides -- The CTRIAX6 element has associated with it three REV sides. The first side is associated with Grid Points G1, G2, and G3. The positive face identification normals point away from the element.

9. Application of boundary conditions to CHBDYE is referenced through the EID. Boundary conditions can reference either the front or back face of the CHBDYE by specifying +EID or -EID respectively. Correspondingly, the back face is minus the normal vector of the front face. Similarly, IVIEWF and RADMIDF are associated with +EID and IVIEWB and RADMIDB with -EID. For radiation problems, if the RADMIDF or RADMIDB is zero, default radiant properties assume perfect black body behavior.

CHBDYG Geometric Surface Element Definition (Grid Form)

Defines a boundary condition surface element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		
	G1	G2	G3	G4	G5	G6	G7	G8	

Example:

CHBDYG	2		AREA4	3	3	2	2		
	100	103	102	101					

Field	Contents
EID	Surface element identification number. (Unique (0 < Integer < 100,000,000) among all elemental entries)
TYPE	Surface type. See Remark **3.** (Character)
IVIEWF	A VIEW entry identification number for the front face. See Remark **2.** for default. (Integer > 0)
IVIEWB	A VIEW entry identification number for the back face. See Remark **2.** for default. (Integer > 0)
RADMIDF	RADM identification number for front face of surface element. See Remark **2.** for default. (Integer > 0)
RADMIDB	RADM identification number for back face of surface element. See Remark **2.** for default. (Integer $\geq$ 0)
Gi	Grid point IDs of grids bounding the surface. (Integer > 0)

Remarks:

1. EID is a unique ID associated with a particular surface element as defined by the grid points.
2. The defaults for TYPE, IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYG entry and the BDYOR entry, then the default is zero.
3. TYPE specifies the kind of element surface; allowed types are: REV, AREA3, AREA4, AREA6, and AREA8. See **Figure 8-28**, **Figure 8-29**, and **Figure 8-30**.

- TYPE = REV

  The “REV” type has two primary grid points that must lie in the x-z plane of the basic coordinate system with x>0. A midside grid point G3 is optional and supports convection or heat flux from the edge of the six-noded CTRIAX6 element. The defined area is a conical section with z as the axis of symmetry. A property entry is required for convection, radiation, or thermal vector flux. Automatic view factor calculations with VIEW data are not supported for the REV option.

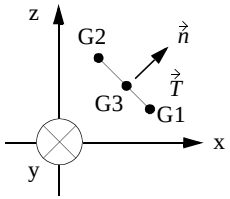


Figure 8-28 Normal Vector for CHBDYG Element of Type “REV”

The unit normal lies in the x-z plane, and is given by

$$\vec{n} = (\vec{e}_y \times \vec{T}) / \left|\vec{e}_y \times \vec{T}\right| .$$

$\vec{e}_y$ is the unit vector in the y direction.

- TYPE = AREA3, AREA4, AREA6, or AREA8

  These types have three and four primary grid points, respectively, that define a triangular or quadrilateral surface and must be ordered to go around the boundary. A property entry is required for convection, radiation, or thermal vector flux.

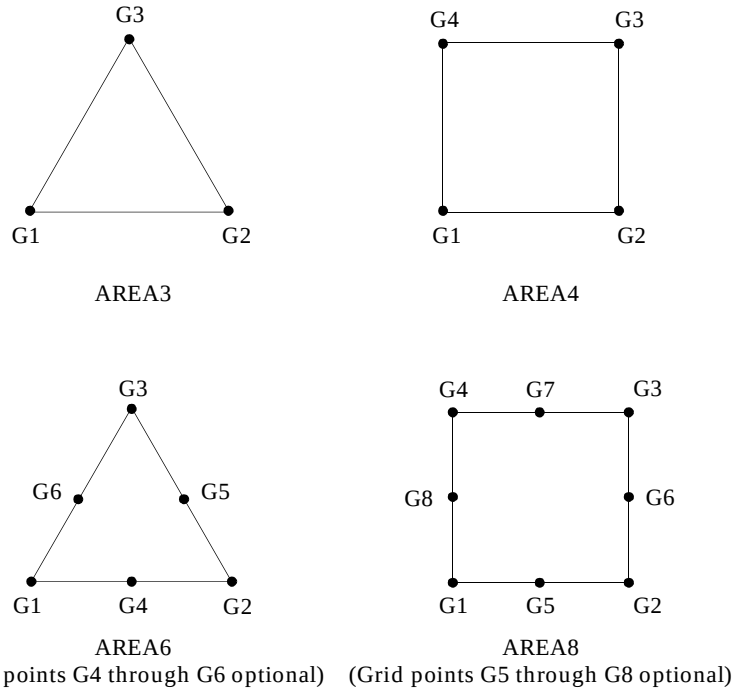


Figure 8-29 TYPE Examples

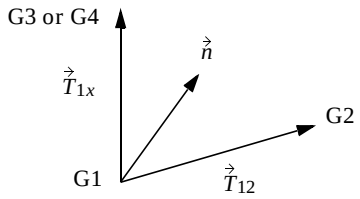


Figure 8-30 Normal Vector for CHBDYG Element of Types "AREAi"

The unit normal vector is given by

$$\vec{n} = \frac{(\vec{T}_{12} \times \vec{T}_{1x})}{\left|\vec{T}_{12} \times \vec{T}_{1x}\right|}$$

(G3 is used for triangles, and G4 is used for quadrilaterals.)

4. For defining the front face, the right-hand rule is used on the sequence G1 to G2 to ... Gn of grid points.

5. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
6. All conduction elements to which any boundary condition is to be applied must be individually identified with one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.
7. See Remark **9.** of CHBDYE for application of boundary conditions using CHBDYG entries and a discussion of front and back faces.

CHBDYP Geometric Surface Element Definition (Property Form)

Defines a boundary condition surface element with reference to a PHBDY entry.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYP	EID	PID	TYPE	IVIEWF	IVIEWB	G1	G2	G0	
	RADMIDF	RADMIDB	GMID	CE	E1	E2	E3		

Example:

CHBDYP	2	5	POINT	2	2	101		500	
	3	3			0.0	0.0	1.0		

Field	Contents
EID	Surface element identification number. (Unique (0 < Integer < 100,000,000) among all element identification numbers.)
PID	PHBDY property entry identification numbers. (Integer > 0)
TYPE	Surface type. See Remark **3.** (Character)
IVIEWF	VIEW entry identification number for the front face. (Integer > 0 or blank)
IVIEWB	VIEW entry identification number for the back face. (Integer > 0 or blank)
G1, G2	Grid point identification numbers of grids bounding the surface. (Integer > 0)
GO	Orientation grid point. (Integer ≥ 0; Default = 0)
RADMIDF	RADM entry identification number for front face. (Integer ≥ 0 or blank)
RADMIDB	RADM entry identification number for back face. (Integer ≥ 0 or blank)
GMID	Grid point identification number of a midside node if it is used with the line type surface element.
CE	Coordinate system for defining orientation vector. (Integer ≥ 0; Default = 0)
Ei	Components of the orientation vector in coordinate system CE. The origin of the orientation vector is grid point G1. (Real or blank)

Remarks:

1. EID is a unique ID associated with a particular surface element as defined by the grid point(s).
2. The defaults for PID, TYPE, IVIEWF, IVIEWB, GO, RADMIDF, RADMIDB, CE, and Ei may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYP entry and the BDYOR entry, then the default is zero.
3. TYPE specifies the kind of element surface; the allowed types are: “POINT,” “LINE,” “ELCYL,” “FTUBE,” and “TUBE.” For TYPE = “FTUBE” and TYPE = “TUBE,” the geometric orientation is completely determined by G1 and G2; the GO, CE, E1, E2, and E3 fields are ignored.

   - TYPE = “POINT”

     TYPE = “POINT” has one primary grid point, requires a property entry, and the normal vector Vi must be specified if thermal flux is to be used.

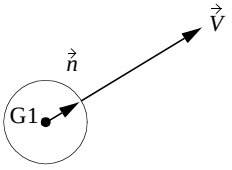


Figure 8-31 Normal Vector for CHBDYP Element of Type “POINT” (See Remarks 4. and 5.)

     The unit normal vector is given by $\vec{n} = \vec{V}/|\vec{V}|$ where $\vec{V}$ is specified in the Ei field and given in the basic system at the referenced grid point. See Remarks **4.** and **5.** for the determination of $\vec{V}$.

   - TYPE = “LINE,” “FTUBE,” or “TUBE”

     The TYPE = “LINE” type has two primary grid points, requires a property entry, and the vector is required. TYPE = “FTUBE” and TYPE = “TUBE” are similar to TYPE = “LINE” except they can have linear taper with no automatic view factor calculations. GMID is an option for the TYPE = “LINE” surface element only and is ignored for TYPE = “FTUBE” and “TUBE”.

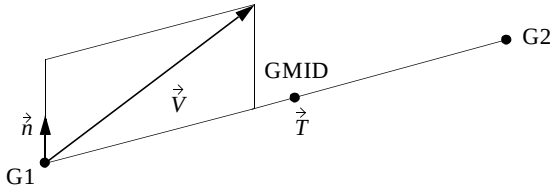


Figure 8-32 Normal Vector for CHBDYP Element with TYPE="LINE", TYPE="FTUBE", or TYPE="TUBE" (See Remarks 4. and 5.)

The unit normal lies in the plane $\vec{V}$ and $\vec{T}$, is perpendicular to $\vec{T}$, and is given by:

$$\vec{n} = \frac{\vec{T} \times (\vec{V} \times \vec{T})}{\left|\vec{T} \times (\vec{V} \times \vec{T})\right|}$$

- TYPE = "ELCYL"

  TYPE = "ELCYL" (elliptic cylinder) has two connected primary grid points and requires a property entry. The vector must be nonzero. Automatic view factor calculations are not available.

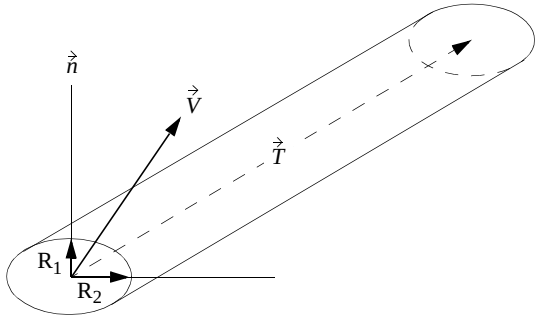


Figure 8-33 Normal Vector for CHBDYP Element of TYPE="ELCYL" (See Remarks 4. and 5.)

The same logic is used to determine $\vec{n}$ as for TYPE = LINE. The "radius" R_1 is in the $\vec{n}$ direction, and R2 is the perpendicular to $\vec{n}$ and $\vec{T}$ (see fields 7 and 8 of PHBDY entry).

4. For TYPE = “POINT,” TYPE = “LINE,” and TYPE = “ELCYL,” geometric orientation is required. The required information is sought in the following order:
    - If GO > 0 is found on the CHBDYP entry, it is used.
    - Otherwise, if a nonblank CE is found on the CHBDYP continuation entry, this CE and the corresponding vectors E1, E2, and E3 are used.
    - If neither of the above, the same information is sought in the same way from the BDYOR entry.
    - If none of the above apply, a warning message is issued.
5. The geometric orientation can be defined by either GO or the vector E1, E2, E3.
    - If GO > zero:

      For a TYPE = “POINT” surface, the normal to the front face is the vector from G1 to GO. For the TYPE = “LINE” surface, the plane passes through G1, G2, GO and the right-hand rule is used on this sequence to get the normal to the front face. For TYPE = “ELCYL” surface the first axis of the ellipse lies on the G1, G2, GO plane, and the second axis is normal to this plane. For TYPE = “FTUBE” or “TUBE” surface, no orientation is required, and GO is superfluous.
    - If GO is zero:

      For a TYPE = “POINT” surface, the normal to the front face is the orientation vector. For the TYPE = “LINE” surface, the plane passes through G1, G2, and the orientation vector; the front face is based on the right-hand rule for the vectors G2-G1 and the orientation vector. For TYPE = “ELCYL” surface, the first axis of the ellipse lies on the G1, G2, orientation vector plane, and the second axis is normal to this plane.
6. The continuation entry is optional.
7. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
8. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP entries.

CHEXA Six-Sided Solid Element Connection

Defines the connections of the six-sided solid element with eight to twenty grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

Example:

CHEXA	71	4	3	4	5	6	7	8	
	9	10	0	0	30	31	53	54	
	55	56	57	58	59	60			

Field	Contents	Type	Default
EID	Element identification number.	(0 < Integer < 100,000,000)	Required
PID	Property identification number of a PSOLID or PLSOLID entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer $\geq$ 0 or blank	Required

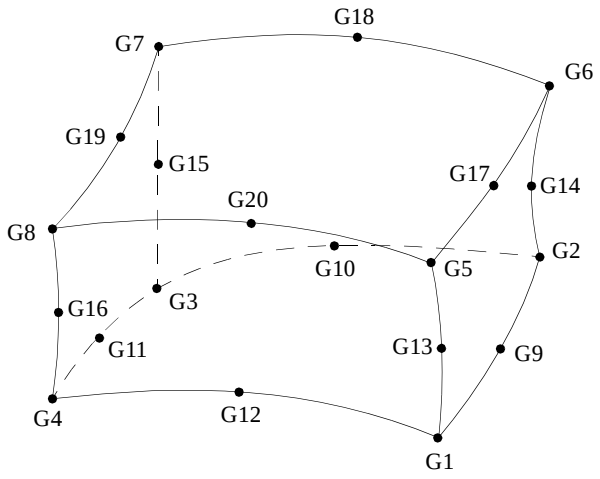


Figure 8-34 CHEXA Element Connection

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the input example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element (with shear correction) in all cases.
4. Components of stress are output in the material coordinate system except for hyperelastic elements, which are output in the basic coordinate system. The material coordinate system is defined on the PSOLID entry.
5. The second continuation is optional.
6. For nonhyperelastic elements, the element coordinate system for the CHEXA element is defined in terms of the three vectors R, S, and T, which join the centroids of opposite faces.

   R vector joins the centroids of faces G4-G1-G5-G8 and G3-G2-G6-G7.

   S vector joins the centroids of faces G1-G2-G6-G5 and G4-G3-G7-G8.

   T vector joins the centroids of faces G1-G2-G3-G4 and G5-G6-G7-G8.

   The origin of the coordinate system is located at the intersection of these vectors. The X, Y, and Z axes of the element coordinate system are chosen as close as possible to the R, S, and T vectors and point in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that if the R, S, and T vectors are described in the element coordinate system a 3 x 3 positive-definite symmetric matrix would be produced.)

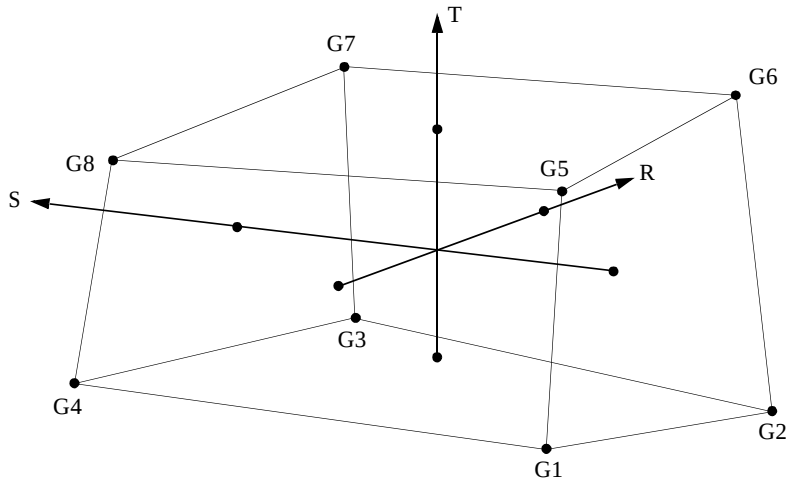


Figure 8-35 CHEXA Element R, S, and T Vectors

7. It is recommended that the edge points be located within the middle third of the edge.
8. For hyperelastic elements, the plot codes are specified under the CHEXAFD element name in "**Item Codes**" on page 873.
9. If a CHEXA element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
    - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.
    - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.
10. By default, all of the twelve edges of the element are considered straight unless:
    - For p-elements there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.
    - For h-elements, any of G9 through G20 are specified.

CINTC Line Interface Element Connection

Defines a line interface element with specified boundaries.

Format:

1	2	3	4	5	6	7	8	9	10
CINTC	EID	TYPE							
	LIST = (BID1(INTP1), BID2(INTP2),...,BIDn(INTPn))								

Example:

CINTC	1001	GRIDLIST							
	LIST=(101,102(Q),-103(Q),104(L))								

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
TYPE	Connectivity. If TYPE = “GRDLIST” or blank (default), the user will specify the boundaries via Bulk Data entry, GMBNDC. See Remark 2. (Character; Default = “GRDLIST”)
BIDi	Boundary curve identification number, referenced to Bulk Data entry, GMBNDC. See Remark 2. (Integer ≠ 0)
INTPi	Interpolation scheme. (Character; Default = “L”) INTP = “L”: Linear interpolation; INTP = “Q”: Quadratic interpolation.

Remarks:

1. Line interface element identification numbers must be unique with respect to all other line interface elements.
2. There must be at least two BIDi specified. If all BIDi are positive, by default, the degrees of freedom associated with the grids on the boundary represented by the first BID will be taken as the independent (n-set), and the degrees of freedom with the grids on the rest of boundaries are taken as the dependent (m-set). If there is a single negative BID, the degrees of freedom associated with the grids on the boundary represented by this BID will be

taken as the independent (n-set), and the rest of the degrees of freedom with other boundaries are used as the dependent (m-set). If there are two or more negative BIDs, the degrees of freedom with the first negative one will be taken as the independent.

3. Forces of multipoint constraints may be recovered with the MPCFORCE Case Control command.
4. The m-set degrees of freedom specified on the boundary grids by this entry may not be specified by other entries that define mutually exclusive sets.

CLOAD Static Load Combination for Superelement Loads (Superposition)

Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOLs 106 or 153).

Format:

1	2	3	4	5	6	7	8	9	10
CLOAD	CID	S	S1	IDV1	S2	IDV2	S3	IDV3	
	S4	IDV4	-etc.-						

Example:

CLOAD	25	1.0	25.0	10	-1.0	101	2.2-1	604	
	-62.0	62							

Field	Contents
CID	Combination identification number. (Integer > 0)
S	Scale factor. (Real)
Si	Scale factors. (Real)
IDVi	Identification numbers of load vectors (EXCITEID of a selected LSEQ entry) calculated for a superelement loads entry. (Integer > 0)

Remarks:

1. The CLOAD entry must be selected in the residual solution subcases of the Case Control with CLOAD = CID and must be used if loads are applied to upstream superelements in SOL 106 or 153.
2. The load vector defined is given by $\{P\} = S\sum_{i} Si\{P_{\text{IDVi}}\}$
3. The IDVi field refers to a previously calculated load vector for the superelement via the LSEQ approach. That is, a LOADSET keyword must have been selected in Case Control that in turn refers to one or more LSEQ entries in the Bulk Data Section. The IDVi refers to the EXCITEID of such LSEQ entries. For more details, see the the Case Control commands "**LSEQ**" on page 1653 Bulk Data entry and the "**LOADSET**" on page 351.
4. In the CID or IDV fields, a CLOAD entry may not reference an identification number defined by another CLOAD entry.

CMASS1 Scalar Mass Connection

Defines a scalar mass element.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS1	EID	PID	G1	C1	G2	C2			

Example:

CMASS1	32	6	2	1					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PMASS entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid or scalar point identification number. (Integer $\geq$ 0)
C1, C2	Component number. (0 $\leq$ Integer $\leq$ 6; blank or zero if scalar point)

Remarks:

1. Scalar points may be used for G1 and / or G2, in which case the corresponding C1 and / or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and / or ground are involved, it is more efficient to use the CMASS3 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must not be coincident.
4. For a discussion of the scalar elements, see the *MSC.Nastran Reference Manual*, Section 5.6.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.
6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

7. Scalar elements input coupled mass matrices when the second pair of fields is entered. When uncoupled point masses are desired input only the first pair of fields. When a coupled mass matrix is requested the submatrix added has M on the diagonal, and -M on the off-diagonal. The element is not checked for internal constraints, which is the user's responsibility if desired. There are instances where elements with internal constraints are desired, although not frequently. To identify the presence of internal constraints caused by coupled mass, inspect GPWG output, OLOAD output due to GRAV loads, and rigid body modes of free structures. Some forms of coupled mass will cause coupling of rigid body translational mass terms in GPWG output, and poor rigid body modes in modal analysis.

CMASS2 Scalar Mass Property and Connection

Defines a scalar mass element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS2	EID	M	G1	C1	G2	C2			

Example:

CMASS2	32	9.25	6	1					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
M	Value of the scalar mass. (Real)
G1, G2	Geometric grid or scalar point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; blank or zero if scalar point)

Remarks:

1. Scalar points may be used for G1 and / or G2, in which case the corresponding C1 and / or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and / or ground are involved, it is more efficient to use the CMASS4 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct. Except in unusual circumstances, one of them will be a grounded terminal with blank entries for Gi and Ci.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide.*
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

6. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. See Remark 7 for "**CMASS1**" on page 1241.

CMASS3 Scalar Mass Connection to Scalar Points Only

Defines a scalar mass element that is connected only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS3	EID	PID	S1	S2					

Example:

CMASS3	13	42	62						

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PMASS entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer $\geq$ 0; S1 $\neq$ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

CMASS4 Scalar Mass Property and Connection to Scalar Points Only

Defines a scalar mass element that is connected only to scalar points, without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS4	EID	M	S1	S2					

Example:

CMASS4	23	14.92		23					

Field	Contents
EID	Unique element identification number. (0 < Integer < 100,000,000)
M	Scalar mass value. (Real)
S1, S2	Scalar point identification numbers. (Integer $\geq$ 0; S1 $\neq$ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate. This is the usual case.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.
4. For a discussion of the scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

COMBWLD (SOL 700) Complex Combined Weld in the LS-DYNA Style for SOL 700 Only

Defines a complex or combined weld in the LS-DYNA style for use in SOL 700 only. Replaces CWELD for SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
COMBWLD	EID	NSID		FILTER	WINDOW	NPR	NPRT	M32	
	TFAIL	EPSF	SIG	BETA	L	W	A	ALPHA	
	GA	GB	NCID	WTYP					

The second and third entries are repeated NPR times (as many welds as are in the combined weld).

Example:

COMBWLD	1002	11		0	0	12	1		
			44000.	2.5	4.0	3.0	2.0	22.5	
	1520	2520		0					
			42000.	2.5	2.0	1.25	0.75	-22.5	
	101	201	1	1					

Field	Contents
EID	Unique element identification number. (Integer > 0, Required, No Default)
NSID	ID of a set number containing the grid points comprising this weld. (Integer > 0, Required, no Default)
CID	ID of a CORDi entry providing the local output coordinate system for this weld. (Integer $\geq$ 0 or blank, blank is the same as zero indicating the basic coordinate system)
Filter	Number of force vectors saved for filtering. This option can eliminate spurious failures due to numerical force spikes but memory will be larger if this option is envoked. Enter 0 for no filtering and N for a simple average of force components divided by N or the maximum number of force vectors that are stored for the time window option WINDOW (Integer > 0, Default = 0)

Field	Contents
WINDOW	Time window for filtering (Real, Default = 0.0 for no filtering)
NPR	Number of individual nodal pairs in this weld (Integer > 0, Required, no Default)
NPRT	Control of weld force output in file RBDOUT (Integer > 0, Default = 1) NPRT=1 data is output NPRT=2 data is not output
TFAIL	Failure time for this weld (Real > 0 or blank, Default = 1.0E20)
EPSF	Effective plastic strain at failure (Real > 0 or blank, Default = blank which means not used) – Used for ductile failures.
SIG	Stress at failure (Real > 0 or blank, Default = blank which means not used) – Used for brittle failures.
BETA	Failure parameter for brittle failure (Real > 0 or blank, Default = blank which means not used) – Used for brittle failures.
L	Length of weld (Real > 0, Required, no Default)
W	Width, W, of flange, see figure (Real > 0, Required, no Default)
A	Width, A, of flange, see figure (Real > 0, Required, no Default)
ALPHA	Weld angle, see figure (Real > 0, Required, no Default)
GA	Grid A ID, see figure (Integer > 0, Required, no Default)
GB	Grid B ID, see figure (Integer > 0, Required, no Default)
NCID	Local coordinate system of weld (Integer $\geq$ 0 or blank - blank is the same as zero indicating the basic coordinate system)
WTYPE	Weld pair type – see figure (Integer > 0, Required, no Default) 0 = fillet weld 1 = butt weld

Remarks:

1. No property entry is needed for the CFILLET entry.
2. The 2nd and 3rd entries are repeated for each portion of the combined weld

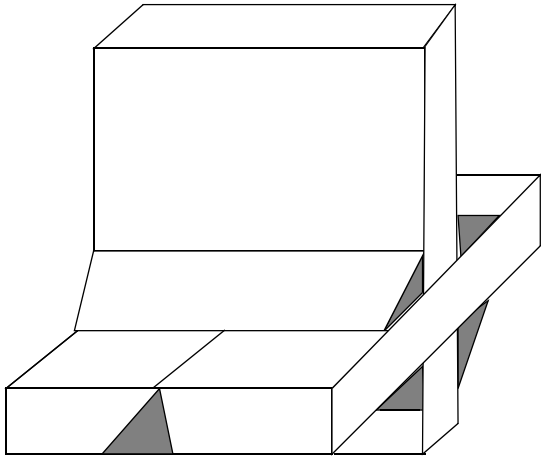


Figure 8-36 A Combined Weld is a Mixture of Fillet and Butt Welds.

CONM1 Concentrated Mass Element Connection, General Form

Defines a 6 x 6 symmetric mass matrix at a geometric grid point.

Format:

1	2	3	4	5	6	7	8	9	10
CONM1	EID	G	CID	M11	M21	M22	M31	M32	
	M33	M41	M42	M43	M44	M51	M52	M53	
	M54	M55	M61	M62	M63	M64	M65	M66	

Example:

CONM1	2	22	2	2.9	6.3				
	4.8	28.6							
		28.6						28.6	

Field	Contents
EID	Unique element identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number for the mass matrix. (Integer $\geq$ 0)
Mij	Mass matrix values. (Real)

Remarks:

1. For a less general means of defining concentrated mass at grid points, see the CONM2 entry description.
2. Element identification numbers should be unique with respect to all other element identification numbers.

CONM2 Concentrated Mass Element Connection, Rigid Body Form

Defines a concentrated mass at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
CONM2	EID	G	CID	M	X1	X2	X3		
	I11	I21	I22	I31	I32	I33			

Example:

CONM2	2	15	6	49.7					
	16.2		16.2			7.8			

Field	Contents
EID	Element identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. For CID of -1; see X1, X2, X3 below. (Integer $\geq$ -1; Default = 0)
M	Mass value. (Real)
X1, X2, X3	Offset distances from the grid point to the center of gravity of the mass in the coordinate system defined in field 4, unless CID = -1, in which case X1, X2, X3 are the coordinates, not offsets, of the center of gravity of the mass in the basic coordinate system. (Real)
lij	Mass moments of inertia measured at the mass center of gravity in the coordinate system defined by field 4. If CID = -1, the basic coordinate system is implied. (For I11, I22, and I33; Real $\geq$ 0.0; for I21, I31, and I32; Real)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For a more general means of defining concentrated mass at grid points, see the CONM1 entry description.
3. The continuation is optional.

4. If CID = -1, offsets are internally computed as the difference between the grid point location and X1, X2, X3. The grid point locations may be defined in a nonbasic coordinate system. In this case, the values of Iij must be in a coordinate system that parallels the basic coordinate system.
5. The form of the inertia matrix about its center of gravity is taken as:

$$\begin{bmatrix} M & & & & & \\ & M & & & \text{symmetric} & \\ & & M & & & \\ & & & I11 & & \\ & & & -I21 & I22 & \\ & & & -I31 & -I32 & I33 \end{bmatrix}$$

where

$$M = \int \rho dV$$

$$I11 = \int \rho (x_2^2 + x_3^2) dV$$

$$I22 = \int \rho (x_1^2 + x_3^2) dV$$

$$I33 = \int \rho (x_1^2 + x_2^2) dV$$

$$I21 = \int \rho x_1 x_2 dV$$

$$I31 = \int \rho x_1 x_3 dV$$

$$I32 = \int \rho x_2 x_3 dV$$

and x_1, x_2, x_3 are components of distance from the center of gravity in the coordinate system defined in field 4. The negative signs for the off-diagonal terms are supplied automatically. A warning message is issued if the inertia matrix is nonpositive definite, since this may cause fatal errors in dynamic analysis modules.

6. If CID $\geq$ 0, then X1, X2, and X3 are defined by a local Cartesian system, even if CID references a spherical or cylindrical coordinate system. This is similar to the manner in which displacement coordinate systems are defined.
7. See "**Grid Point and Coordinate System Definition**" on page 41 of the *MSC.Nastran Reference Guide* for a definition of coordinate system terminology.

CONROD Rod Element Property and Connection

Defines a rod element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	

Example:

CONROD	2	16	17	4	2.69				

Field	Contents
EID	Unique element identification number. (Integer > 0)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)
MID	Material identification number. (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)
C	Coefficient for torsional stress determination. (Real)
NSM	Nonstructural mass per unit length. (Real)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For structural problems, MID must reference a MAT1 material entry.
3. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.

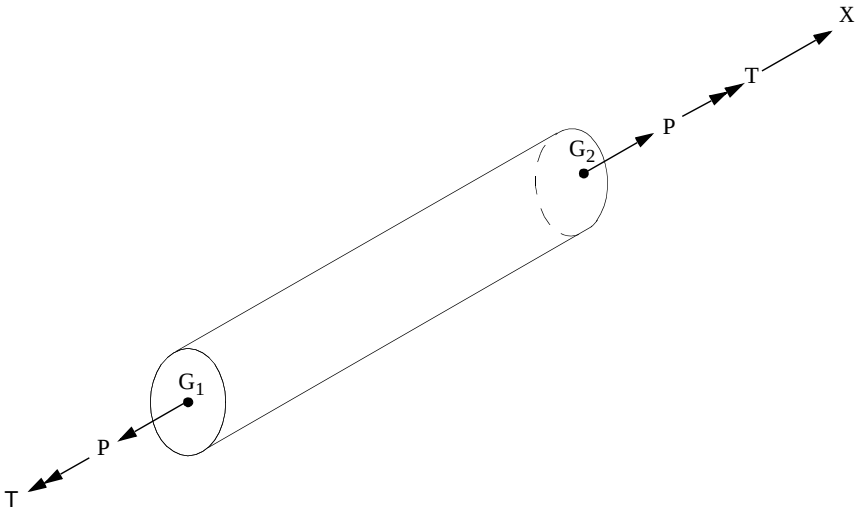


Figure 8-37 CONROD Element Forces and Moments

CONTRLT Thermal Control ELement for Heat Transfer Analysis

Defines the control mechanism for QVECT, QVOL, QBCY3, in heat transfer analysis (SOL 159).

Format:

1	2	3	4	5	6	7	8	9	10
CONTRLT	ID	Sensor	SFORM	CTYPE	Pl	Ph	PTYPE	PZERO	
	DT	Delay	TAUc	TA8					

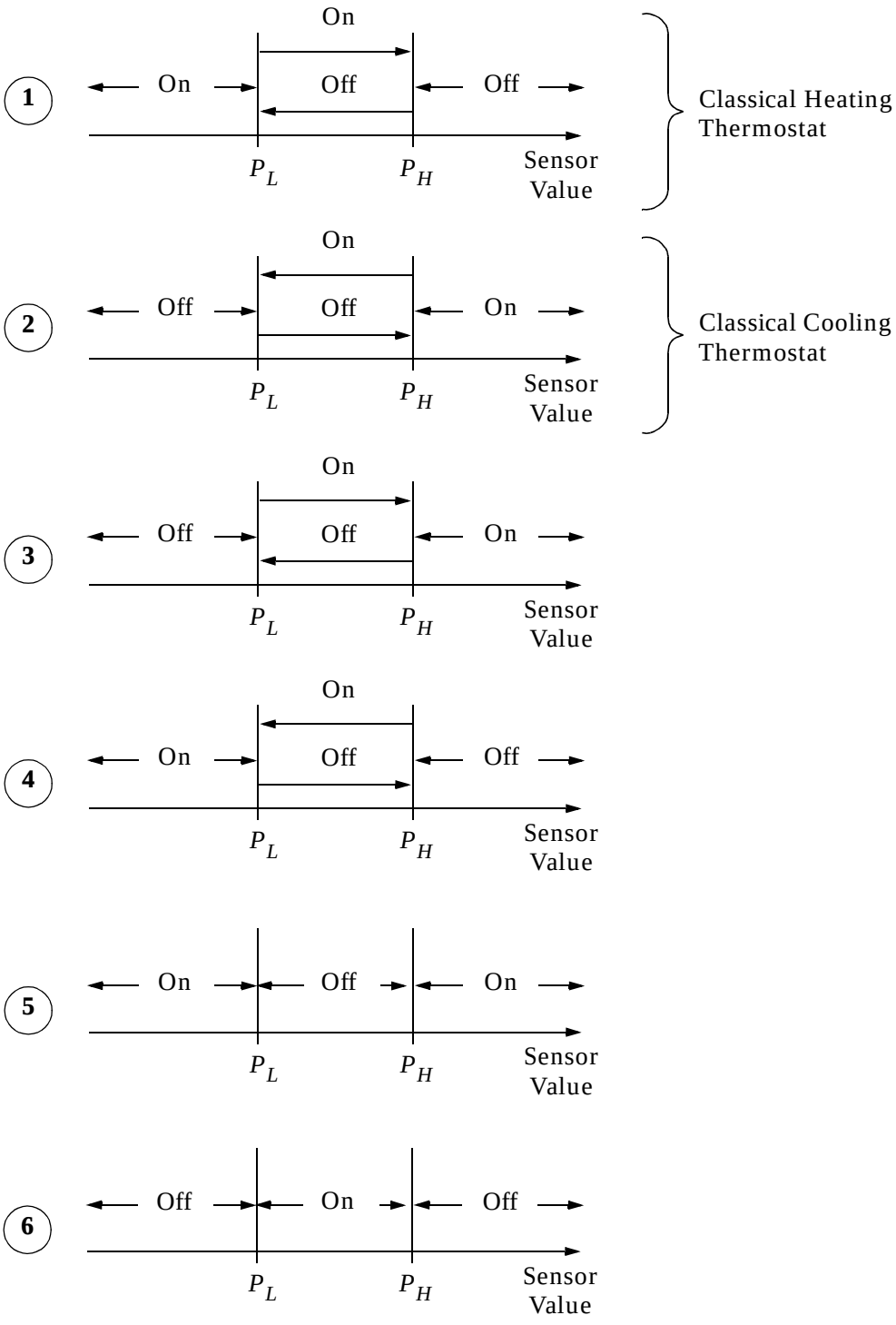
Example:

CONTRLT	100	20	3	68.	73.	1	0.		

Field	Contents
ID	Control node ID as well as CONTROLT ID. See Remark **1.** (Integer > 0; no Default)
Sensor	Grid or scalar point ID of the sensor. See Remark **2.** (Integer > 0; no Default)
SFORM	Sensor output form. See Remark **3.** (Character, T; default = T)
CTYPE	Control type. See Remark **4.** (Character, TSTAT for thermostat; Default = TSTAT)
Pl, Ph	Lower and upper limit value for desired temperature in the thermostat. See Remark **5.** (Real; no default)
PTYPE	Process type. See Remark **5.** (Integer value 1 through 6; no Default)
PZERO	Initial controller value. See Remark **4.** (0. < Real < 1.; Default = 0)
DT	Monitoring time interval, or sampling period. (Real > 0.; Default = 0)
Delay	Time delay after the switch is triggered or time for delayed control action in PID control. (Real < 0.; Default = 0)
TAUc	Decay time constant for actuator response. (Real > 0.; Default = 0)

Remarks:

1. The CONTRLT ID is referenced by CNTRLND entry identified on any of the QVECT, QVOL, QBDY3, Bulk Data entries. If any grid or scalar point ID is the same as the CONTRLT ID, then the combined logic associated with the controller and the control node will be in force for the LBC referenced. Any number of CONTRLT statements may exist in a single model.
2. Sensor point, where a feedback temperature or rate of change of temperature is measured. May be a dependent DOF in a MPC relationship.
3. Sensor output may only be temperature (T)
4. Control type can only be TSTAT. The PZERO field cannot have any other value but 0.0 or 1.0.
5. The upper and lower limit values (Pl and Ph) define a dead band for a thermostat. The available thermostat controller (TSTAT) formats are (PTYPE = 1 through 6).


1
On
On
Off
Off
P_L
P_H
Sensor Value
Classical Heating Thermostat
2
On
Off
Off
On
P_L
P_H
Sensor Value
Classical Cooling Thermostat
3
On
Off
Off
On
P_L
P_H
Sensor Value
4
On
On
Off
Off
P_L
P_H
Sensor Value
5
On
Off
On
P_L
P_H
Sensor Value
6
Off
On
Off
P_L
P_H
Sensor Value

CONV Heat Boundary Element Free Convection Entry

Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

Format:

1	2	3	4	5	6	7	8	9	10
CONV	EID	PCONID	FLMND	CNTRLND	TA1	TA2	TA3	TA4	
	TA5	TA6	TA7	TA8					

Example:

CONV	2	101	3	201	301				

Field	Contents
EID	CHBDYG, CHBDYE, or CHBDYP surface element identification number. (Integer > 0)
PCONID	Convection property identification number of a PCONV entry. (Integer > 0)
FLMND	Point for film convection fluid property temperature. (Integer $\geq$ 0; Default = 0)
CNTRLND	Control point for free convection boundary condition. (Integer $\geq$ 0; Default = 0)
TAi	Ambient points used for convection. (Integer > 0 for TA1 and Integer $\geq$ 0 for TA2 through TA8; Default for TA2 through TA8 is TA1.)

Remarks:

1. The basic exchange relationship can be expressed in one of the following forms:
   - $q = H \cdot (T - \text{TAMB})^{\text{EXPF}}(T - \text{TAMB})$, CNTRLND = 0
   - $q = (H \cdot u_{\text{CNTRLND}})(T - \text{TAMB})^{\text{EXPF}}(T - \text{TAMB})$, CNTRLND $\neq$ 0
   - $q = H(T^{\text{EXPF}} - \text{TAMB}^{\text{EXPF}})$, CNTRLND = 0

- $q = (H \cdot u_{\text{CNTRLND}})(T^{\text{EXPF}} - \text{TAMB}^{\text{EXPF}})$, CNTRLND ≠ 0

  EXPF is specified on the PCONV entry.

  (See "**PCONV**" on page 2191 entry for additional clarification of forms.)

2. The continuation entry is not required.
3. CONV is used with an CHBDYi (CHBDYG, CHBDYE, or CHBDYP) entry having the same EID.
4. The temperature of the film convection point provides the look up temperature to determine the convection film coefficient. If FLMND=0, the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperature, or the ambient temperature, as defined in the FORM field of the PCONV Bulk Data entry.
5. If only one ambient point is specified then all the ambient points are assumed to have the same temperature. If midside ambient points are missing, the temperature of these points is assumed to be the average of the connecting corner points.
6. See the Bulk Data entry, "**PCONV**" on page 2191, for an explanation of the mathematical relationships involved in free convection and the reference temperature for convection film coefficient.

CONVM Heat Boundary Element Forced Convection Entry

Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

Format:

1	2	3	4	5	6	7	8	9	10
CONVM	EID	PCONID	FLMND	CNTMDOT	TA1	TA2			

Example:

CONVM	101	1	201	301	20	21			

Field	Contents
EID	CHBDYP element identification number. (Integer > 0)
PCONID	Convection property identification number of a PCONVM entry. (Integer > 0)
FLMND	Point used for fluid film temperature. (Integer $\geq$ 0; Default = 0)
CNTMDOT	Control point used for controlling mass flow. (Integer > 0)
TA1, TA2	Ambient points used for convection. (Integer > 0 for TA1 and Integer $\geq$ 0 for TA2; Default for TA2 is TA1.)

Remarks:

1. CONVM is used with an CHBDYP entry of type FTUBE having the same EID.
2. The temperature of the fluid film point may be specified to determine the material properties for the fluid. If FLMND=0, the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperatures, or the ambient temperature, as defined in the FORM field of the PCONVM Bulk Data entry.
3. CNTMDOT must be set to the desired mass flow rate (mdot) to effect the advection of energy downstream at an $\text{mdot} \cdot C_p \cdot T$ rate. In addition to the effect that mdot has on the transfer of thermal energy in the streamwise direction, this control point value is also used in computing the tube Reynolds number and subsequently the forced convection heat transfer coefficient if requested. This enables the fluid stream to exchange heat with its surroundings.

4. If only the first ambient point is specified then, the second ambient point is assumed to have the same temperature.
5. See the Bulk Data entry, “**PCONVM**” on page 2195, for an explanation of the mathematical relationships available for forced convection and the reference temperature for fluid material properties.

CORD1C Cylindrical Coordinate System Definition, Form 1

Defines a cylindrical coordinate system using three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1C	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1C	3	16	32	19					

Field	Contents
CIDA, CIDB	Coordinate system identification number. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A; G1B ≠ G2B ≠ G3B)

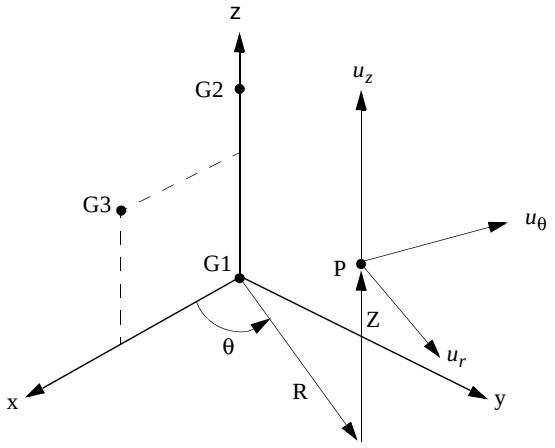


Figure 8-38 CORD1C Definition

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.

3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in **Figure 8-38**) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.
5. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z).
6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in "**Grid Point and Coordinate System Definition**" on page 41 of the *MSC.Nastran Reference Guide*.

CORD1R Rectangular Coordinate System Definition, Form 1

Defines a rectangular coordinate system using three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1R	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1R	3	16	32	19					

Field	Contents
CIDA, CIDB	Coordinate system identification number. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B)

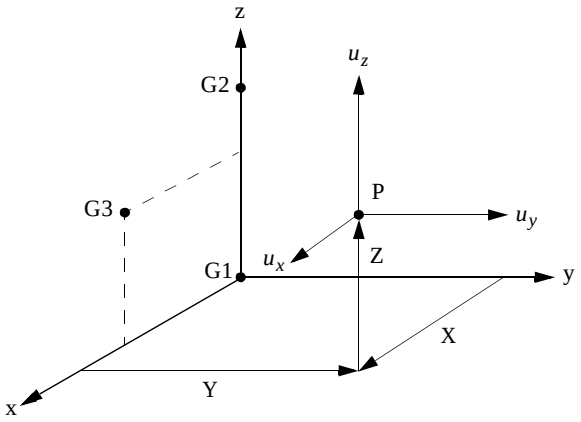


Figure 8-39 CORD1R Definition

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.

3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the x-z plane. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in **Figure 8-39**) in this coordinate system is given by (X, Y, Z).
5. The displacement coordinate directions at P are shown above by (u_x, u_u, u_z).

CORD1RX (SOL 700)

Alternate rectangular coordinate system specification for SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1RX	CID	G1	G2	G3	CID2	G4	G5	G6	

Example:

CORD1RX	1	22	456	457					

Field	Contents	Type	Default
CID	Coordinate-system number.	I > 0	Required
G1, G2, G3	Grid-point numbers G1, G2, and G3. The grid points must be unique.	I > 0	Required
CID2	Optional second coordinate-system number	I > 0	Blank
G4, G5, G6	Grid-point numbers G4, G5, and G6. The grid-point numbers must be unique.	I > 0	Blank

Remark:

1. First node is the origin , 2nd node is the local x-axis , 3rd node is the node on the XY plane.

CORD2RX (SOL 700)

Alternate rectangular coordinate system specification for SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2RX	CID	A1	A2	A3	B1	B2	B3		
	C1	C2	C3						

Example:

CORD2RX	1	22	456	457					
	0.1								

Field	Contents	Type	Default
CID	Coordinate-system number.	I > 0	Required
RID	Reference coordinate system that is defined independent of the new coordinate system.	I > 0	0
A1, A2, A3 B1, B2, B3 C1, C2, C3	Coordinate of three points in the coordinate system referenced by RID.	Real	0.0

Remark:

1. First node is the origin , 2nd node is the local x-axis , 3rd node is the node on the XY plane.

CORD3RX (SOL 700)

Alternate rectangular coordinate system specification for SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
CORD3RX	CID	G1	G2	G3	CID2	G4	G5	G6	

Example:

CORD3RX	1	22	456	457					

Field	Contents	Type	Default
CID	Coordinate-system number.	I > 0	Required
G1, G2, G3	Grid-point numbers G1, G2, and G3 must be unique.	I > 0	0

Remark:

1. First node is the origin , 2nd node is the local x-axis , 3rd node is the node on the XY plane.

CORD1S Spherical Coordinate System Definition, Form 1

Defines a spherical coordinate system by reference to three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1S	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1S	3	16	32	19					

Field	Contents
CIDA, CIDB	Coordinate system identification numbers. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B)

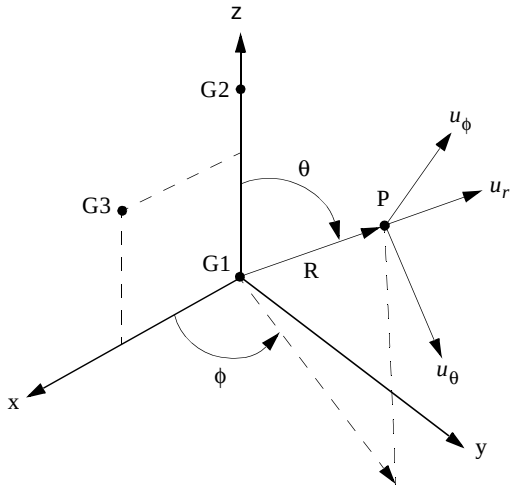


Figure 8-40 CORD1S Definition

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.

3. GiA and GiB must be defined in coordinate systems with a definition that does not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in **Figure 8-40**) in this coordinate system is given by (R, θ, ϕ) where θ and ϕ are measured in degrees.
5. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_ϕ).
6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in "**Grid Point and Coordinate System Definition**" on page 41 of the *MSC.Nastran Reference Guide*.

CORD2C Cylindrical Coordinate System Definition, Form 2

Defines a cylindrical coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2C	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Field	Contents
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default = 0 is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real)

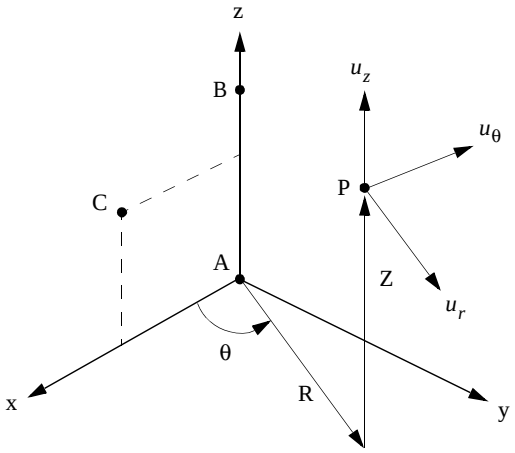


Figure 8-41 CORD2C Definition

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.
5. The location of a grid point (P in **Figure 8-41**) in this coordinate system is given by (R, θ, Z), where θ is measured in degrees.
6. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z).
7. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in "**Grid Point and Coordinate System Definition**" on page 41 of the *MSC.Nastran Reference Guide*.
8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

CORD2R Rectangular Coordinate System Definition, Form 2

Defines a rectangular coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2R	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Field	Contents
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer $\geq$ 0; Default = 0, which is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real)

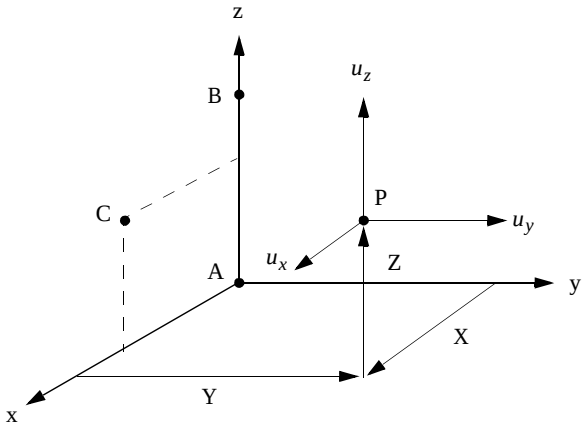


Figure 8-42 CORD2R Definition

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second defines the direction of the z-axis. The third point defines a vector which, with the z-axis, defines the x-z plane. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.
5. The location of a grid point (P in the **Figure 8-42**) in this coordinate system is given by (X, Y, Z).
6. The displacement coordinate directions at P are shown by (u_x, u_y, u_z).
7. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

CORD2S Spherical Coordinate System Definition, Form 2

Defines a spherical coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2S	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2S	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Field	Contents
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer $\geq$ 0; Default = 0 is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3. (Real)

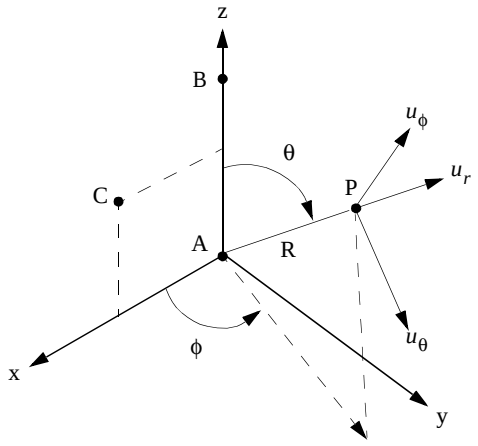


Figure 8-43 CORD2S Definition

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must all be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.
5. The location of a grid point (P in **Figure 8-43**) in this coordinate system is given by (R, θ, ϕ), where θ and ϕ are measured in degrees.
6. The displacement coordinate directions at P are shown above by (u_r, u_θ, u_ϕ).
7. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in "**Grid Point and Coordinate System Definition**" on page 41 of the *MSC.Nastran Reference Guide*.
8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

CORD3G General Coordinate System

Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system. The CORD3G entry is used with the MAT9 entry to orient material principal axes for 3-D composite analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CORD3G	CID	METHOD	FORM	THETAID1	THETAID2	THETAID3	CIDREF		

Example:

CORD3G	100	E313	EQN	110	111	112	0		

Field	Contents
CID	Coordinate system identification number. See Remark 1. (Integer > 0)
METHOD	E313 or S321 for Euler angle rotation in 313 sequence or space-fixed rotation in 321 sequence. See Remark 2. (Character; Default = “E313")
FORM	Specifies the Bulk Data entry which defines angles. FORM = “EQN” for DEQATN entry or FORM = “TABLE” for TABLE3D entry. (Character; Default = “EQN”)
THETAID	Identification number for DEQATN or TABLE3D Bulk Data entry which defines the three angles (in radians) measured from reference coordinates to the general material coordinate system. See Remark 3. (Integer > 0)
CIDREF	Identification number for the coordinate system from which the orientation of the general coordinate system is defined. There is no default. (Integer > 0)

Remarks:

1. CID must be unique with respect to all other coordinate systems. CID cannot be referenced on GRID entries.

2. Three Euler angles specify the rotation of the CORD3G coordinate axes (xyz) with respect to the local Cartesian coordinate axes (XYZ) in CIDREF as follows: first rotate about Z-axis by θ_1, next rotate about rotated x-axis by θ_2, and then rotate about rotated z-axis by θ_3. On the other hand, the space-fixed rotations in 321 sequence specify all the rotations about the fixed coordinate axes: first rotate about Z by θ_1, next about Y by θ_2, then about X by θ_3.
3. The three rotations define a coordinate transformation which transforms position vectors in the reference coordinate system into the general coordinate system.
4. The DEQATN option must have three arguments representing the three axes of CIDREF, although not all arguments are necessarily needed in the equation.

CORD3R (SOL 700) Moving Rectangular Coordinate System

Defines a moving rectangular coordinate system using three points (SOL 700 only).

Format:

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	N1	N2	N3	CID	N1	N2	N3	

Example:

CORD3R	1001	1	144	300					

Field	Contents
CID	Unique coordinate system number. (Integer > 0)
N1, N2, N3	Grid point numbers (must be unique). (Integer > 0)

Remarks:

1. Available in SOL 700 only.
2. Two different coordinate systems may be defined on one entry
3. The grid points must be defined in an independent coordinate system.
4. The first grid point is the origin, the second lies on the z-axis and the third lies in the x-z plane.
5. The position and orientation of the coordinate system is updated as the grid points move.
6. The three grid points must not be collinear.

CPENTA Five-Sided Solid Element Connection

Defines the connections of a five-sided solid element with six to fifteen grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CPENTA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

Example:

CPENTA	112	2	3	15	14	4	103	115	
	5	16	8				120	125	
	130								

Field	Contents	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID or PLSOLID entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer $\geq$ 0 or blank	Required

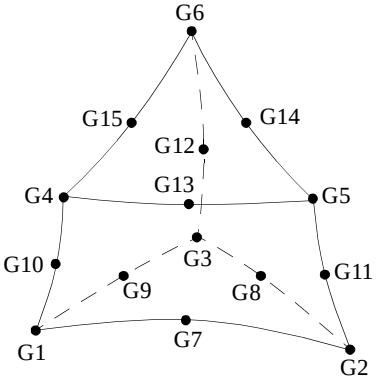


Figure 8-44 CPENTA Element Connection

Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved; i.e., G1, G2, and G3 define a triangular face, G1, G10, and G4 are on the same edge, etc.
3. The edge grid points, G7 to G15, are optional. Any or all of them may be deleted. In the example shown, G10, G11, and G12 have been deleted. The continuations are not required if all edge grid points are deleted.
4. Components of stress are output in the material coordinate system except for hyperelastic elements, which are output in the basic coordinate system.
5. For nonhyperelastic elements the element coordinate system for the CPENTA element is derived accordingly. The origin of the coordinate system is located at the midpoint of the straight line connecting the points G1 and G4. The Z axis points toward the triangle G4-G5-G6 and is oriented somewhere between the line joining the centroids of the triangular faces and a line perpendicular to the midplane. The midplane contains the midpoints of the straight lines between the triangular faces. The X and Y axes are perpendicular to the Z axis and point in a direction toward, but not necessarily intersecting, the edges G2 through G5 and G3 through G6, respectively.

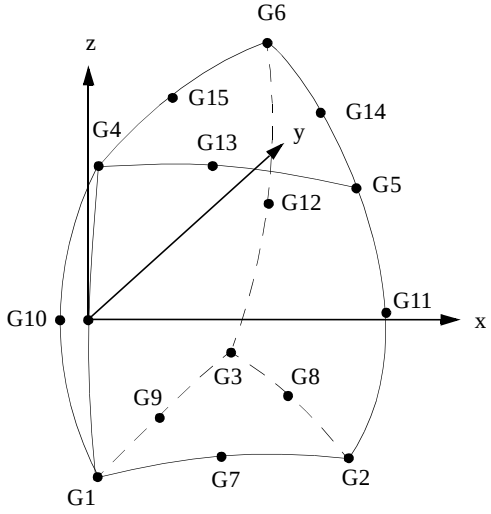


Figure 8-45 CPENTA Element Coordinate System

6. We recommend that the edge grid points be located within the middle third of the edge.

7. For hyperelastic elements, the plot codes are specified under the CPENTAFD element name in "**Item Codes**" on page 873.
8. If a CPENTA element is referenced on a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
    - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.
    - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.
9. By default, all of the nine edges of the element are considered straight unless:
    - For p-elements there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.
    - For h-elements any of G7 through G15 are specified.

CQUAD Fully Nonlinear Plane Strain Element

Defines a plane strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9						

Example:

CQUAD	111	203	31	74	75	32			

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PLPLANE entry. (Integer > 0)
G1, G2, G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)
G5, G6, G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer $\geq$ 0 or blank)
G9	Identification number of center grid point. Optional. (Integer $\geq$ 0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. Grid points G1 to G9 must be numbered as shown and must lie on a plane.
3. It is recommended that the edge points be located within the middle third of the edge.
4. The plot codes are specified under the CQUADFD element name in "**Item Codes**" on page 873.
5. Stresses and strains are output in the coordinate system identified by the CID field of the PLPLANE entry.

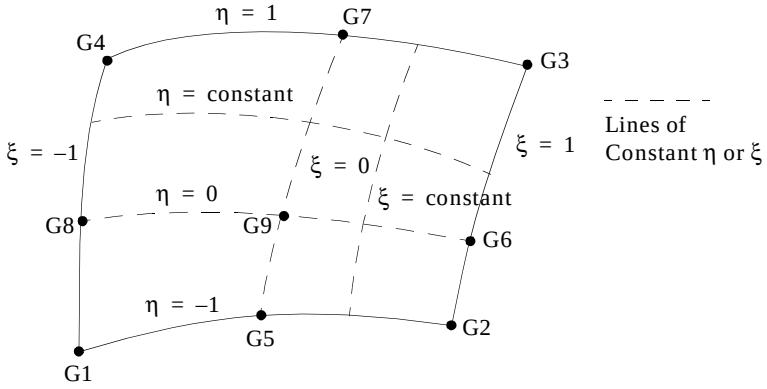


Figure 8-46 CQUAD Element Coordinate System

CQUAD4 Quadrilateral Plate Element Connection

Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
		TFLAG	T1	T2	T3	T4			

Example:

CQUAD4	111	203	31	74	75	32	2.6	0.3	
			1.77	2.04	2.09	1.80			

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL, PCOMP, or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique.)
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. See **Figure 8-48**. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. Use DIAG 38 to print the computed THETA values. MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. (Integer $\geq$ 0; If blank, then THETA = 0.0 is assumed.)

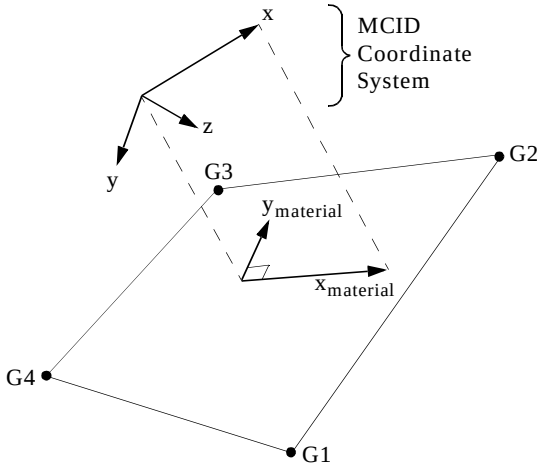


Figure 8-47 MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. ZOFFS is ignored for hyperelastic elements. See Remark **6.** (Real)
TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" is zero or blank, then Ti are actual user specified thicknesses. See Remark **4.** for default. (Real $\geq$ 0.0 or blank, not all zero.) If "TFLAG" is one, then the Ti are fractions relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than 180°.
4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.

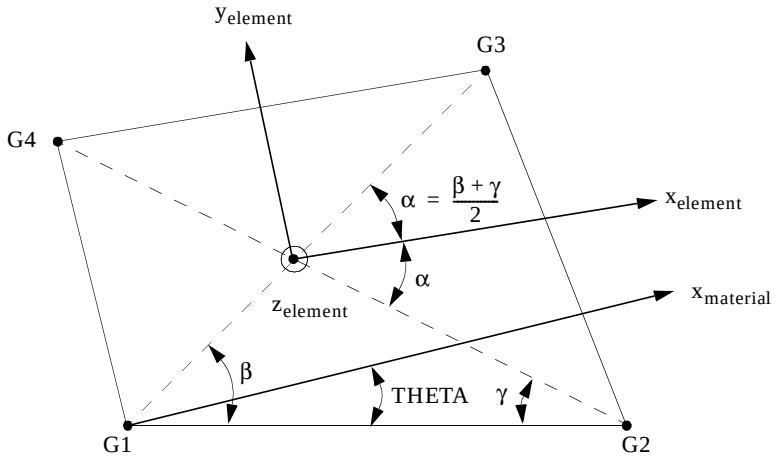


Figure 8-48 CQUAD4 Element Geometry and Coordinate Systems

5. The reference coordinate system for the output of stress, strain and element force depends on the element type.

   - For CQUAD4 elements which are not p-elements and not hyperelastic, the reference coordinate system is the default for output is the element coordinate system. See PARAM,OMID for output in the material system.
   - For CQUAD4 elements referenced by a PSET or PVAL entry, the stresses, strains and element forces are output in the local tangent plane of the element. The local tangents are oriented in a user defined direction which is uniform across a set of elements. By default, the local tangent x-direction is oriented in the positive x-direction of the basic coordinate system. See the Bulk Data entry, "**OUTRCV**" on page 2064 for user defined output coordinate systems.
   - For hyperelastic elements the stress and strain are output according to CID on the PLPLANE entry.

6. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced both by PID.

The specification of offset vectors gives wrong results in solution sequences with geometric nonlinear analysis (SOL 106, 129, 153, 159 and PARAM,LGDISP>0). The differential stiffness is computed correctly in MSC.Nastran 2004 and later versions for (SOLs 5, 16, 105 and 200 and SOL 106 with PARAM,LGDISP≤0).

7. For finite deformation hyperelastic analysis, the plot codes are given by the CQUADFD element name in "**Item Codes**" on page 873.
8. If a CQUAD4 element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
    - If a curved edge of a p-element is shared by an h-element CQUAD4, the geometry of the edge is ignored and set straight.
9. By default, all of the four edges of the element are considered straight unless the element is a p-element and the edge is associated to curved geometry with a FEEDGE or FEFACE entry.

CQUAD8 Curved Quadrilateral Shell Element Connection

Defines a curved quadrilateral shell or plane strain element with eight grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	T1	T2	T3	T4	THETA or MCID	ZOFFS	
	TFLAG								

Example:

CQUAD8	207	3	31	33	73	71	32	51	
	53	72	0.125	0.025	0.030	.025	30.	.03	

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL, PCOMP, or PLPLANE entry. (Integer > 0)
G1, G2, G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)
G5, G6, G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer $\geq$ 0 or blank)
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. See Remark **4.** for default. (Real $\geq$ 0.0 or blank, not all zero.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.
THETA	Material property orientation angle in degrees. See **Figure 8-49**. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)

Field	Contents
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element (see Remark 3.) MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed.)
ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 6. ZOFFS is ignored for hyperelastic elements. (Real)
TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. Grid points G1 to G8 must be numbered as shown in **Figure 8-49**.
3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between $x_{material}$ and the line of constant η.

   The definition of the material coordinate system by projection is used to calculate an angle THETA. Please note that since xi changes directions throughout the element based on element shape, the material coordinate system varies similarly. Because of this an orthotropic or anisotropic material will cause the CQUAD8's stiffness to be biased by both it's shape and grid ordering. Use the QUAD4 element if a constant material coordinate system direction is desired with orthotropic and anisotropic materials.
4. T1, T2, T3 and T4 are optional. If they are not supplied and no TFLAG, then T1 through T4 will be set to the value of T on the PSHELL entry.
5. It is recommended that the midside grid points be located within the middle third of the edge. If the edge point is located at the quarter point, the program may fail with a divide-by-zero error or the calculated stresses will be meaningless.

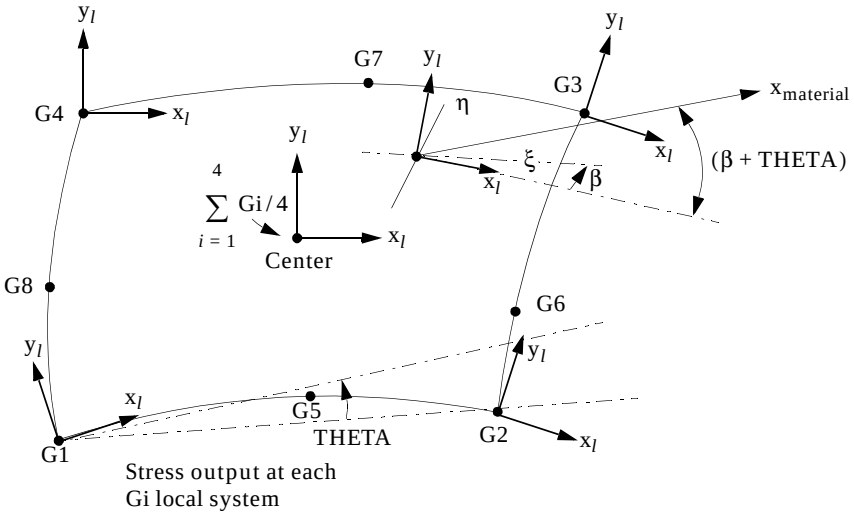
6. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

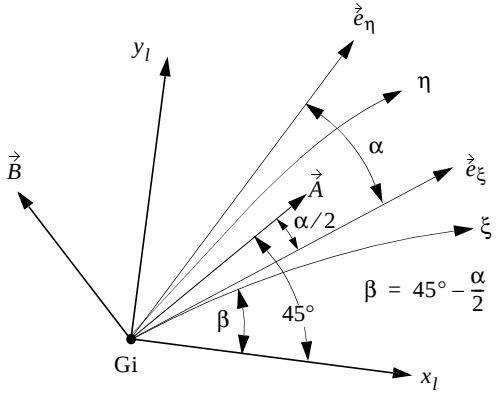
   The specification of offset vectors gives wrong results in solution sequences that compute differential stiffness: linear buckling analysis provided in SOLs 105 and 200; SOLs 103 and 107 through 112 with the STATSUB command; and geometric nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1.

7. If all midside grid points are deleted, then the element will be excessively stiff and the transverse shear forces incorrect. A User Warning Message is printed, and a CQUAD4 element is recommended instead. If the element is hyperelastic, then it is processed identically to the hyperelastic CQUAD4 element.

8. For a description of the element coordinate system, see "**Shell Elements (CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR)**" on page 131 of the *MSC.Nastran Reference Guide*. Stresses and strains are output in the local coordinate system identified by x_l and y_l in **Figure 8-49**. However, for hyperelastic elements the stress and strain are output in the coordinate system identified by the CID field on the PLPLANE entry.

9. For hyperelastic elements the plot codes are specified under the CQUADFD element name in "**Item Codes**" on page 873.





where

$\vec{e}_\eta$ is tangent to η at Gi

$\vec{e}_\xi$ is tangent to ξ at Gi

$\vec{A}$ is formed by bisection of $\vec{e}_\eta$ and $\vec{e}_\xi$

$\vec{B}$ and $\vec{A}$ are perpendicular

y_l is formed by bisection of $\vec{A}$ and $\vec{B}$

x_l is perpendicular to y_l

Figure 8-49 CQUAD8 Element Geometry and Coordinate Systems

CQUADR Quadrilateral Plate Element Connection

Defines an isoparametric membrane and bending quadrilateral plate element. This element has a normal rotational degrees-of-freedom. It is a companion to the CTRIAR element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUADR	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
		TFLAG	T1	T2	T3	T4			

Example:

CQUADR	82	203	31	74	75	32	2.6		
			1.77	2.04	2.09	1.80			

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL or PCOMP entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. See **Figure 8-51**. (Real; Default = 0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. Use DIAG 38 to print the computed THETA values. For SOL 600, only CORD2R is allowed. (Integer ≥ 0; If blank, then THETA = 0.0 is assumed.)
ZOFFS	Offset from the surface of grid point to the element plane. See Remark **8.**

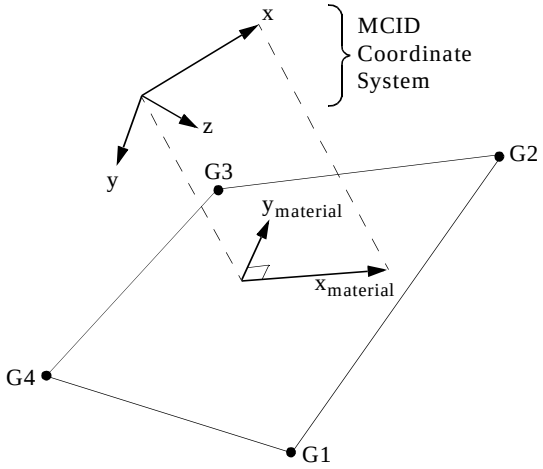


Figure 8-50 MCID Coordinate System Definition

TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. (Real $\geq$ 0.0 or blank, not all zero. See Remark **4.** for default.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All the interior angles must be less than 180°.
4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.
5. Stresses and strains are output in the element coordinate system at the centroid and grid points G1 through G4.
6. The rotational degrees-of-freedom normal to the element are active in the element formulation and must not be constrained unless at a boundary. Inaccurate results will be obtained if they are constrained.

7. The CTRIAR element is the triangular companion to the CQUADR element and should be used instead of CTRlA3 or CTRlA6.

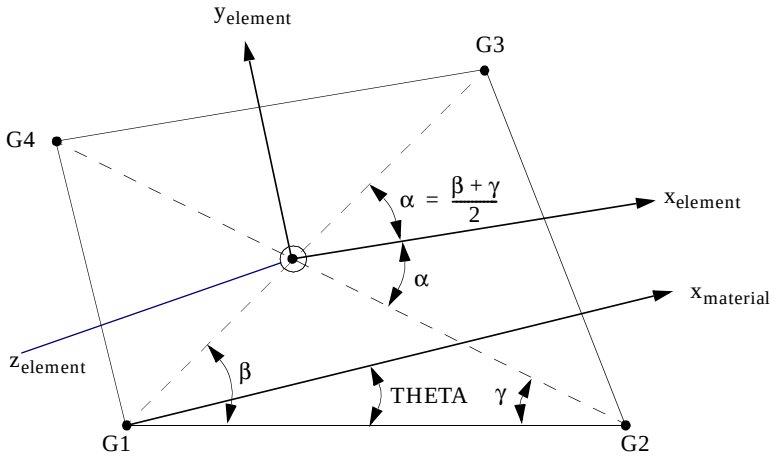


Figure 8-51 CQUADR Element Geometry and Coordinate Systems

8. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced both by PID.

   The specification of offset vectors gives wrong results in solution sequences that compute differential stiffness: linear buckling analysis provided in SOLs 105 and 200; SOLs 103 and 107 through 112 with the STATSUB command; and geometric nonlinear analysis provided in SOLs 106, 129, 153 and 159 with PARAM,LGDISP,1.

CQUADX Fully Nonlinear Axisymmetric Element

Defines an axisymmetric quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CQUADX	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9						

Example:

CQUADX	111	203	31	74	75	32			

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PLPLANE entry. (Integer > 0)
G1, G2 G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)
G5, G6 G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer $\geq$ 0 or blank)
G9	Identification number of center grid point. Optional. (Integer $\geq$ 0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. Gi must be numbered as shown in **Figure 8-52**.
3. It is recommended that the edge points be located within the middle third of the edge.
4. The plot codes are specified under the CQUADXFD element name in "**Item Codes**" on page 873.
5. All Gi must lie on the x-y plane of the basic coordinate system. Stress and strain are output in the basic coordinate system.

6. A concentrated load (e.g., FORCE entry) at Gi is divided by the radius to Gi and then applied as a force per unit circumferential length. For example, in order to apply a load of 100 N / m on the circumference at G1, which is located at a radius of 0.5 m, then the magnitude specified on the static load entry must result in:

$$(100\ \text{N/m}) \cdot (0.5\ \text{m}) = 50\ \text{N}$$

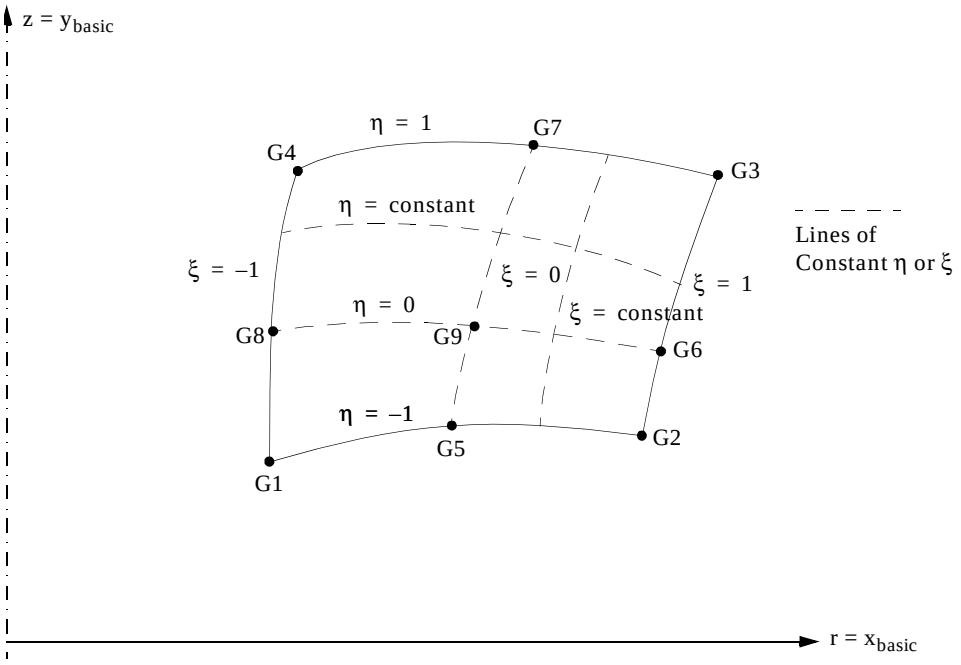


Figure 8-52 CQUADX Element Coordinate System

CRAC2D Two-Dimensional Crack Tip Element

Defines a two-dimensional crack tip element.

Format:

1	2	3	4	5	6	7	8	9	10
CRAC2D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18					

Example:

CRAC2D	114	108	2	5	6	8	7	11	
	12	14	16	17		20	22		

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PRAC2D entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer $\geq$ 0; G11 through G18 may be blank.)

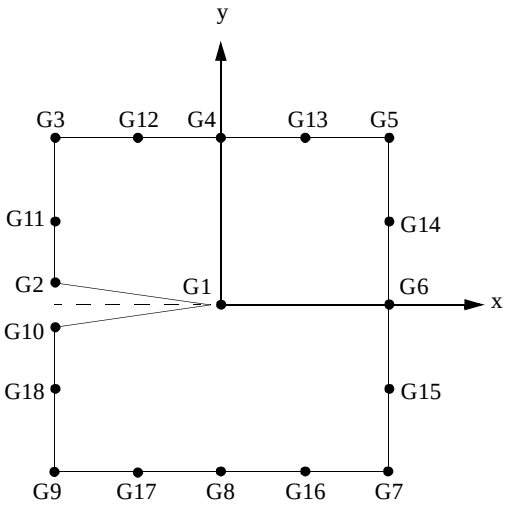
Remarks:

1. The following is a dummy element and requires the presence of one Bulk Data entry of the form:

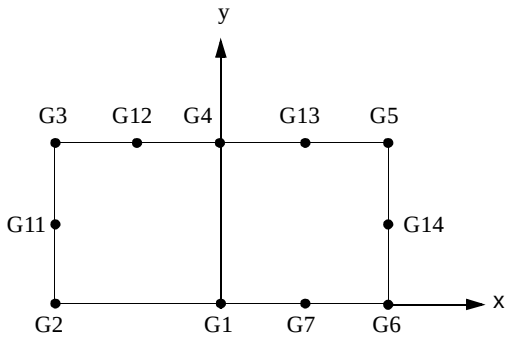
ADUM8	18	0	5	0	CRAC2D				

2. The element should be planar. Significant deviations will produce fatal errors.
3. Grid points G1 through G10 are required while grid points G11 through G18 are optional for the quadrilateral form of the element.
4. The stresses and stress intensity factors are calculated assuming that G2 and G10 are coincident. Deviations from this will produce erroneous results.
5. For the symmetric half-crack option, grid points G1 through G7 are required while grid points G11 through G14 are optional. Grid points G8 through G10 and G15 through G18 must not be present for this option.

6. The ordering conventions for the full-crack and half-crack options are shown in **Figure 8-53**.
7. The stress output is interpreted as shown in "**Crack Tip Elements (CRAC2D, CRAC3D)**" on page 197 of the *MSC.Nastran Reference Guide*.



(a) Full Crack Option



(b) Symmetric Half-Crack Option

Figure 8-53 CRAC2D Element Connection for Full and Symmetric Options

CRAC3D Three-Dimensional Crack Tip Element

Defines a three-dimensional crack tip element.

Format:

1	2	3	4	5	6	7	8	9	10
CRAC3D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20	G21	G22	
	G23	G24	G25	G26	G27	G28	G29	G30	
	G31	G32	G33	G34	G35	G36	G37	G38	
	G39	G40	G41	G42	G43	G44	G45	G46	
	G47	G48	G49	G50	G51	G52	G53	G54	
	G55	G56	G57	G58	G59	G60	G61	G62	
	G63	G64							

Example:

CRAC3D	113	101	2	5	7	8	4	10	
	11	14	15	17		3	6	9	
	12		16		102	105	107	108	
	104	110	111	114	115	117		103	
	106	109	112		116		202	205	
	207	208	204	210	211	214	215	217	
	225	226							

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PRAC3D entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer $\geq$ 0)

Remarks:

1. The following is a dummy element and requires the presence of one Bulk Data entry of the form:

ADUM9	64	0	6	0	CRAC3D				

2. Element identification numbers should be unique with respect to all other element identification numbers.
3. This element, including grid point numbering conventions, is shown in **Figure 8-54** and **Figure 8-55**. Grid points G1 through G10, and G19 through G28 are required; midside and surface grid points G11 through G18, G29 through G36, and G37 through G64 are optional. Either all or none of grid points G37 through G46 should be present. A fatal error message will be issued for partial connectivity.
4. For the symmetric half-crack option Grid Points G1 through G7, and G19 through G25 are required, whereas grid points G11 through G14, G29 through G32, and G37 through G42 are optional. Grid points G8 through G10, G15 through G18, G26 through G28, G33 through G36, G43 through G46, G51 through G55, and G60 through G64 should not be specified to invoke this option.
5. It is recommended that both the faces (formed by grid points G2 through G18 and grid points G20 through G36) and the midplane (formed by grid points G37 through G46 and grid points G37 through G46) be planar. It is also recommended that midside grid points G37 through G46 be located within the middle third of the edges.
6. The midside nodes on both the faces should be defined in pairs. For example, if grid point G11 is not defined, then grid point G29 should not be defined and vice versa.
7. The stresses and stress intensity factors are calculated with the assumptions that grid points G2 and G10, G20 and G28, and G38 and G46 are coincident. Deviation from this condition will produce erroneous results.
8. The stress output is interpreted as shown in "**Crack Tip Elements (CRAC2D, CRAC3D)**" on page 197 of the *MSC.Nastran Reference Guide*.
9. As depicted in **Figure 8-54** and **Figure 8-55**, the element is a right-handed" element. Thus define the vectors $\overline{G1G9}$ and $\overline{G1G3}$, then the cross-product $\overline{G1G9} \times \overline{G1G3}$ points to the face defined by G19, G20, ...

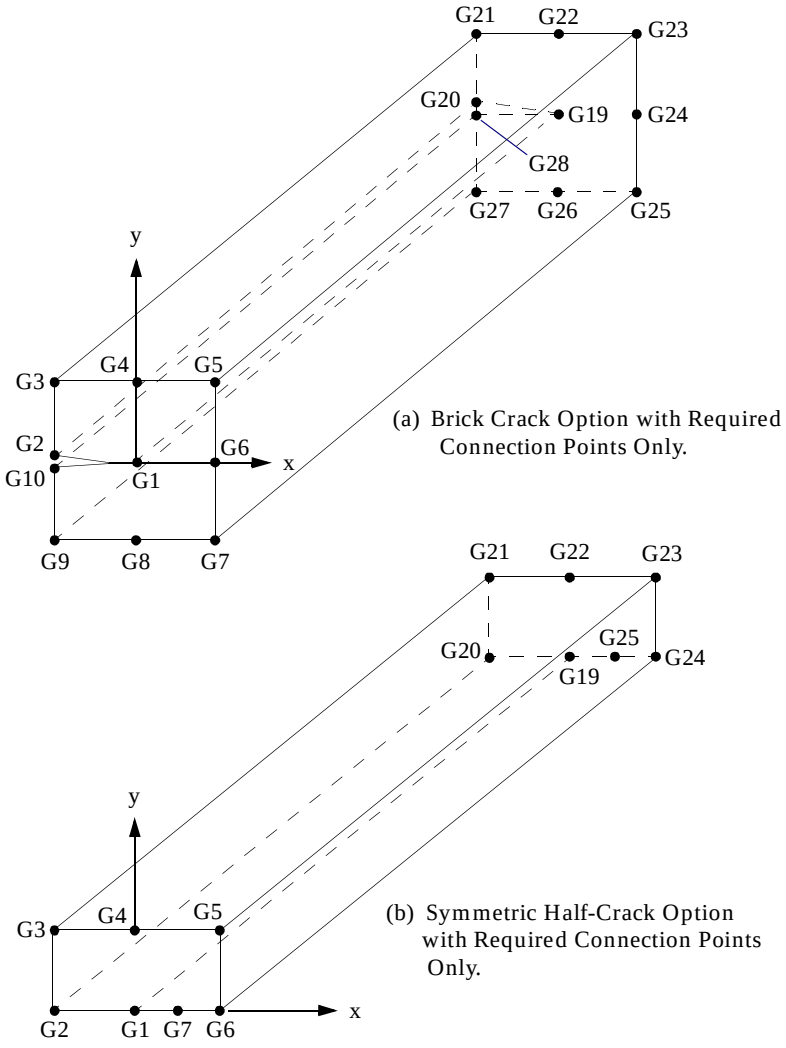
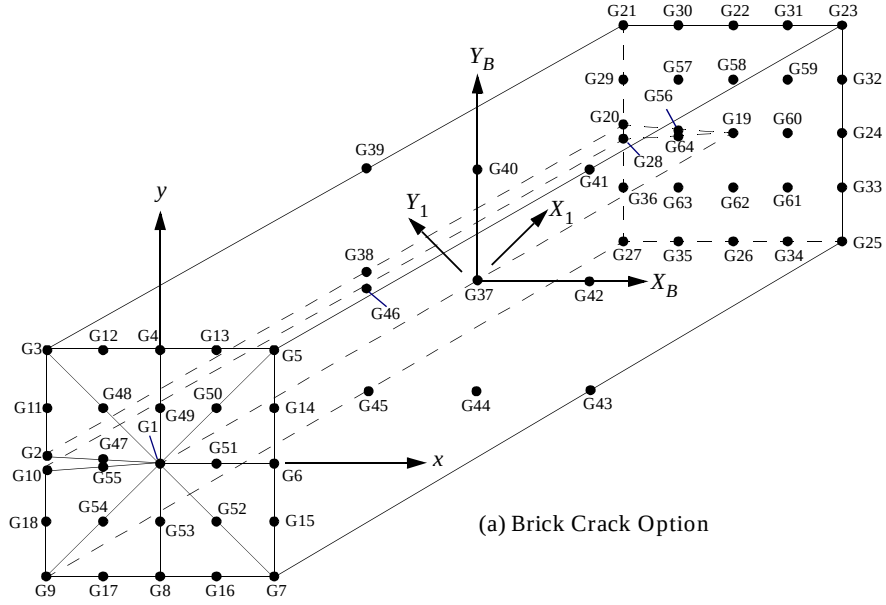
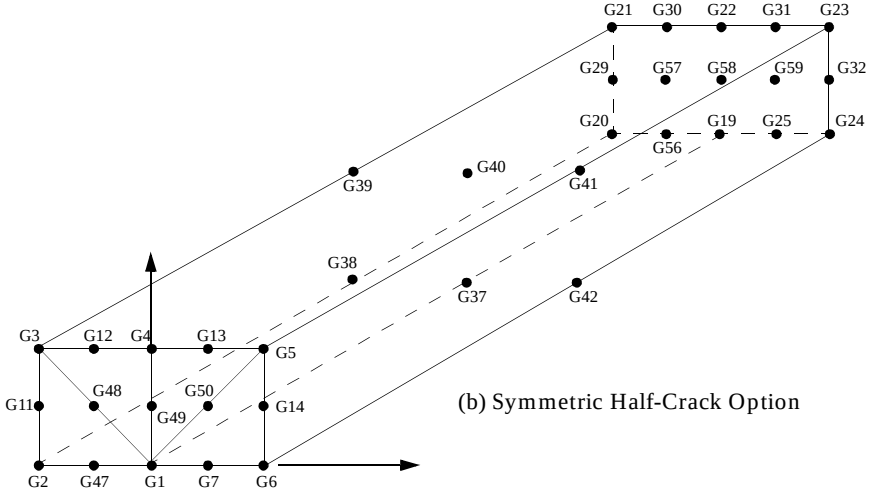


Figure 8-54 CRAC3D Solid Crack Tip Element with Required Connection Points Only



(a) Brick Crack Option



(b) Symmetric Half-Crack Option

Figure 8-55 CRAC2D Solid Crack Tip Element with All Connection Points

CREEP Creep Characteristics

Defines creep characteristics based on experimental data or known empirical creep law. This entry will be activated if a MAT1, MAT2, or MAT9 entry with the same MID is used and the NLPARM entry is prepared for creep analysis. The creep formulation is principally suited for isotropic materials and, in general, when used with anisotropic materials may produce incorrect results. However, slightly anisotropic materials may produce acceptable results.

Format:

1	2	3	4	5	6	7	8	9	10
CREEP	MID	T0	EXP	FORM	TIDKP	TIDCP	TIDCS	THRESH	
	TYPE	a	b	c	d	e	f	g	

Example:

CREEP	8	1100.		CRLAW					
	121	6.985-6	2.444	7.032-4	0.1072	6.73-9	0.1479	3.0	

Field	Contents
MID	Material identification number of a MAT1, MAT2, or MAT9 entry. (Integer > 0)
T0	Reference temperature at which creep characteristics are defined. See Remark **2.** (Real; Default = 0.0)
EXP	Temperature-dependent term, $e^{(-\Delta H/(R \cdot T0))}$, in the creep rate expression. See Remark **2.** (0.0 < Real ≤ 1.0; Default = 1.0E-9)
FORM	Form of the input data defining creep characteristics. (Character: “CRLAW” for empirical creep law, or “TABLE” for tabular input data of creep model parameters.)
TIDKP, TIDCP, TIDCS	Identification number of a TABLES1 entry, which defines the creep model parameters $K_p(\sigma)$, $C_p(\sigma)$, and $C_s(\sigma)$, respectively. See Remarks **3.** through **5.** (Integer > 0)
THRESH	Threshold limit for creep process. Threshold stress under which creep does not occur is computed as THRESH multiplied by Young’s modulus. (0.0 < Real < 1.0E-3; Default = 1.0E-5)

Field	Contents
TYPE	Identification number of the empirical creep law type. See Remark 1. (Integer: 111, 112, 121, 122, 211, 212, 221, 222, or 300)
a through g	Coefficients of the empirical creep law specified in TYPE. Continuation should not be specified if FORM = "TABLE". See Remark 1. (Real)

Remarks:

1. Two classes of empirical creep law are available.

   Creep Law Class 1

   The first creep law class is expressed as:

$$\varepsilon^c(\sigma, t) = A(\sigma)[1 - e^{-R(\sigma)t}] + K(\sigma)t \qquad \text{Eq. 8-1}$$

   Parameters $A(\sigma)$, $R(\sigma)$, and $K(\sigma)$ are specified in the following form, as recommended by Oak Ridge National Laboratory:

Parameter	Function 1	Digit	Function 2	Digit
$A(\sigma)$	$a\sigma^b$	i=1	$ae^{b\sigma}$	i=2
$R(\sigma)$	$ce^{d\sigma}$	j=1	$c\sigma^d$	j=2
$K(\sigma)$	$e \cdot [\sinh(f\sigma)]^g$	k=1	$ee^{f\sigma}$	k=2

   TYPE=ijk where i, j, and k are digits equal to 1 or 2, according to the desired function in the table above. For example, TYPE=122 defines $A(\sigma) = a\sigma^b$, $R(\sigma) = c\sigma^d$, and $K(\sigma) = ee^{f\sigma}$.

Creep Law Class 2

The second creep law class (TYPE=300) is expressed as:

$$\varepsilon^c(\sigma, t) = a\sigma^b t^d \qquad \text{Eq. 8-2}$$

where the values of b and d must be defined as follows:

$$1.0 < b < 8.0$$

and

$$0.2 < d < 2.0$$

The coefficient g should be blank if TYPE = 112, 122, 222, or 212 and c, e, f, and g should be blank if TYPE = 300. The coefficients a through g are dependent on the structural units; caution must be exercised to make these units consistent with the rest of the input data.

2. Creep law coefficients a through g are usually determined by least squares fit of experimental data, obtained under a constant temperature. This reference temperature at which creep behavior is characterized must be specified in the T0 field if the temperature of the structure is different from this reference temperature. The conversion of the temperature input (°F or °C) to °K (degrees Kelvin) must be specified in the PARAM,TABS entry as follows:

   PARAM,TABS,273.16 (If Celsius is used.)

   PARAM,TABS,459.69 (If Fahrenheit is used.)

   When the correction for the temperature effect is required, the temperature distribution must be defined in the Bulk Data entries (TEMP, TEMPP1 and/or TEMPRB), which are selected by the Case Control command TEMP(LOAD) = SID within the subcase.

   From the thermodynamic consideration, the creep rate is expressed as:

$$\dot{\varepsilon}^{C} = \dot{\varepsilon}_{A}(e^{-\Delta H/RT}) \qquad \text{Eq. 8-3}$$

   where:

   ΔH = energy of activation

   R = gas constant (= 1.98 cal/mole °K)

   T = absolute temperature (°K)

   $\dot{\varepsilon}_{A}$ = strain/sec per activation

   If the creep characteristics are defined at temperature T0, the creep rate at temperature T is corrected by a factor

$$\frac{\dot{\varepsilon}^{c}}{\dot{\varepsilon}^{c}_{o}} = \text{EXP}^{\left(\frac{T0}{T}-1\right)} \qquad \text{Eq. 8-4}$$

where:

$\dot{\varepsilon}^{C}$ = corrected creep rate

$\dot{\varepsilon}_{O}^{C}$ = creep rate at T0

$\text{EXP}^{\left(\frac{T0}{T}-1\right)}$ = correction factor

3. If the creep model parameters K_p, C_p, and C_s are to be specified with FORM = "TABLE" then TABLES1 entries (with IDs that appear in TIDXX fields) must be provided in the Bulk Data Section. In this case, the continuation should not be specified.
4. Creep model parameters K_p, C_p, and C_s represent parameters of the uniaxial rheological model as shown in **Figure 8-56**.

   Tabular values (X_i, Y_i) in the TABLES1 entry correspond to (σ_i, K_{pi}), (σ_i, C_{pi}), and (σ_i, C_{si}) for the input of K_p, C_p, and C_s, respectively. For linear viscoelastic materials, parameters K_p, C_p, and C_s are constant and two values of σ_i must be specified for the same value of K_{pi}, C_{pi}, and C_{si}.

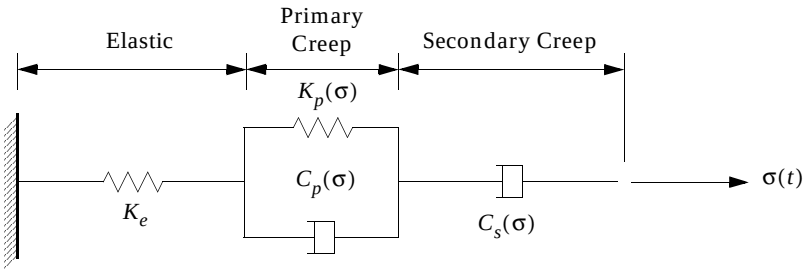


Figure 8-56 CREEP Parameter Idealization

Creep model parameters, as shown in **Figure 8-57** through **Figure 8-59** below, must have positive values. If the table look-up results in a negative value, the value will be reset to zero and a warning message (TABLE LOOK-UP RESULTS IN NEGATIVE VALUE OF CREEP MODEL PARAMETER IN ELEMENT ID=****) will be issued.

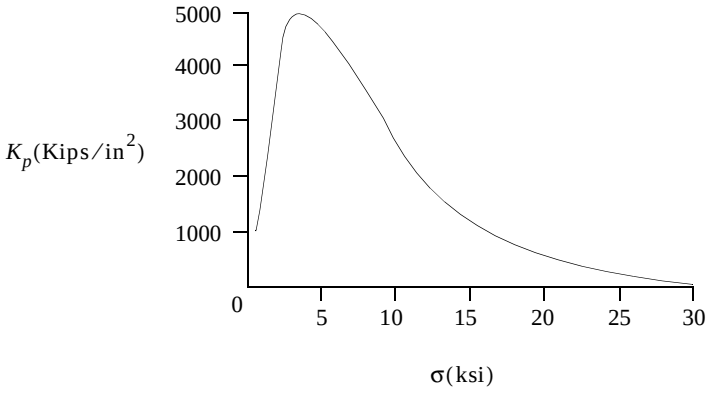


Figure 8-57 K_p Versus σ Example for CREEP

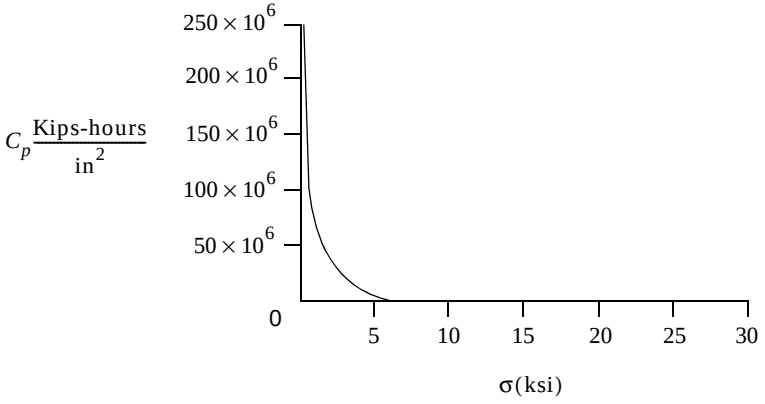


Figure 8-58 C_p Versus σ Example for CREEP

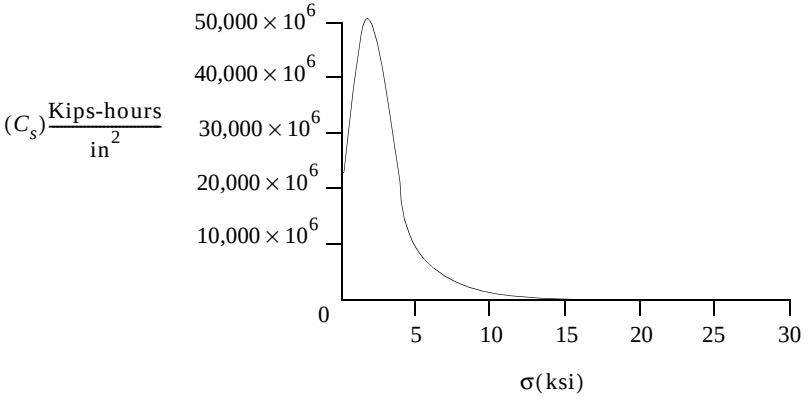


Figure 8-59 C_s Versus σ Example for CREEP

5. Creep analysis requires an initial static solution at $t = 0$, which can be obtained by specifying a subcase that requests an NLPARM entry with DT = 0.0.

CROD Rod Element Connection

Defines a tension-compression-torsion element.

Format:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

Example:

CROD	12	13	21	23					

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PROD entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. See "**CONROD**" on page 1253 for alternative method of rod definition.
3. Only one element may be defined on a single entry.

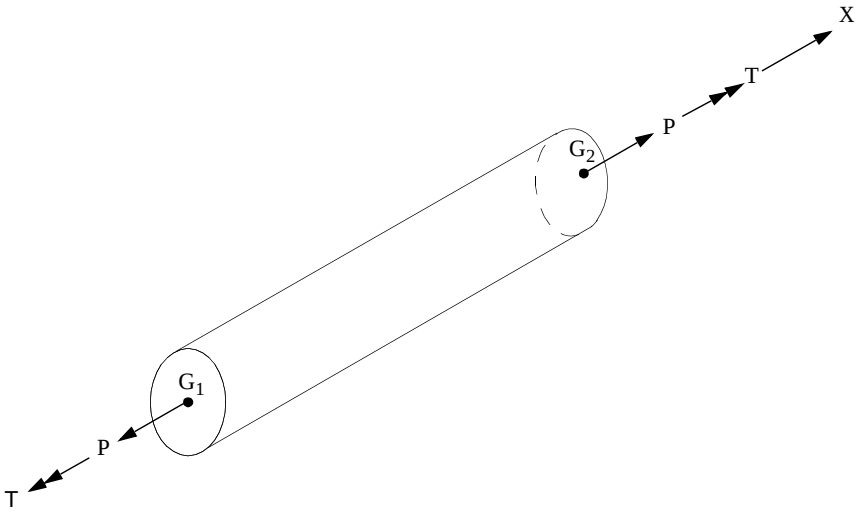


Figure 8-60 CROD Element Internal Forces and Moments

CSET Free Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
CSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

CSET	124	1	5	23	6	16			

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. CSET and BNDFREE entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREEi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREEi entries present or both BSETi/BNDFIXi and CSETi/BNDFREEi entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREEi entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:

- If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
- If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

CSET1 Free Boundary Degrees-of-Freedom, Alternate Form of CSET Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
CSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

CSET1	124	1	5	7	6	9	12	122	
	127								

Alternate Formats and Examples:

CSET1	C	ID1	“THRU”	ID2					
CSET1	3	6	THRU	32					

CSET1		“ALL”							
CSET1		ALL							

Field	Contents
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; For “THRU” option, ID1< ID2)

Remarks:

1. CSET1 and BNDFREE1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREEi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only

CSETi/BNDFREEi entries present or both BSETi/BNDFIXi and CSETi/BNDFREEi entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREEi entries, and any remaining a-set points are placed in the b-set.

3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
    - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
    - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

CSHEAR Shear Panel Element Connection

Defines a shear panel element.

Format:

1	2	3	4	5	6	7	8	9	10
CSHEAR	EID	PID	G1	G2	G3	G4			

Example:

CSHEAR	3	6	1	5	3	7			

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHEAR entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2 ≠ G3 ≠ G4)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than 180°.

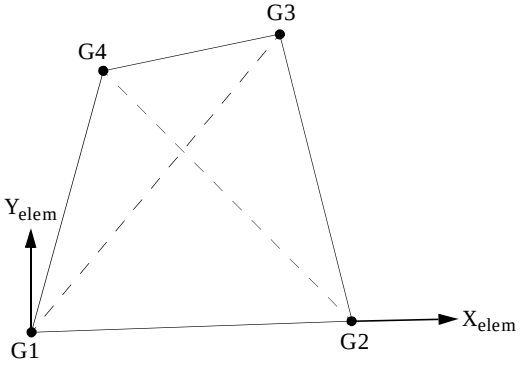


Figure 8-61 CSHEAR Element Connection and Coordinate System

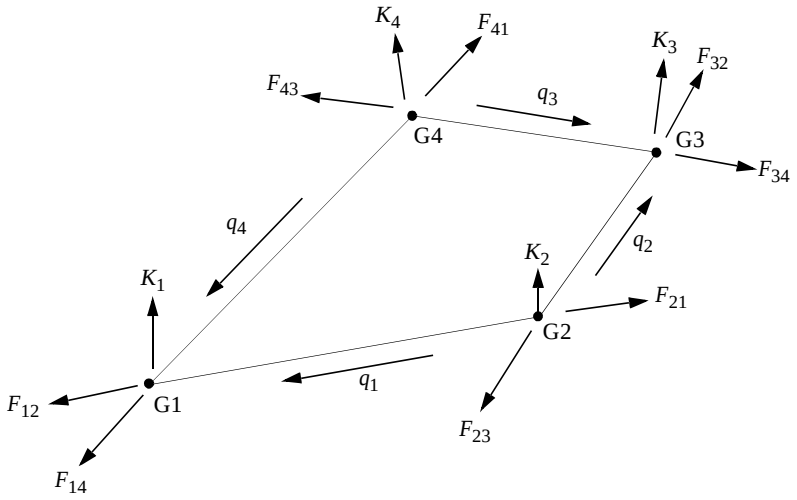


Figure 8-62 CSHEAR Element Corner Forces and Shear Flows

CSLOT3 Three Point Slot Element Connection

Defines an element connecting three points that solves the wave equation in two dimensions. Used in the acoustic cavity analysis for the definition of evenly spaced radial slots.

Format:

1	2	3	4	5	6	7	8	9	10
CSLOT3	EID	IDS1	IDS2	IDS3		RHO	B	M	

Example:

CSLOT3	100	1	3	2		3.0-3		6	

Field	Contents
EID	Element identification number. (Integer > 0)
IDSi	Identification number of connected GRIDS points. (Integer > 0)
RHO	Fluid density in mass units. (Real > 0.0; Default is the value of RHOD on the AXSLOT entry)
B	Fluid bulk modulus. (Real $\geq$ 0.0; Default is the value of BD on the AXSLOT entry)
M	Number of slots in circumferential direction. (Integer $\geq$ 0; Default is the value of MD on the AXSLOT entry)

Remarks:

1. CSLOT3 is allowed only if an AXSLOT entry is also present.
2. This element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO, B, or M are blank, then the RHOD, BD, or MD fields on the AXSLOT entry must be specified.
4. This element generates three plot elements, connecting points IDS1 to IDS2, IDS2 to IDS3, and IDS3 to IDS1.
5. If B=0.0, then the slot is considered to be an incompressible fluid.
6. If M=0, then no matrices for CSLOT3 elements are generated.

CSLOT4 Four Point Slot Element Connection

Defines an element connecting four points that solves the wave equation in two dimensions. Used in acoustic cavity analysis for the definition of evenly spaced radial slots.

Format:

1	2	3	4	5	6	7	8	9	10
CSLOT4	EID	IDS1	IDS2	IDS3	IDS4	RHO	B	M	

Example:

CSLOT4	101	1	3	2	4		6.2+4	3	

Field	Contents
EID	Element identification number. (Integer > 0)
IDSi	Identification number of connected GRIDS points. (Integer > 0)
RHO	Fluid density in mass units. (Real > 0.0; Default is the value of RHOD on the AXSLOT entry.)
B	Fluid bulk modulus. (Real $\geq$ 0.0; Default is the value of BD on the AXSLOT entry.)
M	Number of slots in circumferential direction. (Integer $\geq$ 0; Default is the value of MD on the AXSLOT entry.)

Remarks:

1. This entry is allowed only if an AXSLOT entry is also present.
2. This element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO, B, or M are blank, then the RHOD, BD, or MD fields on the AXSLOT entry must be specified.
4. This element generates four plot elements connecting points IDS1 to IDS2, IDS2 to IDS3, IDS3 to IDS4, and IDS4 to IDS1.
5. If B = 0.0, then the slot is considered to be an incompressible fluid.
6. If M = 0, then no matrices for CSLOT4 elements are generated

CSPOT (SOL 700) Spot Weld in the LS-DYNA Style for SOL 700 Only

Defines a complex or combined weld in the LS-DYNA style for use in SOL 700 only. Replaces CWELD for SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
CSPOT	EID	NSID	CID	FILTER	WINDOW		NPRT		
	TFAIL	EPSF	SN	SS	N	M			

Example:

CSPOT	1001	11	5	0	0		1		
	0.005	.07							

Field	Contents
EID	Unique element identification number. (Integer > 0, required no default)
NSID	ID of a set number containing the grid points comprising this weld (Integer > 0, required, no default)
CID	ID of a CORDi entry providing the local output coordinate system for this weld. (Integer > 0, or blank - blank is the same as zero indicating the basic coordinate system)
FILTER	Number of force vectors saved for filtering. This option can eliminate spurious failures due to numerical force spikes but memory will be larger if this option is envoked. Enter 0 for no filtering and N for a simple average of force components divided by N or the maximum number of force vectors that are stored for the time window option WINDOW. (Integer > 0, Default = 0)
NPRT	Control of weld force output in file RBDOUT. (Integer > 0, Default = 1) NPRT=1 data is output. NPRT=2 data is not output.
TFAIL	Failure time for this weld (Real > 0 or blank, Default = 1.0E20)
EPSF	Effective plastic strain at failure (Real > 0 or blank, Default = blank which means not used). Used for ductile failures.

Field	Contents
SN	Sn, normal force at failure (Real > 0 or blank, Default = blank which means not used) Used for brittle failures
SS	Ss, shear force at failure (Real > 0 or blank, Default = blank which means not used) Used for brittle failures.
N	Exponent n for normal force (real > 0 or blank, Default = blank which means no used) Used for brittle failures.
M	Exponent m for normal force (Real > 0 or blank, Default = blank which means not used) Used for brittle failures

Remarks:

1. No property entry is needed for the CSPOT entry.
2. Spotweld failure due to plastic straining occurs when the effective nodal plastic strain exceeds the input value, ε_{fail}^{p}. This option can model the tearing out of a spotweld from the sheet metal since the plasticity is in the material that surrounds the spotweld, not the spotweld itself. A least squares algorithm is used to generate the nodal values of plastic strains at the nodes from the element integration point values. The plastic strain is integrated through the element and the average value is projected to the nodes via a least square fit. This option should only be used for the material models related to metallic plasticity and can result in slightly increased run times.

   Brittle failure of the spotwelds occurs when:

$$\left(\frac{\max(f_n, 0)}{S_n}\right)^n + \left(\frac{|f_s|}{S_s}\right)^m \geq 1$$

   where f_n and f_s are the normal and shear interface force. Component f_n contributes for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. In **Figure 8-63** the ordering of the nodes is shown for the 2 node and 3 node spotwelds. This order is with respect to the local coordinate system where the local z-axis determines the tensile direction. The nodes in the spotweld may coincide. The failure of the 3 node spotweld may occur gradually with first one node failing and later the second node may fail. For n noded spotwelds the failure is progressive starting with the outer nodes (1 and n) and then moving inward to nodes 2 and n-1. Progressive failure is necessary to preclude failures that would create new rigid bodies.

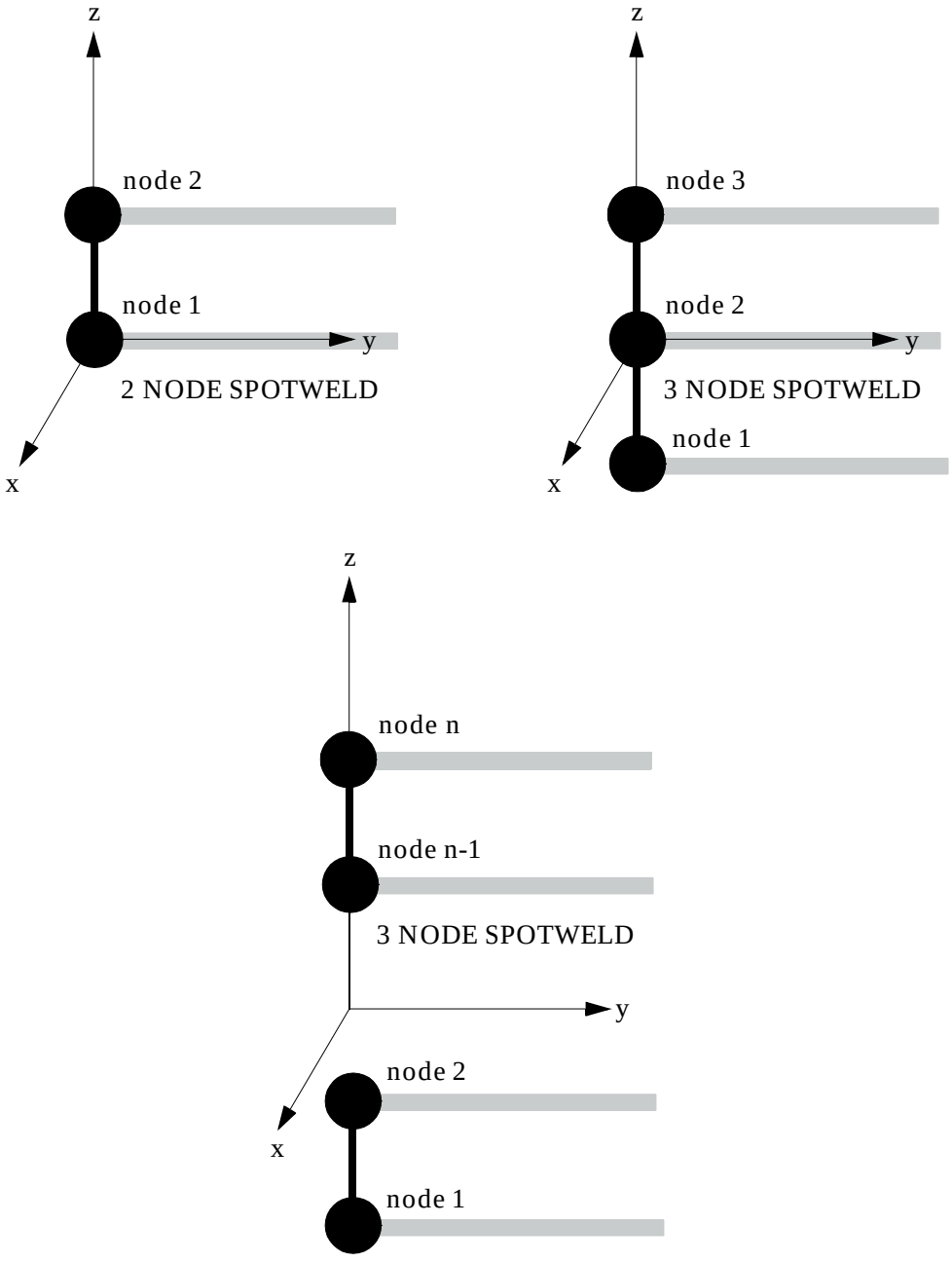


Figure 8-63

Nodal ordering and orientation of the local coordinate system is important for determining spotweld failure.

CSPR (SOL 700)

Springs with offsets for use in SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
CSPR	EID	PID	G1	G2	VID	IOP	S	PF	
	OFST								

Example:

CSPR	1	22	456	457			1.0		
	0.1								

Field	Contents	Type	Default
EID	Element ID. A unique number has to be used.	I > 0	Required
PID	Property ID of PSPRMAT entry	I > 0	Required
N1	Nodal point 1	I > 0	Required
N2	Nodal point 2. If zero, the spring / damper connects node N1 to ground.	I ≥ 0	0
IOP	Type of rectangular coordinate system. 1 CORDR1X 2 CORDR2X 3 CORDR3X	I > 0	1
VID	ID of CORDiX. Orientation option. The orientation option should be used cautiously since forces, which are generated as the nodal points displace, are not orthogonal to rigid body rotation unless the nodes are coincident. The type 6, 3D beam element, is recommended when orientation is required with the absolute value of the parameter SCOOR set to 2 or 3, since this option avoids rotational constraints. EQ.0: the spring / damper acts along the axis from node N1 to N2,	I > 0	0

Field	Contents	Type	Default
	NE.0: the spring/damper acts along the axis defined by the direction of coordinate CORDRiX system.		
S	Scale factor on forces.	R > 0	1.0
PF	Print flag:	I > 0	0
	EQ.0: forces are printed in DEFORC file		
	EQ.1: forces are not printed DEFORC file.		
OFFSET	Initial offset. The initial offset is a displacementor rotation at time zero. For example, a positive offset on a translational spring will lead to a tensile force being developed at time zero.	R > 0	0.0

CSSCHD Aerodynamic Control Surface Schedule Input

Defines a scheduled control surface deflection as a function of Mach number and angle of attack.

Format:

1	2	3	4	5	6	7	8	9	10
CSSCHD	SID	AESID	LALPHA	LMACH	LSCHD				

Example:

CSSCHD	5	ELEV	12	15	25				

Field	Contents
SID	Set identification number. (Integer > 0)
AESID	ID of an AESURF Bulk Data entry to which the schedule is being attached.
LALPHA	ID of an AEFACT Bulk Data entry containing a list of angles of attack (in radians) at which schedule information is provided. (Integer > 0: Default = no angle information provided.
LMACH	ID of an AEFACT Bulk Data entry containing a list of Mach numbers at which schedule information is provided. (Integer > 0; Default = no Mach information provided)
LSCHD	ID of an AEFACT Bulk Data entry which contains the scheduling information. See Remarks **4.** and **5.** (Integer > 0; No default)

Remarks:

1. Control system schedules must be selected with the Case Control command CSSCHD = SID.
2. The AESID cannot appear on an AELINK or TRIM Bulk Data entry for the same subcase.
3. The control surface deflection is computed using a linear interpolation for the Mach number provided on the associated TRIM entry and the angle of attack derived as part of the trim calculation.

4. The LSCHD data are provided as a list of deflections (in radians) as a function of Mach numbers and angles of attack. If there are NMACH Mach numbers and NALPHA angles of attack, the first NALPHA deflections are for the first Mach number, the next NALPHA are for the second Mach number, and so on, until the last NALPHA deflections are for the final Mach number.
5. If LALPHA is blank, LSCHD contains NMACH deflections to define the Mach schedule. If LMACH is blank, LSCHD contains NALPHA deflections to define the angle of attack schedule.
6. LALPHA and LMACH cannot be simultaneously blank. If LALPHA or LMACH are not blank, at least two values of angle of attack or Mach number must be defined in order to perform interpolation.
7. If the Mach number or angle of attack is outside the range specified by the tabulated values, the value at the table end is used. That is, data are not extrapolated.

CSUPER Secondary Superelement Connection

Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.

Format:

1	2	3	4	5	6	7	8	9	10
CSUPER	SSID	PSID	GP1	GP2	GP3	GP4	GP5	GP6	
	GP7	GP8	-etc.-						

Example:

CSUPER	120003	21	3	6	4	10			

Field	Contents
SSID	Coded identification number for secondary superelement. See Remark **1.** (Integer > 0)
PSID	Identification number for referenced primary superelement. See Remark **2.** (Integer > 0 or blank)
GPi	Grid or scalar point identification numbers of the exterior points of the secondary superelement. See Remark **3.** (Integer > 0)

Remarks:

1. The value of SSID is written in the form XXX0000 + n, where n is the referenced secondary superelement identification number and n must be less than 10000 and XXX is a displacement component sign reversal code as follows:

   The sign reversal code specifies the displacement component(s) normal to the plane of the mirror through which the reflection is to be made

   Blank or 0 — no reversal for identical superelement. If PSID is preceded by a minus sign and there is no xxx code on SSID, then a z-reversal mirror is generated.

   1x-reversal
   2y-reversal
   3z-reversal
   } Mirror Images

12x and y-reversal 23y and z-reversal 31z and x-reversal	Identical Images
123x, y and z-reversal	Mirror Images

2. If PSID = 0 or blank, the superelement boundary matrices are obtained from an external source (such as a database or external file). See also PARAM, "**EXTDRUNT**" on page 702.

   If PSID ≠ 0, the secondary superelement is identical to, or is a mirror image of, a primary superelement.

3. For identical or mirror image superelements, the grid point IDs, GPi, may appear in any order. However, if they are not in the same order as the external GRIDs of the primary superelement, then the SEQSEP entry is also required. In case of external superelements, the GRID IDs must be in the order that the terms in the associated matrices occur in.

4. Image superelements and their primaries must be congruent. The identical or mirror image superelement must have the same number of exterior grid points as its primary superelement. The exterior grid points of the image superelement must have the same relative location to each other as do the corresponding points of the primary superelement. The global coordinate directions of each exterior grid point of the image superelement must have the same relative alignment as those of the corresponding grid points of the primary superelement. If congruency is not satisfied because of round-off, then the tolerance may be adjusted with PARAM,CONFAC or DIAG 37.

5. For superelements from an external source, please refer to PARAMS "**EXTDR,**" on page 702, "**EXTDROUT,**" on page 702, "**EXTDRUNT**" on page 702 and "**EXTUNIT**" on page 707.

CSUPEXT Superelement Exterior Point Definition

Assigns exterior points to a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
CSUPEXT	SEID	GP1	GP2	GP3	GP4	GP5	GP6	GP7	

Example:

CSUPEXT	2	147	562	937					

Alternate Format and Example:

CSUPEXT	SEID	GP1	“THRU”	GP2					
CSUPEXT	5	12006	THRU	12050					

Field	Contents
SEID	Identification number of a primary superelement. (Integer > 0)
GPi	Grid or scalar point identification number in the downstream superelement or residual structure. (Integer > 0 or “THRU”; for “THRU” option, GP1 < GP2)

Remarks:

1. Grid or scalar points are connected (that is, are exterior) to a superelement only if they are connected by structural, rigid, or plot elements. MPC entries are not regarded as elements. This entry is a means of providing connectivity for this purpose.
2. Open sets are allowed with the “THRU” option.
3. Scalar points may be interior to the residual structure (SEID = 0) only.
4. This entry may be applied only to the primary superelements. The CSUPER entry is used for secondary superelements (identical image, mirror image, and external superelements).

CTETRA Four-Sided Solid Element Connection

Defines the connections of the four-sided solid element with four to ten grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTETRA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10					

Example:

CTETRA	112	2	3	15	14	4	103	115	
	5	16	8	27					

Field	Contents	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID or PLSOLID entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer ≥ 0 or blank	Required

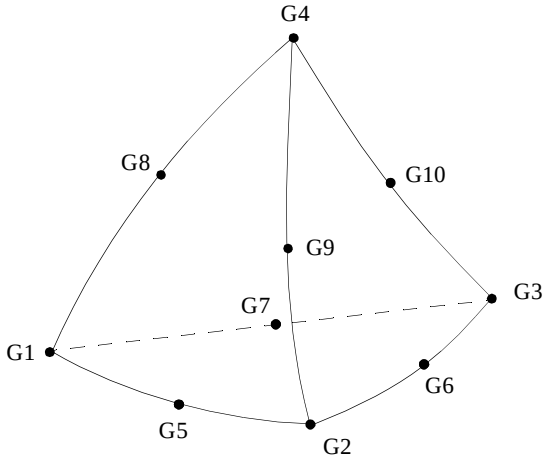


Figure 8-64 CTETRA Element Connection

Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved, i.e., G1, G2, G3 define a triangular face; G1, G8, and G4 are on the same edge, etc.
3. The edge points, G5 to G10, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element in all cases.
4. Components of stress are output in the material coordinate system except for hyperelastic elements, which are output in the basic coordinate system.
5. For nonhyperelastic elements, the element coordinate system is derived from the three vectors R, S, and T, which join the midpoints of opposite edges.

   R vector joins midpoints of edges G1-G2 and G3-G4.

   S vector joins midpoints of edges G1-G3 and G2-G4.

   T vector joins midpoints of edges G1-G4 and G2-G3.

   The origin of the coordinate system is located at G1. The element coordinate system is chosen as close as possible to the R, S, and T vectors and points in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that, if the R, S, and T vectors are described in the element coordinate system, a 3x3 positive definite symmetric matrix would be produced.)

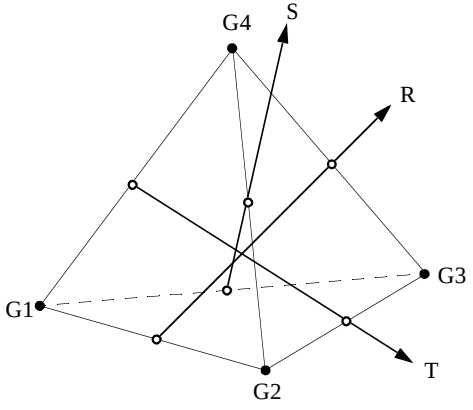


Figure 8-65 CTETRA Element R, S, and T Vectors

6. It is recommended that the edge points be located within the middle third of the edge.
7. For hyperelastic elements, the plot codes are specified under the CTETRAFD element name in “**Item Codes**” on page 873.
8. If a CTETRA element is referenced by a PSET or PVAL entry, then p-version formulation is used and the element can have curved edges.
    - If a curved edge of a p-element is shared by an h-element without midside nodes, the geometry of the edge is ignored and set straight.
    - Elements with midside nodes cannot be p-elements and edges with midside nodes cannot be shared by p-elements.
9. By default, all of the six edges of the element are considered straight unless:
    - For p-elements, there is an FEEDGE or FEFACE entry that contains the two grids of any edge of this element. In this case, the geometry of the edge is used in the element.
    - For h-elements, any of G5 through G10 are specified.

CTRIA3 Triangular Plate Element Connection

Defines an isoparametric membrane-bending or plane strain triangular plate element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
		TFLAG	T1	T2	T3				

Example:

CTRIA3	111	203	31	74	75	3.0	0.98		
			1.77	2.04	2.09				

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL, PCOMP or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers>0, all unique)
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. Use DIAG 38 to print the computed THETA values. MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. (Integer $\geq$ 0; if blank, then THETA = 0.0 is assumed.)

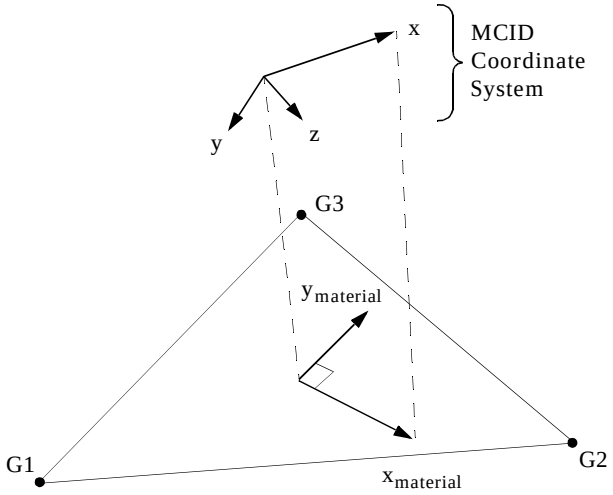


Figure 8-66 MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark **3.** ZOFFS is ignored for hyperelastic elements. (Real)
TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. (Real $\geq$ 0.0 or blank, not all zero. See Remark **4.** for default.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank; not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.

3. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

   The specification of offset vectors gives wrong results in solution sequences with geometric nonlinear analysis (SOL 106, 129, 153, 159 and PARAM,LGDISP>0). The differential stiffness is computed correctly in MSC.Nastran 2004 and later versions for (SOLs 5, 16, 105 and 200 and SOL 106 with PARAM,LGDISP≤0).

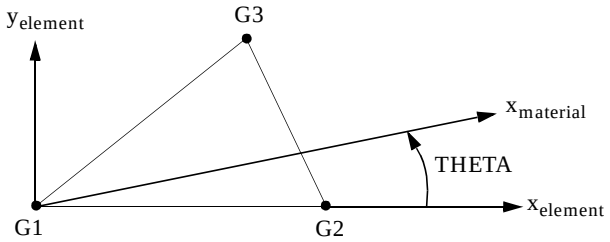


Figure 8-67 CTRIA3 Element Geometry and Coordinate Systems

4. The reference coordinate system for the output of stress, strain and element force depends on the element type.
   - For CTRIA3 elements, which are not p-elements and not hyperelastic, the reference coordinate system for output is the element coordinate system.
   - For CTRIA3 elements referenced by a PSET of PVAL entry, the stresses, strains and element forces are output in the local tangent plane of the element. The local tangents are oriented in a user defined direction which is uniform across a set of elements. By default, the local tangent x-direction is oriented in the positive x-direction of the basic coordinate system.
   - For hyperelastic elements the stress and strain are output according to CID on the PLPLANE entry.
5. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in “**Item Codes**” on page 873.

6. SYSTEM(218), alias T3SKEW, allows the user to control the minimum vertex angle for TRIA3 elements at which USER WARNING MESSAGE 5491 is issued. The default value is 10. degrees.
7. If a CTRIA3 element is referenced by a PSET or PVAL entry, then a p-version formulation is used and the element can have curved edges.
    - If a curved edge of a p-element is shared by an h-element CTRIA3, the geometry of the edge is ignored and set straight.
8. By default, all of the three edges of the element are considered straight unless the element is a p-element and the edges are associated to curved geometry with FEEDGE or FEFACE Bulk Data entries.

CTRIA6 Curved Triangular Shell Element Connection

Defines a curved triangular shell element or plane strain with six grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA6	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA or MCID	ZOFFS	T1	T2	T3	TFLAG			

Example:

CTRIA6	302	3	31	33	71	32	51	52	
	45	.03	.020	.025	.025				

Field	Contents
EID	Element Identification number. (Integer > 0)
PID	Property identification number of PSHELL, PCOMP, or PLPLANE entry. (Integer > 0)
G1, G2, G3	Identification numbers of connected corner grid points. (Unique Integers > 0)
G4, G5, G6	Identification number of connected edge grid points. Optional data for any or all three points. (Integer $\geq$ 0 or blank)
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. (Integer $\geq$ 0; if blank, then THETA = 0.0 is assumed)

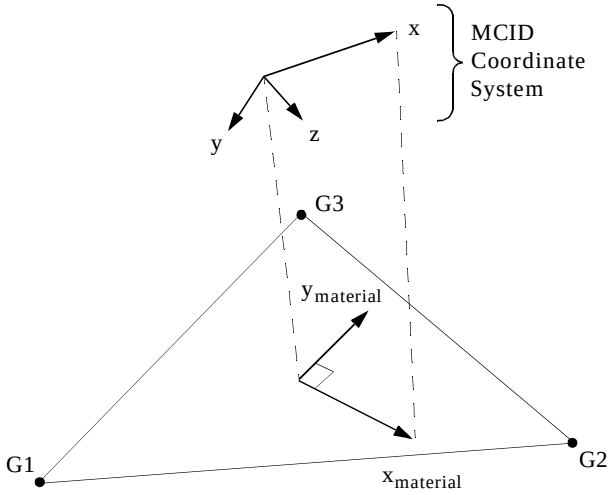


Figure 8-68 MCID Coordinate System Definition

ZOFFS — Offset from the surface of grid points to the element reference plane; see Remark **6.** ZOFFS is ignored for hyperelastic elements. (Real)

Ti — Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. (Real $\geq$ 0.0 or blank, not all zero. See Remark **4.** for default.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

TFLAG — An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element IDs.
2. Grid points G1 through G6 must be numbered as shown in **Figure 8-69**.
3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between $x_{material}$ and the line of constant η.
4. T1, T2, and T3 are optional. If they are not supplied and no TFLAG, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
5. It is recommended that the midside grid points be located within the middle third of the edge.

6. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

   The specification of offset vectors gives wrong results in solution sequences that compute differential stiffness: linear buckling analysis provided in SOLs 105 and 200; SOLs 103 and 107 through 112 with the STATSUB command; and geometric nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1.

7. If all midside grid points are deleted, then the element will be excessively stiff and the transverse shear forces will be incorrect. A User Warning Message is printed. A CTRIA3 element entry is recommended instead. If the element is hyperelastic, then the element is processed identically to the hyperelastic CTRIA3 element.

8. For a description of the element coordinate system, see "**Shell Elements (CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR)**" on page 131 of the *MSC.Nastran Reference Guide*. Stresses and strains are output in the local coordinate system identified by x_l and y_l in **Figure 8-69**. For hyperelastic elements, stresses and strains are output in the coordinate system defined by the CID field on the PLPLANE entry.

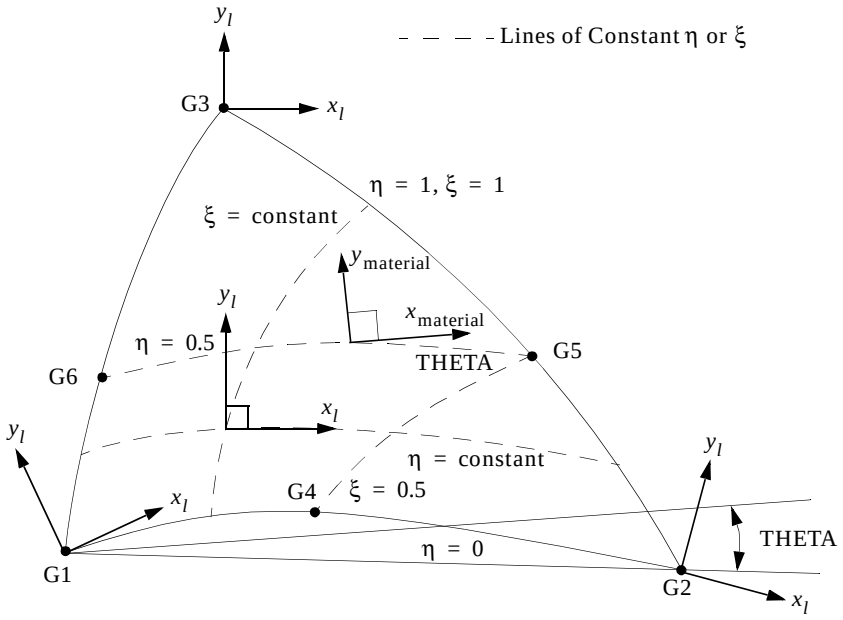


Figure 8-69 CTRIA6 Element Geometry and Coordinate Systems

9. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in "**Item Codes**" on page 873.

CTRIAR Triangular Plate Element Connection

Defines an isoparametric membrane-bending triangular plate element. This element has a normal rotational degrees-of-freedom. It is a companion to the CQUADR element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAR	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
		TFLAG	T1	T2	T3				

Example:

CTRIAR	111	203	31	74	75	3.0			
			1.77	2.04	2.09				

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PSHELL or PCOMP entry. (Integer > 0; Default = EID)
G1, G2, G3	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system (defined by the CORDij entry or zero for the basic coordinate system) onto the surface of the element. Use DIAG 38 to print the computed THETA values. For SOL 600, only CORD2R is allowed. (Integer $\geq$ 0; if blank, then THETA = 0.0 is assumed)
ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 5.

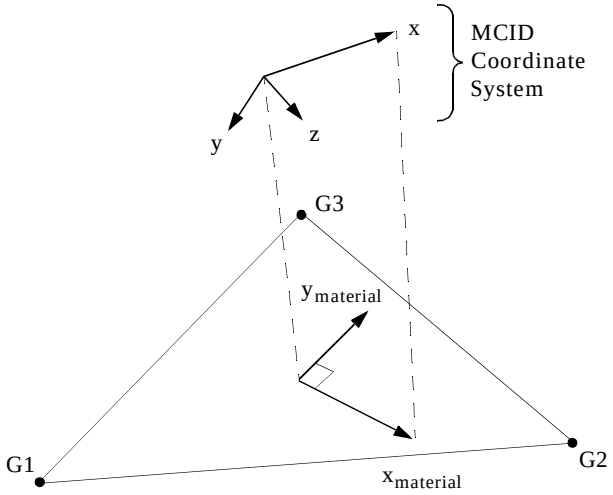


Figure 8-70 MCID Coordinate System Definition

TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. (Real $\geq$ 0.0 or blank, not all zero. See Remark **4.** for default.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
3. Stresses are output in the element coordinate system at the centroid and grid points G1 through G3.
4. The rotational degrees-of-freedom normal to the element are active in the element formulation and must not be constrained unless at a boundary. If they are constrained, then inaccurate results will be obtained.

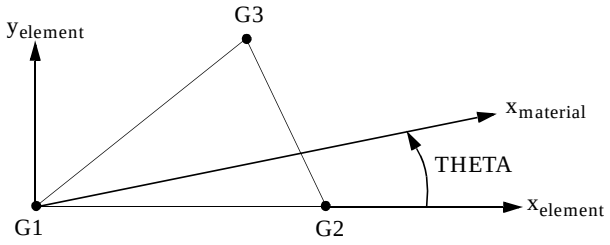


Figure 8-71 CTRIAR Element Geometry and Coordinate Systems

5. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

   The specification of offset vectors gives wrong results in solution sequences that compute differential stiffness: linear buckling analysis provided in SOLs 105 and 200; SOLs 103 and 107 through 112 with the STATSUB command; and geometric nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1.

CTRIAX Fully Nonlinear Axisymmetric Element

Defines an axisymmetric triangular element with up to 6 grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAX	EID	PID	G1	G2	G3	G4	G5	G6	

Example:

CTRIAX	111	203	31	74	75				

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PLPLANE entry. (Integer > 0)
G1, G2, G3	Identification numbers of connected corner grid points. Required data for all three grid points. (Unique Integers > 0)
G4, G5, G6	Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer $\geq$ 0 or blank)

Remarks:

1. Element identification numbers must be unique with respect to all other element IDs of any kind.
2. Gi must be numbered as shown in **Figure 8-72**.
3. It is recommended that the edge points be located within the middle third of the edge.
4. The plot codes are specified under the CTRIAXFD element name in "**Item Codes**" on page 873.
5. The grid points of the axisymmetric element must lie on the x-y plane of the basic coordinate system. Stress and strain are output in the basic coordinate system.

6. A concentrated load (e.g., FORCE entry) at Gi is divided by the radius to Gi and then applied as a force per unit circumferential length. For example, in order to apply a load of 100 N / m on the circumference at G1, which is located at a radius of 0.5 m, then the magnitude specified on the static load entry must result in:

$$(100\ \mathrm{N/m}) \cdot (0.5\mathrm{m}) = 50\mathrm{N}$$

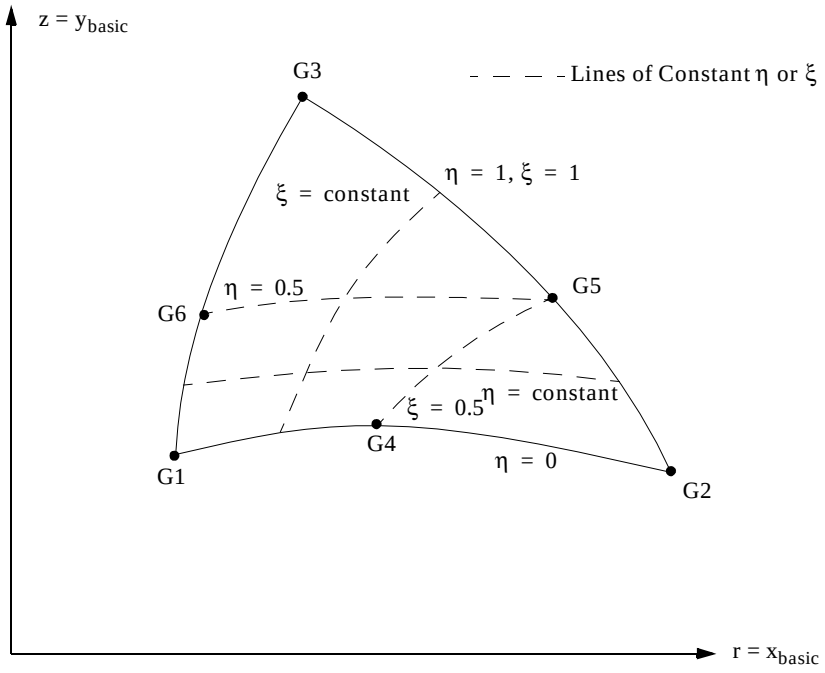


Figure 8-72 CTRIAX Element Coordinate System

CTRIAX6 Axisymmetric Triangular Element Connection

Defines an isoparametric and axisymmetric triangular cross section ring element with midside grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAX6	EID	MID	G1	G2	G3	G4	G5	G6	
	TH								

Example:

CTRIAX6	22	999	10	11	12	21	22	32	
	9.0								

Field	Contents
EID	Element identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
Gi	Grid point identification numbers of connected points (unique Integers > 0, or blank for deleted nodes.)
TH	Material property orientation angle in degrees. (Real; Default = 0.0)

Remarks:

1. The grid points must lie in the x-z plane of the basic coordinate system, with $x = r \geq 0$. The grid points must be listed consecutively beginning at a vertex and proceeding around the perimeter in either direction. Corner grid points G1, G3, and G5 must be present. Any or all edge grid points G2, G4, or G6 may be deleted. Note that the alternate corner-edge grid point pattern is different from the convention used on the CTRIA6 element.
2. For structural problems, the MID may refer to a MAT1 or MAT3 entry.
3. The continuation is optional.
4. Material properties (if defined on a MAT3 entry) and stresses are given in the (r_m, z_m) coordinate system shown in **Figure 8-74**.

5. A concentrated load (e.g., FORCE entry) at Gi is divided by the 2π times the radius to Gi and then applied as a force per unit circumferential length. For example, in order to apply a load of 100 N / m on the circumference at G1 (which is located at a radius of 0.5 m), the magnitude of the load specified on the static load entry must result in:

$$(100\ \text{N/m}) \cdot 2\pi \cdot (0.5\ \text{m}) = 314.159\ \text{N}$$

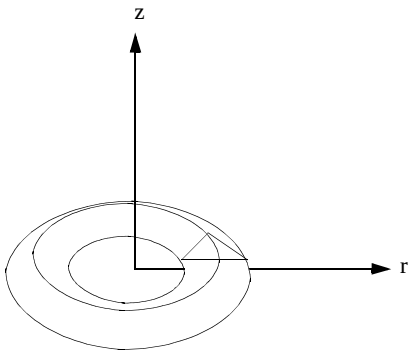


Figure 8-73 CTRIAX6 Element Idealization

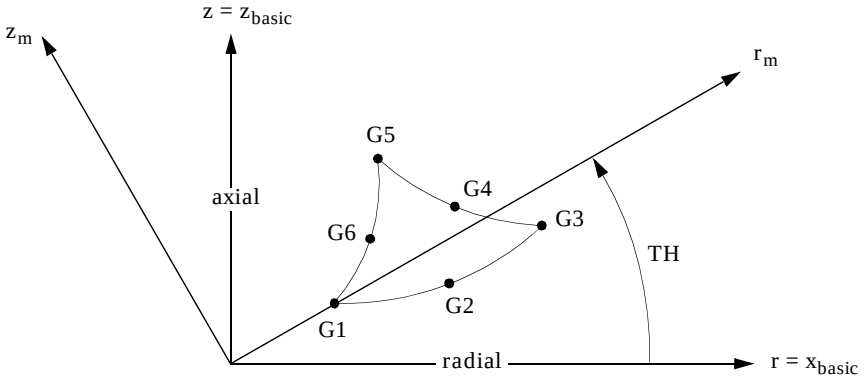


Figure 8-74 CTRIAX6 Element Geometry and Coordinate Systems

CTUBE Tube Element Connection

Defines a tension-compression-torsion tube element.

Format:

1	2	3	4	5	6	7	8	9	10
CTUBE	EID	PID	G1	G2					

Example:

CTUBE	12	13	21	23					

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PTUBE entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Only one tube element may be defined on a single entry.

CVISC Viscous Damper Connection

Defines a viscous damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CVISC	EID	PID	G1	G2					

Example:

CVISC	21	6327	29	31					

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PVISC entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Only one viscous damper element may be defined on a single entry.

CWELD Weld or Fastener Element Connection

Defines a weld or fastener connecting two surface patches or points.

Format PARTPAT:

1	2	3	4	5	6	7	8	9	10
CWELD	EWID	PWID	GS	“PARTPAT”	GA	GB			
	PIDA	PIDB							
	XS	YS	ZS						

Example:

CWELD	101	8	203	PARTPAT					
	21	33							

Alternate formats and examples:

Format ELPAT:

CWELD	EWID	PWID	GS	“ELPAT”	GA	GB			
	SHIDA	SHIDB							
	XS	YS	ZS						

Example:

CWELD	103	5	403	ELPAT					
	309	511							

Format ELEMID:

CWELD	EWID	PWID	GS	“ELEMID”	GA	GB			
	SHIDA	SHIDB							

Example:

CWELD	103	5	403	ELEMID					
	309	511							

Format GRIDID:

CWELD	EWID	PWID	GS	“GRIDID”	GA	GB	SPTYP		
	GA1	GA2	GA3	GA4	GA5	GA6	GA7	GA8	
	GB1	GB2	GB3	GB4	GB5	GB6	GB7	GB8	

Example:

CWELD	7	29	233	GRIDID			QT		
	15	28	31	35	46	51	55	60	
	3	5	8						

Format ALIGN:

CWELD	EWID	PWID		“ALIGN”	GA	GB			

Example:

CWELD	7	29		ALIGN	103	259			

Field	Contents	Type	Default
EWID	CWELD element identification number. See Remark 1.	Integer > 0	Required
PWID	Property identification number of a PWELD entry.	Integer > 0	Required
GS	Identification number of a grid point which defines the location of the connector. See Remarks 2. and 3.	Integer > 0 or blank	
“PARTPAT”	Character string indicating the type of connection. The format of the subsequent entries depends on the type. “PARTPAT”, for example, indicates that the connectivity of surface patch A to surface patch B is defined with two property identification numbers of PSHELL entries, PIDA and PIDB, respectively. The “PARTPAT” format connects up to 3x3 elements per patch. See Remark 4.	Character	Required

Field	Contents	Type	Default
GA, GB	Grid point identification numbers of piercing points on surface A and surface B, respectively. See Remark 5.	Integer > 0 or blank	blank
PIDA, PIDB	Property identification numbers of PSHELL entries defining surface A and B respectively.	Integer > 0	Required for “PARTPAT”
XS, YS, ZS	Coordinates of spot weld location in basic. See Remark 2.	Real	Required if GS and GA are not defined.

For the alternate formats, the field contents are described below:

Field	Contents	Type	Default
“ELPAT”	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The “ELPAT” format connects up to 3x3 elements per patch. See Remark 6.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A and B, respectively.	Integer > 0	Required for “ELPAT”
“ELEMID”	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The “ELEMID” format connects one shell element per patch. See Remark 7.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A and B, respectively.	Integer > 0	Required for “ELEMID”

Field	Contents	Type	Default
“GRIDID”	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two sequences of grid point identification numbers, GAi and GBi, respectively. The “GRIDID” format connects the surface of any element. See Remark **8.**	Character	Required
SPTYP	Character string indicating types of surface patches A and B. SPTYP=”QQ”, “TT”, “QT”, “TQ”, “Q” or “T”. See Remark **9.**	Character	Required for “GRIDID”
GAi	Grid identification numbers of surface patch A. GA1 to GA3 are required. See Remark **10.**	Integer > 0	Required for “GRIDID”
GBi	Grid identification numbers of surface patch B. See Remark **10.**	Integer > 0	
“ALIGN”	Character string indicating that the connectivity of surface A to surface B is defined with two shell vertex grid points GA and GB, respectively. See Remark **11.**	Character	Required
GA, GB	Vertex grid identification number of shell A and B, respectively.	Integer > 0	Required for “ALIGN”

Remarks:

1. CWELD defines a flexible connection between two surface patches, between a point and a surface patch, or between two shell vertex grid points. See **Figure 8-75** through **Figure 8-79**.
2. Grid point GS defines the approximate location of the connector in space. GS is projected on surface patch A and on surface patch B. The resulting piercing points GA and GB define the axis of the connector. GS must have a normal projection on surface patch A and B. GS does not have to lie on the surface patches. GS is ignored for format “ALIGN”. GA is used instead of GS if GS is not specified. For the formats “ELPAT” and “PARTPAT,” if GS and GA are not specified, then XS, YS, and ZS must be specified.

3. The connectivity between grid points on surface patch A and grid points on surface patch B is generated depending on the location of GS and the cross sectional area of the connector. Diagnostic print outs, checkout runs and non default settings of search and projection parameters are requested on the SWLDPRM Bulk Data entry. It is recommended to start with the default settings.
4. The format “PARTPAT” defines a connection of two shell element patches A and B with PSHELL property identification numbers PIDA and PIDB, respectively. The two property identification numbers must be different, see **Figure 8-75**. Depending on the location of the piercing points GA, GB and the size of the diameter D, the number of connected elements per patch ranges from a single element up to 3x3 elements. The diameter D is defined on the PWELD property entry. For this option, shell element patches A and B are allowed to share a common grid.

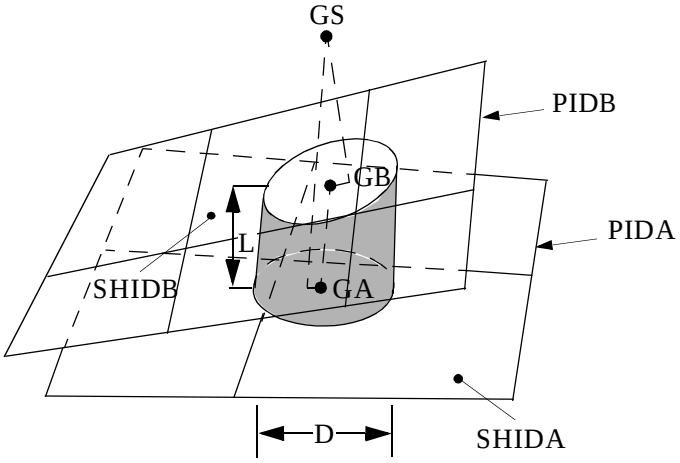


Figure 8-75 Patch to Patch Connection Defined with the Formats PARTPAT or ELPAT

5. The definition of the piercing grid points GA and GB is optional for all formats with the exception of the format “ALIGN”. If GA and GB are given, GS is ignored. If GA and GB are not specified, they are generated from the normal projection of GS on surface patches A and B. For the formats “ELEMID” and “GRIDID,” internal identification numbers are generated for GA and GB starting with 101e+6 by default. The offset number can be changed with PARAM,OSWPPT. If GA and GB are specified, they must lie

on or at least have a projection on surface patches A and B, respectively. The locations of GA and GB are corrected so that they lie on surface patches A and B within machine precision accuracy. The length of the connector is the distance of grid point GA to GB.

6. The format “ELPAT” defines a connection of two shell element patches A and B with shell element identification numbers SHIDA and SHIDB, see **Figure 8-75**. The connectivity is similar to the format “PARTPAT”. Depending on the location of the piercing points GA, GB and the size of the diameter D, the number of connected elements per patch may range from a single element up to 3x3 elements. For this option, shell element patches A and B are allowed to share a common grid.

7. The format “ELEMID” defines a connection of two shell element patches A and B with shell element identification numbers SHIDA and SHIDB, see **Figure 8-76**. The connectivity is restricted to a single element per patch regardless of the location of GA, GB and regardless of the size of the diameter of the connector. In addition, the format “ELEMID” can define a point to patch connection if SHIDB is left blank, see **Figure 8-77**. Then grid GS is connected to shell SHIDA.

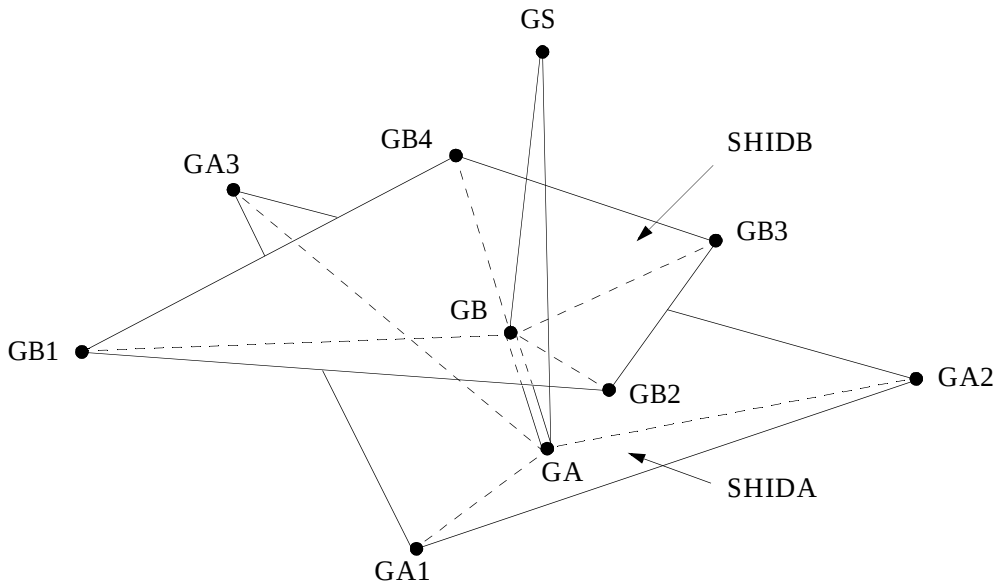


Figure 8-76 Patch to Patch Connection Defined with Format ELEMID or GRIDID

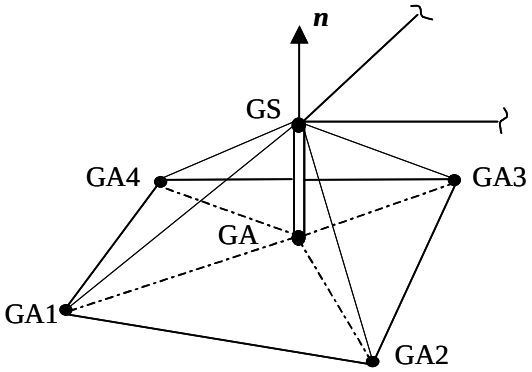


Figure 8-77 Point to Patch Connection Defined with Format ELEMID or GRID.

8. The format “GRIDID” defines a connection of two surface patches A and B with a sequence of grid points GAi and GBi, see **Figure 8-76**. In addition, the format “GRIDID” can define a point to patch connection if all GBi fields are left blank, see **Figure 8-77**. Then grid GS is connected to grids GAi. The grids GAi and GBi do not have to belong to shell elements.

9. SPTYP defines the type of surface patches to be connected. SPTYP is required for the format "GRIDID" to identify quadrilateral or triangular patches. The combinations are:

SPTYP	Description
QQ	Connects a quadrilateral surface patch A (Q4 to Q8) with a quadrilateral surface patch B (Q4 to Q8).
QT	Connects a quadrilateral surface patch A (Q4 to Q8) with a triangular surface patch B (T3 to T6).
TT	Connects a triangular surface patch A (T3 to T6) with a triangular surface patch B (T3 to T6).
TQ	Connects a triangular surface patch A (T3 to T6) with a quadrilateral surface patch B (Q4 to Q8).
Q	Connects the shell vertex grid GS with a quadrilateral surface patch A (Q4 to Q8) if surface patch B is not specified.
T	Connects the shell vertex grid GS with a triangular surface patch A (T3 to T6) if surface patch B is not specified.

10. GAi are required for the format "GRIDID". At least 3 and at most 8 grid IDs may be specified for GAi and GBi, respectively. The rules of the triangular and quadrilateral elements apply for the order of GAi and GBi, see **Figure 8-78**. Missing midside nodes are allowed.

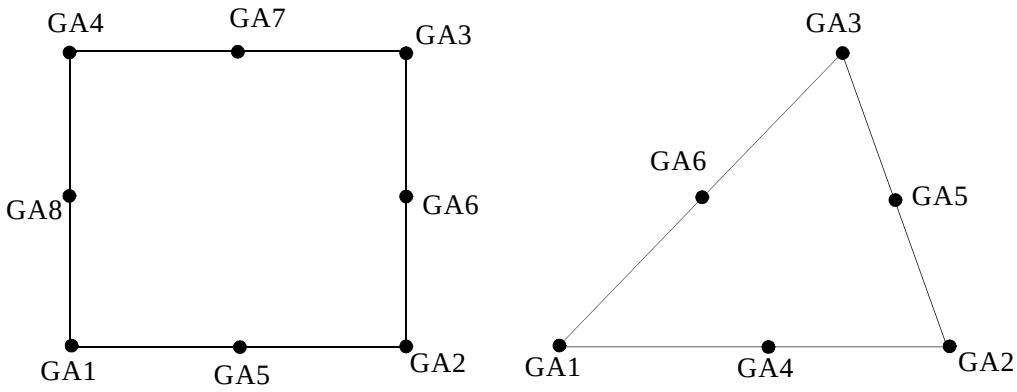


Figure 8-78 Quadrilateral and Triangular Surface Patches defined with Format GRIDID

11. The format "ALIGN" defines a point to point connection, see **Figure 8-79**. GA and GB are required, they must be existing vertex nodes of shell elements. For the other formats, GA and GB are not required. Two shell normals in the direction GA-GB are generated at GA and GB, respectively.

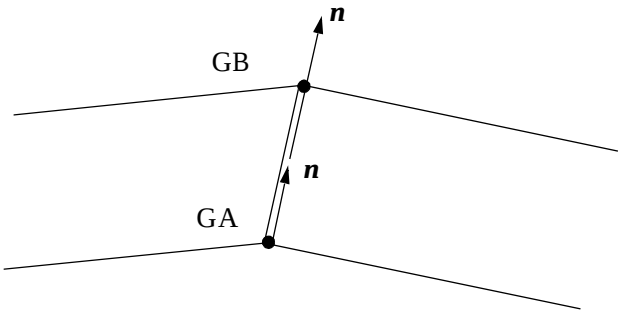


Figure 8-79 Point to Point Connection Defined with Format ALIGN

12. Forces and moments are output in the element coordinate system, see **Figure 8-80**. The element coordinate system is constructed using the following rules:

The element x-axis points from GA to GB.

$$e_1 = \frac{x_B - x_A}{\left\| x_B - x_A \right\|} \quad \text{element x-axis}$$

In case of zero length, the normal of shell A is taken. All vector components are in basic if not noted otherwise.

Find the smallest component j of e_1

$$e_1^j = \min_{i = 1, 2, 3} \left\{ e_1^i \right\}.$$

In case of two equal components we take the one with the smaller i. The corresponding basic vector

$$b_j \text{ , e.g., for } j = 3,\ b_3 = \left\{ \begin{array}{c} 0 \\ 0 \\ 1 \end{array} \right\}$$

provides a good directional choice for e_2. In addition, the vector e_2 must be orthogonal to e_1.

$$\tilde{e_2} = b_j - \frac{e_1^T b_j}{e_1^T e_1} e_1 \qquad e_2 = \frac{\tilde{e_2}}{\left\| \tilde{e_2} \right\|} \quad \text{element y-axis}$$

and e_3 is just the cross product

$$e_3 = e_1 \times e_2 \quad \text{element z-axis}$$

The final transformation matrix is

$$T_{be} = \left[e_1 \middle| e_2 \middle| e_3 \right]$$

13. The output format of the forces and moments including the sign convention is identical to the CBAR element, see "**Element Force Item Codes**" on page 917.

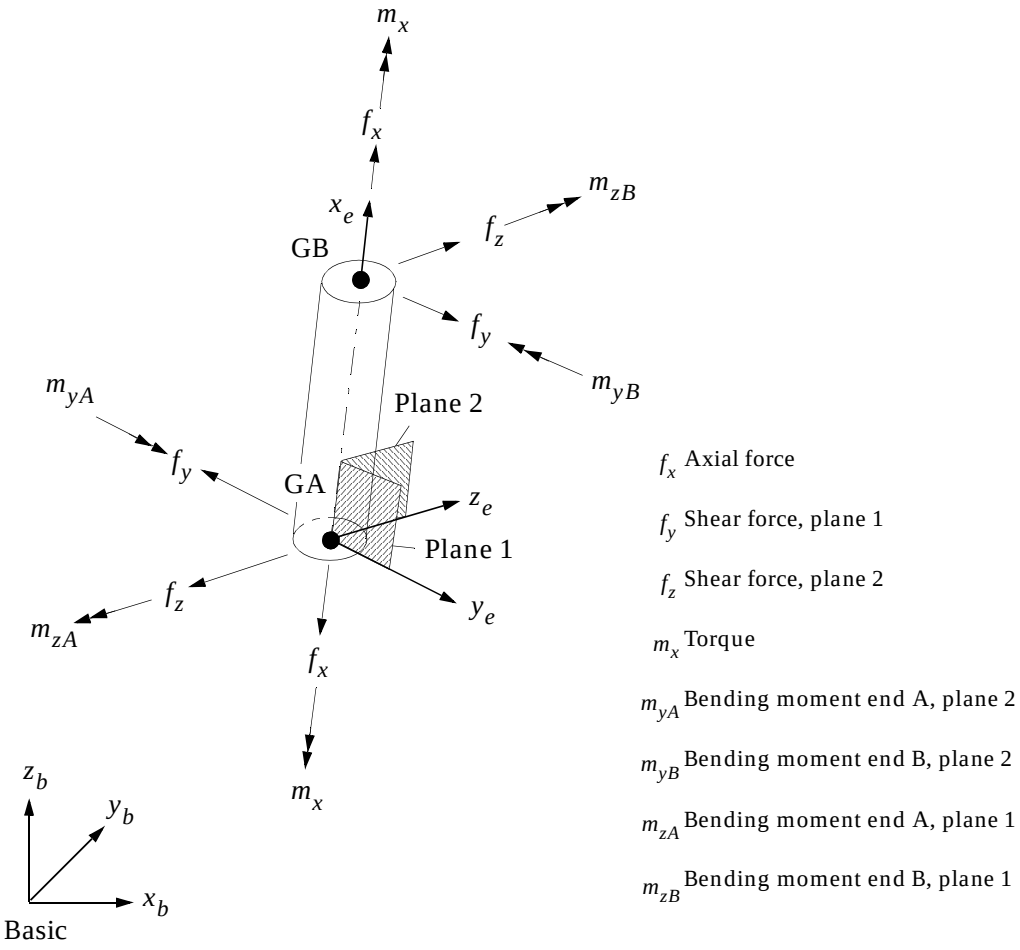


Figure 8-80 Element Coordinate System and Sign Convention of Element Forces

CWSEAM A Shell Patch Seam Connection

Defines a seam element connecting two surface patches.

Format:

1	2	3	4	5	6	7	8	9	10
CWSEAM	EID	PID		“GRIDID”		PIDA	PIDB		
	GS	GE							

Alternate Format:

1	2	3	4	5	6	7	8	9	10
CWSEAM	EID	PID		“XYZ”		PIDA	PIDB		
	XS	YS	ZS	XE	YE	ZE			

Example:

CWSEAM	3	17		GRIDID		2305	7116		
	30024	65467							

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PWSEAM entry. (Integer > 0)
LTYPE	Connectivity search type. (Character) If LTYPE=”GRIDID”, location is defined by GS and GE. If LTYPE=”XYZ”, location is defined by two XYZ locations.
PIDA,PIDB	Property identification numbers of PSHELL entries defining surface A and B respectively. (Integer > 0)
GS, GE	Grid identification numbers of piercing points on surface A and B of the start and end of the seam. (Integer > 0)
XS,YS,ZS	Location of the start of the seam in basic coordinate system. (Real or blank)
XE,YE,ZE	Location of the end of the seam in basic coordinate system. (Real or blank)

Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. GS and GE define the start and end points of the seam element. At these points and using the value W specified on the PWSEAM entry, surface patches A and B are determined. Points projected onto the surface patches A and B from GS and GE are used to determine the auxiliary points that form faces of a CHEXA element. These auxiliary points are then connected to the physical grids of the patches. The total number of unique physical grids ranges from a possibility of 6 to 32 grids. The auxiliary points must have a projection on patches A and B, but they do not have to lie on patch A or B.
3. A maximum of three shell elements of patch A and three shell elements of patch B can be connected with one CWSEAM element, see **Figure 8-81**.

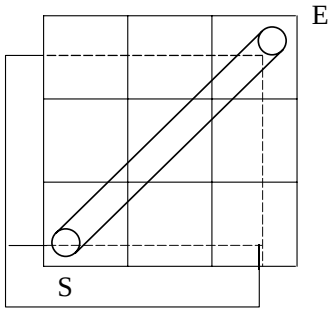


Figure 8-81 Connected Shell Elements for a CWSEAM Element

CYAX Grid Points on Axis of Symmetry

Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CYAX	G1	G2	G3	G4	G5	G6	G7	G8	
	G9	G10	-etc.-						

Example:

CYAX	27	152	THRU	160	192	11			

Field	Contents
Gi	A list of grid points on the axis of symmetry. (Integer > 0 or Character “THRU”)

Remarks:

1. The displacement coordinate system (see CD field on GRID entry) for a grid point lying on the axis of symmetry must be a rectangular system with the z-component of motion aligned with the axis of symmetry. The positive axis of symmetry is defined so that the azimuthal angle from positive side 1 to side 2 of a segment is in the same direction as the angle from T1 to T2 for the axis point. This is consistent with left- or right-hand rule.
2. If the dihedral symmetry option (STYPE = “DIH” on the CYSYM entry) is selected, the y-axis must be perpendicular to side 1.
3. Grid points lying on the axis of symmetry may be constrained by SPCs but not by MPCs. If the number of segments is three or more, SPCs must be applied to both components 1 and 2 or to neither, and SPCs must be applied to both components 4 and 5 or to neither in order to satisfy symmetry. In addition, the degrees-of-freedom (not constrained by SPCs, if any) at these grid points must be in the analysis set (a-set). If all degrees-of-freedom of grid points on the axis of symmetry are constrained by SPCs (including heat transfer, where there is only one degree-of-freedom), the grid point should not be listed on the CYAX entry.

4. Grid points lying on the axis of symmetry must not be defined on side 1 or on side 2 by means of a CYJOIN entry.
5. The word “THRU” must not appear in fields 2 or 9.

CYJOIN Cyclic Symmetry Boundary Points

Defines the boundary points of a segment in cyclic symmetry problems.

Format:

1	2	3	4	5	6	7	8	9	10
CYJOIN	SIDE	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

CYJOIN	1	T2	7	9	16	THRU	33	64	
	72	THRU	89						

Field	Contents
SIDE	Side identification. (Integer 1 or 2)
C	Type of coordinate system used on boundaries of dihedral or axisymmetry problems. See Remark **3.** (Character: “T1", “T2", “T3", “R”, “C”, “S”)
Gi	Grid or scalar point identification numbers. (Integer > 0 or Character “THRU”)

Remarks:

1. CYJOIN entries are used only for cyclic symmetry problems. The CYSYM entry must be used to specify rotational, dihedral, or axisymmetry.
2. For rotational or axisymmetry problems, there must be one logical entry for SIDE = 1 and one for SIDE = 2. The two lists specify grid points to be connected; therefore, both lists must have the same length.
3. For dihedral problems, side 1 refers to the boundary between segments and side 2 refers to the middle of a segment. For dihedral and/or AXI type of symmetry, the grid point degree-of-freedom that is normal to the boundary must be specified in field 3 as “T1", “T2", or “T3" (“R”, rectangular, and “C”, cylindrical, are the same as “T2" while “S”, spherical, is the same as “T3"). For scalar and extra points with one degree-of-freedom, these should be specified as blank, “T2", or “T3" if they are to have the same sign, and “T1", if the two connected points are to be opposite in sign.

4. All components of displacement at boundary points are connected to adjacent segments except those constrained by SPCi, MPC, or OMITi entries.
5. The points on the axis of symmetry of the model, defined in the CYAX entry must not be defined as a side 1 or side 2 point by means of this entry.
6. The word “THRU” may not appear in fields 4 or 9 of the parent entry and fields 2 or 9 on the continuation entries.
7. All grid points that are implicitly or explicitly referenced must be defined.
8. For rotational and axisymmetry problems, the displacement coordinate systems must be consistent between sides 1 and 2. This is best satisfied by the use of a spherical or cylindrical coordinate system.

CYSUP Fictitious Supports for Cyclic Symmetry

Defines fictitious supports for cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CYSUP	GID	C							

Example:

CYSUP	16	1245							

Field	Contents
GID	Grid point identification number. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

Remarks:

1. Components of motion defined on this entry may not appear on SPC, SPC1, OMIT, OMIT1 entries, or in rigid elements or multipoint constraints as dependent degrees-of-freedom.
2. Supports are applied at the grid point identified in field 2 to prevent rigid body motions in static analysis, or to define rigid body modes in eigenvalue analysis. All degrees-of-freedom should be at a single grid point. In other words, there can only be one such supported grid point in the model. The supports are applied only to the cyclic components of order k=0 or k=1. In order to satisfy conditions of symmetry, certain restrictions are placed on the location of the grid point and the orientation of its displacement coordinate system, as shown in the following table:

Symmetry Option (STYPE on the CYSYM entry)	ROT	ROT	DIH	DIH	DIH
Number of Segments, N	2	$\geq$3	1	2	$\geq$3

Location of Grid Point	See Note c.	See Note d.	Side 1	Side 1	Side 1
Special Restrictions on Displacement Coordinate System	See Notes a. and e.	See Note b.	None	See Note a.	See Note b.

Notes:

a. T3 axis must be parallel to axis of symmetry.

b. Displacement coordinate system at the referenced grid point must be cylindrical with z-axis along the axis of symmetry.

c. Any location except on side 2.

d. Any location except on the axis of symmetry or on side 2.

e. If the grid point is on the axis of symmetry, the displacement coordinate system must be rectangular.

3. If the number of segments, N, is 1 (in the case of DIH symmetry) or 2 (in the case of ROT or AXI symmetry), it is important that the rotational components referenced in field 3 be elastically connected to the structure. If $N \geq 2$ (in the case of DIH symmetry) or $N \geq 3$ (in the case of ROT or AXI symmetry), it is not important, because in this case the supports for rigid body rotation are actually applied to translational motions.
4. If $N \geq 3$, supports will be applied to both the 1 and 2 (inplane-translational) components, if either is referenced, and to both the 4 and 5 (out-of-plane rotational) components, if either is referenced. If component 6 is supported, component 2 should not appear on OMIT or OMIT1 entries.
5. The restrictions noted in Remarks 2. and 4. are related to symmetry requirements. For $N \geq 3$, symmetry requires that the supports be symmetrical (or antisymmetrical), with respect to any plane passing through the axis of symmetry. For the DIH options, N = 1 and N = 2, symmetry requires that the supports be symmetrical (or antisymmetrical) with respect to the plane(s) of symmetry. For the ROT option, N = 2, symmetry requires that a support be either parallel or perpendicular to the axis of symmetry.
6. GID must be a grid point and not a scalar point.

CYSYM Cyclic Symmetry Parameters

Defines parameters for cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CYSYM	NSEG	STYPE							

Example:

CYSYM	6	ROT							

Field	Contents
NSEG	Number of segments. (Integer > 0)
STYPE	Symmetry type. (Character: “ROT”, “DIH”, or “AXI”)

Remarks:

1. STYPE = “AXI” is a special case of STYPE = “ROT” used to model axisymmetric structures.
2. If STYPE = “AXI”, then all grid points must lie on side 1, side 2, or the axis. Also, plate elements with midside grid points may not be defined. See “**Additional Topics**” on page 555 of the *MSC.Nastran Reference Guide*.

D2R0000 (SOL 700) Deformable to Rigid

Defines materials to be switched to rigid at the start of the calculation.

Format:

1	2	3	4	5	6	7	8	9	10
D2R0000	PID	MRB							

Example:

D2R0000	12	0							

Field	Contents
PID	Property ID of the deformable part, which is switched to a rigid material. (I > 0, Default = Required)
MRB	Property ID of the master rigid body to which the deformable part is merged. If zero, the deformable part becomes either an independent or master rigid body. (I $\geq$ 0, Default = 0)

Remark:

Corresponds to LS-Dyna entry *DEFORMABLE_TO_RIGID

D2RAUTO (SOL 700)

Defines a set of parts to be switched to rigid or to deformable at some stage in the calculation.

Format:

1	2	3	4	5	6	7	8	9	10
D2RAUTO	SWSET	CODE	TIME 1	TIME 2	TIME 3	ENTNO	RELSW	PAIRED	

Continuation Line (enter one time):

	NRBF	NCSJ	RWF	DTMAX	D2R	R2D			

Continuation Line (repeated D2R times):

	PID	MRB							

Continuation Line (repeated R2D times):

	IPID								

Example:

1	2	3	4	5	6	7	8	9	10
D2RAUTO	100	1	.005	.008	.00002	2			
	0	2	2	.00002	2	3			
	400	1400							
	401	1401							
	1001								
1	1002								
	1003								

Field	Contents
SWSET	Set number for this automatic switch set. Must be unique. (Integer, no Default)
CODE	Activation switch code. Defines the test to activate the automatic material switch of the part (Integer, Default = 0, see Remark 1.)

Field	Contents
	EQ.0: switch takes place at time 1,
	EQ.1: switch takes place between time 1 and time 2 if rigid wall force (specified below) is zero,
	EQ.2: switch takes place between time 1 and time 2 if contact surface force (specified below) is zero,
	EQ.3: switch takes place between time 1 and time 2 if rigid wall force (specified below) is non-zero,
	EQ.4: switch takes place between time 1 and time 2 if contact surface force (specified below) is non-zero.
TIME 1	Switch will not take place before this time (Real, Default = 0.0)
TIME 2	Switch will not take place after this time (Real, Default = 1.E20)
	EQ.0 Time 2 set to 1.0e20.
TIME 3	Delay period. After this part switch has taken place, another automatic switch will not take place for the duration of the delay period. If set to zero a part switch may take place immediately after this switch (Real, Default = 0.0)
ENTNO	Rigid wall/contact surface number for switch codes 1, 2, 3, 4 (Integer, Default = 0, see Remarks **1.**, **2.**)
RELSW	Related switch set. The related switch set is another automatic switch set that must be activated before this part switch can take place (Integer, Default = 0)
	EQ.0: no related switch set.
PAIRED	Define a pair of related switches (Integer, Default = 0, see Remark **3.**)
	EQ. 0: not paired
	EQ. 1: paired with switch set RELSW and is the Master switch.
	EQ.-1: paired with switch set RELSW and is the Slave switch.
NRBF	Flag to delete or activate nodal rigid bodies (Integer, Default = 0, see Remark **4.**)
	'If nodal rigid bodies or generalized, weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities:
	EQ.0: no change,

Field	Contents
	EQ.1: delete,
	EQ.2: activate.
NCSF	Flag to delete or activate nodal constraint set (Integer, Default = 0, see Remark 4.)
	If nodal constraint / spotweld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities:
	EQ.0: no change,
	EQ.1: delete,
	EQ.2: activate.
RWF	Flag to delete or activate rigid walls (Integer, Default = 0, see Remark 4.)
	EQ.0: no change,
	EQ.1: delete,
	EQ.2: activate.
DTMAX	Maximum permitted time step size after switch (Integer, Default = 0)
D2R	Number of deformable parts to be switched to rigid plus number of rigid parts for which new master / slave rigid body combinations will be defined (Integer, Default = 0)
	EQ.0: no parts defined.
R2D	Number of rigid parts to be switched to deformable (Integer, Default = 0)
	EQ.0: no parts defined.
PID	Property ID of the part, which is switched to a rigid material (Integer, no Default)
MRB	Property ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body (Integer, Default = 0)
IPID	Property ID of the part, which is switched to a deformable Material (Integer, no Default)

Remarks:

1. Only surface to surface and node to surface contacts can be used to activate an automatic part switch.

2. Contact surface and rigid wall numbers are the order in which they are defined in the file. The first rigid wall and the first contact surface encountered in the input file will have an entity number of 1.
3. Switch sets may be paired together to allow a pair of switches to be activated more than once. Each pair of switches should use consistent values for CODE, i.e. 1&3 or 2&4. Within each pair of switches the related switch, RELSW, should be set to the ID of the other switch in the pair. The Master switch (PAIRED = 1) will be activated before the Slave switch (PAIRED = -1).
4. If the delete switch is activated, ALL corresponding constraints are deactivated regardless of their relationship to a switched part. By default, constraints which are directly associated with a switched part are deactivated / activated as necessary.

Note: This entry is mapped into LS-Dyna * DEFORMABLE_T

D2RINER (SOL 700)

Inertial properties can be defined for the new rigid bodies that are created when the deformable parts are switched. These can only be defined in the initial input if they are needed in a later restart. Unless these properties are defined, the new rigid body properties will be recomputed from the finite element mesh. The latter requires an accurate mesh description. **When rigid bodies are merged to a master rigid body, the inertial properties defined for the master rigid body apply to all members of the merged set.**

Format:

1	2	3	4	5	6	7	8	9	10
D2RINER	PID								
	XC	YC	ZC	TM					
	IXX	IXY	IXZ	IYY	IYZ	IZZ			

Example:

D2RINER	300								
	100.	200.	300.	5					
	2.0	.02	.04	5.0	.05	5.0			

Field	Contents
PID	Property ID (Integer, no Default)
XC	x-coordinate of center of mass (Real, no Default)
YC	y-coordinate of center of mass (Real, no Default)
ZC	z-coordinate of center of mass (Real, no Default)
TM	Translational mass (Real, no Default)
IXX	I_{xx}, xx component of inertia tensor (Real, no Default)
IXY	I_{xy} (Real, no Default)
IXZ	I_{xz} (Real, no Default)
IYY	I_{yy} (Real, no Default)
IYZ	I_{yz} (Real, no Default)
IZZ	I_{zz} (Real, no Default)

Note: This entry is mapped to LS-DYNA * DEFORMABLE_TO_RIGID_INERTIA

DAMPGBL (SOL 700)

Defines Values to Use for Dynamic Relaxation for SOL 700 Only

Defines parameters to be used for static analysis simulation using Dynamic Relaxation for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
DAMPGBL	LCID	VLDMP	STX	STY	STZ	SRX	SRY	SRZ	

Example:

DAMPGBL	2		1.01	1.02	1.03	0.0	0.0	0.0	

Field	Contents
LCID	Integer $\geq$ 0 ID of TABLED1 defining system damping constant for dynamic relaxiation. The damping force applied to each grid is f=d(t)mv where d(t) is described in TABLED1. (Default = 0)
VALDMP	Real $\geq$ 0.0, System damping constant if LCID is not used. Will apply for all grids all the time. (Default = 0.0)
STX	Real $\geq$ 0.0, Scale factor for basic coordinate system x direction damping forces. (Default = 0.0)
STY	Real $\geq$ 0.0, Scale factor for basic coordinate system y direction damping forces. (Default = 0.0)
STZ	Real $\geq$ 0.0, Scale factor for basic coordinate system z direction damping forces. (Default = 0.0)
SRX	Real $\geq$ 0.0, Scale factor for basic coordinate system rotation about x direction damping forces. (Default = 0.0)
SRY	Real $\geq$ 0.0, Scale factor for basic coordinate system rotation about y direction damping forces. (Default = 0.0)
SRZ	Real $\geq$ 0.0, Scale factor for basic coordinate system rotation about z direction damping forces. (Default = 0.0)

Remark:

1. This entry corresponds to LS-DYNA's *DAMPING GLOBAL entry.

DAREA Load Scale Factor

Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with ACSRCE, RLOADi and TLOADi entries.

Format:

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

Example:

DAREA	3	6	2	8.2	15	1	10.1		

Field	Contents
SID	Identification number. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer 1 through 6 for grid point; blank or 0 for extra or scalar point.)
Ai	Scale (area) factor. (Real)

Remarks:

1. One or two scale factors may be defined on a single entry.
2. Refer to RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entries for the formulas that define the scale factor Ai in dynamic analysis.
3. Component numbers refer to the displacement coordinate system.
4. In dynamic analysis, DAREA entries may be used with LSEQ Bulk Data entries if LOADSET is specified in Case Control. The LSEQ and static load entries will be used to internally generate DAREA entries.
5. If DAREA is referenced by a GUST entry, Pi must be defined. However, it is only used if selected through a DLOAD Case Control command. WG from the GUST entry is used instead of Ai when requested via a GUST entry.
6. All DAREA entries corresponding to all grid and scalar points are automatically converted internally by the program to equivalent FORCE / MOMENT / SLOAD entries (as appropriate) *if there are no LSEQ Bulk Data entries*.

7. In superelement analysis, DAREA may be used to specify loads not only on the interior points of the residual, but also on the interior points of upstream superelements *if there are no LSEQ Bulk Data entries.*
8. In static analysis, DAREA entries may be used *only if there are no LSEQ Bulk Data entries.* They are ignored if there are any LSEQ Bulk Data entries.

DCONADD Design Constraint Set Combination

Defines the design constraints for a subcase as a union of DCONSTR entries.

Format:

1	2	3	4	5	6	7	8	9	10
DCONADD	DCID	DC1	DC2	DC3	DC4	DC5	DC6	DC7	
	DC8	-etc.-							

Example:

DCONADD	10	4	12						

Field	Contents
DCID	Design constraint set identification number. (Integer > 0)
DCi	DCONSTR entry identification number. (Integer > 0)

Remarks:

1. The DCONADD entry is selected by a DESSUB or DESGLB Case Control command.
2. All DCi must be unique from other DCi.

DCONSTR Design Constraints

Defines design constraints.

Format:

1	2	3	4	5	6	7	8	9	10
DCONSTR	DCID	RID	LALLOW /LID	UALLOW /UID	LOWFQ	HIGHFQ			

Example:

DCONSTR	10	4	1.25						

Field	Contents
DCID	Design constraint set identification number. (Integer > 0)
RID	DRESPi entry identification number. (Integer > 0)
LALLOW / LID	Lower bound on the response quantity or the set identification ID of a number of a TABLEDi entry that supplies the lower bound as a function of frequency. (Real, Default = -1.0E20)
UALLOW / UID	Upper bound on the response quantity or the set identification ID of a number of a TABLEDi entry that supplies the upper bound as a function of frequency. (Real, Default = 1.0E20)
LOWFQ	Low end of frequency range in Hertz (Real $\geq$ 0.0, Default = 0.0). See Remark **8.**
HIGHFQ	High end of frequency range in Hertz (Real $\geq$ LOWFQ, Default = 1.0E+20). See Remark **8.**

Remarks:

1. The DCONSTR entry may be selected in the Case Control Section by the DESSUB or DESGLB command.
2. DCID may be referenced by the DCONADD Bulk Data entry.
3. For a given DCID, the associated RID can be referenced only once.
4. The units of LALLOW and UALLOW must be consistent with the referenced response defined on the DRESPi entry. If RID refers to an “EIGN” response, then the imposed bounds must be expressed in units of eigenvalue, $(\text{radian}/\text{time})^2$. If RID refers to a “FREQ” response, then the imposed bounds must be expressed in cycles / time.

5. LALLOW and UALLOW are unrelated to the stress limits specified on the MATi entry.
6. Constraints are computed as follows:

$$g = \frac{\text{LALLOW} - r}{\text{GNORM}} \text{ for lower bound constraints}$$

$$g = \frac{r - \text{UALLOW}}{\text{GNORM}} \text{ for upper bound constraints}$$

$$\text{GNORM} = \begin{cases} |\text{LALLOW}| \text{ for lower bounds if } |\text{LALLOW}| > \text{GSCAL} \\ |\text{UALLOW}| \text{ for upper bounds if } |\text{UALLOW}| > \text{GSCAL} \\ \text{GSCAL otherwise} \end{cases}$$

   GSCAL is specified on the DOPTPRM entry (Default = 0.001)
7. As Remark **6.** indicates, small values of UALLOW and LALLOW require special processing and should be avoided. Bounds of exactly zero are particularly troublesome. This can be avoided by using a DRESP2 entry that offsets the constrained response from zero.
8. LOWFQ and HIGHFQ fields are functional only for RTYPE with a ‘FR’ or a ‘PSD’ prefix, e.g., FRDISP or on DRESP2 or DRESP3 entries that inherit the frequency value from these RTYPES. The bounds provided in LALLOW and UALLOW are applicable to a response only when the value of forcing frequency of the response falls between the LOWFQ and HIGHFQ. If the ATTB field of the DRESP1 entry is not blank, LOWFQ and HIGHFQ are ignored.
9. LID and UID are optional inputs that identify tabular input to specify the lower and upper bounds as a function of frequency. They are applicable to the response identified in Remark **8.**

DDVAL Discrete Design Variable Values

Define real, discrete design variable values for discrete variable optimization.

Format:

1	2	3	4	5	6	7	8	9	10
DDVAL	ID	DVAL1	DVAL2	DVAL3	DVAL4	DVAL5	DVAL6	DVAL7	

Alternate Format:

DDVAL	ID	DVAL1	“THRU”	DVAL	“BY”	INC			

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	DVAL8	DVAL9	DVAL10	DVAL11	-etc.-				

Continuation Entry Format 2:

	DVAL8	“THRU”	DVAL9	“BY”	INC				

Example:

1	2	3	4	5	6	7	8	9	10
DDVAL	110	0.1	0.2	0.3	0.5	0.6	0.4		
	.7	thru	1.0	by	0.05				
	1.5	2.0							

Field	Contents
ID	Unique discrete value set identification number (Integer > 0)
DVALi	Discrete values (Real, or “THRU” or “BY”)
INC	Discrete value increment (Real)

Remarks:

1. DDVAL entries must be referenced by a DESVAR entry in the DDVAL field (field 8).
2. Trailing fields on a DDVAL record can be left blank if the next record is of type DVALi "THRU" DVALj "BY" INC. Also fields 7 - 9 must be blank when the type DVALi "THRU" DVALj "BY" INC is used in fields 2 - 6 and fields 8 - 9 must be blank when the type DVALi "THRU" DVALj "BY" INC is used in fields 3 - 7 for the first record. Embedded blanks are not permitted in other cases.
3. The DVALi sequence can be random.
4. The format DVALi “THRU” DVALj “BY” INC defines a list of discrete values, e.g., DVALi, DVALi+INC, DVALi+2.0*INC, ..., DVALj. The last discrete DVALj is always included, even if the range is not evenly divisible by INC.

DEFORM Static Element Deformation

Defines enforced axial deformation for one-dimensional elements for use in statics problems.

Format:

1	2	3	4	5	6	7	8	9	10
DEFORM	SID	EID1	D1	EID2	D2	EID3	D3		

Example:

DEFORM	1	535	.05	536	-.10				

Field	Contents
SID	Deformation set identification number. (Integer > 0)
EIDi	Element number. (Integer > 0)
Di	Deformation. (Real; positive value represents elongation.)

Remarks:

1. The referenced element must be one-dimensional (CROD, CONROD, CTUBE, CBAR, CBEAM).
2. Deformation sets must be selected in the Case Control Section with DEFORM = SID.
3. One to three enforced element deformations may be defined on a single entry.
4. The DEFORM entry, when called by the DEFORM Case Control command, is applicable to linear static, inertia relief, differential stiffness, and buckling (Solutions 101, 105, 114, and 200) and will produce fatal messages in other solution sequences. Use SPCD to apply enforced displacements in solution sequences for which DEFORM does not apply.

DEFUSET Degree-of-Freedom Set Name Definition

Defines new names for degree-of-freedom sets.

Format:

1	2	3	4	5	6	7	8	9	10
DEFUSET	OLD1	NEW1	OLD2	NEW2	OLD3	NEW3	OLD4	NEW4	

Example:

DEFUSET	U2	X	U4	Y	U3	Z			

Field	Contents
OLDi	Default set name. (One to four characters)
NEWi	New set name. (One to four characters)

Remarks:

1. From one to four set names may be specified on a single entry.
2. OLDi must refer to any of the set names given in "**Degree-of-Freedom Sets**" on page 939. It is recommended that OLDi refer only to the set names U1 through U6. If sets PA or PS are referenced, a user fatal message is issued.
3. All NEWi names must be unique with respect to all other set names.
4. The DEFUSET entry is optional since default set names exist for all displacement sets.
5. The DEFUSET entry must be present in the Bulk Data Section in all restarts.

DELAY Dynamic Load Time Delay

Defines the time delay term τ in the equations of the dynamic loading function.

Format:

1	2	3	4	5	6	7	8	9	10
DELAY	SID	P1	C1	T1	P2	C2	T2		

Example:

DELAY	5	21	6	4.25	7	6	8.1		

Field	Contents
SID	Identification number of the DELAY entry. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer 1 through 6 for grid point, blank or 0 for extra point or scalar point.)
Ti	Time delay τ for designated point Pi and component Ci. (Real)

Remarks:

1. One or two dynamic load time delays may be defined on a single entry.
2. SID must also be specified on a RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entry. See those entry descriptions for the formulas that define the manner in which the time delay τ is used.
3. A DAREA, LSEQ or static load entry should be used to define a load at Pi and Ci.
4. In superelement analysis, DELAY entries may only be applied to loads on points in the residual structure.

DEQATN Design Equation Definition

Defines one or more equations for use in design sensitivity or p-element analysis.

Format:

1	2	3	4	5	6	7	8	9	10
DEQATN	EQID	EQUATION							
	EQUATION (Cont.)								

Example:

DEQATN	14	F1(A, B, C, D, R) = A + B · *C* – (D ** 3 + 10.0) + sin(PI(1) · R)	
	+ A**2/(B – C) ; F = A + B – F1 · D		

Field	Contents
EQID	Unique equation identification number. (Integer > 0)
EQUATION	Equation(s). See Remarks. (Character)

Remarks:

1. EQUATION is a single equation or a set of nested equations and is specified in fields 3 through 9 on the first entry and may be continued on fields 2 through 9 on the continuation entries. On the continuation entries, no commas can appear in columns 1 through 8. All data in fields 2 through 9 must be specified in columns 9 through 72. The large-field format is not allowed.

   A single equation has the following format:

   variable-1 (x1, x2, ..., xn)=expression-1

   A set of nested equations is separated by semicolons and has the format:

   variable-1 (x1, x2, ..., xn)=expression-1;
   variable-2=expression-2;
   variable-3=expression-3;

   etc.

   variable-m=expression-m

Expression-i is a collection of constants, real variables, and real functions, separated by operators, and must produce a single real value. (x1, x2, ..., xn) is the list of all the variable names (except variable-i) that appear in all expressions. Variable-i may be used in subsequent expressions. The last equation, variable-m=expression-m, provides the value that is returned to the Bulk Data entry that references EQID; e.g., DRESP2. The example above represents the following mathematical equations:

$$F1 = A + B \cdot C - (D^3 + 10) + \sin(\pi \cdot R) + \frac{A^2}{B - C}$$

$$F = A + B + F1 \cdot D$$

where SIN and PI are intrinsic functions. See Remark **4.**

2. EQUATION may contain embedded blanks. EQUATION must contain less than 32,000 nonblank characters. If more characters are required for use with a DRESP2 entry, the DRESP2 can be divided into two or more DRESP2 entries with a master DRESP2 referencing subsequent DRESP2s.
3. The syntax of the expressions follows FORTRAN language standards. The allowable arithmetic operations are shown in **Table 8-4** in the order of execution precedence. Parenthesis are used to change the order of precedence. Operations within parentheses are performed first with the usual order of precedence being maintained within the parentheses.

Table 8-4 DEQATN Entry Operators

Operator	Operation	Sample Expressions	Interpreted As
-, +	Negative or Positive immediately preceded by exponentiation	X * * -Y	X * * (-Y)
* *	Exponentiation	-X * * Y	(-X * * Y)
-, +	Negative or Positive	-X-Y	(-X)-Y
*, /	Multiplication or Division	X * Y-Z	(X * Y)-Z
+, -	Addition or Subtraction	X+Y	X+Y

4. The expressions may contain intrinsic functions. **Table 8-5** contains the format and descriptions of functions that may appear in the expressions. The use of functions that may be discontinuous must be used with caution because they can cause discontinuous derivatives. These are ABS, DIM, MAX, MIN, and MOD. For examples and further details see the *MD Nastran 2006 DMAP Programmer's Guide*.

Table 8-5 DEQATN Entry Functions

Format	Description	Mathematical Expressions
ABS(x)	absolute value	$\|x\|$
ACOS(x)	arccosine	$\cos^{-1} x$
ACOSH(x)	hyperbolic arccosine	$\cosh^{-1} x$
ASIN(x)	arcsine	$\sin^{-1} x$
ASINH(x)	hyperbolic arcsine	$\sinh^{-1} x$
ATAN(x)	arctangent	$\tan^{-1} x$
ATAN2(x,y)	arctangent of quotient	$\tan^{-1} (x / y)$
ATANH(x)	hyperbolic arctangent	$\tanh^{-1} x$
ATANH2(x,y)	hyperbolic arctangent of quotient	$\tanh^{-1} (x / y)$
AVG(X_1, X_2, .., Xn)	average	$\frac{1}{n}\sum_{i=1}^{n} X_i$
COS(x)	cosine	$\cos x$
COSH(x)	hyperbolic cosine	$\cosh x$

Table 8-5 DEQATN Entry Functions (continued)

Format	Description	Mathematical Expressions
DB(P, PREF)	sound pressure in decibel	$20.0 \cdot \log\left(\frac{P}{PREF}\right)$
DBA(P, PREF, F)	sound pressure in decibel (perceived)	$20.0 \cdot \log\left(\frac{P}{PREF}\right) + 10.0 \cdot \log(Ta1) + 10.0 \cdot \log(Ta2)$
DIM(x,y)	positive difference	x-MIN(x,y)
EXP(x)	exponential	e^x
INVDB(DB, PREF)	inverse Db	$10^{\left(\frac{DB}{20.0} + \log PREF\right)}$
INVDBA(DBA, PREF, F)	inverse Dba	$10^{\left(\frac{DBA - 10.0 \cdot \log(Ta1) - 10.0 \cdot \log(Ta2)}{20.0}\right)}$
LOG(x)	natural logarithm	$\log_e x$
LOG10(x)	common logarithm	$\log_{10} x$
LOGX(x,y)	base x logarithm	$\log_x y$
MAX(x_1, x_2, ...)	maximum	maximum of x_1, etc.
MIN(x_1, x_2, ...)	minimum	minimum of x_1, etc.
MOD(x,y)	remainder (modulo)	$x - y \cdot (\mathrm{INT}(x / y))$
PI(x)	multiples of pi (π)	$x \cdot \pi$
RSS(X_1, X_2, ..., Xn)	square root of sum of squares	$\sqrt{\sum_{i=1}^{n} X_i^2}$

Table 8-5 DEQATN Entry Functions (continued)

Format	Description	Mathematical Expressions
SIN(x)	sine	sin x
SINH(x)	hyperbolic sine	sinh x
SQRT(x)	square root	$\sqrt{x}$
SSQ(X_1, X_2, ..., Xn)	sum of squares	$\sum_{i=1}^{n} X_i^2$
SUM(X_1, X_2, ..., Xn)	summation	$\sum_{i=1}^{n} X_i$
TAN(x)	tangent	tan x
TANH(x)	hyperbolic tangent	tanh x

where:

$X_1, X_2, .., Xn, P$ = structure responses or acoustic pressure

PREF = reference pressure

F = forcing frequency

DB = acoustic pressure in Decibel

DBA = perceived acoustic pressure in Decibel

$$Ta1 = \frac{K3 \cdot F^4}{(F^2 + P2^2)(F^2 + P3^2)}$$

$$Ta2 = \frac{K1 \cdot F^4}{(F^2 + P1^2)^2 (F^2 + P4^2)^2}$$

$K1$ = 2.242882e+16

$K3$ = 1.562339

$P1$ = 20.598997

$P2 = 107.65265$

$P3 = 737.86223$

$P4 = 12194.22$

5. If the DEQATN entry is referenced by the:
   - DVCREL2, DVMREL2, or DVPREL2 entry, then X_i represents the DVIDj and LABLk fields.
   - DRESP2 entry, then X_i represents the DVIDj, LABLk, NRm, Gp, DPIPq, DCICr, DMIMs, DPI2Pt, DCI2Cu, DMI2Mv, and NRRw fields in that order.
   - GMLOAD, GMBC, or TEMPF entries, then
     X_1 represents x in the basic coordinate system,
     X_2 represents y in the basic coordinate system, and
     X_3 represents z in the basic coordinate system.
   - GMCURV entry, then
     X_1 represents line parameter u.
   - GMSURF entry, then
     X_1 represents surface parameter u and
     X_2 represents surface parameter v.
6. If the DEQATN entry is referenced by the GMLOAD, GMBC, TEMPF, GMCURV, or GMSURF entries and your computer has a short word length (e.g., 32 bits / word), then EQUATION is processed with double precision and constants may be specified in double precision; e.g., 1.2D0. If your machine has a long word length (e.g., 64 bits / word) then EQUATION is processed in single precision and constants must be specified in single precision; e.g., 1.2. If the DEQATN entry is referenced by DRESP2, DVCREL2, DVMREL2 or DVPREL2 entries, constants must be specified in single precision regardless of your machine's word length.
7. The DMAP logical operators NOT, AND, OR, XOR, and XQV cannot be used as X_i names.
8. Input errors on the DEQATN entry often result in poor messages. Substituting a "[" for a parenthesis or violating the restriction against large field format are examples. Known messages are UFM 215, SFM 233 and UFM 5199. If any of these messages are encountered then review the DEQATN entry input.

9. Intrinsic functions MAX and MIN are limited to <100 arguments. If more arguments are desired, the functions may be concatenated.

10. Arithmetic is carried out using the type of the input data. For example, in the expression:

$$X = A**(1/2)$$

both values in the exponent are integers so that the value returned for the exponent is calculated using integer arithmetic or $1/2 = 0$. In this case 1/2 should be replaced by (.5).

DESVAR Design Variable

Defines a design variable for design optimization.

Format:

1	2	3	4	5	6	7	8	9	10
DESVAR	ID	LABEL	XINIT	XLB	XUB	DELXV	DDVAL		

Example:

DESVAR	2	BARA1	35.0	10.	100.	0.2			

Field	Contents
ID	Unique design variable identification number. (Integer > 0)
LABEL	User-supplied name for printing purposes. (Character)
XINIT	Initial value. (Real, XLB $\leq$ XINIT $\leq$ XUB)
XLB	Lower bound. (Real, Default = -1.0E+20)
XUB	Upper bound. (Real, Default = +1.0E+20)
DELXV	Fractional change allowed for the design variable during approximate optimization. (Real > 0.0, for Default see Remark 2.)
DDVAL	ID of a DDVAL entry that provides a set of allowable discrete values. (Blank or Integer > 0; Default=blank for continuous design variables. See Remark 3.)

Remarks:

1. DELXV can be used to control the change in the design variable during one optimization cycle.
2. If DELXV is blank, the default is taken from the specification of the DELX parameter on the DOPTPRM entry. If DELX is not specified, then the default is 0.5.
3. If the design variable is to be discrete (Integer>0 in DDVAL field), and if either of the XLB and / or XUB bounds are wider than those given by the discrete list of values on the corresponding DDVAL entry, XLB and / or XUB will be replaced by the minimum and maximum discrete values.

DIVERG Divergence Analysis Data

Defines Mach numbers (m) for a divergence analysis in SOLs 144 and 200.

Format:

1	2	3	4	5	6	7	8	9	10
DIVERG	SID	NROOT	M1	M2	M3	M4	M5	M6	
	M7	M8	-etc.-						

Example:

DIVERG	70	2	.5	.8	.9				

Field	Contents
SID	Unique set identifier. (Integer > 0)
NROOT	Number of divergence roots that are to be output and their eigenvectors printed. (Integer; Default = 1)
Mi	Mach number. (Real $\geq$ 0.0)

Remarks:

1. The DIVERG entry is referenced in Case Control by “DIVERG = SID”.
2. The NROOT lowest divergence dynamic pressures are printed. If there are fewer than NROOT pressures, all available dynamic pressures are printed.
3. Mi values must be distinct.
4. A blank Mach number field terminates the input.

DLINK Multiple Design Variable Linking

Relates one design variable to one or more other design variables.

Format:

1	2	3	4	5	6	7	8	9	10
DLINK	ID	DDVID	C0	CMULT	IDV1	C1	IDV2	C2	
	IDV3	C3	-etc.-						

Example:

DLINK	10	2	0.1	0.33	2	2.0	6	-1.0	
	8	7.0							

Field	Contents
ID	Unique identification number. (Integer > 0)
DDVID	Dependent design variable identification number. (Integer > 0)
C0	Constant term. (Real; Default = 0.0)
CMULT	Constant multiplier. (Real; Default = 1.0)
IDVi	Independent design variable identification number. (Integer > 0)
Ci	Coefficient i corresponding to IDVi. (Real)

Remarks:

1. DLINK defines the relationship

$$\text{DDVID} = C0 + \text{CMULT} \sum_{i} \text{Ci} \cdot \text{IDVi}$$

2. This capability provides a means of linking physical design variables such as element thicknesses to nonphysical design variables such as the coefficients of interpolating functions.
3. CMULT provides a simple means of scaling the Ci. For example if Ci = 1/7, 2/7, 4/7, etc. is desired, then CMULT = 1/7 and Ci = 1, 2, 4, etc., may be input.

4. An independent IDVi must not occur on the same DLINK entry more than once.
5. ID is for user reference only.
6. If a design variable is specified as dependent on a DLINK entry, then it cannot be specified as independent on another DLINK entry.

DLOAD Dynamic Load Combination or Superposition

Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.

Format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-	*					

Examples:

DLOAD	17	1.0	2.0	6	-2.0	7	2.0	8	
	-2.0	9							

Field	Contents
SID	Load set identification number. (Integer > 0)
S	Scale factor. (Real)
Si	Scale factors. (Real)
Li	Load set identification numbers of RLOAD1, RLOAD2, TLOAD1, TLOAD2, and ACSRC entries. (Integer > 0)

Remarks:

1. Dynamic load sets must be selected in the Case Control Section with DLOAD = SID.
2. The load vector being defined by this entry is given by

$$\{P\} = S\sum_{i} Si\{P_i\}$$

3. Each Li must be unique from any other Li on the same entry.
4. SID must be unique from all TLOADi and RLOADi entries.
5. Nonlinear transient load sets (NOLINi entries) may not be specified on DLOAD entries. NOLINi entries are selected separately in the Case Control Section by the NONLINEAR command.

6. A DLOAD entry may not reference a set identification number defined by another DLOAD entry.
7. TLOAD1 and TLOAD2 loads may be combined only through the use of the DLOAD entry.
8. RLOAD1 and RLOAD2 loads may be combined only through the use of the DLOAD entry.

DMI Direct Matrix Input

Defines matrix data blocks. Generates a matrix of the following form:

$$[\text{NAME}] = \begin{bmatrix} X_{11} & X_{12} & \cdots & X_{1n} \\ X_{21} & X_{22} & \cdots & X_{2n} \\ \cdot & \cdot & \cdot & \cdot \\ X_{m1} & \cdots & \cdots & X_{mn} \end{bmatrix}$$

where the elements X_{ij} may be real $(X_{ij} = A_{ij})$ or complex $(X_{ij} = A_{ij} + iB_{ij})$. The matrix is defined by a single header entry and one or more column entries. Only one header entry is required. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	"0"	FORM	TIN	TOUT		M	N	

Column Entry Format for Real Matrices:

DMI	NAME	J	I1	A(I1,J)	A(I1+1,J)		-etc.-	I2	
	A(I2,J)		-etc.-						

Column Entry Format for Complex Matrices:

DMI	NAME	J	I1	A(I1,J)	B(I1,J)	A(I1+1,J)	B(I1+1,J)	-etc.-	
	I2	A(I2,J)	B(I2,J)	-etc.-					

Example of a Real Matrix:

DMI	BBB	0	2	1	1		4	2	
DMI	BBB	1	1	1.	3.	5.			
DMI	BBB	2	2	6.	4	8.			

$$\text{BBB} = \begin{bmatrix} 1.0 & 0.0 \\ 3.0 & 6.0 \\ 5.0 & 0.0 \\ 0.0 & 8.0 \end{bmatrix}$$

Example of a Complex Matrix:

DMI	QQQ	0	2	3	3		4	2	
DMI	QQQ	1	1	1.0	2.0	3.0	0.0	3	
	5.0	6.0							
DMI	QQQ	2	2	6.0	7.0	4	8.0	9.0	

$$[QQQ] = \begin{bmatrix} 1.0 + 2.0i & , & 0.0 + 0.0i \\ 3.0 + 0.0i & , & 6.0 + 7.0i \\ 5.0 + 6.0i & , & 0.0 + 0.0i \\ 0.0 + 0.0i & , & 8.0 + 9.0i \end{bmatrix}$$

Field	Contents
NAME	Name of the matrix. See Remark **1.** Name is used to reference the data block in the DMAP sequence. (One to eight alphanumeric characters, the first of which must be alphabetic.)
FORM	Form of matrix, as follows: (Integer) 1 = Square matrix (not symmetric) 2 = General rectangular matrix 3 = Diagonal matrix (M=number of rows, N = 1) 4 = Lower triangular factor 5 = Upper triangular factor 6 = Symmetric matrix 8 = Identity matrix (M=number of rows, N = M)
TIN	Type of matrix being input, as follows: (Integer) 1 = Real, single precision (one field used / element) 2 = Real, double precision (one field used / element) 3 = Complex, single precision (two fields used / element) 4 = Complex, double precision (two fields used / element)

Field	Contents
TOUT	Type of matrix being output, as follows: (Integer) 0 = Set by precision cell 1 = Real, single precision 2 = Real, double precision 3 = Complex, single precision 4 = Complex, double precision
M	Number of rows in NAME. (Integer > 0)
N	Number of columns in NAME. Except for FORM 3 and 8. (Integer > 0)
"0"	Indicates the header entry.
J	Column number of NAME. (Integer > 0)
I1, I2, etc.	Row number of NAME, which indicates the beginning of a group of nonzero elements in the column. See Remark **13.** (Integer > 0)
A(Ix,J)	Real part of element (see TIN). (Real)
B(Ix,J)	Imaginary part of element (see TIN). (Real)

Remarks:

1. In order to use the DMI feature, the user must write a DMAP, or make alterations to a solution sequence that includes the DMIIN module. See the *MD Nastran 2006 DMAP Programmer's Guide*. All of the rules governing the use of data blocks in DMAP sequences apply.
2. The total number of DMIs and DTIs may not exceed 1000.
3. Field 3 of the header entry must contain an integer of zero (0).
4. For symmetric matrices, the entire matrix must be input.
5. Only nonzero terms need be entered.
6. Leading and trailing zeros in a column do not have to be entered. However, a blank field between nonzero fields on this entry is not equivalent to a zero. If a zero input is required, the appropriate type zero must be entered (i.e., 0.0 or 0.0D0).
7. Complex input must have both the real and imaginary parts entered if either part is nonzero; i.e., the zero component must be input explicitly.

8. If A(Ix,J) is followed by "THRU" in the next field and an integer row number "IX" after the THRU, then A(lx,J) will be repeated in each row through IX. The "THRU" must follow an element value. For example, the entries for a real matrix RRR would appear as follows:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	J	I1	A(I1,J)			I1	A(I2,J)	
DMI	RRR	1	2	1.0	THRU	10	12	2.0	

These entries will cause the first column of the matrix RRR to have a zero in row 1, the values 1.0 in rows 2 through 10, a zero in row 11, and 2.0 in row 12.

9. Each column must be a single logical entry. The terms in each column must be specified in increasing row number order.
10. The "FORM" options 4, 5, and 8 are nonstandard forms and may be used only in conjunction with the modules indicated in **Table 8-6**.

Table 8-6 DMI FORM Options

FORM	Matrix Description	Modules			
		ADD	FBS	MATPRN	MPYAD
4	Lower Triangular Factor		X	X	
5	Upper Triangular Factor		X	X	
8	Identity	X	X	X	X

11. Form 3 matrices are converted to Form 6 matrices, which may be used by any module.
12. Form 7 matrices may not be defined on this entry.
13. I1 must be specified. I2, etc. are not required if their matrix elements follow the preceding element in the next row of the matrix. For example, in the column entry for column 1 of QQQ, neither I2 nor I3 is specified.
14. The DMIG entry is more convenient for matrices with rows and columns that are referenced by grid or scalar point degrees-of-freedom.

DMIAX Direct Matrix Input for Axisymmetric Analysis

Defines axisymmetric (fluid or structure) related direct input matrix terms.

The matrix is defined by a single header entry and one or more column entries. Only one header entry is required. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIAX	NAME	“0"	IFO	TIN	TOUT				

Column Entry Format:

DMIAX	NAME	GJ	CJ	NJ					
	G1	C1	N1	A1	B1				
	G2	C2		-etc.-					

Example:

DMIAX	B2PP	0	1	34					
DMIAX	B2PP	32							
	1027	3		4.25+6	2.27+3				

Field	Contents
NAME	Name of the matrix. See Remark **2.** (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix: (Integer) 1 = Square matrix 2 = General rectangular matrix 6 = Symmetric matrix
TIN	Type of matrix being input: (Integer) 1 = Real, single precision (One field is used per element.) 3 = Complex, single precision (Two fields are used per element.)

Field	Contents
TOUT	Type of matrix that will be created: (Integer) 1 = Real, single precision 2 = Real, double precision 3 = Complex, single precision 4 = Complex, double precision
GJ, Gi	Grid, scalar, RINGFL fluid point, PRESPT pressure point, FREEPT free surface displacement, or extra point identification number. (Integer > 0)
CJ, Ci	Component number for GJ or Gi grid point (0 ≤ Integer ≤ 6; Blank or zero if GJ or Gi is a scalar, fluid, or extra point.)
NJ, Ni	Harmonic number of RINGFL point. Must be blank if a point type other than RINGFL is used. A negative number implies the "sine" series; a positive number implies the "cosine" series. (Integer)
Ai, Bi	Real and imaginary parts of matrix element; row (Gi, Ci, Ni) column (GJ, CJ, NJ). If the matrix is real (TIN = 1), then Bi must be blank.

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. Matrices defined on this entry may be used in dynamics by selection with the Case Control commands K2PP = NAME, B2PP = NAME, or M2PP = NAME for $[K^2_{pp}]$, $[B^2_{pp}]$, or $[M^2_{pp}]$, respectively. See "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide*.
3. Field 3 or the header entry must contain an integer 0.
4. For symmetric matrices, either the upper or the lower triangle terms may be specified, but not both.
5. Only nonzero terms need be entered.
6. If any DMIAX entry is changed or added on restart then a complete re-analysis may be performed. Therefore, DMIAX entry changes or additions are not recommended on restart.

DMIG Direct Matrix Input at Points

Defines direct input matrices related to grid, extra, and/or scalar points. The matrix is defined by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIG	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

Example:

DMIG	STIF	0	1	3	4				
DMIG	STIF	27	1		2	3	3.+5	3.+3	
	2	4	2.5+10	0.	50		1.0	0.	

Field	Contents
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix input. IFO = 6 must be specified for matrices selected by the K2GG, M2GG, and B2GG Case Control commands. (Integer) 1 = Square 9 or 2 = Rectangular 6 = Symmetric
TIN	Type of matrix being input: (Integer) 1 = Real, single precision (One field is used per element.) 2 = Real, double precision (One field is used per element.) 3 = Complex, single precision (Two fields are used per element.) 4 = Complex, double precision (Two fields are used per element.)

Field	Contents
TOUT	Type of matrix that will be created: (Integer) 0 = Set by precision system cell (Default) 1 = Real, single precision 2 = Real, double precision 3 = Complex, single precision 4 = Complex, double precision
POLAR	Input format of Ai, Bi. (Integer=blank or 0 indicates real, imaginary format; Integer > 0 indicates amplitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. See Remarks **5.** and **6.** (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer $\leq$ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ $\leq$ 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

Remarks:

1. Matrices defined on this entry may be used in dynamics by selection in the Case Control with K2PP = NAME, B2PP = NAME, M2PP = NAME for $[K_{pp}]$, $[B_{pp}]$, or $[M_{pp}]$, respectively. Matrices may also be selected for all solution sequences by K2GG = NAME, B2GG = NAME, and M2GG = NAME. The g-set matrices are added to the structural matrices before constraints are applied, while p-set matrices are added in dynamics after constraints are applied. Load matrices may be selected by P2G = NAME for dynamic and superelement analyses.
2. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column follows. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but input of an element of the matrix more than once will produce a fatal message.

3. Field 3 of the header entry must contain an integer 0.
4. For symmetric matrices (IFO = 6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both below and above the diagonal.
5. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIG matrices is always either p-set or g-set size, depending on the context.) The GJ term is used for the column index. The CJ term is ignored.
6. If NCOL is not used for rectangular matrices, two different conventions are available:
    - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
    - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered non-null column (in internal sort). Trailing null columns of the g- or p-size matrix will be truncated.
7. The matrix names must be unique among all DMIGs.
8. TIN should be set consistent with the number of decimal digits required to read the input data adequately. For a single-precision specification on a short-word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double-field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a “D” type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
9. On long-word machines, almost all matrix calculations are performed in single precision and on short-word machines, in double precision. It is recommended that DMIG matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the

machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIG input to be used on any machine. If TOUT is contrary to the machine type specified (for example, a TOUT of 1 on a short-word machine), unreliable results may occur.

10. If any DMIG entry is changed or added on restart then a complete re-analysis is performed. Therefore, DMIG entry changes or additions are not recommended on restart.

DMIG,UACCEL Direct Matrix Input of Enforced Static Acceleration

Defines rigid body accelerations in the basic coordinate system.

Format:

1	2	3	4	5	6	7	8	9	10
DMIG	UACCEL	“0"	“9"	TIN				NCOL	
DMIG	UACCEL	L			G1	C1	X1		
	G2	C2	X2		G3	C3	X3		

Example:

DMIG	UACCEL	0	9	1				4	
DMIG	UACCEL	2			2	3	386.4		
DMIG	UACCEL	3			2	4	3.0		
DMIG	UACCEL	4			2	6	1.0		

Field	Contents
TIN	Type of matrix being input. (Integer 1 or 2) 1 = Real, single precision (One field is used per element.) 2 = Real, double precision (One field is used per element.)
NCOL	Number of columns, see Remark 2. Default is the number of columns specified. (Integer > 0)
L	Load sequence number. (Integer > 0)
Gi	Grid point identification number of a single reference point. (Integer > 0)
Ci	Component number for Gi in the basic coordinate system. See Remark 4. (0 < Integer $\leq$ 6)
Xi	Value of enforced acceleration term in the basic coordinate system. (Real)

Remarks:

1. DMIG,UACCEL is an optional entry when PARAM,INREL,-1 is specified in SOLs 101 or 200. If DMIG,UACCEL is present, the loads applied to the structure are the sum of the conventional applied loads plus the inertia loads resulting from the rigid body accelerations defined on this entry. If it is not present, conventional inertia relief calculations are performed.
2. The load sequence number interpretation depends on the value of the NCOL field. The recommended method is to set it equal to the number of loading conditions. The load sequence number L is then the sequence number of the subcase to which the applied acceleration will be applied.
3. The grid point identification number listed on Gi defines a single grid point on the model where loads will be applied to cause the enforced acceleration state. Gi must also appear on a SUPORT Bulk Data entry. It must also appear on a PARAM,GRDPNT entry. In superelement analysis, it must be a residual structure point exterior to all superelements.
4. The Xi value is the enforced acceleration at grid point Gi. The translation and rotation components are in consistent units and will be applied in the basic coordinate system regardless of the displacement coordinate system specified for Gi (CD field on GRID entry).
5. Only nonzero terms need be entered.
6. See "**Superelement Analysis**" on page 470 of the *MSC.Nastran Reference Guide* for the theoretical basis of inertia relief with superelements.
7. If any DMIG entry is changed or added on restart then a complete re-analysis is performed. Therefore, DMIG entry changes or additions are not recommended on restart.

DMIJ Direct Matrix Input at js-Set of the Aerodynamic Mesh

Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the interference elements of a CAERO2, use DMIJI or DMI.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIJ	NAME	“0”	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIJ	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

Example:

DMIJ	ALPH1	0	9	2	0			1	
DMIJ	ALPH1	1	1		1	1	.1		
	2	1	.1						

Field	Contents
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix being input. (Integer)
	1 = Square
	9 or 2 = Rectangular
	6 = Symmetric
TIN	Type of matrix being input: (Integer)
	1 = Real, single precision (One field is used per element)
	2 = Real, double precision (One field is used per element)
	3 = Complex, single precision (Two fields are used per element)

Field	Contents		
	4	=	Complex, double precision (Two fields are used per element)
TOUT	Type of matrix being created: (Integer)		
	0	=	Set by precision system cell (Default)
	1	=	Real, single precision
	2	=	Real, double precision
	3	=	Complex, single precision
	4	=	Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)		
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)		
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)		
CJ	Component number for grid point GJ. (0 < Integer $\leq$ 6; blank or zero if GJ is a scalar or extra point.)		
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)		
Ci	Component number for Gi for a grid point. (0 < CJ $\leq$ 6; blank or zero if Gi is a scalar or extra point.)		
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)		

Remarks:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. Also, DMIJ may also be used for the W2GJ and FA2J entries. Again, a single column is required. If both DMI and DMIJ are specified for W2GJ or FA2J, the DMI entry will be used. DMI may NOT be used for AEDW and AEPRESS.
2. The js-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIJ data will be partitioned to the j-set,

not reduced. No warnings are issued about truncated data.The j-set DOF's for each aerodynamic element / grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X		
CAERO2-Y		X			
CAERO2-Z			X		
CAERO2-ZY		X	X		

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.
4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices (very rare in the j-set!) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. the number of columns in the matrix is NCOL. (The number of rows in all DMIJ matrices is always the js-set size--the union of the j-set and the permanently SPC'd partition). The GJ term is used for the column index. the CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
    - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).

- If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Training null columns of the js-size matrix will be truncated.

8. The matrix names must be unique among all DMIJ.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, not that a double precision specification requires a “D” type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIJ matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIJ input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIJ entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIJ entry changes or additions are not recommended on restart.

DMIJI Direct Matrix Input at js-Set of the Interference Body

Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the slender elements of a CAERO2, or a CAERO1, 3, 4 or 5 use DMIJ or DMI.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIJI	NAME	“0”	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIJ	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

Example:

DMIJI	ALPH1	0	9	2	0			1	
DMIJI	ALPH1	1	1		1	1	.1		
	2	1	.1						

Field	Contents
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix being input. (Integer)
	1 = Square
	9 or 2 = Rectangular
	6 = Symmetric
TIN	Type of matrix being input: (Integer)
	1 = Real, single precision (One field is used per element)
	2 = Real, double precision (One field is used per element)
	3 = Complex, single precision (Two fields are used per element)

Field	Contents		
	4	=	Complex, double precision (Two fields are used per element)
TOUT	Type of matrix being created: (Integer)		
	0	=	Set by precision system cell (Default)
	1	=	Real, single precision
	2	=	Real, double precision
	3	=	Complex, single precision
	4	=	Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)		
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)		
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)		
CJ	Component number for grid point GJ. (0 < Integer $\leq$ 6; blank or zero if GJ is a scalar or extra point.)		
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)		
Ci	Component number for Gi for a grid point. (0 < CJ $\leq$ 6; blank or zero if Gi is a scalar or extra point.)		
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)		

Remarks:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. DMI may NOT be used for AEDW and AEPRESS.
2. The js-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIJI data will be partitioned to the j-set, not reduced. No warnings are issued about truncated data. The j-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO2-Y		X			
CAERO2-Z			X		
CAERO2-ZY		X	X		

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.
4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices (very rare in the j-set!) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. the number of columns in the matrix is NCOL. (The number of rows in all **DMIJI** matrices is always the js-set size--the union of the j-set and the permanently SPC'd partition). The GJ term is used for the column index. the CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
   - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
   - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Training null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIJI.

9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, not that a double precision specification requires a “D” type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIJ matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same **DMIJI** input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIJ entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIJ entry changes or additions are not recommended on restart.

DMIK Direct Matrix Input at ks-Set of the Aerodynamic Mesh

Defines direct input matrices related to physical (displacement) degrees-of-freedom (ks-set) of aerodynamic grid points. These include WKK, WTFACT and input forces associated with AEFORCE entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIK	NAME	“0”	IFO	TIN	TOUT	POLAR		NCOL	

Column Entry Format:

DMIK	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

Example:

DMIK	ALPH1	0	9	2	0			1	
DMIK	ALPH1	1	1		1	1	1.0		
	2	1	1.0						

Field	Contents
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix being input. (Integer)
	1 = Square
	9 or 2 = Rectangular
	6 = Symmetric
TIN	Type of matrix being input: (Integer)
	1 = Real, single precision (One field is used per element)
	2 = Real, double precision (One field is used per element)
	3 = Complex, single precision (Two fields are used per element)
	4 = Complex, double precision (Two fields are used per element)

Field	Contents
TOUT	Type of matrix being created: (Integer) 0 = Set by precision system cell (Default) 1 = Real, single precision 2 = Real, double precision 3 = Complex, single precision 4 = Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer $\leq$ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ $\leq$ 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

Remarks:

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEFORCE entries. In that paradigm, a single column is required. Also, DMIK may also be used for the WKK and WTFACT entries. If both DMI and DMIK are specified for WKK or WTFACT, the DMI entry will be used. DMI may NOT be used for AEFORCE.
2. The ks-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIK data will be partitioned to the k-set, not reduced. No warnings are issued about truncated data. The k-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X	X	
CAERO2-Y		X		X	
CAERO2-Z			X		X
CAERO2-ZY		X	X	X	X

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.
4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIK matrices is always the ks-set size--the union of the k-set and the permanently SPC'd partition). The GJ term is used for the column index. The CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
    - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
    - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Training null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIK.

9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, not that a double precision specification requires a “D” type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIK matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIK input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIK entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIK entry changes or additions are not recommended on restart.

DOPTPRM Design Optimization Parameters

Overrides default values of parameters used in design optimization.

Format:

1	2	3	4	5	6	7	8	9	10
DOPTPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	PARAM4	VAL4	
	PARAM5	VAL5	-etc.-						

Example:

DOPTPRM	IPRINT	5	DESMAX	10					

Field	Contents
PARAMi	Name of the design optimization parameter. Allowable names are given in **Table 8-7**. (Character)
VALi	Value of the parameter. (Real or Integer, see **Table 8-7**.)

Remark:

1. Only one DOPTPRM entry is allowed in the Bulk Data Section.
2. OPTCOD specifies which optimization code to be used in SOL 200 and METHOD specifies which optimization method to be used.

Table 8-7 PARAMi Names and Descriptions

Name	Description, Type, and Default Value
APRCOD	Approximation method to be used. 1 = Direct Linearization; 2=Mixed Method based on response type; 3 = Convex Linearization. APRCOD = 1 is recommended for shape optimization problems. (Integer 1, 2, or 3; Default = 2)
CONV1	Relative criterion to detect convergence. If the relative change in objective between two optimization cycles is less than CONV1, then optimization is terminated. (Real > 0.0; Default = 0.001 for sizing / shape optimization; Default = 0.0001 for topology optimization)

Table 8-7 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value
CONV2	Absolute criterion to detect convergence. If the absolute change in objective between two optimization cycles is less than CONV2, then optimization is terminated. (Real > 0.0; Default = 1.0E-20)
CONVDV	Relative convergence criterion on design variables. (Real > 0.0; Default = 0.001 for sizing / shape optimization; Default = 0.0001 for topology optimization)
CONVPR	Relative convergence criterion on properties. (Real > 0.0; Default = 0.001)
CT	Constraint tolerance. Constraint is considered active if current value is greater than CT. (Real < 0.0; Default = -0.03)
CTMIN	Constraint is considered violated if current value is greater than CTMIN. (Real > 0.0; Default = 0.003)
DABOBJ	Maximum absolute change in objective between ITRMOP consecutive iterations (see ITRMOP) to indicate convergence at optimizer level. F0 is the initial objective function value. (Real > 0.0; Default = MAX[0.001*ABS(F0), 0.0001])
DELB	Relative finite difference move parameter. (Real > 0.0; Default = 0.0001)
DELOBJ	Maximum relative change in objective between ITRMOP consecutive iterations to indicate convergence at optimizer level. (Real > 0.0; Default = 0.001)
DELP	Fractional change allowed in each property during any optimization design cycle. This provides constraints on property moves. (Real > 0.0; Default = 0.2)
DELX	Fractional change allowed in each design variable during any optimization cycle. (Real > 0.0; Default = 0.5 for sizing / shape optimization; Default = 0.2 for topology optimization)
DESMAX	Maximum number of design cycles (not including FSD cycle) to be performed. (Integer $\geq$ 0; Default = 5 for sizing / shape optimization; Default = 30 for topology optimization)

Table 8-7 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
DISCOD	Discrete Processing Method: (Integer 0, 1, 2, 3 or 4; Default = 1)	
	0	No Discrete Optimization
	1	Design of Experiments
	2	Conservative Discrete Design
	3	Round up to the nearest design variable
	4	Round off to the nearest design variable
DISBEG	Design cycle ID for discrete variable processing initiation. Discrete variable processing analysis is carried out for every design cycle after DISBEG. (Integer≥0, default = 0=the last design cycle)	
DOBJ1	Relative change in objective attempted on the first optimization iteration. Used to estimate initial move in the one-dimensional search. Updated as the optimization progresses. (Real > 0.0; Default = 0.1)	
DPMAX	Maximum fraction of change on designed property (Default = 0.5) , used by Trust Region Method.	
DXMAX	Maximum fraction of change on design variable (Default = 1.0), used by Trust Region Method.	
DX1	Maximum relative change in a design variable attempted on the first optimization iteration. Used to estimate the initial move in the one dimensional search. Updated as the optimization progresses. (Real > 0.0; Default = 0.01)	
DX2	Absolute change in a design variable attempted on the first optimization iteration. (Real > 0.0; Default = 0.2*MAX[X(I)])	
ETA1 (η_1)	the cutting ratio 1 (Default = 0.01), used by Trust Region Method.	
ETA2 (η_2)	the cutting ratio 2 (Default = 0.25), used by Trust Region Method.	
ETA3 (η_3)	the cutting ratio 3 (Default = 0.7), used by Trust Region Method.	
GMAX	Maximum constraint violation allowed at the converged optimum. (Real > 0.0; Default = 0.005)	
GSCAL	Constraint normalization factor. See Remarks under the DSCREEN and DCONSTR entries. (Real > 0.0; Default = 0.001)	

Table 8-7 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
IGMAX	If IGMAX = 0, only gradients of active and violated constraints are calculated. If IGMAX > 0, up to NCOLA gradients are calculated including active, violated, and near active constraints. (Integer > 0; Default = 0)	
IPRINT	Print control during approximate optimization phase. Increasing values represent increasing levels of optimizer information. (0 ≤ Integer ≤ 7; Default = 0)	
	0	No output (Default)
	1	Internal optimization parameters, initial information, and results
	2	Same, plus objective function and design variables at each iterations
	3	Same, plus constraint values and identification of critical constraints
	4	Same, plus gradients
	5	Same, plus search direction
	6	Same, plus scaling factors and miscellaneous search information
	7	Same, plus one dimensional search information
IPRNT1	If IPRNT1 = 1, print scaling factors for design variable vector. (Integer 0 or 1; Default = 0)	
IPRNT2	If IPRNT2 = 1, print miscellaneous search information. If IPRNT2 = 2, turn on print during one-dimensional search process. (Warning: This may lead to excessive output.) (Integer 0, 1, or 2; Default = 0)	
ISCAL	Design variables are rescaled every ISCAL iterations. Set ISCAL = -1 to turn off scaling. (Integer; Default=NDV (number of design variables))	
ITMAX	Maximum number of iterations allowed at optimizer level during each design cycle. (Integer; Default = 40)	

Table 8-7 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
ITRMOP	Number of consecutive iterations for which convergence criteria must be satisfied to indicate convergence at the optimizer level. (Integer; Default = 2)	
ITRMST	Number of consecutive iterations for which convergence criteria must be met at the optimizer level to indicate convergence in the Sequential Linear Programming Method. (Integer > 0; Default = 2)	
IWRITE	FORTRAN unit for print during approximate optimization phase. Default value for IWRITE is set to the FORTRAN unit for standard output. (Integer>0, Default=6 or value of SYSTEM(2).)	
JTMAX	Maximum number of iterations allowed at the optimizer level for the Sequential Linear Programming Method. This is the number of linearized subproblems solved. (Integer $\geq$ 0; Default = 20)	
JPRINT	Sequential Linear Programming subproblem print. If JPRINT > 0, IPRINT is turned on during the approximate linear subproblem. (Default = 0)	
JWRITE	If JWRITE > 0, file number on which iteration history will be written. (Integer > 0; Default = 0)	
METHOD	Optimization Method: (Integer > 0; Default = 1)	
	= 1	Modified Method of Feasible Directions for both MSCADS and DOT.
	= 2	Sequential Linear Programming for both MSCADS and DOT
	= 3	Sequential Quadratic Programming for both MSCADS and DOT
	= 4	SUMT method for MSCADS
	= IJK	See Remark **1.**
OPTCOD	OPTCOD. (Character; Default = Blank)	
	=	Blank (taken from system cell number 413; Default = MSCADS for Design Optimization Option and BIGDOT for Topology Optimization Option)
	=	“MSCADS” : MSCADS is used
	=	“DOT” : DOT is used

Table 8-7 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
	=	"BIGDOT" : BIGDOT is used
P1	Print control items specified for P2. (Integer ≥ 0; Default = 0) Initial results are always printed prior to the first approximate optimization. If an optimization task is performed, final results are always printed for the final analysis unless PARAM,SOFTEXIT,YES is specified. These two sets of print are not controllable.	
	n	Print at every n-th design cycle.
P2	Items to be printed according to P1: (Integer; Default = 1)	
	0	No print.
	1	Print objective and design variables. (Default for sizing / shape optimization) Print objective. (Default for topology optimization) P2 ≥ 8 Print design variables for topology optimization
	2	Print properties.
	4	Print constraints.
	8	Print responses.
	16	Print weight as a function of a material ID (note that this is not a design quantity so that only inputs to the approximate design are available).
	n	Sum of desired items. For example, P2 = 10 means print properties and responses.
P2CALL	Maximum number of retained constraints of all categories to be printed per category. This single parameter can be used in place of the individual parameters P2CBL, P2CC, P2CDDV, P2CM, P2CP and P2CR. If any of these six parameters are non-zero, the P2CALL value is overridden for that constraint type. (Integer > 0, default is to print all retained constraints.)	
P2CBL	Maximum number of constraints on beam library dimensions to be printed. (Integer ≥ 0, default is to print all beam library constraints.)	
P2CC	Maximum number of constraints on connectivity properties to be printed. (Integer ≥ 0, default is to print all connectivity property constraints.)	

Table 8-7 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value
P2CDDV	Maximum number of constraints on dependent design variables to be printed. (Integer ≥ 0, default is to print all dependent design variable constraints.)
P2CM	Maximum number of constraints on material properties to be printed. (Integer ≥ 0, default is to print all material property constraints.)
P2CP	Maximum number of constraints on element properties to be printed. (Integer ≥ 0, default is to print all element property constraints.)
P2CR	Maximum number of constraints on design responses to be printed. (Integer ≥ 0, default is to print all retained design response constraints.)
P2RSET	ID of a SET1 Bulk Data entry to identify the set of retained responses (DRESP1, DRESP2 and / or DRESP3) to be printed. (Integer, Default is to print all responses associated with printed constraints. If P2CR is > 0, the set associated P2RSET > 0 will be printed independent of the responses associated with the printed constraint. If P2CR > 0 and PR2SET = -1, all retained responses will be printed.
PENAL	Penalty parameter used to transform an infeasible approximate optimization task to a feasible one. Setting this parameter to; e.g., 2.0 may improve optimizer performance when the starting design is infeasible. (Real, Default = 0.0)
PLVIOL	Flag for handling of property limit violation. By default, the job will terminate with a user fatal message if the property derived from design model (DVPRELi, DVMRELi, DVCRELi) exceeds the property limits. Setting PLVIOL to a non-zero number will cause the program to issue a user warning message by ignoring the property limits violation and proceed with the analysis. (Integer; Default=0)
PTOL	Maximum tolerance on differences allowed between the property values on property entries and the property values calculated from the design variable values on the DESVAR entry (through DVPRELi relations). PTOL is provided to trap ill-posed design models. (The minimum tolerance may be specified on user parameter DPEPS. See “**Parameters**” on page 659) (Real > 0.0; Default = 1.0E+35)

Table 8-7 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value
STPSCL	Scaling factor for shape finite difference step sizes, to be applied to all shape design variables. (Real > 0.0; Default = 1.0)
TCHECK	Topology Filtering (Checkerboarding) options (Integer 0 or 1) 1 Filtering algorithm is on for topology optimization (Default) 0 No filtering algorithm
UPDFAC1	Updating factor 1 (Default = 2.0), used by Trust Region Method.
UPDFAC2	Updating factor 2 (Default = 0.5), used by Trust Region Method.

Additional Remarks:

1. METHOD = IJK enables a user selectable optimization strategy as documented in Vanderplaats, G. N., ADS -- A Fortran Program for Automated Design Synthesis -- Version 1.10, NASA CR 177985, 1985. The I selects one of ten available strategy options:

I	ADS Strategy Option
0	None -- Go directly to the optimizer
1	Sequential unconstrained minimization using the exterior penalty function method
2	Sequential unconstrained minimization using the linear extended interior penalty function method
3	Sequential unconstrained minimization using the quadratic extended interior penalty function method
4	Sequential unconstrained minimization using the cubic extended interior penalty function method
5	Augmented Lagrange multiplier method
6	Sequential linear programming
7	Method of centers
8	Sequential quadratic programming
9	Sequential convex programming

The J selects one of five available optimizer options:

1	Fletcher-Reeves algorithm for unconstrained minimization
2	Davidon-Fletcher-Powell (DFP) variable metric method for unconstrained minimization
3	Broydon-Fletcher-Goldfarb-Shanno (BFGS) variable metric method for unconstrained minimization
4	Method of feasible directions for constrained minimization
5	Modified method of feasible directions for constrained minimization

And K selects one of eight available one-dimensional search strategies:

1	Find the minimum of an unconstrained function using the Golden Section method
2	Find the minimum of an unconstrained function using the Golden Section method followed by polynomial interpolation
3	Find the minimum of an unconstrained function by first finding bounds and then using the Golden Section method followed by polynomial interpolation
4	Find the minimum of an unconstrained function by polynomial interpolation / extrapolation without first finding bounds on the solution
5	Find the minimum of a constrained function using the Golden Section method
6	Find the minimum of a constrained function using the Golden Section method followed by polynomial interpolation
7	Find the minimum of a constrained function by first finding bounds and then using polynomial interpolation
8	Find the minimum of a constrained function by polynomial interpolation / extrapolation without first finding bounds on the solution

DPHASE Dynamic Load Phase Lead

Defines the phase lead term θ in the equation of the dynamic loading function.

Format:

1	2	3	4	5	6	7	8	9	10
DPHASE	SID	P1	C1	TH1	P2	C2	TH2		

Example:

DPHASE	4	21	6	2.1	8	6	7.2		

Field	Contents
SID	Identification number of DPHASE entry. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integers 1 through 6 for grid points; zero or blank for extra or scalar points)
THi	Phase lead θ in degrees. (Real)

Remarks:

1. One or two dynamic load phase lead terms may be defined on a single entry.
2. SID must be referenced on a RLOADi entry. Refer to the RLOAD1 or RLOAD2 entry for the formulas that define how the phase lead θ is used.
3. A DAREA, LSEQ or static load entry should be used to define a load at Pi and Ci.
4. In superelement analysis, DPHASE entries may only be applied to loads on points in the residual structure.

DRESP1 Design Sensitivity Response Quantities

Defines a set of structural responses that is used in the design either as constraints or as an objective.

Format:

1	2	3	4	5	6	7	8	9	10
DRESP1	ID	LABEL	RTYPE	PTYPE	REGION	ATTA	ATTB	ATT1	
	ATT2	-etc.-							

Example:

DRESP1	1	DX1	STRESS	PROD	2	3		102	
	103								

Field	Contents
ID	Unique entry identifier. (Integer > 0)
LABEL	User-defined label. (Character)
RTYPE	Response type. See **Table 8-8**. (Character)
PTYPE	Element flag (PTYPE = "ELEM") or property entry name. Used with element type responses (stress, strain, force, etc.) to identify the property type, since property entry IDs are not unique across property types. (Character: "ELEM", "PBAR", "PSHELL", etc.)
REGION	Region identifier for constraint screening. See Remark **10.** for defaults. (Integer > 0)
ATTA, ATTB, ATTi	Response attributes. See **Table 8-8**. (Integer > 0 or Real or blank)

Table 8-8 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
WEIGHT	Row Number ($1 \le ROW \le 6$) See Remark 24.	Column Number ($1 \le COL \le 6$)	SEIDi or All or blank. See Remark 12.
VOLUME	Blank	Blank	SEIDi or ALL or blank. See Remark 12.
FRMASS (see Remarks 28. & 29.)	Blank	Blank	Blank
COMP (see Remark 28.)	Blank	Blank	Blank
EIGN	Normal Modes Mode Number	Approximation Code. See Remark 19.	Blank
CEIG	Complex Eigenvalue Mode Number (Integer>0)	ALPHA or OMEGA (Default=ALPHA)	Blank
FREQ	Normal Modes Mode Number See Remark 18.	Approximation Code. See Remark 19.	Blank
LAMA	Buckling Mode Number	Approximation Code. See Remark 19.	Blank
DISP	Displacement Component	Blank or Mode Number	Grid ID
STRAIN	Strain Item Code	Blank or Mode Number	Property ID (PID) or Element ID (EID)
ESE	Strain Energy Item Code See Remark 21.	Blank or Mode Number	Property ID (PID) or Element ID (EID)

Table 8-8 Design Sensitivity Response Attributes (continued)

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
STRESS	Stress Item Code	Blank or Mode Number	Property ID (PID) or Element ID (EID)
FORCE	Force Item Code	Blank or Mode Number	Property ID (PID) or Element ID (EID)
SPCFORCE	SPC Force Component	Blank	Grid ID
CSTRAIN	Strain Item Code	LAMINA Number (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
CSTRESS	Stress Item Code	LAMINA Number (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
CFAILURE	Failure Criterion Item Code	LAMINA Number (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
CSTRAT	Composite Stress Ratio Item Code	LAMINA Number (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
TOTSE (Total Strain Energy)	Blank	Blank or Mode Number	SEIDi or All or blank. See Remark 12.
GPFORCE	Grid Point (see Remark 25.)	Blank	Element ID
GPFORCP	Grid Point (see Remark 26.)	Blank	Grid ID connected to ATTA grid to specify orientation.
ABSTRESS	Strress item code	Blank	Property ID (PID) or Element ID (EID)
FRDISP	Displacement Component	Frequency Value. (Blank, Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Grid ID

Table 8-8 Design Sensitivity Response Attributes (continued)

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
PRES	Acoustic Pressure Component (= 1 or 7)	Frequency Value. (Blank, Real ≥ 0.0 or Character) See Remarks **15.** and **20.**	Grid ID
FRVELO	Velocity Component	Frequency Value. (Blank, Real ≥ 0.0 or Character) See Remarks **15.** and **20.**	Grid ID
FRACCL	Acceleration Component	Frequency Value. (Blank, Real ≥ 0.0 or Character) See Remarks **15.** and **20.**	Grid ID
FRSPCF	SPC Force Component	Frequency Value. (Blank, Real ≥ 0.0 or Character) See Remarks **15.** and **20.**	Grid ID
FRSTRE	Stress Item Code	Frequency Value. (Blank, Real ≥ 0.0 or Character) See Remarks **15.** and **20.**	Property ID (PID) or Element ID (EID)
FRFORC	Force Item Code	Frequency Value. (Blank, Real ≥ 0.0 or Character) See Remarks **15.** and **20.**	Property ID (PID) or Element ID (EID)
PSDDISP	Displacement Component (see Remark **27.**)	Frequency Value. (Blank, Real ≥ 0.0 or Character). See Remarks **15.** and **20.**	Grid ID
PSDVELO	Velocity Component (see Remark **27.**)	Frequency Value (Blank, Real ≥ 0.0 or Character). See Remarks **15.** and **20.**	Grid ID

Table 8-8 Design Sensitivity Response Attributes (continued)

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
PSDACCL	Acceleration Component (see Remark 27.)	Frequency Value. (Blank, Real ≥ 0.0 or Character). See Remarks 15. and 20.	Grid ID
RMSDISP	Displacement Component	RANDPS ID	Grid ID
RMSVELO	Velocity Component	RANDPS ID	Grid ID
RMSACCL	Acceleration Component	RANDPS ID	Grid ID
TDISP	Displacement Component	Time Value. (Blank, Real, or Character) See Remarks 16. and 20.	Grid ID
TVELO	Velocity Component	Time Value. (Blank, Real, or Character) See Remarks 16. and 20.	Grid ID
TACCL	Acceleration Component	Time Value. (Blank, Real, or Character) See Remarks 16. and 20.	Grid ID
TSPCF	SPC Force Component	Time Value. (Blank, Real, or Character) See Remarks 16. and 20.	Grid ID
TSTRE	Stress Item Code	Time Value. (Blank, Real, or Character) See Remarks 16. and 20.	Property ID (PID) or Element ID (EID)
TFORC	Force Item Code	Time Value. (Blank, Real, or Character) See Remarks 16. and 20.	Property ID (PID) or Element ID (EID0
TRIM	AESTAT or AESURF Entry ID	Blank	Blank

Table 8-8 Design Sensitivity Response Attributes (continued)

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
STABDER	AESTAT or AESURF Entry ID	Restraint Flag. (Integer 0 or 1) See Remark 13.	Component
FLUTTER	Blank	Blank	See Remark 14.

Remarks:

1. Stress, strain, and force item codes can be found in "**Item Codes**" on page 873. For stress or strain item codes that have dual meanings, such as von Mises or maximum shear, the option specified in the Case Control Section will be used; i.e., STRESS(VONM) or STRESS(MAXS).
2. RTYPE = "CSTRESS", "CSTRAIN", "CFAILURE", and "CSTRAT" are used only with the PCOMP entry. "CSTRESS" and "CSTRAIN" item codes are described under Table 1 (Element Stress / Strain Item Codes) in "**Item Codes**" on page 873. "CFAILURE" and "CSTRAT" item codes are described under Table 2 (Element Force Item Codes) in "**Item Codes**" on page 873. Only force item codes that refer to failure indices of direct stress and interlaminar shear stress are valid.

   The CFAILURE and CSTRAT response types requires the following specifications on the applicable entries:

   - Failure theory in the FT field on PCOMP entry.
   - Allowable bonding shear stress in the SB field on PCOMP entry.
   - Stress limits in the ST, SC, and SS fields on all MATi entries.
3. ATTB can be used for responses of weight composite laminae, dynamics, real and complex eigenvalues, and stability derivatives. For other responses, this field must be blank.
4. All grids associated with a DRESP1 entry are considered to be in the same region for screening purposes. Only up to NSTR displacement constraints (see DSCREEN entry) per group per load case will be retained in the design optimization phase.
5. DRESP1 identification numbers must be unique with respect to DRESP2 identification numbers.
6. If PTYPE = "ELEM", the ATTi correspond to element identification numbers.

7. If RTYPE = "DISP", “SPCFORCE”, “GPFORCE”, "TDISP", "TVELO", "TACCL" or "TSPCF", multiple component numbers (any unique combination of the digits 1 through 6 with no embedded blanks) may be specified on a single entry. Multiple response components may not be used on any other response types.
8. If RTYPE = "FRDISP", "FRVELO", "FRACCL", or "FRSPCF" only one component number may be specified in the ATTA field. Numbers 1 through 6 correspond to real (or magnitude) components and 7 through 12 imaginary (or phase) components. If more than one component for the same grid is desired, then a separate entry is required.
9. Real / imaginary representation is the default for complex response types. Magnitude / phase representation must be requested by the corresponding Case Control command; e.g., DlSP(PHASE) = ALL for FRDISP type responses.
10. REGION is used for constraint screening. The NSTR field on DSCREEN entries gives the maximum number of constraints retained for each region per load case.

    IF RTYPE = "WEIGHT", "VOLUME", "LAMA", "EIGN", "FREQ", “CEIG”, “TOTSE”, “RMSDISP”, “RMSVELO”, “RMSACCL”, no REGION identification number should be specified. If the region field is left blank for a grid response, one region is created for each DRESP1 ID. If the region field is left blank for an element response, one region is created for each property ID invoked. Usually, the default value is appropriate.

    If the REGION field is not blank, all the responses on this entry as well as all responses on other DRESP1 entries that have the same RTYPE and REGION identification number will be grouped into the same region.
11. REGION is valid only among the same type of responses. Responses of different types will never be grouped into the same region, even if they are assigned the same REGION identification number by the user.
12. If RTYPE = "WEIGHT", "VOLUME", or “TOTSE” field ATTi = "ALL" implies total weight / volume / total strain energy of all superelements except external superelements, 0 implies residual only and i implies SEID=i. Default="ALL”
13. RTYPE = "STABDER" identifies a stability derivative response. ATTB is the restraint flag for the stability derivative. ATTB = 0 means unrestrained, and ATTB = 1 means restrained. For example, ATTA = 4000, ATTB = 0, and ATT1 = 3 reference the unrestrained C_z derivative for the AESTAT (or AESURF) entry ID = 4000.

14. RTYPE = "FLUTTER" identifies a set of damping responses. The set is specified by ATTi:

    ATT1 = Identification number of a SET1 entry that specifies a set of modes.

    ATT2 = Identification number of an FLFACT entry that specifies a list of densities.

    ATT3 = Identification number of an FLFACT entry that specifies a list of Mach numbers.

    ATT4 = Identification number of an FLFACT entry that specifies a list of velocities.

    If the flutter analysis is type PKNL, it is necessary to put PKNL in the PTYPE field of this entry.

15. For RTYPE = "FRXXXX", “PSDXXXX” and “PRES” a real value for ATTB specifies a frequency value in cycles per unit time. If a real ATTB value is specified, then the responses are evaluated at the closest excitation frequency. The default for ATTB is all excitation frequencies. See Remark **20.** for additional ATTB options. The OFREQ Case Control command has no affect on the selection of the frequencies.

16. For RTYPE = "TDISP", "TVELO", "TACCL", "TSPCF", "TFORC", and "TSTRE", ATTB specifies a time value. If ATTB is specified, then the responses are evaluated at the closest time selected by the OTIME command. The default for ATTB is all time steps selected by the OTIME command.

17. Intermediate station responses on CBAR elements due to PLOAD1 and/or CBARAO entries may not be defined on the DRESP1 entry.

18. RTYPE = "EIGN" refers to normal modes response in terms of eigenvalue (radian/time)**2 while RTYPE = "FREQ" refers to normal modes response in terms of natural frequency or units of cycles per unit time.

19. For RTYPE = LAMA, EIGN or FREQ, the response approximation used for optimization can be individually selected using the ATTB field when APRCOD = 1 is being used.

    For RTYPE = LAMA, ATTB = blank or 1 selects direct linearization, ATTB = 2 = inverse linearization.

    For RTYPE = EIGEN or FREQ, ATTB = blank = Rayleigh Quotient Approximation, = 1 = direct linearization, = 2 = inverse approximation.

    The default Rayleigh Quotient Approximation should be preferred in most cases.

20. Character input for ATTB is available for RTYPE of FRXXXX, PSDXXXX, TXXXX and PRES. The character input represents a mathematical function and the options for character input are SUM, AVG, SSQ, RSS, MAX and MIN. The expression of mathematical function is shown as follows:

$$SUM(X_1, X_2, \ldots, Xn) = \sum_{i=1}^{n} X_i$$

$$AVG(X_1, X_2, \ldots, Xn) = \sum_{i=1}^{n} X_i / n$$

$$SSQ(X_1, X_2, \ldots, Xn) = \sum_{i=1}^{n} X_i^2$$

$$RSS(X_1, X_2, \ldots, Xn) = \sqrt{\sum_{i=1}^{n} X_i^2}$$

$$MAX(X_1, X_2, \ldots, Xn) = \text{Maximum value among } X_i (i=1 \text{ to } n)$$

$$MIN(X_1, X_2, \ldots, Xn) = \text{Minimum value among } X_i (i=1 \text{ to } n)$$

where X_i is the response for a forcing frequency or time step. For example

DRESP1,10,DX1,FRSTRE,ELEM,,3,AVG,10

yields a response which is equal to the average stress for element 10 across all forcing frequencies. **NOTE:** that the response computed is considered a type 2 response. Therefore, if referenced on a DRESP2, the ID of such DRESP1 (ATTB with character input) must be listed following DRESP2 keyword.

21. Element strain energy item codes can be found under **Table 6-6** in "**Element Strain Energy Item Codes**" on page 938. Only element strain energy and element strain energy density can be referenced on a DRESP1 entry.

22. For RTYPE=RMSDISP, RMSVELO, or RMSACCL the ATTB specifies the appropriate RANDPS ID.

23. Input other than 1 or 7 of ATTA field, acoustic pressure component, for PRES response type will be reset to 1 (if less than 7) or 7 (if greater than 6 and less than 13).

24. Design response weight is obtained from Grid Point Weight Generator for a reference point GRDPNT (see parameter GRDPNT). If GRDPNT is either not defined, equal to zero, or not a defined grid point, the reference point is taken as the origin of the basic coordinate system. Fields ATTA and ATTB refer to the row and column numbers of the rigid body weight matrix, which is partitioned as follows:

$$[W] = \begin{bmatrix} W_x & W_{12} & W_{13} & W_{14} & W_{15} & W_{16} \\ W_{21} & W_y & W_{23} & W_{24} & W_{25} & W_{26} \\ W_{31} & W_{32} & W_z & W_{34} & W_{35} & W_{36} \\ W_{41} & W_{42} & W_{43} & I_x & W_{45} & W_{46} \\ W_{51} & W_{52} & W_{53} & W_{54} & I_y & W_{56} \\ W_{61} & W_{62} & W_{63} & W_{64} & W_{65} & I_z \end{bmatrix}_{6 \times 6}$$

The default values of ATTA and ATTB are 3, which specifies weight in the Z direction. Field ATT1 = “ALL” implies total weight of all superelements except external superelements. SEIDi refers to a superelement identification number. SEIDi = “0” refers to the residual superelement. The default of ATT1 is blank which is equivalent to “ALL”.

25. For RTYPE = GPFORCE, the PTYPE field is used to designate the GRID ID at which the force is defined. Output that is produced using PARAM NOELOF > 0 is not supported for the DRESP1 entry.

26. For RTYPE = GPFORCP, the PTYPE field is blank. The grid point force is for the sum of all elements from the GRID ID listed in ATTA to the GRID (orient ID) listed in ATTi. This response corresponds to that produced with PARAM NOELP > 0. It is not necessary to set PARAM NOELP > 0 to compute the GPFORCP response.

27. For RTYPE = PSDXXXX, the PTYPE field specifies the RANDPS ID.

28. RTYPE=COMP (compliance of structures = $P^T u$) and FRMASS (mass fraction of designed elements) entries are used for topology optimization only.

29. RTYPE=FRMASS is the mass divided by the mass calculated if all design variables are 1.0. FRMASS is calculated for designed elements only. FRMASS = 1.0 if all design variables are 1.0.

DRESP2 Design Sensitivity Equation Response Quantities

Defines equation responses that are used in the design, either as constraints or as an objective.

Format:

1	2	3	4	5	6	7	8	9	10
DRESP2	ID	LABEL	EQID or FUNC	REGION	METHOD	C1	C2	C3	
	"DESVAR"	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	"DTABLE"	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						
	"DRESP1"	NR1	NR2	NR3	NR4	NR5	NR6	NR7	
		NR8	-etc.-						
	"DNODE"	G1	CMM	G2	CMP2	G3	CMP3		
		G4	C4	etc.					
	"DVPREL1"	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7	
		DPIP8	DPIP9	-etc.-					
	"DVCREL1"	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7	
		DCIC8	DCIC9	-etc.-					
	"DVMREL1"	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7	
		DMIM8	DMIM9	-etc.-					
	"DVPREL2"	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7	
		DPI2P8	DPI2P9	-etc.-					
	"DVCREL2"	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7	
		DCI2C8	DCI2C9	-etc.-					
	"DVMREL2"	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7	
		DMI2M8	DMI2M9	-etc.-					
	"DRESP2"	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7	
		NRR8	-etc.-						

Example:

DRESP2	1	LBUCK	5	3					
	DESVAR	101	3	4	5	1	205	209	
		201							
	DTABLE	PI	YM	L					
	DRESP1	14	1	4	22	6	33	2	
	DNODE	14	1	4	1	22	3		
		2	1	43	1				
	DVPREL1	101	102						
	DVCREL1	201	202						
	DVMREL1	301							
	DVPREL2	401	402						
	DVCREL2	501							
	DVMREL2	601	602	603					
	DRESP2	50	51						

Field	Contents
ID	Unique identification number. (Integer > 0)
LABEL	User-defined label. (Character)
EQID	DEQATN entry identification number. (Integer > 0)
FUNC	Function to be applied to the arguments. See Remark **8.** (Character)
REGION	Region identifier for constraint screening. See Remark **5.** (Integer > 0)
METHOD	When used with FUNC = BETA, METHOD = MIN indicates a minimization task while MAX indicates a maximization task. (Default = MIN) When used with FUNCT = MATCH, METHOD = LS indicated a least squares while METHOD = BETA indicated minimization of the maximum difference. (Default = LS)
Ci	Constants used when FUNC = BETA or FUNC = MATCH in combination with METHOD = BETA. See Remark **8.** (Real, Defaults: C1 = 100., C2 = .005)
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)

Field	Contents
“DTABLE”	Flag indicating that the labels for the constants in a DTABLE entry follow. (Character)
LABLj	Label for a constant in the DTABLE entry. (Character)
“DRESP1"	Flag indicating DRESP1 entry identification numbers. (Character)
NRk	DRESP1 entry identification number. (Integer > 0)
“DNODE”	Flag indicating grid point and component identification numbers. (Character)
Gm	Identification number for any grid point in the model. (Integer > 0)
Cm	Component number of grid point Gm. ($1 \leq$ Integer ≤ 3)
“DVPREL1”	Flag indicating DVPREL1 entry identification number. (Character)
DPIPi	DVPREL1 entry identification number. (Integer > 0)
“DVCREL1”	Flag indicating DVCREL1 entry identification number. (Character)
DCICi	DVCREL1 entry identification number. (Integer > 0)
“DVMREL1”	Flag indicating DVPREL2 entry identification number. (Character)
DMIMi	DVMREL1 entry identification number. (Integer > 0)
“DVPREL2”	Flag indicating DVPREL2 entry identification number. (Character)
DPI2Pi	DVPREL2 entry identification number. (Integer > 0)
“DVCREL2”	Flag indicating DVCREL2 entry identification number. (Character)
DCI2Ci	DVCREL2 entry identification number. (Integer > 0)
“DVMREL2”	Flag indicating DVMREL2 entry identification number. (Character)
DMI2Mi	DVMREL2 entry identification number. (Integer > 0)
“DRESP2”	Flag indicating other DRESP2 entry identification number. (Character)
NRRk	DRESP2 entry identification number. (Integer > 0)

Remarks:

1. DRESP2 entries may only reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, and DVMREL2 entries. They may also reference other DRESP2 entries. However, a DRESP2 entry cannot reference itself directly or recursively.

2. a) If the referenced DRESP1 entries span subcases, the DRSPAN Case Control command is required to identify DRESP1 IDs for each subcase. DRESP2 entries that span subcases must be invoked above the subcase level by DESGLB on DESOBJ commands.
   b) Referenced DRESP entries that span superelements are supported automatically.
   c) Referenced DRESP2 entries cannot span subcases or superelements.
3. DRESP2 entries must have unique identification numbers with respect to DRESP1 entries.
4. The “DESVAR”, “DTABLE”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, DVCREL2”, “DVMREL2”, and "DRESP2" flags in field 2 must appear in the order given above. Any of these words, along with the identification numbers associated with them, may be omitted if they are not involved in this DRESP2 relationship. However, at least one of these ten types of arguments must exist.
5. The REGION field follows the same rules as for the DRESP1 entries. DRESP1 and DRESP2 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP2 entry in the same region.
6. The variables identified by DVIDi, LABLj, NRk, the Gm, CMPM pairs, DPIPi, DCICm, DMIMn, DPI2Po, DCI2Cp, DMI2Mq, and NRRu are assigned (in that order) to the variable names (x1, x2, x3, etc.) specified in the left-hand side of the first equation on the DEQATN entry referenced by EQID. In the example below,

   DESVARs 101 and 3 are assigned to arguments A and B.
   DTABLEs PI and YM are assigned to arguments C and D.
   Grid 14, Component 1 is assigned to argument R.

DRESP2	1	LBUCK	5	3					
	DESVAR	101	3						
	DTABLE	PI	YM						
	DNODE	14	1						
DEQATN	5	F1(A, B, C, D, R)=A+B*C-(D**3+10.0)+sin(C*R)							

7. (Gm, Cm) can refer to any grid component and is no longer limited to a designed grid component.

8. The FUNC attributes can be used in place of the EQID and supports the functions shown in the following table:

Function	Description
SUM	Sum of the arguments
AVG	Average of the arguments
SSQ	Sum of the squares of the arguments
RSS	Square root of the sum of the squares of the arguments
MAX	The maximum value of the argument list
MIN	The minimum value of the argument list
BETA	Minimize the maximum response. See Remark **10.**
MATCH	Match analysis results with user specified values. See Remark **11.**

When EQID has character input, the DEQATN entry is no longer needed. The functions are applied to all arguments on the DRESP2 regardless of the type. See Remark **20.** of the DRESP1 entry for an explanation of these functions.

9. The number of arguments of a DEQATN can be more than the number of values defined on the DRESP2 if the DRESP1s referenced have RTYPE with ‘FR’ or ‘PSD’ prefix. Arguments are still positional. The extra arguments in the DEQATN must appear at the end of the argument list. The discrepancy is resolved internally with the forcing frequency(ies) associated with DRESP1s. An example is shown as follows:

DRESP1	10	FDISP1	FRDISP			1	10.	1001	
DRESP1	20	FDISP2	FRDISP			1	20.	1001	
DRESP2	30	AVGFD	100						
	DRESP1	10	20						
DEQATN	100	AVG(D1,D2,F1,F2) = (D1 / F1+D2 / F2)*0.5							

In the above example, the DEQATN has two more additional terms than have been defined on the DRESP2. The first additional term is the forcing frequency (in hertz) of the first DRESP1 ID on the DRESP2. The second additional term is the forcing frequency of second DRESP1 ID in the list. When all DRESP1s involved have the same frequency, the user is not required to name all the additional terms in the argument list of DEQATN.

10. FUNC = BETA facilitates a design task where the objective is to minimize the maximum response. It does this by creating the following design task:

$$\text{Minimize } \phi = C_1 X_\beta$$

$$\text{Subject to } g = \frac{r_j - \gamma X_\beta}{C_3} \le 0$$

where γ is determined from

$$C_2 = (r_{jmax} - \gamma X_\beta) / C_3$$

User input parameters C_1, C_2, C_3 therefore have the following meaning:

C_1 weights the spawned design variable, X_β, to create the objective. Since X_β starts at 1.0, C_1 is the initial objective.

C_2 sets the initial value of the maximum constraint created by this process. The default values of 0.005 is equal to DOPTPRM parameter GMAX.

C_3 is an offset value to avoid dividing by zero when creating constraints.

Note: DRESP2s that use FUNC = BETA can only be invoked by a DESOBJ Case Control command.

11. FUNC = MATCH a design task where the objective is to minimize the difference between analysis results, r_j, that are associated with DRESP1s and target values, r_j^T, that are input using DTABLE data

When METHOD = LS, a least square minimization is performed where the objective is

$$\phi_j = \sum_{j=1}^{m} \left(\frac{r_j - r_j^T}{r_j^T} \right)^2$$

When METHOD = BETA, the design task becomes one of minimizing the maximum normalized difference between the analysis and target values

$$\frac{r_j - r_j^T}{\left| r_j^T \right|}$$

in the same manner as outlined in Remark **10.**

Note: DRESP2s that use FUNC = MATCH can only be invoked by a DESOBJ Case Control command. Analysis results and target values are obtained from an equal number of DRESP1, DTABLE pairs.

DRESP3

Defines an external response using user-supplied routine(s).

Format:

1	2	3	4	5	6	7	8	9	10
DRESP3	ID	LABEL	GROUP	TYPE	REGION				
	"DESVAR"	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	etc.						
	"DTABLE"	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	etc.						
	"DRESP1"	NR1	NR2	NR3	NR4	NR5	NR6	NR7	
		NR8	etc.						
	"DNODE"	G1	C1	G2	C2	G3	C3		
		G4	C4	etc.					
	"DVPREL1"	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7	
		DPIP8	DPIP9	etc.					
	"DVCREL1"	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7	
		DCIC8	DCIC9	-etc.-					
	"DVMREL1"	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7	
		DMIM8	DMIM9	-etc.-					
	"DVPREL2	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7	
		DPI2P8	DPI2P9	-etc.-					
	'DCREL2"	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7	
		DCI2C8	DCI2C9	-etc.-					
	"DVMREL2"	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7	
		DMI2M8	DMI2M9	-etc.-					
	"DRESP2"	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7	
		NRR8	-etc.-						
	"USRDATA"	String							
		-etc.-							

Example:

1	2	3	4	5	6	7	8	9	10
DRESP3	1	LBUCK	TAILWNG	BUCK					
	DESVAR	101	3	4	5	1	205	209	
		201							
	DTABLE	PI	YM	L					
	DRESP1	14	1	4	22	6	33	2	
	DNODE	14	1	4	1	22	3		
		2	1	43	1				
	DVPREL1	101	102						
	DVCREL1	201	202						
	DVMREL1	301							
	DVPREL2	401	402						
	DVCREL2	501							
	DVMREL2	601	602	603					
	DRESP2	50	51						
	USRDATA	Constants: 12345.6 789.0 99.							

Field	Contents
ID	Unique identification number. (Integer > 0)
LABEL	User-defined label. (Character)
GROUP	Group name the external response type belongs to (Character). See Remark **2.**
TYPE	External response type (Character). See Remark **3.**
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	Flag indicating that the labels for the constants in a DTABLE entry follow. (Character)
LABLj	Label for a constant in the DTABLE entry. (Character)
“DRESP1"	Flag indicating DRESP1 entry identification numbers. (Character)
NRk	DRESP1 entry identification number. (Integer > 0)
“DNODE”	Flag signifying that the following fields are grid points.
Gm	Grid point identification number. (Integer > 0)

Field	Contents
Cm	Degree-of-freedom number of grid point Gm. (1 ≤ Integer ≤ 3)
“DVPREL1”	Flag indicating DVPREL1 entry identification number. (Character)
DPIPi	DVPREL1 entry identification number. (Integer > 0)
“DVCREL1”	Flag indicating DVCREL1 entry identification number. (Character)
DCICi	DVCREL1 entry identification number. (Integer > 0)
“DVMREL1”	Flag indicating DVMREL1 entry identification number. (Character)
DMIMi	DVMREL1 entry identification number. (Integer > 0)
“DVPREL2”	Flag indicating DVPREL2 entry identification number. (Character)
DPI2Pi	DVPREL2 entry identification number. (Integer > 0)
“DVCREL2”	Flag indicating DVCREL2 entry identification number. (Character)
DCI2Ci	DVCREL2 entry identification number. (Integer > 0)
“DVMREL2”	Flag indicating DVMREL2 entry identification number. (Character)
DMI2Mi	DVMREL2 entry identification number (Integer > 0)
“DRESP2”	Flag indicating other DRESP2 entry identification number. (Character)
NRRk	DRESP2 entry identification number. (Integer > 0)
“USRDATA”	Flag indicating user input data (Character). See Remark **8.**

Remarks:

1. DRESP3 entries may reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, DVMREL2 and DRESP2 entries. However, a DRESP3 entry cannot reference another DRESP3 entry.
2. The group name must be referenced by an FMS CONNECT entry.
3. Multiple types of external responses can be defined in one group. Each type name identifies a specific external response evaluated in the user-supplied routines. See “**Building and Using the Sample Programs**” on page 255 of the *MD Nastran 2006 Installation and Operations Guide* for a discussion of how to incorporate external responses.

4. a) Referenced DRESP2 entries cannot span subcases or superelements.
   b) If referenced DRESP1 entries span subcases, the DRSPAN Case Control command is required to identify the DRESP1 IDs for each subcase. DRESP3 entries that span subcases must be invoked above the subcase level by DESGLB or DESOBJ commands.
   c) Referenced DRESP1 entries that span superelements are supported automatically.
5. DRESP3 entries must have unique identification numbers with respect to DRESP2 and DRESP1 entries.
6. The “DESVAR”, “DTABLE”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1", “DVPREL2”, DVCREL2”, “DVMREL2”, “DRESP2”, and “USRDATA” keywords on the continuation entries must appear in the order given above. Any of these words, along with the subsequent data associated with them, may be omitted if they are not involved in this DRESP3 relationship. However, at least one of these types of arguments must exist.
7. The REGION field follows the same rules as for the DRESP1 entries. DRESP1 and DRESP3 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP3 entry in the same region.
8. The data in the USRDATA field is character string based. It provides a convenient way to pass constants to the external response server routines. The maximum number of characters allowed is 32000.

DSCREEN Design Constraint Screening Data

Defines screening data for constraint deletion.

Format:

1	2	3	4	5	6	7	8	9	10
DSCREEN	RTYPE	TRS	NSTR						

Example:

DSCREEN	STRESS	-0.7	2						

Field	Contents
RTYPE	Response type for which the screening criteria apply. See Remark **3.** (Character)
TRS	Truncation threshold. (Real; Default = -0.5)
NSTR	Maximum number of constraints to be retained per region per load case. See Remark **3.** (Integer > 0; Default = 20)

Remarks:

1. Grid responses associated with one particular load case are grouped by the specification of DRESP1 entries. From each group, a maximum of NSTR constraints are retained per load case.
2. Element responses are grouped by the property; i.e., all element responses for one particular load case belonging to the set of PIDs specified under ATTi on a DRESPi entry are regarded as belonging to the same region. In superelement sensitivity analysis, if the property (PID) is defined in more than one superelement, then separate regions are defined. A particular stress constraint specification may be applied to many elements in a region generating many stress constraints, but only up to NSTR constraints per load case will be retained.
3. For aeroelastic responses, that is RTYPE = “TRIM”, “STABDER”, and “FLUTTER”, the NSTR limit is applied to all DRESP1 IDs that are the same RTYPE and have the same REGION specified.
4. For responses that are not related to grids or elements, that is RTYPE = WEIGHT”, “VOLUME”, “EIGN”, “FREQ”, “LAMA”, CEIG”, FRMASS, COMP, and TOTSE”, NSTR is not used. TRS is still applicable.

5. The RTYPE field is set to EQUA if constraints that are associated with DRESP2 entries are to be screened. The RTYPE field is set to DRESP3 if constraints that are associated with DRESP3 entries are to be screened. If the REGION field on the DRESP2 or DRESP3 is blank, one region is established for each DRESP2/DRESP3 entry.
6. If a certain type of constraint exists but no corresponding DSCREEN entry is specified, all the screening criteria used for this type of constraint will be furnished by the default values.
7. Constraints can be retained only if they are greater than TRS. See the Remarks under the "**DCONSTR**" on page 1380 entry for a definition of constraint value.
8. Constraint screening is applied to each superelement.

DTABLE Table Constants

Defines a table of real constants that are used in equations (see DEQATN entry).

Format:

1	2	3	4	5	6	7	8	9	10
DTABLE	LABL1	VALU1	LABL2	VALU2	LABL3	VALU3	LABL4	VALU4	
	LABL5	VALU5	LABL6	VALU6	LABL7	VALU7	LABL8	VALU8	
		-etc.-							

Example:

DTABLE	PI	3.142	H	10.1	E	1.0E6			
	G	5.5E5	B	100.					

Field	Contents
LABLi	Label for the constant. (Character)
VALUi	Value of the constant. (Real)

Remarks:

1. Only one DTABLE entry may be specified in the Bulk Data Section.
2. LABLi are referenced by the LABi on the DRESP2, DRESP3, DVCREL2, DVMREL2, or DVPREL2 entries.
3. Trailing blank fields are permitted at the end of each line of ABLi/VALUi pairs, but intermediate blanks are not. (See the example above for permitted trailing blanks.)

DTI Direct Table Input

Defines table data blocks.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	NAME	"0"	T1	T2	T3	T4	T5	T6	
	V01	V02	-etc.-						
DTI	NAME	IREC	V1	V2	V3	V4	V5	V6	
	V7	V8	V9	V10	-etc.-	"ENDREC"			

Example: (The first logical entry is the header entry.)

DTI	XXX	0	3	4	4096		1	0	
	1.2	2.3							
DTI	XXX	1	2.0	-6	ABC	6.000	-1	2	
	4	-6.2	2.9	1	DEF	-1	ENDREC		

Field	Contents
NAME	Any character string that will be used in the DMAP sequence to reference the data block. See Remark **1.** (Character; the first character must be alphabetic.)
Ti	Trailer values. (Integer $\geq$ 0; Default = 32767)
IREC	Record number. (Integer > 1)
V0i, Vi	Value. (Integer, Real, Character or blank)
"ENDREC"	Flags the end of the string of values (V0i or Vi) that constitute record IREC. (Character)

Remarks:

1. The user defines the data block and therefore must write a DMAP (or ALTER a solution sequence), which includes the DTIIN modules, in order to use the DTI feature. See the *MD Nastran 2006 DMAP Programmer's Guide*. All of the rules governing the use of data blocks in DMAP sequences apply.
2. All fields following ENDREC must be blank.

3. The entry using IREC = 0 is called the header entry and is an optional entry. The values T1 through T6 go to a special record called the trailer. Other values on the optional continuation go to the header record. If the header entry or the trailer is not specified, T1 through T6 = 32767. On this entry, “ENDREC” may be used only if there is at least one continuation.
4. In addition to the optional header entry, there must be one logical entry for each record in the table. Null records require no entries.
5. “ENDREC” is used to input blank values at the end of a record. If “ENDREC” is not specified, the string for a record ends with the last nonblank field.
6. The maximum number of DMI and DTI data blocks is 1000.
7. If Ti is not an integer, a machine-dependent error will be issued that may be difficult to interpret.

DTI,ESTDATA Superelement Estimation Data Overrides

Provides override data for time and space estimation for superelement processing operations.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	ESTDATA	“0"							
	kd1	vd1	kd2	vd2	-etc.-				

The next entries are repeated for any superelement for which estimate data overrides are desired. IREC must be incremented by 1.

DTI	ESTDATA	IREC	SEFLAG	SEID	k1	v1	k2	v2	
	k3	v3	-etc.-						

Example:

DTI	ESTDATA	0							
	NOMASS	-1							
DTI	ESTDATA	1	SE	10	C1	5.5	C3	4.5	
	C7	7.3							

Field	Contents
kdi	Keyword for estimation parameter. (Character from **Table 8-9**.)
vdi	Value assigned to the estimation parameter kdi. (The type given in **Table 8-9**.)
IREC	Record number beginning with 1. (Integer>0)
SEFLAG	SEFLAG = “SE” or “SEID” indicates the next field containing a superelement identification number. (Character)
SEID	Superelement identification number. (Integer>0)
ki	Keyword for override of estimation parameter for indicated superelement. (Character from **Table 8-9**.)
vi	Value for keyword ki. (Type depends on ki as shown in the **Table 8-9**.)

Table 8-9 DTI,ESTDATA Input Parameter Descriptions

Input Parameters				Meaning and Comments
Keyword	Type	Default Value	Math Symbol	
CRMS*	Real	-1.0	C	Number of active columns in $[K_{oo}]$.
FCRMS*	Real	0.10		If FCRMS $\leq$ 0.0, FCRMS is used (c/o).
C1	Real	6.0	c_1	Average number of degrees-of-freedom per grid point in o-set.
C3	Real	8.0	c_3	Average number of connections per grid point.
C4	Real	0.15	c_4	I/O time (seconds) per block transferred.
C5	Real	6.0	c_5	Average number of effective degrees-of-freedom per grid point in a-set.
C6	Real	1.2	c_6	Total CPU factor.
C7	Real	6.0	c_7	Number of equivalent KGG data blocks for space estimation.
WF	Real	-1.0	W	If WF $\leq$ 0.0 then use available working storage in units of single-precision words.
NOMASS	Integer	1		If NOMASS $\neq$ 1 then exclude mass terms from estimates.
TSEX	Real	0.5 (min)		Threshold limit for CPU.
SSEX	Real	50.0 (blocks)		Threshold limit for space.
TWALLX	Real	5.0 (min)		Threshold limit for wall time.
BUFSIZ	Integer	Machine Buffsize	B	Buffsize. See "**The NASTRAN Statement (Optional)**" on page 8 of the *MSC.Nastran Reference Guide*.

Table 8-9 DTI,ESTDATA Input Parameter Descriptions (continued)

Input Parameters				Meaning and Comments
Keyword	Type	Default Value	Math Symbol	
ML	Real	Machine Loop Time	M	Arithmetic time for the multiply/add loop. See the *MD Nastran Configuration and Operations Guide.*
CONIO	Integer	Machine I/O ratio		I/O count/CPU equivalence
PREC	Integer	1 or 2		Machine Word Length (1 = long, 2 = short). See "**The NASTRAN Statement (Optional)**" on page 8 of the *MSC.Nastran Reference Guide.*
NLOADS	Integer	1	N_L	Number of loading conditions
SETYPE	Character	"T"		Superelement type (T = Tip)
CMAX	Real	-1.0	C_{max}	Maximum bandwidth

Parameters Obtained from SEMAP

NGI	Number of interior grid points.
NPE	Number of exterior grid points.
NS	Number of scalar points
NE	Number of elements.

Derived Parameters

$O = C1 + NGI$	Size of o-set.
$A = C5(NPE - NS) + NS$	Size of a-set.
T = BUFFSIZE/PREC	Number of matrix terms in a buffer.

Estimation Equations

For each superelement, estimates of CPU time and disk space are made using the following equations.

Table 8-10 Equations Used for CPU Time and Disk Space Estimate

Printout Symbol	Math Symbol	Equations
TD	T_1	$T_1 = 1/2 \cdot M \cdot O \cdot C^2$
TFBS	T_2	$T_2 = 2 \cdot M \cdot C \cdot O \cdot a$
TMAA	T_3	$T_3 = M \cdot O \cdot a^2$ (set to 0.0 if NOMASS ≠ +1)
TSE	T_{SE}	$T_{SE} = C_6(T_1 + T_2 + T_3)$
SLOO	S_1	$S_1 = O \cdot C \cdot \frac{\text{PREC}}{b}$
SGO	S_2	$S_2 = O \cdot a \cdot \frac{\text{PREC}}{B}$
SKGG	S_3	$S_3 = 36(NG_i + NG_e - NS)(c_3 + 1.0)\left(\frac{\text{PREC}}{B}\right)$
SSE	S_{SE}	$S_{SE} = S_1 + S_2 + c_7 \cdot S_3$
PASSES	p	FBS passes $= p = a \cdot O \cdot \frac{\text{PREC}}{WF}$
BKSTRN	BT	Blocks Transferred $= BT = 2 \cdot p \cdot S_1 + S_2 + p \cdot S_2$. (Last term omitted if NOMASS ≠ +1)
TWALL	T_W	Wall Time $= T_W = T_{SE} + c_4 \cdot BT$

Remarks:

1. In the superelement solution sequences, this data is stored automatically.
2. The header record continuation entries are optional if no global override data is to be specified. In this case, the complete header entry is optional.

- Active column data can come from one of several places. The value for CRMS is determined as follows:
- RMS from the entry when IREC > 0 and field 4 is “SE”.
- RMS from entries with IREC = 0.
- Computed bandwidth when PARAM,OLDSEQ is specified.
- If FCRMS is specified when IREC > 0 and field 4 is “SE”, then CRMS = FCRMS · O .
- If FCRMS is specified when IREC = 0, then CRMS = FCRMS · O .
- CRMS = 0.1 · O .

3. If CMAX is not specified, then it is defaulted to CRMS.
4. In the example above, mass terms are excluded for all superelements and new values are given for parameters C1, C3, and C7 for Superelement 10 only.
5. The estimates for TSEX, SSEX, and TWALLX are not printed unless at least one estimate exceeds the threshold.

DTI,INDTA Stress, Strain and / or Force Sort / Filter Item Code Override

Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	INDTA	“0"							

To specify / override items for a sort of stress quantities:

DTI	INDTA	“1"	B1	C1	B2	C2	“ENDREC”		

To specify / override items for a sort of force quantities:

DTI	INDTA	“2"	B1	C1	B2	C2	“ENDREC”		

Examples:

DTI	INDTA	0							

To specify / override items for a sort of stress quantities:

DTI	INDTA	1	64	18	75	18	ENDREC		

To specify / override items for a sort of force quantities:

DTI	INDTA	2	34	2	2	4	ENDREC		

Field	Contents
Bi	Element type identification number. See the table in “**Item Codes**” on page 873 for allowable values. (Integer > 0)
Ci	Item code identification number for the stress, strain, or force quantity on which the sort or filter is to be performed. See the table in the “**Item Codes**” on page 873 for allowable values. (Integer)

Remarks:

1. This table is recognized only in SOLs 101, 103, 105, 106, 108, 109, 111, 112, 114, 115, 144, 153, and for stress quantities only. One or more of the user parameters S1, S1G, or S1M must be specified with a value greater then or equal to zero in order to request sorting and / or filtering. See user parameter S1 in "**Parameters**" on page 659. In order to sort force or strain quantities, a DMAP Alter is required.
2. If the Ci value is -1, the element type will be suppressed on the output file. An example of this feature could be as follows: If an element type is to be sorted on two different values and output twice, this can be accomplished by two calls to the STRSORT module with two unique DTI tables. However, other element types will be printed twice. This additional print can be suppressed by setting their sort codes to -1.
3. **Table 8-11** lists the elements currently that are sortable. In addition, the element type identification number, the default stress output quantity, and the associated stress code identification numbers are provided. If this entry is not specified, then the stresses are sorted based on the default quantity given in **Table 8-11**.

   The following should be noted:

   a. The element type identification number is used internally by the program to differentiate element types.

   b. The stress code identification number is merely the word number in the standard printed output for the stress quantity of interest. For example, the thirteenth word of stress output for the CHEXA element is the octahedral shear stress. For this element type, the element identification number and the grid point ID each count as a separate word. Stress codes for the elements are tabulated in "**Item Codes**" on page 873.

   c. By default, stress sorting for the membrane and plate elements will be performed on the Hencky-von Mises stress. For maximum shear stress, the STRESS (MAXS) Case Control command should be specified.

Table 8-11 Sortable Elements

Element	Element Type ID Number	Default Stress Output Quantity and Identification Number	
		Quantity	Stress Code ID Number
CBAR	34	Maximum stress at end B	14
CBEAM	2	Maximum stress at end B	108
CBEND	69	Maximum stress at end B	20
CONROD	10	Axial stress	2
CELAS1	11	Stress	2
CELAS2	12	Stress	2
CELAS3	13	Stress	2
CHEXA	67	Hencky-von Mises or Octahedral stress	13
CQUAD4	33	Maximum shear or Hencky-von Mises stress at Z_2	17
CQUAD4*	144	Maximum shear or Hencky-von Mises stress at Z_2	19
CQUAD8	64	Maximum shear or Hencky-von Mises stress at Z_2	19
CQUADR	82	Maximum shear or Hencky-von Mises stress at Z_2	19
CPENTA	68	Octahedral stress	13
CROD	1	Axial stress	2
CSHEAR	4	No default	---
CTETRA	39	No default	---
CTRIA3	74	Maximum shear or Hencky-von Mises stress at Z_2	17
CTRIA6	75	Maximum shear or Hencky-von Mises stress at Z_2	19

Table 8-11 Sortable Elements (continued)

Element	Element Type ID Number	Default Stress Output Quantity and Identification Number	
		Quantity	Stress Code ID Number
CTRIAR	70	Maximum shear or Hencky-von Mises stress at Z_2	19
CTRIAX6	53	No default	---
CTUBE	3	Axial stress	2

*CORNER output

DTI,SETREE Superelement Tree Definition

Defines a superelement tree that determines the superelement processing order.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	SETREE	“1"	SEUP1	SEDOWN1	SEUP2	SEDOWN2	SEUP3	SEDOWN3	
	SEUP4	SEDOWN4	SEUP5	SEDOWN5	-etc.-				

Example:

DTI	SETREE	1	1	14	2	14	3	14	
	4	14	14	0					

Field	Contents
SEUPi	Identification number of the superelement upstream from SEDOWNi. (Integer > 0)
SEDOWNi	Identification number of the superelement into which SEUPi is assembled. (Integer $\geq$ 0)

Remarks:

1. SETREE entries or the DTI,SETREE entry are required for multi-level superelement configurations.
2. If an DTI,SETREE entry is provided, then SETREE entries are not required.
3. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.
4. If a superelement is not referenced on the DTI,SETREE or SETREE entry, then the manner in which it is handled depends on the type of that superelement. If it is a PART superelement, then the residual will be regarded as its downstream superelement and the undefined superelement will therefore be placed immediately above the residual in the tree. If it is a Main Bulk Data superelement, then it will also be handled like an undefined PART superelement as above *if all of its exterior points belong to the residual*. However, if one or more of its exterior points do not belong to the residual, then the program will terminate with a user fatal error complaining that one of more of the superelements are not in the same path.

5. If this entry is not present, the superelement tree and the processing order are determined automatically.
6. A superelement identification may appear only once in a SEUPi field.
7. On restart, if a superelement identification does not appear in a SEUPi field, its matrices will not be assembled, even though they may be present in the database.
8. See the *MSC.Nastran Superelement User's Guide* for a description of user-designated trees.
9. This entry is stored in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.
10. In the example above, the following superelement tree is defined:

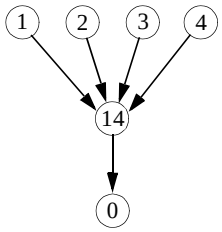


Figure 8-82 Sample Superelement Tree

DTI,SPECSEL Response Spectra Input Correlation Table

Correlates spectra lines specified on TABLED1 entries with damping values.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	SPECSEL	RECNO		TYPE	TIDl	DAMP1	TID2	DAMP2	
	TID3	DAMP3	TID4	DAMP4	TID5	DAMP5	-etc.-		

Example:

DTI	SPECSEL	1		A	1	.02	2	.04	
	3	.06							
DTI	SPECSEL	3		V	4	.01			

Field	Contents
RECNO	Spectrum number. (Integer > 0)
TYPE	Type of spectrum. (Character: “A” for acceleration, “V” for velocity, or “D” for displacement.)
TIDi	TABLED1 entry identification number. (Integer > 0)
DAMPi	Damping value assigned to TIDi. (Real)

Remarks:

1. The RECNO is the number of the spectrum defined by this entry. It is referenced on DLOAD Bulk Data entries.
2. The TIDi, DAMPi pairs list the TABLEDl entry, which defines a line of the spectrum and the damping value assigned to it. The damping value is in the units of fraction of critical damping.
3. This entry is placed in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.

DTI,SPSEL Response Spectra Generation Correlation Table

Correlates output requests with frequency and damping ranges.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	SPSEL	RECNO	DAMPL	FREQL	G1	G2	G3	G4	
	G5	G6	G7	-etc.-					

Example:

DTI	SPSEL	1	2	1	11	12			
DTI	SPSEL	2	4	3	1	7	11	12	
	13	14							

Field	Contents
DAMPL	Identification number of the FREQ, FREQ1, or FREQ2 Bulk Data entry that specifies the list of damping values. (Integer > 0)
FREQL	Identification number of the FREQi Bulk Data entry that specifies the list of frequencies. (Integer > 0)
Gi	Grid point number where response spectra will be calculated. (Integer > 0)
RECNO	Record number of spectra to be generated. (Sequential integer beginning with 1.)

Remarks:

1. This table is used in SOLs 109 and 112.
2. Damping values are in the units of fraction of critical damping.
3. Output of response spectra requires the use of the XYPLOT...SPECTRA(RECNO)/Gi... command, where Gi is restricted to the grid points listed on the (RECNO) record of this entry.
4. See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide* for example problems using this feature.
5. The SPSEL table is stored in the database automatically in SOLs 109 and 112. Once stored, the Bulk Data entry may be removed from the input file.

DVBSHAP Design Variable to Boundary Shapes

Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.

Format:

1	2	3	4	5	6	7	8	9	10
DVBSHAP	DVID	AUXMOD	COL1	SF1	COL2	SF2	COL3	SF3	

Example:

DVBSHAP	4	1	1	1.6					

Field	Contents
DVID	Design variable identification number of a DESVAR entry. (Integer > 0)
AUXMOD	Auxiliary model identification number. (Integer > 0)
COLi	Load sequence identification number from AUXMODEL Case Control command. (Integer > 0)
SFi	Scaling factor for load sequence identification number. (Real; Default = 1.0)

Remarks:

1. Design variable DVID must be defined on a DESVAR entry.
2. Multiple references to the same DVID and / or COLi will result in the vector addition of the referenced boundary shape vectors.
3. Multiple DVBSHAP entries may be specified.

DVCREL1 Design Variable to Connectivity Property Relation

Defines the relation between a connectivity property and design variables.

Format:

1	2	3	4	5	6	7	8	9	10
DVCREL1	ID	TYPE	EID	CPNAME	CPMIN	CPMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	COEF3	-etc.-		

Example:

DVCREL1	5	CQUAD4	1	ZOFFS		1.0			
	1	1.0							

Field	Contents
ID	Unique identification number. (Integer > 0)
TYPE	Name of an element connectivity entry, such as “CBAR”, CQUAD4”, etc. (Character)
EID	Element Identification number. (Integer > 0)
CPNAME	Name of connectivity property, such as “X1”, X2”, “X3”, “ZOFFS”, etc. (Character)
CPMIN	Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark **4.** (Real)
CPMAX	Maximum value allowed for this property. See Remark **4.** (Real; Default =1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation or keyword = “PVAL”. (If i = 1, Real or Character; if i > 1, Real)

Remarks:

1. The relationship between the connectivity property and design variables is given by:

$$CP_j = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc), ZOFFS (case insensitive) must be specified on the CPNAME field.
4. The default values for CPMIN and CPMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the associated DESVAR entry will keep the designed property within meaningful bounds.
5. When the character input is used, only a single design variable can be referenced on a DVCREL1 entry and “PVAL” is specified on the COEF1 field. If a DVCREL1 entry references more than one design variable with the PVAL option, a user fatal error message will be issued.

DVCREL2 Design Variable to Connectivity Property Relation

Defines the relation between a connectivity property and design variables with a user-supplied equation.

Format:

1	2	3	4	5	6	7	8	9	10
DVCREL2	ID	TYPE	EID	CPNAME	CPMIN	CPMAX	EQID		
	"DESVAR"	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	"DTABLE"	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						

Example:

DVCREL2	1	CBAR	100	X1	0.05	1.0	100		
	DESVAR	1001							
	DTABLE	X10							

Field	Contents
ID	Unique identification number. (Integer > 0)
TYPE	Name of an element connectivity entry, such as "CBAR", CQUAD4", etc. (Character)
EID	Element Identification number. (Integer > 0)
CPNAME	Name of connectivity property, such as "X1", X2", "X3", "ZOFFS", etc. (Character)
CPMIN	Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-15. Otherwise, it is -1.0E35. (Real)
CPMAX	Maximum value allowed for this property. (Real; Default =1.0E+20)
EQID	DEQATN entry identification number. (Integer > 0)
"DESVAR"	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation. (Real)

Field	Contents
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE entry. (Character)

Remarks:

1. The variable identified by DVIDi and LABLi correspond to variable names (x1, x2, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x1 through xN (where $N = m + n$) are assigned in the order DVID1, DVID2, ..., DVIDm, LABL1, LABL2, ..., LABLn.
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc.), ZOFFS (case insensitive) must be specified on the CPNAME field.

DVGRID Design Variable to Grid Point Relation

Defines the relationship between design variables and grid point locations.

Format:

1	2	3	4	5	6	7	8	9	10
DVGRID	DVID	GID	CID	COEFF	N1	N2	N3		

Example:

DVGRID	3	108	5	0.2	0.5	0.3	1.0		

Field	Contents
DVID	DESVAR entry identification number. (Integer > 0)
GID	Grid point (GRID) or geometric point (POINT) identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer $\geq$ 0; Default = 0)
COEFF	Multiplier of the vector defined by Ni. (Real; Default = 0.0)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni $\neq$ 0.0)

Remarks:

1. A CID of zero or blank (the default) references the basic coordinate system.
2. Multiple references to the same grid ID and design variable result in vectorial addition of the participation vectors defined by CID, COEFF, and Ni. There is no restriction on the number of DVGRID entries that may reference a given grid (GID) or design variable (DVID).
3. The coordinate update equation is given as

$$\{g\}_i - \{g\}_i^0 = \sum_i \text{COEFF}_j (X_{\text{DVIDj}} - X^0_{\text{DVIDj}}) \{N\}_j$$

where $\{g\}_i$ is the location of the *i-th* grid, $[g_x g_y g_z]^T$.

The vector $\{N\} = [N_x N_y N_z]^T$ is determined from CID and Ni. Note that it is a change in a design variable from its initial value X^0, and not the absolute value of the design variable itself, that represents a change in a grid point location, $\{g\}_i - \{g\}_i^0$.

4. The DVGRID entry defines the participation coefficients (basis vectors) of each design variable for each of the coordinates affected by the design process in the relationship

$$\{\Delta g\}_i = \sum_j \{T\}_{ij} \cdot \Delta X_j$$

5. DVGRID entries that reference grid points on MPCs or RSSCON entries produce incorrect sensitivities. Often the sensitivities are 0.0 which may result in a warning message indicating zero gradients which may be followed by UFM 6499. Other rigid elements produce correct results.

DVMREL1 Design Variable to Material Relation

Defines the relation between a material property and design variables.

Format:

1	2	3	4	5	6	7	8	9	10
DVMREL1	ID	TYPE	MID	MPNAME	MPMIN	MPMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	COEF3	-etc.-		

Example:

DVMREL1	5	MAT1	1	RHO	0.05	1.0			
	1	1.0							

Field	Contents
ID	Unique identification number. (Integer > 0)
TYPE	Name of a material property entry, such as “MAT1”, MAT2”, etc. (Character)
MID	Material Identification number. (Integer > 0)
MPNAME	Name of material property, such as “E” or “RHO”. (Character)
MPMIN	Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark **4.** (Real)
MPMAX	Maximum value allowed for this property. See Remark **4.** (Real; Default = 1.0E+20)
C0	Constant term of relation. (Real, Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation or keyword = “PVAL”. (If i = 1, Real or Character; if i > 1, Real)

Remarks:

1. The relationship between the material property and design variables is given by:

$$MP_i = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. The fifth field of the entry, MPNAME, only accepts string characters. It must be the same as the name that appears in the "**Bulk Data Entries**" on page 945 for various material properties. For example, if the isotropic material density is to be designed, RHO (case insensitive) must be specified on the MPNAME field.
4. The default value for MPMIN and MPMAX are not applied when the linear property is a function of a single design variable and C0=0.0. It is expected that the limits applied to the DESVAR entry will keep the designed property within reasonable bounds.
5. When the character input is used, only a single design variable can be referenced on a DVMREL1 entry and "PVAL" is specified on the COEF1 field. If a DVMREL1 entry references more than one design variable with the PVAL option, a user fatal error message will be issued.

DVMREL2 Design Variable to Material Relation

Defines the relation between a material property and design variables with a user-supplied equation.

Format:

1	2	3	4	5	6	7	8	9	10
DVMREL2	ID	TYPE	MID	MPNAME	MPMIN	MPMAX	EQID		
	DESVAR	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	DTABLE	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						

Example:

DVMREL2	5	MAT1	1	E	0.05	1.0	100		
	DESVAR	1	2						
	DTABLE	E0							

Field	Contents
ID	Unique identification number. (Integer > 0)
TYPE	Name of a material property entry, such as “MAT1”, MAT2”, etc. (Character)
MID	Material Identification number. (Integer > 0)
MPNAME	Name of material property, such as “E” or “RHO”. (Character)
MPMIN	Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-15. Otherwise, it is -1.0E35. (Real)
MPMAX	Maximum value allowed for this property. (Real; Default = 1.0E+20)
EQID	DEQATN entry identification number. (Integer > 0)
DESVAR	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)

Field	Contents
DTABLE	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE entry. (Character)

Remarks:

1. The variables identified by DVIDi and LABLi correspond to variable names (x1, x2, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x1 through xN (where $N = m + n$) are assigned in the order DVID1, DVID2, ..., DVIDm, LABL1, LABL2, ..., LABLn.
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. The fifth field of the entry, MPNAME, only accepts string characters. It must be the same as the name that appears in the “**Bulk Data Entries**” on page 945 for various material properties. For example, if the isotropic material density is to be designed, RHO (case insensitive) must be specified on the MPNAME field

DVPREL1 Design Variable to Property Relation

Defines the relation between an analysis model property and design variables.

Format:

1	2	3	4	5	6	7	8	9	10
DVPREL1	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	-etc.-			

Example:

DVPREL1	12	PBAR	612	6	0.2	3.0			
	4	0.25	20	20.0	5	0.3			

Field	Contents
ID	Unique identification number. (Integer > 0)
TYPE	Name of a property entry, such as “PBAR”, “PBEAM”, etc. (Character)
PID	Property entry identification number. (Integer > 0)
PNAME/FlD	Property name, such as “T”, “A”, or field position of the property entry, or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referred to by field position. (Character or Integer ≠ 0)
PMIN	Minimum value allowed for this property. If PMIN references a property that can only be positive, then the default value for PMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 6. (Real)
PMAX	Maximum value allowed for this property. (Real; Default = 1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation or keyword = “PVAL”. (If i = 1, Real or Character; if i > 1, Real)

Remarks:

1. The relationship between the analysis model property and design variables is given by:

$$P_j = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. TYPE="PBEND" is not supported. TYPE="PBARL" or "PBEAML" support only PNAME and not FID.
4. FID may be either a positive or a negative number. If FID > 0, it identifies the field position on a property entry. If FID < 0, it identifies the word position of an entry in the element property table. For example, to specify the area of a PBAR, either PNAME=A, FID=+4 or FID=-3 can be used. In general, use of PNAME is recommended.
5. Designing PBEAML or PBEAM requires specification of both property name and station. **Table 8-12** shows several examples.

Table 8-12

PTYPE	Property Name	END A	END B	i-th Station
PBEAML	DIM1	DIM1 or DIM1(A)	DIM1(B)	DIM1(i)
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM or PBEAML entry can be referenced by a DVPREL1. For example, referencing an END B property name on a DVPREL1 entry when the referenced PBEAM does not explicitly specify the END B station, is not allowed.

6. The default values of PMIN and PMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the DESVAR entry will keep the designed property within reasonable bounds.
7. When the character input is used, only a single design variable can be referenced on a DVPREL1 entry and "PVAL" is specified on the COEF1 field. If a DVPREL1 entry references more than one design variable with the PVAL option, a user fatal error message will be issued.

DVPREL2 Design Variable to Property Relation

Defines the relation between an analysis model property and design variables with a user-supplied equation.

Format:

1	2	3	4	5	6	7	8	9	10
DVPREL2	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	EQID		
	"DESVAR"	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	"DTABLE"	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						

Example:

DVPREL2	13	PBAR	712	5	0.2	4			
	DESVAR	4	11	13	5				
	DTABLE	PI	YM						

Field	Contents
ID	Unique identification number. (Integer > 0)
TYPE	Name of a property entry, such as PBAR, PBEAM, etc. (Character)
PID	Property entry identification number. (Integer > 0)
PNAME/FID	Property name, such as "T", "A", or field position of the property entry, or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referred to by field position. (Character or Integer ≠ 0)
PMIN	Minimum value allowed for this property. If FID references a stress recovery location field, then the default value for PMIN is -1.0+35. PMIN must be explicitly set to a negative number for properties that may be less than zero (for example, field ZO on the PCOMP entry). (Real; Default = 1.E-15)
PMAX	Maximum value allowed for this property. (Real; Default = 1.0E20)
EQID	DEQATN entry identification number. (Integer > 0)
"DESVAR"	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)

Field	Contents
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE entry. (Integer > 0)

Remarks:

1. The variables identified by DVIDi and LABLi correspond to variable names (x1, x2, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x1 through xN (where N = m+n) are assigned in the order DVID1, DVID2, ..., DVIDn, LABL1, LABL2, ..., LABLm.
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. FID may be either a positive or a negative number. If FID > 0, it identifies the field position on a property entry. If FID < 0, it identifies the word position of an entry in EPT. For example, to specify the area of a PBAR, either PNAME=A, FID = +4 or FID = -3 may be used. In general, use of PNAME is recommended.
4. TYPE = “PBEND” is not supported. TYPE=”PBARL” or “PBEAML” support only PNAME and not FID.
5. Designing PBEAM requires specification of both property name and station. **Table 8-13** shows one example.

Table 8-13

PTYPE	Property Name	END A	END B	i-th Station
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM entry can be referenced by a DVPREL2. For example, referencing an END B property name on a DVPREL2 entry when the referenced PBEAM does not explicitly specify the END B station, is not allowed.

DVSHAP Design Variable to Basis Vector(s)

Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.

Format:

1	2	3	4	5	6	7	8	9	10
DVSHAP	DVID	COL1	SF1	COL2	SF2	COL3	SF3		

Example:

DVSHAP	2	1	2.0	4	1.0				

Field	Contents
DVID	Design variable identification number on the DESVAR entry. (Integer > 0)
COLi	Column number of the displacement matrix. See Remark 2. (1 ≤ Integer ≤ maximum column number in the displacement matrix.)
SFi	Scaling factor applied to the COLi-th column of the displacement matrix. (Real; Default = 1.0)

Remarks:

1. DVID must be defined on a DESVAR entry.
2. COLi must be a valid column number in the displacement matrix.
3. Multiple references to the same DVID and/or COLi will result in a linear combination of displacement vectors. In the example above, the shape basis vector is a linear combination of the fourth column and twice the second column.
4. The displacement matrix must have been created by MD Nastran and be available on a database, which is attached via the DBLOCATE FMS statement shown below:

```
ASSIGN    DISPMAT=' physical filename of MASTER DBset '
DBLOCATE  DATABLK=(UG/UGD,GEOM1/GEOM1D,GEOM2/GEOM2D) ,
          LOGICAL=DISPMAT
```

DYCHANG (SOL 700)

For a SOL 700 restart analysis, change certain solution options.

Format BOUNDARY:

Defines an arbitrary number of entries giving the nodal ID and the additional translational displacement boundary condition code. Previous boundary condition codes will continue to be imposed, i.e., a fixed node cannot be freed with this option.

Format:

1	2	3	4	5	6	7	8	9	10
DYCHANG	ID	"BOUNDARY"			BCC				
	C1	C2	THRU	C3	BY	C4			

Example:

DYCHANG	101	BOUNDARY			12				
	101	102	THRU	112					

Field	Contents
ID	Identification number of the DYCHANG entry – Not presently used (Integer)
"BOUNDARY"	Change the displacement boundary condition (Character, Required)
BCC	New translational boundary condition: Component number of global coordinate (any unique combination of the digits 1 through 3 with no embedded blanks). Combinations are allowed, e.g., 12, 123. (Integer, Required)
Ci	Nodal point ID. THRU indicates a range of grid points. BY is the increment to be used within this range. (Integer, Required)

Alternate Formats and Examples:

Format RBCNSTR:

Allows translational and rotational boundary conditions on a rigid body to be changed. Also, see RBE2D.

Format:

1	2	3	4	5	6	7	8	9	10
DYCHANG	ID	"RBCNSTR"		PID	BCC				

Example:

DYCHANG	101	RBCNSTR		200	1245				

Field	Contents
ID	Identification number of the DYCHANG entry – Not presently used (Integer)
"RBCNSTR"	Change translational and rotational boundary conditions on a rigid body (Character, Required)
PID	Property ID of the rigid body. (Integer, Required)
BCC	Translational and rotational constraint: Component number of global coordinate (any unique combination of the digits 1 through 6 with no embedded blanks). Combinations are allowed, e.g., 12, 456. (Integer, Required)

Format TABLED1:

Allows a load curve to be redefined. *The new load curve must contain the same number of points as the curve it replaces.* The curve should be defined in a TABLED1. Any offsets and scale factors are ignored.

Format:

1	2	3	4	5	6	7	8	9	10
DYCHANG	ID	"TABLED1"		LCID					

Example:

DYCHANG	101	TABLED1		1000					

Field	Contents
ID	Identification number of the DYCHANG entry – Not presently used (Integer)
LCID	Load curve ID (Integer, Required)
“TABLED1”	Redefine a load curve (Character, Required)

Format VELND:

Allow the velocity of nodal points to be changed at restart. Undefined nodes will have their nodal velocities reset to zero if OPTION is blank. However, if OPTION=ONLY, then only the specified nodes will have their nodal velocities modified.

Format:

1	2	3	4	5	6	7	8	9	10
DYCHANG	ID	“VELND”	OPTION						
	VX	VY	VZ	VXR	VYR	VZR			
	C1	C2	THRU	C3	BY	C4			

Example:

DYCHANG	101	VELND	ONLY						
		50.							
	1101	1105	THRU	1290	BY	1			

Field	Contents
ID	Identification number of the DYCHANG entry – Not presently used (Integer)
“VELND”	Change the velocity of nodal points (Character, Required)
OPTION	BLANK Undefined nodes have their nodal velocities reset to zero ONLY Only the specified nodes have their nodal velocities modified (Character)
VX	Translational velocity in x-direction. . (Real, default = 0.0).
VY	Translational velocity in y-direction. . (Real, default = 0.0).

Field	Contents
VZ	Translational velocity in z-direction. . (Real, default = 0.0).
VXR	Rotational velocity about the x-axis. . (Real, default = 0.0).
VYR	Rotational velocity about the y-axis. . (Real, default = 0.0).
VZR	Rotational velocity about the z-axis. . (Real, default = 0.0).
Ci	Nodal point ID. THRU indicates a range of grid points. BY is the increment to be used within this range. (Integer, Required)

Remarks:

1. If both VELND and VELZERO options are defined then all velocities will be reset to zero.

Format VELZERO:

Resets the velocities to zero at the start of the restart. For this option, no any further input is necessary.

Format:

1	2	3	4	5	6	7	8	9	10
DYCHANG	ID	"VELZERO"							

Example:

DYCHANG	101	VELZERO							

Field	Contents
ID	Identification number of the DYCHANG entry – Not presently used (Integer)
"VELZERO"	Resets the velocities to zero at the start of the restart (Character, Required)

Format VELRB:

Allows the velocity components of a rigid body to be changed at restart.

Format:

1	2	3	4	5	6	7	8	9	10
DYCHANG	ID	“VELRB”		PID					
	VX	VY	VZ	VXR	VYR	VZR			

Example:

DYCHANG	101	VELRB		400					
			60.						

Field	Contents
ID	Identification number of the DYCHANG entry – Not presently used (Integer)
“VELRB”	Change the velocity components of a rigid body (Character, Required)
PID	Property ID of rigid body (Integer, Required)
VX	Translational velocity in x-direction. (Real, default = 0.0).
VY	Translational velocity in y-direction. (Real, default = 0.0).
VZ	Translational velocity in z-direction. (Real, default = 0.0).
VXR	Rotational velocity about the x-axis. (Real, default = 0.0).
VYR	Rotational velocity about the y-axis. (Real, default = 0.0).
VZR	Rotational velocity about the z-axis. (Real, default = 0.0).

Remarks:

1. Rotational velocities are defined about the center of mass of the rigid body.
2. Rigid bodies not defined in this section will not have their velocities modified.

DYDELEM (SOL 700)

Deletes properties or element using a list for SOL 700 restarts.

Format:

1	2	3	4	5	6	7	8	9	10
DYDELEM	ID	TYPE							
	C1	C2	THRU	C3	BY	C4			

Example 1:

DYDELEM	101	BEAM							
	1	2							

Example 2:

DYDELEM	102	SHELL							
	7103	7104	THRU	7166	BY	2			

Field	Contents
ID	Identification number of the DYDELEM card – Not presently used (Integer)
TYPE	BEAM, SHELL, SOLID, TSHELL, PROP (Character, Required)
Ci	Element ID or Property ID. THRU indicates a range of elements or properties. BY is the increment to be used within this range. (Integer, Required)

DYPARAM (SOL 700)

Bulk data parameters for Dytran-LsDyna with extra fields (SOL 700 only).

Format:

1	2	3	4	5	6	7	8	9	10
DYPARAM	F1	F2	F3	F4	F5	F6	F7	F8	

Example:

1	2	3	4	5	6	7	8	9	10
DYPARAM	LSDYNA	BINARY	D3PLOT	.002					

Field	Contents
Fi	Desired entry for field i in the Dytran-lsdyna file.

Remarks:

1. These entries are passed directly to Dytran-lsdyna with the first two characters (DY) removed. In the example the entry would be

   PARAM,LSDYNA,BINARY,.002, ,2,0.01

2. For the MD Nastran release the following parameters are useful:

   DYPARAM,LSDYNA,BINARY

   DYPARAM*,LSDYNA,DB-EXTENT

   DYPARAM,LSDYNA,OUTPUT

   DYPARAM,ELDLTH

   DYPARAM,LSDYNA,RELAX

   DYPARAM*,LSDYNA,HOURGLASS

3. Details of the Dytran-lsdyna parameters can be found in the *MD Nastran SOL 700 User's Guide.*

DYRELAX (SOL 700)

Define controls for dynamic relaxation for restart runs.

Format:

1	2	3	4	5	6	7	8	9	10
DYRELAX	ID	NRCYCK	DRTOL	DRFCTR	DRTERM	TSSFDR	IRELAL	EDTTL	
	IDRFLG								

Example:

DYRELAX	102	1000	0.002	0.6					

Field	Contents
ID	Identification number of the DYRELAX entry – Not presently used (Integer)
NRCYCK	Number of iterations between convergence checks, for dynamic relaxation option (Integer, default = 250).
DRTOL	Convergence tolerance for dynamic relaxation option (Real, default = 0.001).
DRFCTR	Dynamic relaxation factor (Real, default = .995).
DRTERM	Optional termination time for dynamic relaxation. Termination occurs at this time or when convergence is attained (Real, default = infinity).
TSSFDR	Scale factor for computed time step during dynamic relaxation. If zero, the value is set to TSSFAC. After converging, the scale factor is reset to TSSFAC. (Real, default =0.0)
IRELAL	Automatic control for dynamic relaxation option based on algorithm of Papadrakakis [Papadrakakis 1981]. (Integer, default = 0).
EDTTL	Convergence tolerance on automatic control of dynamic relaxation. (Real, default = 0.0).
IDRFLG	Dynamic relaxation flag for stress initialization: EQ.0: not active, EQ.1: dynamic relaxation is activated. (Integer, default = 0).

Remarks:

1. If a dynamic relaxation relaxation analysis is being restarted at a point before convergence was obtained, then NRCYCK, DRTOL, DRFCTR, DRTERM and TSSFDR will default to their previous values, and IDRFLG will be set to 1.
2. If dynamic relaxation is activated after a restart from a normal transient analysis MD Nastran SOL 700 continues the output of data as it would without the dynamic relaxation being active. This is unlike the dynamic relaxation phase at the beginning of the calculation when a separate database is not used. Only load curves that are flagged for dynamic relaxation are applied after restarting.

DYRIGSW (SOL 700)

Defines materials to be switched from rigid to deformable and deformable to rigid in a restart. It is only possible to switch materials on a restart if part switching was activated in the time zero analysis. See **D2R0000** for details of part switching.

Format:

1	2	3	4	5	6	7	8	9	10
DYRIGSW	ID	TYPE							
	PID	MRB							

Example 1:

DYRIGSW	101	D2R							
	1	101							

DYRIGSW	102	R2D							
	200								

Field	Contents
ID	Identification number of the DYRIGSW entry – Not presently used (Integer)
TYPE	D2R Deformable to rigid part switch R2D Rigid to deformable part switch (Character, required)
PID	Property ID of the part which is switched to a rigid material for D2R. Property ID of the part which is switched to a deformable material for R2D. (Integer, required)
MRB	Property ID of the master rigid body to which the part is merged. If zero, the part becomes either an independent or master rigid body. Only used for D2R. (Integer, default = 0)

DYTERMT (SOL 700)

Stop a SOL 700 analysis depending on specified displacement conditions.

Caution: The inputs are different for the nodal and rigid body stop conditions. The nodal stop condition works on the global coordinate position, while the body stop condition works on the relative global translation. The number of termination conditions cannot exceed the maximum of 10 or the number specified in the original analysis.

The analysis terminates for NODE when the current position of the node specified reaches either the maximum or minimum value (stops 1, 2 or 3), or picks up force from any contact surface (stop 4). For BODY the analysis terminates when the center of mass displacement of the rigid body specified reaches either the maximum or minimum value (stops 1, 2 or 3) or the displacement magnitude of the center of mass is exceeded (stop 4). If more than one condition is input, the analysis stops when any of the conditions is satisfied. ***This input completely overrides the existing termination conditions defined in the time zero run.***

Termination by other means is controlled by the RESTART entry.

Format:

1	2	3	4	5	6	7	8	9	10
DYTERMT	ID	TYPE							
	NID / PID	STOP	MAXC	MINC					

Example 1:

DYTERMT	102	NODE							
	1	4							

DYTERMT	102	BODY							
	200	1	0.45						

Field	Contents
ID	Identification number of the DYTERMT entry – Not presently used (Integer)
TYPE	NODE Stop the job depending on nodal conditions BODY Stop the job depending on rigid body conditions (Character, Required)
NID / PID	Node ID for NODE; Property ID of rigid body for BODY (Integer, Required)
STOP	Stop criterion: (Integer, Required) EQ.1: global x direction, EQ.2: global y direction, EQ.3: global z direction, EQ.4: stop if node touches contact surface. (for NODE) stop if displacement magnitude is exceeded. (for BODY)
MAXC	Maximum (most positive) coordinate, options 1, 2 and 3 above only. (for NODE) Maximum (most positive) displacement, options 1, 2, 3 and 4: EQ:0.0. MAXC set to 1.0e21 (for BODY)
MINC	Minimum (most negative) coordinate, options 1, 2 and 3 above only. EQ:0.0. MINC set to -1.0e21 (for BODY only)

DYTIMHS (SOL 700) Choose Types and Form of SOL 700 Output

Specifies various types of time history output and form of the output for SOL 700. (Gridpoint and Element data will be output on a binary jid.dytr.d3tht file, while the additional time history data will be output to either jid.dytr.binout (KIND=BINARY) or jid.dytr.dbout (KIND=ASCII) file(s). The data is written at the intervals specified by DTOUT on this entry.) If different KIND or DTOUT values are needed, enter the DYTIMHS entry as many times as necessary. If they are the same, enter the main DYTIMHS entry once and use continuation entries to define all requested output. When specifying multiple DYTIMHS cards for gridpoints and elements, the smallest DTOUT will be used for those TYPEs. This because both these TYPEs are written to the same jid.dytr.d3tht file..

Format:

1	2	3	4	5	6	7	8	9	10
DYTIMHS	KIND	DTOUT							
	TYPE	I1	I2	I3	I4	I5	I6	I7	
		I8	I9						
	TYPE	TYPE	TYPE	TYPE	TYPE	I1	I2	TYPE	

Example:

DYTIMHS	ASCII	1.OE-6							
	GRID	100	101	200	210	250	260	300	
		400	500	550					
	ELEM	1100	1101	2200	2210	2250	2260	4300	
		4400	6500	6550					
	ABSTAT	GLSTAT	MATSUM	RCFORC	RWFORC	SPCFORC	SWFORC		

Alternate Format:

1	2	3	4	5	6	7	8	9	10
DYTIMHS	KIND	DTOUT							
	TYPE	I1	To	I2	By	I3			
	TYPE	I1	To	I2	By	I3			
	TYPE	TYPE	TYPE	TYPE	TYPE	TYPE	TYPE	TYPE	

Example:

DYTIMHS	BINARY	1.0E-4							
	GRID	100	To	110	By	2			
	GRID	1000	To	2000	By	200			
	ELEM	2000							
	ABSTAT	GLSTAT	MATSUM	RCFORC	RWFORC	SPCFORC	SWFORC		

Field	Contents
KIND	Type of the file to generate options are: BINARY ASCII BOTH (Character, Default = Binary)
DTOUT	Output time interval (seconds). (Real, no Default)
TYPE	A character string designating the output types. Select from: GRID (Gridpoint data is written to jid.dytr.d3tht) ELEM (ELement data is written to jid.dytr.d3tht) ABSTAT (Airbag statistics) Volume Pressure Internal energy Input mass flow rate Output mas flow rate MassTemperature Density

Field	Contents	
	GLSTAT	(Global data)
		Kinetic energy Internal energy Total energy Ratio Stonewall-energy Spring & damper energy Hourglass energy Damping energy Sliding interface energy External work x,y,z velocity Time step Element id controlling the time step
	MATSUM	(Material energies)
		Kinetic energy Internal energy Hourglass energy x,y,z momentum x,y,z rigid body velocity Total kinetic energy Total internal energy Total hourglass energy
	RCFORC	(Resultant interface forces)
		x,y,z force
	RWFORC	(Wall forces)
		Normal x,x,z force
	SPCFORC	(SPC reaction forces)
		x,y,z force x,y,z moment
	SWFORC	(Nodal constriant reaction forces - sportwelds & rivets)

Field	Contents
	Axial force Shear force
I1, I2, I3, ...	Element number or grid ID for the requested items. These are only specified for TYPE=GRID and TYPE=ELEMENT. (Integer, no Default)

ECHOOFF Deactivate Printed Echo

Marks the point or points in the input file to deactivate printed echo of the input file.

Format:

1	2	3	4	5	6	7	8	9	10
ECHOOFF									

Example:

ECHOOFF									

Remarks:

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in the Case Control command, “**ECHO**” on page 278.
2. The companion to this entry is the ECHOON entry.

ECHOON Activate Printed Echo

Marks the point or points in the input file to activate printed echo of the input file.

Format:

1	2	3	4	5	6	7	8	9	10
ECHOON									

Example:

ECHOON									

Remarks:

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in the Case Control command, “**ECHO**” on page 278.
2. The companion to this entry is the ECHOOFF entry.

EIGB Buckling Analysis Set

Defines data needed to perform buckling analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGB	SID	METHOD	L1	L2	NEP	NDP	NDN		
	NORM	G	C						

Example:

EIGB	13	INV	0.1	2.5	2	1	1		
	MAX								

Field	Contents
SID	Set identification number. (Unique Integer > 0)
METHOD	Method of eigenvalue extraction. (Character: “INV” for inverse power method or “SINV” for enhanced inverse power method.)
L1, L2	Eigenvalue range of interest. (Real, L1 < L2)
NEP	Estimate of number of roots in positive range not used for METHOD = “SINV”. (Integer > 0)
NDP, NDN	Desired number of positive and negative roots. (Integer>0; Default = 3*NEP)
NORM	Method for normalizing eigenvectors. (Character: “MAX” or “POINT”; Default = “MAX”) MAX - Normalize eigenvectors to the unit value of the largest component in the analysis set. (Default). POINT - Normalize eigenvectors to the unit value of the component defined in G and C fields. The value for NORM defaults to MAX if the defined component is zero.
G	Grid or scalar point identification number. Required only if NORM=“POINT”. (Integer > 0)
C	Component number. Required only if NORM=“POINT” and G is a geometric grid point. ($1 \leq$ Integer ≤ 6)

Remarks:

1. The EIGB entry must be selected with the Case Control command METHOD = SID.
2. Each eigenvalue is the factor by which the prebuckling state of stress is multiplied to produce buckling in the shape defined by the corresponding eigenvector.
3. The continuation entry is optional. If the continuation is not specified, than NORM = “MAX” normalization is performed.
4. See *The Nastran Theoretical Manual*, Sections 10.3.6 and 10.4.2.2, for a discussion of convergence criteria.
5. If NORM = “MAX”, components that are not in the analysis set may have values larger than unity.
6. The SINV method is an enhanced version of the INV method. It uses Sturm sequence techniques to ensure that all roots in the range have been found. It is generally more reliable and more efficient than the INV method.
7. Convergence is achieved at 10^{-6}.
8. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the F1 and F2 fields are left blank (for EIGRL, L1 and L2 on EIGB) an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If ND is set to 1 (on EIGRL; NDN and NDP on EIGB) there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.

EIGC Complex Eigenvalue Extraction Data

Defines data needed to perform complex eigenvalue analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD	NORM	G	C	E	ND0		

The following continuation is repeated for each desired search region. (J = 1 to n, where n is the number of search regions.

	ALPHAAJ	OMEGAAJ	ALPHABJ	OMEGABJ	LJ	NEJ	NDJ		

Alternate Format for Continuation Entry for Block Complex Lanczos:

	ALPHAAJ	OMEGAAJ	MBLKSZ	IBLKSZ	KSTEPS		NJi		

Examples:

EIGC	14	CLAN							
		+5.6					4		
		-5.5					3		

EIGC	15	HESS					6		

Field	Contents	
SID	Set identification number. (Unique Integer > 0)	
METHOD	Method of complex eigenvalue extraction. (Character: "INV," "HESS," or "CLAN")	
	INV	Inverse power.
	HESS	Upper Hessenberg. See Remarks **2.** and **3.**
	CLAN	Complex Lanczos. See Remark **9.**
NORM	Method for normalizing eigenvectors. See Remark **8.** (Character: "MAX" or "POINT"; Default = "MAX")	
	MAX	Normalize the component having the largest magnitude to a unit value for the real part and a zero value for the imaginary part.

Field	Contents	
	POINT	Normalize the component defined in fields 5 and 6 to a unit value for the real part and a zero value for the imaginary part. The value for NORM defaults to "MAX" if the magnitude of the defined component is zero.
G	Grid or scalar point identification number. Required if and only if NORM = "POINT". (Integer > 0)	
C	Component number. Required if and only if NORM="POINT" and G is a geometric grid point. ($0 \le$ Integer ≤ 6)	
E	Convergence criterion. (Real $\ge$ 0.0. Default values are: 10^{-4} for METHOD = "INV", 10^{-15} for METHOD = "HESS", E is machine dependent for METHOD = "CLAN".)	
MBLKSZ	Maximum block size. See Remark **12.** (Default = 7, Real $\ge$ 0.0)	
IBLKSZ	Initial block size. See Remark **12.** (Default = 2, Real $\ge$ 0.0)	
KSTEPS	Frequency of solve. (Default = 5, Integer > 0)	

Table 8-14 Relationship Between METHOD Field and Other Fields

Field	METHOD Field		
	HESS	INV	CLAN
NDj (Integer > 0)	Desired number of eigenvalues and eigenvectors. (No default)	Desired number of roots and eigenvectors in j-th search region. (Default = 3*NEj)	Desired number of roots and eigenvectors to be extracted at j-th shift point. (No default)
ALPHAAj OMEGAAj Real and imaginary parts of Aj in radians per unit time. (Real).	Not used	End point Aj of j-th search region in complex plane. (Default = 0.0)	j-th shift point. (Default = 0.0)

Table 8-14 Relationship Between METHOD Field and Other Fields

Field	METHOD Field		
	HESS	INV	CLAN
ALPHABj OMEGABj Real and imaginary parts of Bj in radians per unit time. (Real).	Not used	End point Bj of j-th search region in complex plane. (Default = 0.0)	See alternate definition below.
Lj (Real > 0.0)	Not used	Width of j-th search region. (Default = 1.0)	See alternate definition below.
NEj (Integer > 0)	Not used	Estimated number of roots in j-th search region.	Not used
MBLKSZ For block CLAN only			Maximum Block Size Default = 7
IBLCKSZ For block CLAN only			Initial Block Size Default = 2

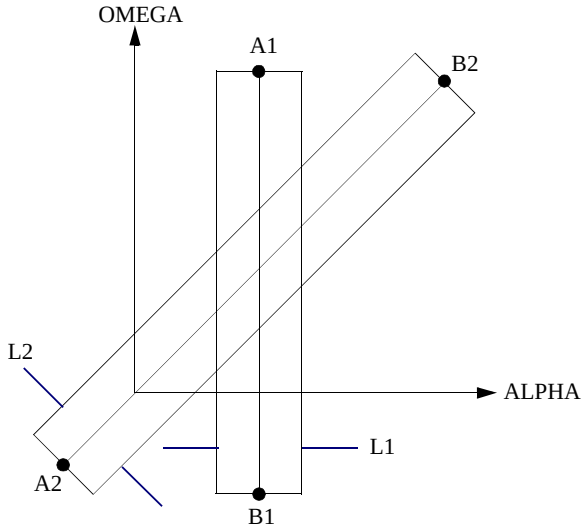


Figure 8-83 Sample Search Regions

Remarks:

1. The EIGC entry must be selected in the Case Control Section with the command CMETHOD = SID. Methods of solution are also controlled by SYSTEM(108) on the NASTRAN statements (described in "**nastran Command and NASTRAN Statement**" on page 1).

SYSTEM(108)	Specification
0 (default)	QZ HESS method or CLAN block Lanczos, as selected by EIGC entry or equivalent PARAM input. Default value.
1	Force Householder QR (Hessenberg with spill, mass matrix must be nonsingular)
2	Force old single vector complex Lanczos
4	Force new block complex Lanczos
8	Debugging output for Lanczos methods
16	Turn off block reduction in block complex Lanczos
32	Turn off block augmentation in block complex Lanczos
64	Turn of full orthogonality in block complex Lanczos
128	Turn off preprocessing of initial vectors in block complex Lanczos

SYSTEM(108)	Specification
256	Force LR method (Hessenberg, no spill, mass matrix must be invertible)
512	Force QZ method
65536	Use semi-algebraic sort on imaginary part of roots (pre-V70.6 sort)

The word “force” above implies that the method selected by the system cell will be used even when another method has been selected on an EIGC entry. Sums of these values will produce two or more actions at once, when feasible. As the numbers get larger, the function is more developer-oriented than user-oriented.

2. When using METHOD = "HESS", the following should be noted:

   The "HESS" method is generally more reliable and economical for small and moderate-size problems. It computes ND eigenvalues and eigenvectors.

3. The EIGC entry may or may not require continuations as noted below.

   - For the "HESS" method, continuations are not required; and their contents are ignored when present, except for ND1. However, it is recommended that continuations are not used.
   - For the "CLAN" method when the continuation entry is not used a shift is calculated automatically. When a shift is input on the first continuation entry it is used as the initial shift. Only one shift is used. Data on other continuation entries is ignored.
   - For METHOD = "INV", each continuation defines a rectangular search region. Any number of regions may be used and they may overlap. Roots in overlapping regions will not be extracted more than once.
   - For all methods, if no continuation is present, then ND0 must be specified on the first entry. If a continuation is present, then NDj must be specified on the continuation and not on the first entry.

4. The units of ALPHAAJ, OMEGAAJ, ALPHABJ, and OMEGABJ are radians per unit time.
5. See *The NASTRAN Theoretical Manual*, Sections 10.4.4.5 and 10.4.4.6, for a discussion of convergence criteria and the search procedure with the INV method.

6. DIAG 12 prints diagnostics for the inverse power method, the complex Lanczos method and the QZ HESS method.
7. If METHOD = "HESS" and the LR or QR methods (non-default methods) are selected by system cell 108 the mass matrix must be nonsingular. The default QA method does not require a nonsingular mass matrix.
8. The normalized eigenvectors may be output with the SDISPLACEMENT and / or DISPLACEMENT Case Control commands.
9. When using METHOD = CLAN, the following should be noted. The modern CLAN method (default for METHOD entry of CLAN) has been enhanced to include a block complex Lanczos approach. This method is more reliable and will not accept inaccurate roots which the old method had a tendency to do. Thus, given the same input, the new method may often accept fewer roots. For continuity the old method has been maintained and may be selected by setting SYSTEM(108).
10. The SVD method is provided for DMAP applications. If used in solution 107 or 110, and mass or damping terms are present, a user fatal exit is taken. See the MSC Web site for the Flight Loads Product examples on the use of the SVD method. The SVD operation decomposes the input stiffness matrix K into the factors U, S, and V. U and V are collections of vectors of orthogonal functions. S is a rectangular matrix with terms on the diagonal of its left partition. The factors satisfy the equation K = U*S*V', where " ' " implies complex conjugate transpose. The ND1 value has a meaning for the SVD functions which differs from eigensolution.

ND1	OUTPUT
>0	All vectors of U and V are output.
=0	U and V are returned in a purged state.
<0	S is returned as a square matrix whose number of columns is equal to the minimum number of rows or columns of the input matrix. U and V are truncated to be commensurate with S. This is a method to reduce the costs of solving very rectangular input matrices by providing a partial solution for the most interesting vectors.

11. For DMAP applications there are input parameters, not present in the solution sequences, that may be used to replace the function of the EIGC and CMETHOD entries. See the MSC.Software Web site for details.

12. The MBLKSZ and IBKLSZ parameters are integers in concept, but must be input at real numbers (that is, with a decimal point.) They represent maximum sizes, and may be reduced internally for small size problems.

EIGP Poles in Complex Plane

Defines poles that are used in complex eigenvalue extraction by the Determinant method.

Format:

1	2	3	4	5	6	7	8	9	10
EIGP	SID	ALPHA1	OMEGA1	M1	ALPHA2	OMEGA2	M2		

Example:

EIGP	15	-5.2	0.0	2	6.3	5.5	3		

Field	Contents
SID	Set identification number. (Integer > 0)
ALPHAi,OMEGAi	Coordinates of point in complex plane. (Real)
Mi	Multiplicity of complex root at pole defined by point at ALPHAi and OMEGAi. (Integer > 0)

Remarks:

1. The EIGP entry defines poles in the complex plane that are used with an associated EIGC entry having the same set number.
2. The units of ALPHAi and OMEGAi are radians per unit time.
3. Poles are used only in the determinant method. (METHOD = “DET” on the EIGC entry).
4. One or two poles may be defined on a single entry.
5. See *The NASTRAN Theoretical Manual,* Section 10.3.4, for details.

EIGR Real Eigenvalue Extraction Data

Defines data needed to perform real eigenvalue analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	F1	F2	NE	ND			
	NORM	G	C						

Example:

EIGR	13	LAN				12			

Field	Contents
SID	Set identification number. (Unique Integer > 0)
METHOD	Method of eigenvalue extraction. (Character)
	Modern Methods:
	LAN Lanczos Method
	AHOU Automatic selection of HOU or MHOU method. See Remark 13.
	Obsolete Methods:
	INV Inverse Power method.
	SINV Inverse Power method with enhancements.
	GIV Givens method of tridiagonalization.
	MGIV Modified Givens method.
	HOU Householder method of tridiagonalization.
	MHOU Modified Householder method.
	AGIV Automatic selection of METHOD = “GIV” or “MGIV”. See Remark 13.
NORM	Method for normalizing eigenvectors. (Character: “MASS,” “MAX,” or “POINT”; Default = “MASS”)
	MASS Normalize to unit value of the generalized mass. (Default)
	MAX Normalize to unit value of the largest component in the analysis set.

Field	Contents	
	POINT	Normalize to a positive or negative unit value of the component defined in fields 3 and 4. The POINT option is not supported for METH=LAN. (Defaults to “MASS” if defined component is zero.)
G	Grid or scalar point identification number. Required only if NORM = “POINT”. (Integer > 0)	
C	Component number. Required only if NORM = “POINT” and G is a geometric grid point. ($1 \leq$ Integer ≤ 6)	

Table 8-15 Relationship Between METHOD Field and Other Fields for Obsolete Methods

Field	METHOD Field	
	INV or SINV	GIV, MGIV, HOU, or MHOU
F1, F2	Frequency range of interest. F1 must be input. If METHOD = “SINV” and ND, is blank, then F2 must be input. See also Remark 21. (Real $\geq$ 0.0)	Frequency range of interest. If ND is not blank, F1 and F2 are ignored. If ND is blank, eigenvectors are found with natural frequencies that lie in the range between F1 and F2. (Real $\geq$ 0.0; F1 < F2)
NE	Estimate of number of roots in range (Required for METHOD = “INV”). Not used by “SINV” method. (Integer > 0)	Not used.
ND	Desired number of roots. If this field is blank and METHOD = “SINV”, then all roots between F1 and F2 are searched and the limit is 600 roots. (Integer > 0, Default is $3 \cdot$ NE for METHOD = “INV” only.)	Desired number of eigenvectors. If ND is zero, the number of eigenvectors is determined from F1 and F2. If all three are blank, then ND is automatically set to one more than the number of degrees-of-freedom listed on SUPORTi entries. (Integer $\geq$ 0; Default = 0)

Remarks:

1. The EIGR entry must be selected with the Case Control command METHOD = SID.
2. See "**Real Eigenvalue Analysis**" on page 37 of the *MSC.Nastran 2006 Basic Dynamics* for a discussion of method selection.
3. The units of F1 and F2 are cycles per unit time.
4. The continuation entry is optional. If the continuation entry is not specified, then mass normalization is performed.
5. The contemporary methods are LAN and AHOU. The other methods are in a maintenance-only status, with no enhancements planned for them. They may be eliminated in a future release of MD Nastran.
6. The LAN method is the most general-purpose method, and may be used on both small- and large-size problems. It takes advantage of sparsity of input matrices, leading to greater efficiency on large-size problems. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20 degrees-of-freedom when the LAN method is selected, the method is switched to AHOU. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry. The NE, G, and C fields are ignored for the LAN method. The NORM field may be set to MASS (the default value) or NORM. The conventions used when both the Fi and ND fields are specified are described in Table 1 of the EIGRL entry description. The EIGRL entry is an alternate method to select the LAN method. It has several other input options for special cases. When both and EIGRL and EIGR have the same SID and that SID is selected by a METHOD command the EIGRL entry takes precedence.
7. The AHOU method is competitive with the LAN method when there are small, dense matrices and many eigenvectors are required. This most commonly occurs when static or dynamic reduction is performed. The AHOU method does not take advantage of matrix sparsity, so that computation cost rises with the cube of the number of DOFs. The AHOU method responds to all permitted values for all the other fields except NE, which is ignored.
8. All methods require a positive semi-definite (psd) mass matrix for stable solutions. The mass matrix may be tested for this condition for all methods of solution by setting SYSTEM(303). A value of "-4" should be sufficient to identify problem matrices. A fatal error exit is taken when it is not met. All MD Nastran metric elements are designed to produce psd mass matrices.

CMASSi elements, DMIG matrices selected by the M2GG command, and matrices input via INPUTT4 are special methods that allow addition of non-psd terms by use of non-metric element input. If none of this type of special input is present and the fatal error exit is taken you may have encountered an error in a metric element. Contact your local MSC technical representative for corrective action in this case.

9. The LAN and AHOU methods allow singular but positive semi-definite mass matrices.
10. The tridiagonal methods include the xGIV and xHOU methods, where "x" is described in the following comments. All tridiagonal methods compute all eigenvalues, and the number of eigenvectors specified by the Fi and Nd fields, as described in Table 13.
11. If "x" is blank (for example, the HOU method is selected) the mass matrix must be non-singular.
12. If "x" is M (for example, the MHOU method is selected) the mass matrix may be singular. A modified, shifted problem is solved in an inverse basis with this method. Some precision in the solution and longer computation time is exchanged for a more stable solution.
13. If "x" is A (for example, the AHOU method is selected) an attempt is made to solve the problem without shifting, in the interest of cost reduction and maximum precision. If the mass matrix is determined to be poorly conditioned for inversion the basis is automatically shifted with the modified method.
14. If NORM = "MAX", components that are not in the analysis set may have values larger than unity.
15. If NORM = "POINT", the selected component should be in the analysis set (a-set). (The program uses NORM = "MAX" when it is not in the analysis set.) The displacement value at the selected component will be positive or negative unity.
16. The "SINV" method is an enhanced version of the "INV" method. It uses Sturm sequence number techniques to make it more likely that all roots in the range have been found. It is generally more reliable and more efficient than the "INV" method.
17. For the "INV" and "SINV" methods, convergence is achieved at 10^{-6}. Convergence is tested by other criteria for the other methods.

18. For the “SINV” method only, if F2 is blank, the first shift will be made at F1, and only one eigensolution above F1 will be calculated. If there are no modes below F1, it is likely that the first mode will be calculated. If there are modes below F1 (including rigid body modes defined by SUPORT entries), a mode higher than the first mode above F1 may be calculated.
19. When F1, F2, and ND are all zero or blank, ND is reset to 1. A User Warning Message is produced for this condition, which is interpreted as likely to be due to an inadvertent omission by the user.
20. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the F1 and F2 fields are left blank (for EIGRL, L1 and L2 on EIGB) an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If ND is set to 1 (on EIGRL; NDN and NDP on EIGB) there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
21. F2 must be specified if DOMAINSOLVER ACMS or DOMAINSOLVER MODES is also specified in the Executive Control Section.

EIGRL Real Eigenvalue Extraction Data, Lanczos Method

Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

Format:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND	MSGLVL	MAXSET	SHFSCL	NORM	
	option_1 = value_1 option_2 = value_2, etc.								

Example:

EIGRL	1	0.1	3.2	10					
	NORM=MAX NUMS=2								

Field	Contents
SID	Set identification number. (Unique Integer > 0)
V1, V2	For vibration analysis: frequency range of interest. For buckling analysis: eigenvalue range of interest. See Remark **4.** (Real or blank, $-5 \times 10^{16} \le V1 < V2 \le 5. \times 10^{16}$)
ND	Number of roots desired. See Remark **4.** (Integer > 0 or blank)
MSGLVL	Diagnostic level. ($0 \le$ Integer ≤ 4; Default = 0)
MAXSET	Number of vectors in block or set. Default is machine dependent. See Remark **14.**
SHFSCL	Estimate of the first flexible mode natural frequency. See Remark **10.** (Real or blank)
NORM	Method for normalizing eigenvectors (Character: "MASS" or "MAX")
	MASS Normalize to unit value of the generalized mass. Not available for buckling analysis. (Default for normal modes analysis.)
	MAX Normalize to unit value of the largest displacement in the analysis set. Displacements not in the analysis set may be larger than unity. (Default for buckling analysis.)

Field	Contents
ALPH	Specifies a constant for the calculation of frequencies (Fi) at the upper boundary segments for the parallel method based on the following formula. See Remark 13. (Integer > 0.0; Default = 1.0): $$Fi = (V2 - V1)\frac{1-\text{ALPH}^{i}}{1. - \text{ALPH}^{\text{NUMS}}}$$
NUMS	Number of frequency segments for the parallel method. (Integer > 0; Default = 1)
Fi	Frequency at the upper boundary of the i-th segment. See Remark 13. (Real or blank; $V1 < F1 < F2 < \ldots F15 < V2$)
option_i= value_i	Assigns a value to the fields above except for SID. ALPH, NUMS, and Fi must be specified in this format. V1, V2, ND, MSGLVL, MAXSET, SHFSCL, and NORM may be specified in this format as long as their corresponding field is blank in the parent entry.

Remarks:

1. Real eigenvalue extraction data sets must be selected with the Case Control command METHOD = SID.
2. The units of V1 and V2 are cycles per unit time in vibration analysis, and are eigenvalues in buckling analysis. Each eigenvalue is the factor by which the prebuckling state of stress is multiplied to produce buckling in the shape defined by the corresponding eigenvector.
3. NORM = “MASS” is ignored in buckling analysis and NORM = “MAX” will be applied.
4. The roots are found in order of increasing magnitude; that is, those closest to zero are found first. The number and type of roots to be found can be determined from Table 8-16.

Table 8-16 Number and Type of Roots Found with EIGRL Entry

V1	V2	ND	Number and Type of Roots Found
V1	V2	ND	Lowest ND or all in range, whichever is smaller.
V1	V2	blank	All in range
V1	blank	ND	Lowest ND in range [V1,+∞]
V1	blank	blank	Lowest root in range [V1,+∞]

Table 8-16 Number and Type of Roots Found with EIGRL Entry (continued)

V1	V2	ND	Number and Type of Roots Found
blank	blank	ND	Lowest ND roots in [-∞,+∞]
blank	blank	blank	Lowest root. See Remark 11.
blank	V2	ND	Lowest ND roots below V2
blank	V2	blank	All below V2

5. In vibration analysis, if V1 < 0.0, the negative eigenvalue range will be searched. (Eigenvalues are proportional to Vi squared; therefore, the negative sign would be lost.) This is a means for diagnosing improbable models. In buckling analysis, negative V1 and/or V2 require no special logic.
6. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.
7. MSGLVL controls the amount of diagnostic output during the eigenvalue extraction process. The default value of zero suppresses all diagnostic output. A value of one prints eigenvalues accepted at each shift. Higher values result in increasing levels of diagnostic output.
8. MAXSET is used to limit the maximum block size. It is otherwise set by the region size or by ND with a maximum size of 15. It may also be reset if there is insufficient memory available. The default value is recommended.
9. In vibration analysis, if V1 is blank, all roots less than zero are calculated. Small negative roots are usually computational zeroes which indicate rigid body modes. Finite negative roots are an indication of modeling problems. If V1 is set to zero, negative eigenvalues are not calculated.
10. A specification for SHFSCL may improve performance, especially when large mass techniques are used in enforced motion analysis. Large mass techniques can cause a large gap between the rigid body and flexible frequencies. If this field is blank, a value for SHFSCL is estimated automatically.
11. On occasion, it may be necessary to compute more roots than requested to ensure that all roots in the range have been found. However, this method will not output the additional roots.

12. NASTRAN SYSTEM(146) provides options for I/O in sparse method only:

Table 8-17 SYSTEM(146) Options

SYSTEM(146)	Description
2	Increase memory reserved for sparse method by approximately 100%.
3	Increase memory reserved for sparse method by approximately 300%.
4	Increase memory reserved for sparse method by approximately 400%.

13. For the distributed parallel method, the frequency range between V1 and V2 may be subdivided into segments that can then be analyzed in parallel. V1 and V2 must be specified for the parallel method. NUMS must be specified greater than 1 to take advantage of the parallel method. NUMS may also be specified on the NUMSEG keyword of the NASTRAN statement. Currently, NUMSEG must equal the number of processors and by default NUMSEG is set to the number of processors requested by the DMP keyword. If both are specified, then NUMS takes precedence.

    The upper frequencies of each segment may be generated automatically by ALPH or specified directly in Fi. If both are specified, then Fi takes precedence over ALPH as long as they are consistent. ALPH if multiplied by 100 may also be specified on FRQSEQ keyword of the NASTRAN statement.

14. Increasing MAXSET may improve performance for large problems where a large number of eigenvalues are being found. The default is 7 on all machines except CRAY which is 12. SYSTEM(263) may be set in an rcfile to effectively modify the default; however the setting on the EIGRL entry always takes precedence.

15. SYSTEM(196), keyword SCRSAVE, controls reuse of scratch files when segment logic is invoked. SYSTEM(196) is useful only when multiple frequency segments are requested on a Lanczos run. (Multiple frequency segments can be requested via the NUMS field in the EIGRL entry and by SYSTEM(197).) Each frequency segment requires a minimum of three scratch files. When multiple frequency segments are used on a single processor computer then each frequency segment is solved serially. In this case, it makes sense to let segment #2 use the scratch files which were used by segment #1 since work for segment #1 has been completed (otherwise it

wouldn't be working on #2). Similarly, when work for segment #2 is finished, segment #3 should be able to use #2's scratch files. SYSTEM(196)=1 allows such file reuse and is considered a safe default on Version 70 and later systems.

16. The new buckling shift logic in Version 70.5 tends to shift to 1.0 first. The logic may have difficulty finding the lowest ND roots if a problem requests a small number of roots (ND) when there are thousands of roots below 1. In this case either the loading should be scaled, SHFSCL specified, or a smaller frequency range requested.
17. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20 degrees-of-freedom when the LAN method is selected, the method is switched to AHOU. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry.
18. V2 must be specified if DOMAINSOLVER ACMS or DOMAINSOLVER MODES is also specified in the Executive Control Section.

ELIST Element List

Defines a list of CQUAD4 and CTRIA3 structureal elements for virtual fluid mass.

Format:

1	2	3	4	5	6	7	8	9	10
ELIST	LID	E1	E2	E3	E4	E5	E6	E7	
	E8	E9	E10	-etc.-					

Example:

ELIST	3	51	-62	68	THRU	102	122		

Field	Contents
LID	Identification number of list. (Integer > 0)
Ei	Identification number of a structural element. See Remark **1.** for the meaning of the negative sign. The string “THRU” may be used to indicate that all existing elements between those referenced in the preceding and succeeding fields are in the list. (Integer ≠ 0 or “THRU”)

Remarks:

1. If the ELIST entry is referenced by field 6 of an MFLUID entry, the wetted side of the element is determined by the presence or absence of a minus sign preceding the element’s ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element’s positive normal as determined by applying the right-hand rule to the sequence of its corner points. If the “THRU” option is used, then immediately preceding and succeeding elements must have the same sign.
2. Large open “THRUs” should be avoided.
3. The word “THRUs” must not appear in field 2 or 9 on the parent entry or on any continuations.
4. If any ELIST entry is changed or added on restart then a complete re-analysis may be performed. Therefore, ELIST entry changes or additions are not recommended on restart.

ENDDATA Bulk Data Delimiter

Designates the end of the Bulk Data Section.

Format:

ENDDATA

Remark:

1. ENDDATA is optional.

ENDDYNA (SOL 700) Defines the End of Direct Text to Dytran-lsdyna

All entries between TODYNA and ENDDYNA will be passed directly by MD Nastran to Dytran-lsdyna. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
ENDDYNA									

Example:

TODYNA									
MAT1	345	29.0E6	0.285	0.0004					
ENDDYNA									

Field	Contents
TODYNA	
MAT1	
ENDDYNA	

Remarks:

See TODYNA for details of how this entry is used.

EOSPOL (SOL 700) Polynomial Equation of State for Solids

Defines the properties of a polynomial equation of state where the pressure p is defined as follows:

In compression ($\mu > 0$),

$$p = a_1\mu + a_2\mu^2 + a_3\mu^3 + (b_0 + b_1 + b_2\mu^2 + b_3\mu^3)\rho_0 e$$

In tension ($i < 0$),

$$p = a_1\mu + (b_0 + b_1\mu)\rho_0 e$$

where:

$\mu = \eta - 1$

$\eta = \rho / \rho_0$

ρ = overall material density

ρ_0 = reference density

E = specific internal energy per unit mass

Format:

1	2	3	4	5	6	7	8	9	10
EOSPOL	EID	A1	A2	A3	B0	B1	B2	B3	
	100	80.E6							

Example:

	HVL	VISC							
	1.1								

Field	Contents
EID	Unique equation of state member. (Integer > 0, Required)
A1	Coefficient a1 or Bulk Modulus. (Real, Default = 0.0)
A2	Coefficient a2. (Real, Default = 0.0)
A3	Coefficient a3. (Real, Default = 0.0)

Field	Contents
B0	Coefficient b0. (Real, Default = 0.0)
B1	Coefficient b1. (Real, Default = 0.0)
B2	Coefficient b2. (Real, Default = 0.0)
B3	Coefficient b3. (Real)
HVL	Hydrodynamic volume limit. (Real > 1.0, Default = 1.1)
VISC	Viscosity coefficient.

Remarks:

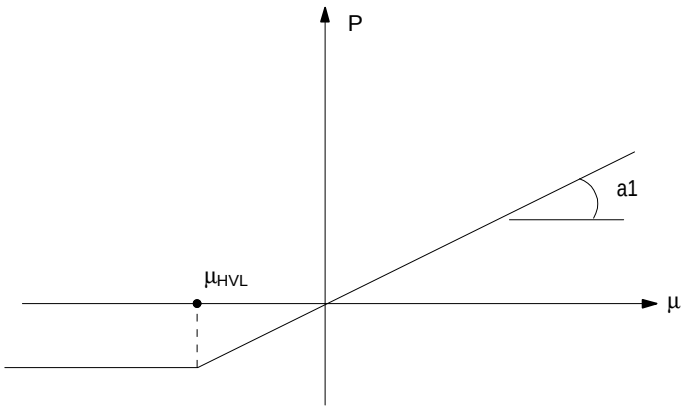
1. The equation of state is used for the Johnson Cook (model 15) and Ramberg-Osgood (model 80) only
2. When the relative volume (ρ_0/ρ) exceeds HVL, the pressure is cut off to

$$P_{HVL} = f(\mu_{HVL})$$

with

$$\mu_{HVL} = \frac{1}{HVL} - 1$$

e.g., for $p = a1 \cdot \mu$, the pressure behavior is as follows:



3. When the PARAM,HVLFAIL is set to YES, the elements where the relative volume (ρ_0/ρ) exceeds HVL fail completely. Their stress state is zero.

EPOINT Extra Point List

Defines extra points for use in dynamics problems.

Format:

1	2	3	4	5	6	7	8	9	10
EPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	

Example:

EPOINT	3	18	1	4	16	2			

Alternate Format and Example:

EPOINT	ID1	“THRU”	ID2						
EPOINT	17	THRU	43						

Field	Contents
IDi	Extra point identification number. (1000000 > Integer > 0; for “THRU” option, ID1 < ID2).

Remarks:

1. All extra point identification numbers must be unique with respect to all other structural, scalar, and fluid points for direct methods of solution. For modal methods, they must be larger than the number of eigenvectors retained for analysis.
2. EPOINT is used to define coordinates used in transfer function definitions (see the TF and DMIG entries).
3. If the alternate format is used, extra points ID1 through ID2 are also defined to be extra points.
4. See the *MSC.Nastran Dynamics Users Guide* for a discussion of extra points.

EXTRN Partitioned External Superelement Connection

Defines a boundary connection for an external superelement.

Format:

1	2	3	4	5	6	7	8	9	10
EXTRN	GID1	C1	GID2	C2	GID3	C3	GID4	C4	
	-etc.-		GID6	“THRU”	GID7	C6	-etc.-		

Example:

EXTRN	1001	123	1120	123456	1201	123			

Field	Contents
GIDi	Grid identification number to which the exterior superelement matrices will be connected.
Ci	Component numbers. (Integer 0, blank, or 1 for scalar points; Integers 1 through 6 with no embedded blanks for grids.)

Remarks:

1. EXTRN can only be specified in partitioned Bulk Data Sections and is ignored in the main Bulk Data Section.
2. Connection grids must be specified in the partitioned Bulk Data Section following BEGIN SUPER = SEID.
3. “THRU” may be specified only in fields 3, 5, or 7.
4. Pairs of blank fields may be entered to allow easier modification of the EXTRN entry.

FBODYLD Equilibrated Free-Body Applied Load Case Definition

Defines an equilibrated free-body applied load case.

Format:

1	2	3	4	5	6	7	8	9	10
FBODYLD	NAMEL	FBODYSB							
	LABEL	LABEL	LABEL	LABEL	LABEL	LABEL	LABEL	LABEL	

Example:

FBODYLD	WINGLD	WINGSB							
	LOAD ON	THE	RIGHT	WING					

Field	Contents
NAMEL	User defined name identifying the load case. (Character, required)
FBODYSB	Name of a FBODYSB Bulk Data entry that defines the subsystem for this load. (Character, required)
LABEL	A string comprising no more than 64 characters (fields 2 through 9) that identifies and labels the load case. (Character, optional)

Remarks:

1. NAMEL must be unique.
2. The Label is optional.

FBODYSB Equilibrated Free-Body Subsystems Definition

Defines an equilibrated free-body subsystem.

Format:

1	2	3	4	5	6	7	8	9	10
FBODYSB	NAMES	GRIDSET	ELEMSET	XFLAG					
	LABEL	LABEL	LABEL	LABEL	LABEL	LABEL	LABEL	LABEL	

Example:

FBODYSB	WING	1	1	ADM					
	RIGHT	WING							

Field	Contents
NAMES	User defined name identifying the submodel. (Character, required)
GRIDSET	Identification number of a SET1 entry that has a list of Grid Point Force grids to include in defining the subsystem. (Integer > 0)
ELEMSET	Identification number of a SET1 entry that has a list of elements to include in the system (Integer > 0 or blank)
XFLAG	Exclusion flag. Exclude the indicated Grid Point Force types. Default = blank (no type excluded). S = SPC forces M = MPC forces A, L, or P = applied loads D = DMIG's (and any other type not described above).
Label	An optional string of up to 64 characters (fields 2 through 9) that identifies the subsystem.

Remarks:

1. Only those Grid Point Forces which have both an included grid point and element (or Grid Point Force type) will be taken into account.
2. If ELEMSET is blank, no contributions are made from the set of elements attached to the grid.
3. Fictitious grids or elements do not produce error or warning messages.

4. The XFLAG data can be any combination of the letters S,M,A,L,P and D (e.g., MAD).
5. The continuation is optional.

FEEDGE Finite Element Edge Definition

Defines a finite element edge and associates it with a curve.

Format:

1	2	3	4	5	6	7	8	9	10
FEEDGE	EDGEID	GRID1	GRID2	CIDBC	GEOMIN	ID1	ID2		

Example:

FEEDGE	101	123	547		GMCURV	12			

Field	Contents	Type	Default
EDGEID	Unique identification number.	Integer > 0	Required
GRIDi	Identification number of end GRIDs defining this edge.	Integer > 0	Required
CIDBC	Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined. See Remark 1.	Integer ≥ 0	
GEOMIN	Type of entry referenced by IDi; “GMCURV” or “POINT”. See Remark 2.	Character	POINT
IDi	Identification number of a POINT or GMCURV entry. See Remarks 2., 3., and 4.	Integer ≥ 0	

Remarks:

1. If CIDBC is not blank then it overrides the CIDBC specified on the GMSURF or FEFACE entries for this particular edge. A fatal message will be issued when more than one CIDBC is associated with any entity.
2. The Bulk Data entries referenced by ID1 and ID2 depends on the GEOMIN field:

GEOMIN	ID1	ID2
POINT	POINT	POINT
GMCURV	GMCURV	not applicable

3. When GEOMIN = “GMCURV”

   - FEEDGE associates the finite element model and the geometric information.
   - GRID1 and GRID2 are the end points of the edge, and the edge is on the CURVID curve. A locally parametric cubic curve is fit to the geometric curve such that the two have the same tangent at GRIDi (C1 continuous).

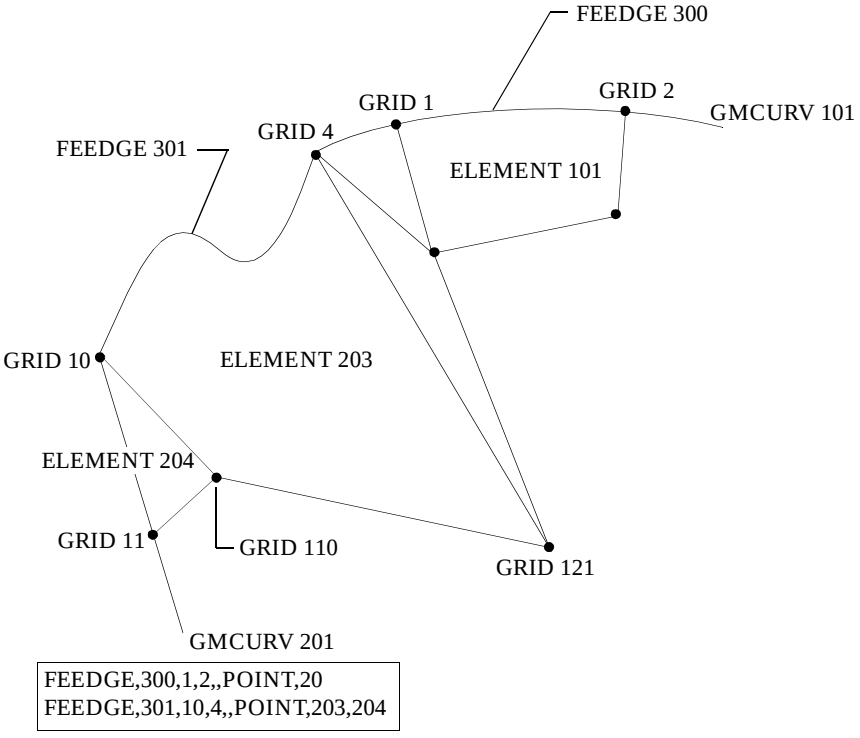


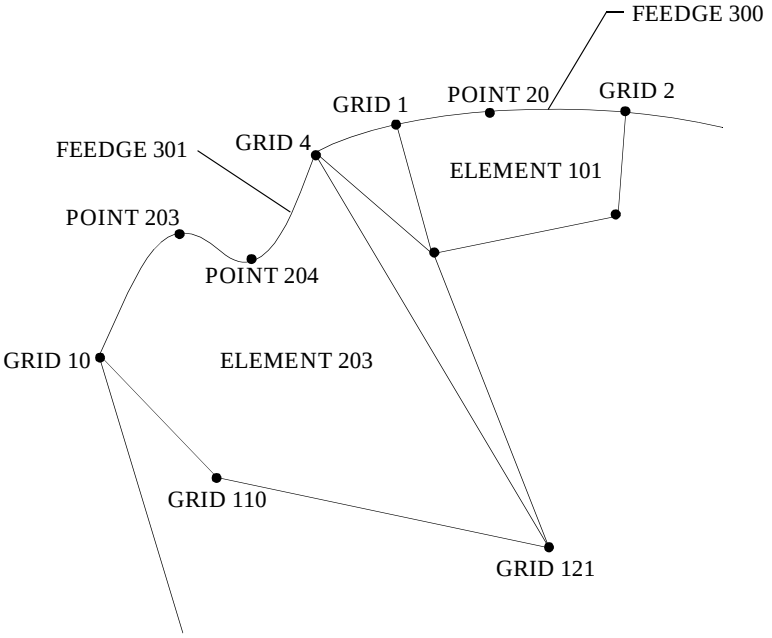
Figure 8-83 Specifying Geometry Using GEOMIN=GMCURV Method

4. When GEOMIN = “POINT”

   - The edge passes through the points defined on the POINT entries referenced by ID1 and ID2.
   - The shape of the edge is selected as follows:

ID1	ID2	Shape of the FEEDGE
Blank or 0	Blank or 0	Linear
>0	Blank or 0	Quadratic

ID1	ID2	Shape of the FEEDGE
>0	>0	Cubic
Blank or 0	>0	Not allowed




```
FEEDGE,300,1,2,,POINT,20
FEEDGE,301,10,4,,POINT,203,204
```

Figure 8-84 Specifying Geometry Using GEOMIN=POINT Method

5. A local coordinate system can be associated with an edge using the GMCORD entry.
6. The hierarchy set to resolve the conflicts arising in the Global System input data is described under Remark 10 of the "**GMBC**" on page 1576 entry description.

FEFACE Finite Element Face Definition

Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.

Format:

1	2	3	4	5	6	7	8	9	10
FEFACE	FACEID	GRID1	GRID2	GRID3	GRID4	CIDBC	SURFID		

Example:

FEFACE	101	123	547	243	295	12			

Field	Contents	Type	Default
FACEID	Unique identification number. See Remark 1.	Integer > 0	Required
GRIDi	Identification number of end GRIDs defining a triangular or quadrilateral face. See Remark 2.	Integer > 0	Required
CIDBC	Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined. See Remark 3.	Integer ≥ 0	Remark 3.
SURFID	Alternate method used to specify the geometry of the edges of the face. See Remarks 4. and 5.	Integer ≥ 0	0

Remarks:

1. An FEFACE entry is required if any of the following situations exist:
   - The geometry of the surface defined by SURFID is to be used by a finite element;
   - CIDBC is specified for a face or surface; or
   - If loads or constraints or enforced boundary conditions are applied to a surface.
2. The shape (geometry) of the face is defined by the shape of the edges. The points defined by GRIDi must be specified in either a clockwise or counterclockwise order.

3. If CIDBC is not blank, then it overrides the CIDBC specified on the GMSURF entry for this particular face. A fatal message will be issued when more than one CIDBC is associated with any entity.
4. When SURFID is blank or 0, the edges will be considered linear unless there is an FEEDGE entry for the given edge.
5. When SURFID > 0,
   - FEFACE associates the finite element model and the geometric information specified on the GMSURF entry.
   - GRIDi defines a finite element face (clockwise or counter clockwise in order) that is on the SURFID surface.
   - For the edges of this face, which are not defined by an FEEDGE entry, locally parametric cubic curves are fit to the geometric surface such that the two have the same tangent at GRIDi (C1 continuous).
6. Whenever a given edge of a face is common to two or more surfaces (i.e., lies on the intersecting curve), then the user must supply GMCURV and FEEDGE entries in order to resolve the conflict in the input geometry. A fatal message is issued if an edge is not uniquely defined.

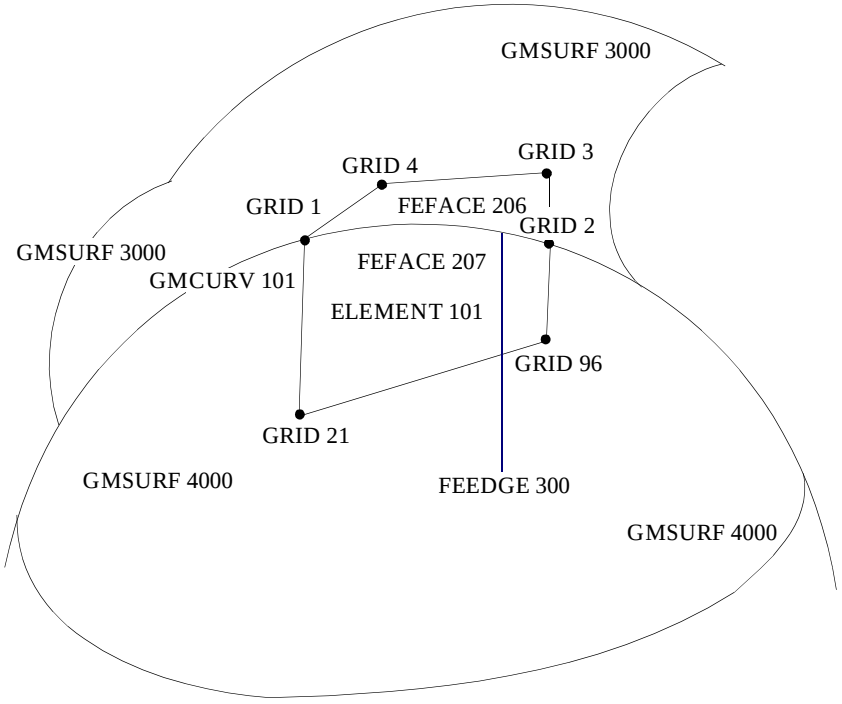


Figure 8-85 Face Edge Common to Two Surfaces

7. The hierarchy set to resolve the conflicts arising in the Global System input data is described under Remark 10 of the "**GMBC**" on page 1576 entry description.

FLFACT Aerodynamic Physical Data

Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.

Format:

1	2	3	4	5	6	7	8	9	10
FLFACT	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	-etc.-						

Example:

FLFACT	97	.3	.7	3.5					

Alternate Format and Example:

FLFACT	SID	F1	“THRU”	FNF	NF	FMID			
FLFACT	201	.200	THRU	.100	11	.133333			

Field	Contents
SID	Set identification number. (Unique Integer > 0)
Fi	Aerodynamic factor. (Real)
FNF	Final aerodynamic factor. (Real)
NF	Number of aerodynamic factors. (Integer > 0)
FMID	Intermediate aerodynamic factors. See Remark 4. (Real)

Remarks:

1. Only the factors selected by a FLUTTER entry will be used.
2. Embedded blank fields are not allowed in the first format above.
3. The factors must be specified in the order in which they are to be used within the looping of flutter analysis.
4. $FMID$ must lie between $F1$ and FNF; otherwise, $FMID$ will be set to $(F1 + FNF)/2$. Then

$$Fi = \frac{F1(FNF - FMID)(NF - 1) + FNF(FMID - F1)(i-1)}{(FNF - FMID)(NF - 1) + (FMID - F1)(i - 1)}$$

where $i = 1, 2, ..., NF$

The use of FMID (middle factor selection) allows unequal spacing of the factors.

$FMID = \frac{2 \cdot F1 \cdot FNF}{F1 + FNF}$ gives equal values to increments of the reciprocal of Fi.

5. If method = PK and this entry specifies velocities, then the velocities must be non-zero. Input of negative values produces eigenvector results at a velocity equal to the positive value of the input. Input of positive values provide eigenvalues results without eigenvectors.

FLSYM Axisymmetric Symmetry Control

Defines the relationship between the axisymmetric fluid and a structural boundary having symmetric constraints. The purpose is to allow fluid boundary matrices to conform to structural symmetry definitions.

Format:

1	2	3	4	5	6	7	8	9	10
FLSYM	M	S1	S2						

Example:

FLSYM	12	S	A						

Field	Contents
M	Number of symmetric sections of structural boundary around the circumference of the fluid being modeled by the set of structural elements. (Even Integers ≥ 2)
S1, S2	Description of boundary constraints used on the structure at the first and second planes of symmetry. (Character: “S” means symmetric, “A” means antisymmetric.)

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. Only one FLSYM entry is allowed.
3. This entry is not required if there are no planes of symmetry.
4. First plane of symmetry is assumed to be at $\phi = 0$. Second plane of symmetry is assumed to be at $\phi = 360° / M$.
5. Symmetric and antisymmetric constraints for the structure must, in addition, be provided by the user.
6. The solution is performed for those harmonic indices listed on the AXIF entry that are compatible with the symmetry conditions.
7. For example, if FLSYM is used to model a quarter section of structure at the boundary, M = 4. If the boundary constraints are “SS”, the compatible cosine harmonics are 0, 2, 4, ..., etc. If “SA” is used, the compatible cosine harmonics are 1, 3, 5, ..., etc.

FLUTTER Aerodynamic Flutter Data

Defines data needed to perform flutter analysis.

Format:

1	2	3	4	5	6	7	8	9	10
FLUTTER	SID	METHOD	DENS	MACH	RFREQ	IMETH	NVALUE / OMAX	EPS	

Example:

FLUTTER	19	K	119	219	319	S	5	1.-4	

Field	Contents
SID	Set identification number. (Integer > 0)
METHOD	Flutter analysis method. (Character: “K” for K-method, “PK” for PK method, “PKNL” for PK method with no looping, “PKS” for PK sweep method, “PKNLS” for PK sweep method with no looping, “KE” for the K-method restricted for efficiency.) See Remark **9.**
DENS	Identification number of an FLFACT entry specifying density ratios to be used in flutter analysis. (Integer > 0)
MACH	Identification number of an FLFACT entry specifying Mach numbers (m) to be used in flutter analysis. (Integer > 0)
RFREQ (or VEL)	Identification number of an FLFACT entry specifying reduced frequencies (k) to be used in flutter analysis; for the “PKx” methods, the velocities FLFACT entry is specified in this field. (Integer > 0)
IMETH	Choice of interpolation method for aerodynamic matrix interpolation. Used in the “K” and “KE” methods only. See Remark **6.** (Character: “L” = linear, “S” = surface; Default = “L”.)
NVALUE	Number of eigenvalues beginning with the first eigenvalue for output and plots. [Integer > 0; Default is the number of modal degrees-of-freedom (u_h).]
OMAX	For the PKS and PKNLS methods, OMAX specifies the maximum frequency, in Hz., to be used in he flutter sweep. (Real > 0.0, Default = maximum normal mode eigenfrequency)

Remarks:

1. The FLUTTER entry must be selected with the Case Control command FMETHOD = SID.
2. The density is given by ENS · RHOREF, where RHOREF is the reference value specified on the AERO entry and DENS is the density ratio specified on the FLFACT entry.
3. The reduced frequency is given by $k = (REFC \cdot \omega / 2 \cdot V)$, where REFC is given on the AERO entry, ω is the circular frequency, and V is the velocity. If $k = 0.0$, as specified on the FLFACT entry, then only the K-method may be specified and the Inverse Power method of eigenvalue extraction (INV on the EIGC entry) must be used. Aeroelastic divergence analysis is more appropriately performed using one of the "PKx" methods.
4. For the PK and PKNL methods, an eigenvalue is accepted when:

   $$|k - k_{estimate}| < EPS \qquad \text{for } k_{estimate} < 1.0$$

   $$|k - k_{estimate}| < EPS \cdot k_{estimate} \qquad \text{for } k_{estimate} \geq 1.0$$

5. When one of the "PKx" methods is selected, physical displacements will only be generated for the velocities on the FLFACT that are specified as negative values of the requested velocity. Also, structural damping as specified on the GE field of MATi entries is ignored.
6. If IMETH = "L", a linear interpolation is performed on reduced frequencies at the Mach numbers specified on the FLFACT entry using the MKAEROi entry Mach number that is closest to the FLFACT entry Mach number. For IMETH = "S", a surface interpolation is performed across Mach numbers and reduced frequencies. For the "PKx" methods, linear interpolation is always performed.
7. For the "K", "KE", "PK", and "PKS" methods, all combinations of the FLFACT entry are analyzed. For the "PKNL" and "PKNLS" methods, only ordered pairs are analyzed; i.e., $(\rho_1, M_1, V_1), (\rho_2, M_2, V_2) \ldots (\rho_n, M_n, V_n)$. For the PKNL and PKNLS methods, equal number of densities, Mach numbers and velocities must be specified.
8. "K" and "KE" methods are not supported for design sensitivity and optimization.

9. The PKS and PKNLS methods determine flutter eigenvalues by performing a sweep of equally spaced reduced frequencies ranging from $k_{est} = 0.0$ through $_{est} = \pi \cdot \text{CREF} \cdot \text{OMAX}/\text{Velocity}$. The number of intervals is calculated using NINT = INT(1.0/EPS).

FORCE Static Force

Defines a static concentrated force at a grid point by specifying a vector.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

Example:

FORCE	2	5	6	2.9	0.0	1.0	0.0		

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer $\geq$ 0; Default = 0)
F	Scale factor. (Real)
Ni	Components of a vector measured in coordinate system defined by CID. (Real; at least one Ni $\neq$ 0.0. unless F is zero)

Remarks:

1. The static force applied to grid point G is given by

$$\vec{f} = F\vec{N}$$

where $\vec{N}$ is the vector defined in fields 6, 7 and 8. The magnitude of $\vec{f}$ is equal to F times the magnitude of $\vec{N}$.

2. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD = SID. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
3. A CID of zero or blank (the default) references the basic coordinate system.
4. For scalar points see SLOAD.

FORCE1 Static Force, Alternate Form 1

Defines a static concentrated force at a grid point by specification of a magnitude and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

Example:

FORCE1	6	13	-2.93	16	13				

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of the force. (Real)
G1, G2	Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident.)

Remarks:

1. The static force applied to grid point G is given by

$$\vec{f} = F\vec{n}$$

where $\vec{n}$ is a unit vector parallel to a vector from G1 to G2.

2. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD=SID. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "**FOLLOWK**" on page 710). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if

geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

FORCE2 Static Force, Alternate Form 2

Defines a static concentrated force at a grid point by specification of a magnitude and four grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE2	SID	G	F	G1	G2	G3	G4		

Example:

FORCE2	6	13	-2.93	16	13	17	13		

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of the force. (Real)
Gi	Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident; G3 and G4 cannot be coincident.)

Remarks:

1. The direction of the force is parallel to the cross product of vectors from G1 to G2 and G3 to G4.
2. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD=SID. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter “**FOLLOWK**” on page 710). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

FORCEAX Conical Shell Static Force

Defines a static concentrated force on a conical shell ring.

Format:

1	2	3	4	5	6	7	8	9	10
FORCEAX	SID	RID	HID	S	FR	FP	FZ		

Example:

FORCEAX	1	2	3	2.0	0.1	0.2	0.3		

Field	Contents
SID	Load set identification number. (Integer > 0)
RID	RINGAX entry identification number. (Integer > 0)
HID	Harmonic identification number or a sequence of harmonics. See Remark 5. (Integer ≥ 0)
S	Scale factor for the force. (Real)
FR, FP, FZ	Force components in r, ϕ, z directions. (Real)

Remarks:

1. FORCEAX is allowed only if an AXIC entry is also present.
2. Axisymmetric shell loads must be selected with the Case Control command LOAD = SID.
3. A separate entry is needed for the definition of the force associated with each harmonic.
4. See "**Conical Shell Element (RINGAX)**" in Chapter 3 of the *MSC.Nastran Reference Guide* for further discussion.
5. If a sequence of harmonics is to be placed in HID, the form is as follows: "Sn1Tn2" where n1 is the start of the sequence and n2 is the end of the sequence (e.g., for harmonics 0 through 10, the field would contain "S0T10").

FREEPT Fluid Free Surface Point

Defines the location of points on the surface of a fluid for recovery of surface displacements in a gravity field.

Format:

1	2	3	4	5	6	7	8	9	10
FREEPT	IDF		IDP1	PHI1	IDP2	PHI2	IDP3	PHI3	

Example:

FREEPT	3		301	22.5	302	90.0	303	370.0	

Field	Contents
IDF	RINGFL entry identification number. (Integer > 0)
IDPi	Free surface point identification number. (Integer > 0)
PHIi	Azimuthal position on fluid point (RINGFL entry) in the fluid coordinate system. (Real)

Remarks:

1. FREEPT is allowed only if an AXIF entry is also present.
2. All free surface point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. The free surface points are used for the identification of output data only.
4. Three points may be defined on a single entry.
5. The referenced fluid point (IDF) must be included in a free surface list (FSLIST entry).
6. Output requests for velocity and acceleration can be made at these points.

FREQ Frequency List

Defines a set of frequencies to be used in the solution of frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	F10	-etc.-					

Example:

FREQ	3	2.98	3.05	17.9	21.3	25.6	28.8	31.2	
	29.2	22.4	19.3						

Field	Contents
SID	Set identification number. (Integer > 0)
Fi	Frequency value in units of cycles per unit time. (Real $\geq$ 0.0)

Remarks:

1. Frequency sets must be selected with the Case Control command FREQUENCY = SID.
2. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|,$$

   where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.
3. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ1 Frequency List, Alternate Form 1

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and the number of increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ1	SID	F1	DF	NDF					

Example:

FREQ1	6	2.9	0.5	13					

Field	Contents
SID	Set identification number. (Integer > 0)
F1	First frequency in set. (Real $\geq$ 0.0)
DF	Frequency increment. (Real > 0.0)
NDF	Number of frequency increments. (Integer > 0; Default = 1)

Remarks:

1. FREQ1 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and DF are cycles per unit time.
3. The frequencies defined by this entry are given by

$$f_i = F1 + DF \cdot (i-1)$$

where i = 1 to (NDF + 1).

4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$\left|f_N - f_{N-1}\right| < \text{DFREQ} \cdot \left|f_{MAX} - f_{MIN}\right|,$$

where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ2 Frequency List, Alternate Form 2

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, final frequency, and the number of logarithmic increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ2	SID	F1	F2	NF					

Example:

FREQ2	6	1.0	8.0	6					

Field	Contents
SID	Set identification number. (Integer > 0)
F1	First frequency. (Real > 0.0)
F2	Last frequency. (Real > 0.0, F2 > F1)
NF	Number of logarithmic intervals. (Integer > 0; Default = 1)

Remarks:

1. FREQ2 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and F2 are cycles per unit time.
3. The frequencies defined by this entry are given by

$$f_i = F1 \cdot e^{(i-1)d}$$

where $d = \frac{1}{NF}\ln\frac{F2}{F1}$ and $i = 1, 2, \ldots, (NF + 1)$

In the example above, the list of frequencies will be 1.0, 1.4142, 2.0, 2.8284, 4.0, 5.6569 and 8.0 cycles per unit time.

4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$\left|f_N - f_{N-1}\right| < \text{DFREQ} \cdot \left|f_{MAX} - f_{MIN}\right|,$$

where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ3 Frequency List, Alternate 3

Defines a set of excitation frequencies for modal frequency-response solutions by specifying number of excitation frequencies between two modal frequencies.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ3	SID	F1	F2	TYPE	NEF	CLUSTER			

Example:

FREQ3	6	20.0	200.0	LINEAR	10	2.0			

Field	Contents
SID	Set identification number. (Integer > 0)
F1	Lower bound of modal frequency range in cycles per unit time. (Real $\geq$ 0.0 for TYPE = LINEAR and Real = 0.0 for TYPE = LOG)
F2	Upper bound of modal frequency range in cycles per unit time. (Real > 0.0, F2 $\geq$ F1, Default = F1)
TYPE	LINEAR or LOG. Specifies linear or logarithmic interpolation between frequencies. (Character; Default = "LINEAR")
NEF	Number of excitation frequencies within each subrange including the end points. The first subrange is between F1 and the first modal frequency within the bounds. The second subrange is between first and second modal frequencies between the bounds. The last subrange is between the last modal frequency within the bounds and F2. (Integer > 1, Default = 10)
CLUSTER	Specifies clustering of the excitation frequency near the end points of the range. See Remark **6.** (Real > 0.0; Default = 1.0)

Remarks:

1. FREQ3 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ3 entries must be selected with the Case Control command FREQUENCY = SID.

3. In the example above, there will be 10 frequencies in the interval between each set of modes within the bounds 20 and 2000, plus 10 frequencies between 20 and the lowest mode in the range, plus 10 frequencies between the highest mode in the range and 2000.
4. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.
5. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter, with a default of 10^{-5}. f_{MAX} an f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

6. CLUSTER is used to obtain better resolution near the modal frequencies where the response varies the most. CLUSTER > 1.0 provides closer spacing of excitation frequency towards the ends of the frequency range, while values of less than 1.0 provide closer spacing towards the center of the frequency range. For example, if the frequency range is between 10 and 20, NEF = 11, TYPE = "LINEAR"; then, the excitation frequencies for various values of CLUSTER would be as shown in **Table 8-18**.

$$\hat{f}_k = \frac{1}{2}(\hat{f}_1 + \hat{f}_2) + \frac{1}{2}(\hat{f}_2 - \hat{f}_1)|\xi|^{1/\text{CLUSTER}} \cdot \text{SIGN}(\xi)$$

where

ξ = $-1 + 2(k-1)/(\text{NEF}-1)$ is a parametric coordinate between -1 and 1

k = varies from 1 to NEF (k = 1, 2, ..., NEF)

$\hat{f}_1$ = is the lower limit of the frequency subrange

$\hat{f}_2$ = is the upper limit o the subrange

$\hat{f}_k$ = is the k-th excitation frequency

$\hat{f}$ = is the frequency, or the logarithm of the frequency, depending on the value specified for TYPE

Table 8-18 CLUSTER Usage Example

Excitation Frequency Number	ξ	CLUSTER				
		c=0.25	c-0.50	c-1.0	c-2.0	c-4.0
		Excitation Frequencies in Hertz				
1	-1.0	10.00	10.0	10.0	10.00	10.00
2	-0.8	12.95	11.8	11.0	10.53	10.27
3	-0.6	14.35	13.2	12.0	11.13	10.60
4	-0.4	14.87	14.2	13.0	11.84	11.02
5	-0.2	14.99	14.8	14.0	12.76	11.66
6	0.0	15.00	15.0	15.0	15.00	15.00
7	0.2	15.01	15.2	16.0	17.24	18.34
8	0.4	15.13	15.8	17.0	18.16	18.98
9	0.6	15.65	16.8	18.0	18.87	19.40
10	0.8	17.05	18.2	19.0	19.47	19.73
11	1.0	20.00	20.0	20.0	20.00	20.00

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ4 Frequency List, Alternate Form 4

Defines a set of frequencies used in the solution of modal frequency-response problems by specifying the amount of "spread" around each natural frequency and the number of equally spaced excitation frequencies within the spread.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ4	SID	F1	F2	FSPD	NFM				

Example:

FREQ4	6	20.0	200.0	0.30	21				

Field	Contents
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real $\geq$ 0.0, Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0, F2 > F1, Default = 1.0E20)
FSPD	Frequency spread, +/- the fractional amount specified for each mode which occurs in the frequency range F1 to F2. (1.0 > Real > 0.0, Default = 0.10)
NFM	Number of evenly spaced frequencies per "spread" mode. (Integer > 0; Default = 3; If NFM is even, NFM + 1 will be used.)

Remarks:

1. FREQ4 applies only to modal frequency-response solutions (SOLs 111, 146, and 200 and is ignored in direct frequency-response solutions.
2. FREQ4 entries must be selected with the Case Control command FREQUENCY = SID.
3. There will be NFM excitation frequencies between $(1 - FSPD) \cdot f_N$ and $(1 + FSPD) \cdot f_N$, for each natural frequency in the range F1 to F2.
4. In the example above there will be 21 equally spaced frequencies across a frequency band of $0.7 \cdot f_N$ to $1.3 \cdot f_N$ for each natural frequency that occurs between 20 and 2000. See **Figure 8-86** for the definition of frequency spread.

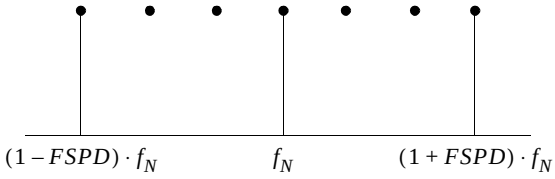


Figure 8-86 Frequency Spread Definition

Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

5. The frequency spread can be used also to define the half-power bandwidth. The half-power bandwidth is given by $2 \cdot \xi \cdot f_N$, where ξ is the damping ratio. Therefore, if FSPD is specified equal to the damping ratio for the mode, NFM specifies the number of excitation frequency within the half-power bandwidth. See **Figure 8-87** for the definition of half-power bandwidth.

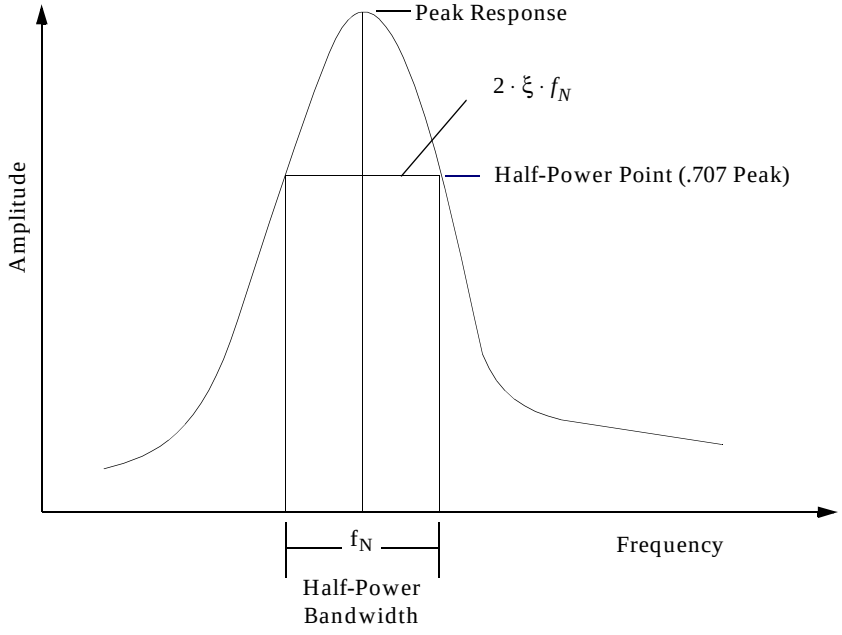


Figure 8-87 Half-Power Bandwidth Definition

6. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

7. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

$$\left|f_N - f_{N-1}\right| < \text{DFREQ} \cdot \left|f_{MAX} - f_{MIN}\right|$$

where DFREQ is a user parameter with a default of 10^{-5}. The values f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

8. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
9. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FREQ5 Frequency List, Alternate Form 5

Defines a set of frequencies used in the solution of modal frequency-response problems by specification of a frequency range and fractions of the natural frequencies within that range.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ5	SID	F1	F2	FR1	FR2	FR3	FR4	FR5	
	FR6	FR7	-etc.-						

Example:

FREQ5	6	20.0	200.0	1.0	0.6	0.8	0.9	0.95	
	1.05	1.1	1.2						

Field	Contents
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real $\geq$ 0.0; Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0, F2 > F1, Default = 1.0E20)
FRi	Fractions of the natural frequencies in the range F1 to F2. (Real > 0.0)

Remarks:

1. FREQ5 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ5 entries must be selected with the Case Control command FREQUENCY = SID.
3. The frequencies defined by this entry are given by

$$f_i = FRi \cdot f_{N_i}$$

where f_{N_i} are the natural frequencies in the range F1 through F2.

4. In the example above, the list of frequencies will be 0.6, 0.8, 0.9, 0.95, 1.0, 1.05, 1.1, and 1.2 times each natural frequency between 20 and 2000. If this computation results in excitation frequencies less then F1 and greater than F2, those computed excitation frequencies are ignored.

   Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

5. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

6. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if

   $$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

   where DFREQ is a user parameter with a default of 10^{-5}. The values f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.

8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

FSLIST Free Surface List

Defines the fluid points (RINGFL entry) that lie on a free surface boundary.

Format:

1	2	3	4	5	6	7	8	9	10
FSLIST	RHO	IDF1	IDF2	IDF3	IDF4	IDF5	IDF6	IDF7	
	IDF8	IDF9	-etc.-						

Examples:

1	2	3	4	5	6	7	8	9	10
FSLIST	1.0-4	1	3	5	4	2	7	6	
	8	9	10	11	AXIS				

Field	Contents
RHO	Mass density at the surface. (Real > 0.0; the default is taken from DRHO on the AXIF entry.
IDFi	Identification number of RINGFL entry. (Integer > 0 or Character = “AXIS” in first and/or last field only).

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. The order of the points must be sequential with the fluid on the right with respect to the direction of travel.
3. The word “AXIS” defines an intersection with the polar axis of the fluid coordinate system.
4. If the fluid density varies along the boundary, there must be one FSLIST entry for each interval between fluid points.

GBAG (SOL 700) Gas-Bag Pressure Definition

Defines the pressure within an enclosed volume.

Format:

1	2	3	4	5	6	7	8	9	10
GBAG	GID	BSID		-	-	-	-	-	
		-	-	-	-	-	-	-	
		-	-	-	-	-	-	-	
	CPGAS	GPGASV	RGAS	PENV		REVERSE	CHECK	PINIT	
	TINIT	TENV							
		-	-	-	-	-	-	-	
		-	-	-	-	-	-	-	

Examples:

1	2	3	4	5	6	7	8	9	10
GBAG	101	37							
	CONSTANT	1.0	293.0	10000.		YES	YES	50000	
	297.0	297.0							

Field	Contents		
GID	Unique gas-bag ID.	I > 0	Required
BSID	Identification number of a BSURF, BCPROP or BCMATL, BSURF, BCPROP, BCMATL can only reference shell elements.	I > 0	Required
CPGAS	The variation of the specific heat constant at constant pressure. CONSTANT The specific heat is constant and specified in CPGASV.	C	CONSTANT
CPGASV	The specific heat of the gas.	R	Required

Field	Contents		
RGAS	Gas constant of the inflowing gas.	R	Required
PENV	Environmental pressure surrounding the gas bag.	R	Required
REVERSE	Normal auto-reverse switch. ON The normals of the SURFACE are automatically reversed if necessary so that they point in the same direction and provide a positive volume. OFF The normals are not automatically reversed.	C	ON
CHECK	Normal checking switch. ON The normals of the SURFACE are checked to see if they all point in the same direction and provide a positive volume. OFF The normals are not checked. If REVERSE is set to ON, CHECK is automatically set to ON.	C	ON
PINIT	Initial pressure inside the gas bag.	R	PENV
TINIT	Initial temperature inside the gas bag. See Remark 4.	R	Required.
TENV	Environmental Temperature. The value is required when heat transfer is used.	R > 0	Required.

Remarks:

1. The collection of shell elements referenced must form a closed volume.
2. The pressure in the gas bag is applied to all the faces of the surface.
3. TINIT is the temperature of the gas inside the volume at *time* = 0. At *time* = 0, the mass of the gas inside the gas bag is calculated as

$$m = \frac{P_{init}V}{RT_{init}}$$

where, P_{init} the initial pressure, V the volume, R the gas constant, and T_{init} the initial gas temperature.

4. GBAG is a subset of AIRBAG, an entry which is not yet available in SOL 700. The blank lines and fields in GBAG are represent entries that are not blank when AIRBAG is incorporated.

GENEL General Element

Defines a general element.

Format:

1	2	3	4	5	6	7	8	9	10
GENEL	EID		UI1	CI1	UI2	CI2	UI3	CI3	
	UI4	CI4	UI5	CI5	-etc.-				

UIm -- The last item in the UI list will appear in one of fields 2, 4, 6, or 8.

	“UD”		UD1	CD1	UD2	CD2	-etc.-		

UDn -- The last item in the UD list will appear in one of fields 2, 4, 6, or 8.

	“K” or “Z”	KZ11	KZ21	KZ31	-etc.-	KZ22	KZ32		
	-etc.-		KZ33	KZ43	-etc.-				

KZmm -- The last item in the K or Z matrix will appear in one of fields 2 through 9.

	“S”	S11	S12	-etc.-		S21	-etc.-		

Smn -- The last item in the S matrix will appear in one of fields 2 through 9.

Example:

GENEL	629		1	1	13	4	42	0	
	24	2							
	UD		6	2	33	0			
	Z	1.0	2.0	3.0	4.0	5.0	6.0	7.0	
	8.0	9.0	10.0						
	S	1.5	2.5	3.5	4.5	5.5	6.5	7.5	
	8.5								

Field	Contents
EID	Unique element identification number. (Integer > 0)
Uli, Cli UDj, CDj	Identification numbers of degrees-of-freedom in the UI or UD list, in sequence corresponding to the [K], [Z], and [S] matrices. UIi and UDi are grid point numbers, and CIi and CDj are the component numbers. If a scalar point is given, the component number is zero. (Integer ≥ 0)
KZij	Values of the [K] or [Z] matrix ordered by columns from the diagonal, according to the UI list. (Real)
Sij	Values of the [S] matrix ordered by rows according to the UD list. (Real)
"UD", "K", "Z", and "S"	Character strings that indicate the start of data belonging to the UD list or the [K], [Z], or [S] matrices.

Remarks:

1. The stiffness approach:

$$\begin{Bmatrix} f_1 \\ \hline f_d \end{Bmatrix} = \left[\begin{array}{c|c} K & -KS \\ \hline -S^T K & S^T KS \end{array}\right] \begin{Bmatrix} u_i \\ \hline u_d \end{Bmatrix}$$

The flexibility approach:

$$\begin{Bmatrix} u_i \\ \hline f_d \end{Bmatrix} = \left[\begin{array}{c|c} Z & S \\ \hline -S^T & O \end{array}\right] \begin{Bmatrix} f_i \\ \hline u_d \end{Bmatrix} \begin{Bmatrix} u_i \\ \hline f_d \end{Bmatrix}$$

where

$$\{u_i\} = [u_{i1}, u_{i2}, \ldots, u_{im}]^T$$

$$\{u_d\} = [u_{d1}, u_{d2}, \ldots, u_{dn}]^T$$

$$[KZ] = [K] \text{ or } [Z] = \begin{bmatrix} KZ11 & \cdots & \cdots & \cdots \\ KZ21 & KZ22 & \cdots & \cdots \\ KZ31 & KZ32 & \cdots & \cdots \\ \vdots & \vdots & \vdots & \vdots \\ KZ_{m1} & \cdots & \cdots & KZ_{mm} \end{bmatrix} \text{ and } [KZ]^T = [KZ]$$

$$[S] = \begin{bmatrix} S11 & \cdots & S_{1n} \\ S21 & \cdots & \cdots \\ S31 & \cdots & \cdots \\ \vdots & \vdots & \vdots \\ S_{m1} & \cdots & S_{mn} \end{bmatrix}$$

The required input is the $\{u_i\}$ list and the lower triangular portion of [*K*] or [*Z*]. Additional input may include the $\{u_d\}$ list and [*S*]. If [*S*] is input, $\{u_d\}$ must also be input. If $\{u_d\}$ is input but [*S*] is omitted, [*S*] is internally calculated. In this case, $\{u_d\}$ must contain six and only six degrees-of-freedom.

The forms shown above for both the stiffness and flexibility approaches assume that the element is a free body with rigid body motions that are defined by $\{u_i\} = [S]\{u_d\}$. See "**General Element Capability (GENEL)**" in Chapter 3 of the *MSC.Nastran Reference Guide*the for further discussion.

2. When the stiffness matrix K is input, the number of significant digits should be the same for all terms.
3. Double-field format may be used for input of K or Z.
4. The DMIG entry or the INPUTT4 module offer alternative methods for inputting large matrices.
5. The general element entry in the example above defines the following:

$$[u_i] = [1\text{–}1,\ 13 - 4,\ 42,\ 24 - 2]^T$$

$$\{u_d\} = [6 - 2,\ 33]^T$$

where i-j means the j-th component of grid point i. Points 42 and 33 are scalar points.

$$[Z] = \begin{bmatrix} 1.0 & 2.0 & 3.0 & 4.0 \\ 2.0 & 5.0 & 6.0 & 7.0 \\ 3.0 & 6.0 & 8.0 & 9.0 \\ 4.0 & 7.0 & 9.0 & 10.0 \end{bmatrix} \quad [S] = \begin{bmatrix} 1.5 & 2.5 \\ 3.5 & 4.5 \\ 5.5 & 6.5 \\ 7.5 & 8.5 \end{bmatrix}$$

GMBC General Enforced Displacement Definition

Defines enforced displacements for GRID, FEEDGE, GMCURV, FEFACE, and GMSURF entries.

Format:

1	2	3	4	5	6	7	8	9	10
GMBC	LID	SPCID	C	ENTITY	ID	METHOD	FIELD1	FIELD2	
	FIELD3	FIELD4	-etc.-						

Example:

GMBC	127	1	2	GMCURV	1	QUAD	1.	2.	
	1.0								

Field	Contents	Type	Default
LID	LOAD set identification number. See Remark 2.	Integer ≥ 0	Required
SPCID	SPC set identification number. See Remark 2.	Integer > 0	Required
C	Component number in the output coordinate system (global). See Remarks 3. and 4.	$0 \leq$ Integer ≤ 6	Required
ENTITY	Entity that the enforced displacement is applied to (Specify GRID, FEEDGE, GMCURV, FEFACE, or GMSURF). See Remarks.	Character	Required
ID	ID of the entity selected above. See Remarks.	Integer >0	Required
METHOD	Method used to supply the data (EQUATION or TABLE, CONSTANT, LINEAR, QUAD, CUBIC). See Remarks.	Character	Remark
FIELDi	Enforced displacement data. See Remarks.	Integer or Real	Required

Remarks:

1. GMBC is the recommended entry for specifying boundary conditions and must be selected with Case Control command SPC = SPCID.
2. LID and SPCID refer to Case Control commands for specifying loads and boundary conditions. Whenever there are several nonzero enforced motion vectors supplied, the most efficient processing of the data (single decomposition of the stiffness matrix) is accomplished by specifying both LID and SPCID.

LID	Result
> 0	Generate SPC entries with zero displacements and SPCD entries with non-zero displacements.
0	Generate SPC entries with non-zero displacements only

3. The components of motion specified by C (field 4) of all degrees-of-freedom in the output coordinate system (Global System) associated with an entity will be constrained.
4. If C = 0 is specified then the degrees-of-freedom will be taken out of the constraint set. In this case the method field is not required.
5. The component is a single integer (1 or 2 or 3 etc.). Use multiple GMBC entries to enforce constraints on multiple components.
6. If METHOD = “EQUATION”, “TABLE”, or “CONSTANT” then FIELD1 is:

METHOD	FIELD1	Type
EQUATION	ID of a DEQATN entry defining the displacement value as a function of location	Integer > 0
TABLE	ID of a TABLE3D entry defining the displacement value as a function of location	Integer > 0
CONSTANT	Value of enforced displacement	Real

7. When METHOD = CONSTANT, a constant displacement is specified for the FEEDGE, GMCURV, FEFACE, and GMSURF entities.

8. If ENTITY = “FEEDGE” the METHOD field can be used to specify, linear, quadratic, and cubic displacements. FIELD1 and FIELD2 correspond to GRID1 and GRID2 on the FEEDGE entry. The values in FIELD3 and FIELD4 are:

Applying Linear, Quadratic, and Cubic Displacements to an FEEDGE		
METHOD	FIELD3	FIELD4
LINEAR	blank	blank
QUAD	Value at 1/2 chord length	blank
CUBIC	Value at 1/3 chord length	Value at 2/3 chord length

9. If ENTITY = “FEFACE” the METHOD field specifies linear or quadratic displacements. The values of FIELDi are location specific:

   - Quadrilateral FEFACE

METHOD	FIELD1 through FIELD4	FIELD5 through FIELD8	FIELD9	Displacement Function
LINEAR	Value at GRID1, 2, 3, 4	blank	blank	Linear
QUAD	Value at GRID1, 2, 3, 4	Value at mid side of EDGE1, 2, 3, 4	Value at middle of FEFACE	Quadratic

   - Triangular FEFACE

METHOD	FIELD1 through FIELD3	FIELD4 through FIELD6	Displacement Function
LINEAR	Value at GRID1, 2, 3	blank	Linear
QUAD	Value at GRID1, 2, 3	Value at mid side of EDGE1, 2, 3	Quadratic

10. In general, the hierarchy set to resolve the conflicts arising in the enforced displacement input data is: GRIDs followed by FEEDGEs followed by GMCURVs followed by FEFACEs followed by GMSURFs. This means that:

- In general the program does not allow the user to supply multiple values of enforced displacements for the same component (C).
- Displacement values specified for each component of a given GMSURF entry are applied to the same component of all GRID, FEEDGE, and FEFACE degrees-of-freedom that lie within the GMSURF.
- Displacement values specified for each component of different GMSURF entries are averaged and applied to the same component of all GRID, FEEDGE degrees-of-freedom that are shared by (that are common to) the multiple GMSURFs.
- Displacement values specified for a given FEFACE entry are applied to all GRID, FEEDGE, and FEFACE degrees-of-freedom that lie within the FEFACE. This data overrides the data that is specified for all the components of the given GRID, FEEDGE and FEFACE degrees-of-freedom that lie within the FEFACE by using GMSURF entries.
- Displacement values specified for each component of different FEFACE entries are averaged and applied to the same component of all GRID, FEEDGE degrees-of-freedom that are shared by (that are common to) the multiple FEFACEs. This data overrides the data that is specified for all the components of the given FEEDGE and GRIDs by using GMSURF entries.
- Displacement values specified for each component of a given GMCURV entry are applied to the same component of all GRID, FEEDGE, degrees-of-freedom that lie within the GMCURV. This data overrides the data for all the components that are specified for the given FEEDGE and GRIDs by using GMSURF or FEFACE entries.
- Displacement values specified for each component of different GMCURV entries are averaged and applied to the same component of all GRID degrees-of-freedom that are shared by (that are common to) the multiple GMCURVs. This data overrides the data for all the components that are specified for the given GRIDs by using GMSURF or FEFACE entries.

- Displacement values specified for each component of a given FEEDGE entry are applied to the same component of all GRID, FEEDGE, degrees-of-freedom that lie within the FEEDGE. This data overrides the data for all the components that is specified for the given FEEDGE and GRIDs by using GMCURV or FEFACE or GMSURF entries.
- Displacement values specified for each component of different FEEDGE entries are averaged and applied to the same component of all GRID degrees-of-freedom that are shared by (that are common to) the multiple FEEDGEs. This data overrides the data for all the components that are specified for the given GRIDs by using GMCURV or FEFACE or GMSURF entries.
- Grids have the highest priority, i.e., any value / property specified using a GRID entry overrides all other information associated with that GRID. If multiple entries are used for a given GRID, e.g., multiple SPCs, then the existing rules govern (SPCs are combined, FORCE is added, SPCDs for the same component are not allowed).
- It is important to recall that these displacements are assumed to be in the Global Coordinate System and that the interconsistency of the output coordinate systems of the various GRIDs, FEEDGEs, FEFACEs is not checked.
- If an entity is specified on both a GMBC and GMSPC entry then the GMSPC specification will be ignored.

11. For the example in **Figure 8-88**,

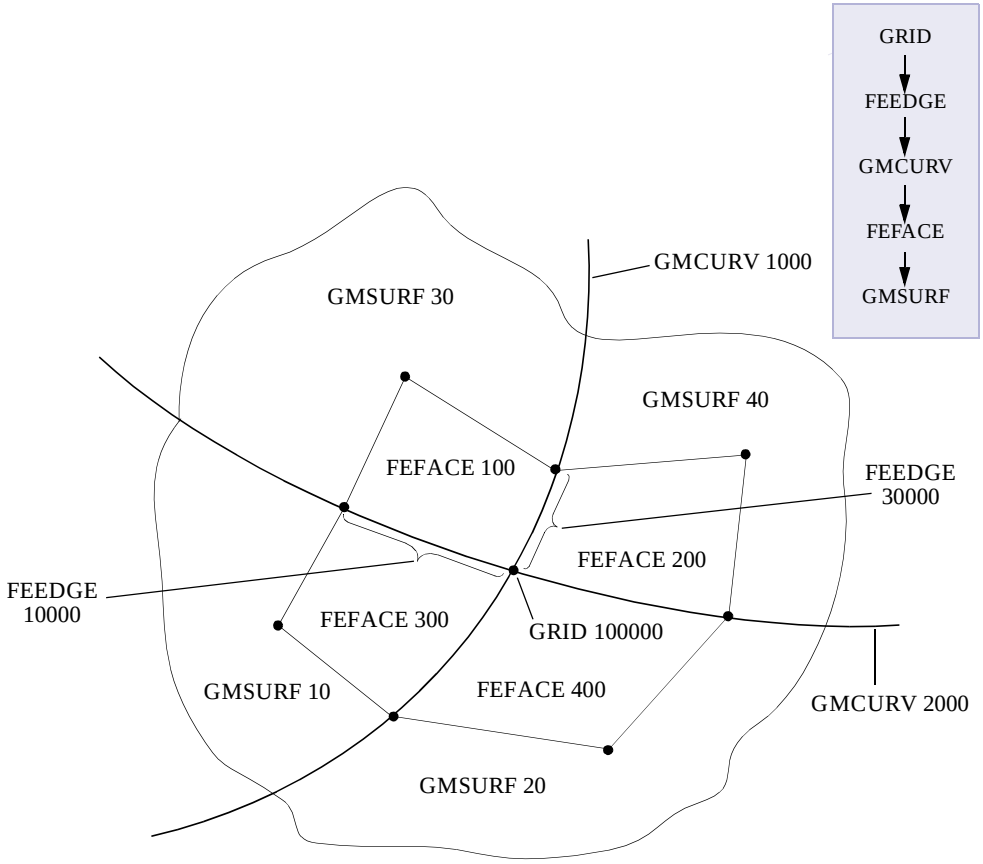


Figure 8-88 Use of Multiple Surface and Curves

- The enforced displacement for GRID 100000 can be specified using SPCD, GMBC referring to an FEEDGE, GMBC referring to a GMCURV, GMBC referring to a FEFACE, and GMBC referring to a GMSURF. **Table 8-19** describes the outcome of using these different methods:

Table 8-19 Enforced Displacement Used for GRID 10000

When Specified Using	Action
SPCD	Overrides all other information supplied for all components.
Single GMBC (FEEDGE)	Overrides information supplied for all components using GMBC(GMCURV) GMBC(FEFACE), GMBC(GMSURF) entries.
Multiple GMBC (FEEDGE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMCURV) GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (GMCURV)	Overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Multiple GMBC (GMCURV)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (FEFACE)	Overrides information supplied for all components using GMBC(GMSURF) entries.
Multiple GMBC (FEFACE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMSURF) entries.
Single GMBC (GMSURF)	Values are applied.
Multiple GMBC (GMSURF)	Values are averaged on a component basis.

- The enforced displacement for GRID DOFs and edge DOFs belonging to FEEDGE 10000 can be specified using GMBC referring to an FEEDGE, GMBC referring to a GMCURV, GMBC referring to a FEFACE, and GMBC referring to a GMSURF. **Table 8-20** describes the outcome of using these different methods:

Table 8-20 Enforced Displacement Used for FEEDGE 10000

When Specified Using	Action
Single GMBC (FEEDGE)	Overrides information supplied for all components using GMBC(GMCURV), GMBC(FEFACE), GMBC(GMSURF) entries.
Multiple GMBC (FEEDGE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMCURV), GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (GMCURV)	Overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Multiple GMBC (GMCURV)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (FEFACE)	Overrides information supplied for all components using GMBC(GMSURF) entries.
Multiple GMBC (FEFACE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMSURF) entries.
Single GMBC (GMSURF)	Values are applied.
Multiple GMBC (GMSURF)	Values are averaged on a component basis.

- The enforced displacement for the GRID DOFs and edge DOFs belonging to GMCURV 1000 can be specified using GMBC referring to a GMCURV, GMBC referring to a FEFACE, and GMBC referring to a GMSURF. **Table 8-21** describes the outcome of using these different methods:

Table 8-21 Enforced Displacement Used for GMCURV 1000

When Specified Using	Action
Single GMBC (GMCURV)	Overrides information supplied for all components using GMBC(FEFACE), GMBC(GMSURF) entries.
Single GMBC (FEFACE)	Overrides information supplied for all components using GMBC(GMSURF) entries.
Multiple GMBC (FEFACE)	Values are averaged on a component basis. The resulting value overrides information supplied for all components using GMBC(GMSURF) entries.
Single GMBC (GMSURF)	Values are applied.
Multiple GMBC (GMSURF)	Values are averaged on a component basis.

- The enforced displacement for the GRID DOFs, the edge DOFs, and the face DOFs belonging to FEFACE 300 can be specified using GMBC referring to a FEFACE and GMBC referring to a GMSURF. **Table 8-22** describes the outcome of using these different methods:

Table 8-22 Enforced Displacement Used for FEFACE 300

When Specified Using	Action
Single GMBC (FEFACE)	Overrides information supplied for all components using GMBC(GMSURF) entries.
Single GMBC (GMSURF)	Values are applied.

GMBNDC Geometric Boundary - Curve

Defines a geometric boundary consisting of either h- or p-element edges along a curve interface. The boundary may consist of edges of shell, beam, or solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
GMBNDC	BID	GRIDI	GRIDF						
	ENTITY	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	-etc.-							

Examples:

GMBNDC	1	101	106						
	GMCURV	1							

GMBNDC	1	101	106						
	FEEDGE	11	12	13	14	15			

GMBNDC	1	101	106						
	GRID	102	103	104	105				

Field	Contents
BID	Boundary identification number to be referenced by a GMINTC entry. (Integer > 0)
GRIDI	Initial grid identification number for boundary. (Integer > 0)
GRIDF	Final grid identification number for boundary. (Integer > 0)
ENTITY	Entity type for defining boundary. (Character)
IDi	Entity identification numbers for boundary of subdomain. (Integer > 0)

Remarks:

1. All boundary identification numbers must be unique.
2. GRIDI and GRIDF define the end points of the boundary.
3. For each boundary, one of the entity types GMCURV, FEEDGE, or GRID is required.

4. For the GMCURV entity type, if there are multiple paths on the GMCURV from the GRIDI to the GRIDF, such as two segments of a circle, the FEEDGE or GRID method must be used instead to uniquely define the path.
5. For the GRID entity type, the entities should be listed in order from the GRIDI to the GRIDF. The GRIDI and GRIDF need not be repeated in the IDi list.
6. If more than one boundary references the same GMCURV entry with the same GRIDI and GRIDF, then the FEEDGE or GRID entity type must be used instead for each to uniquely identify the boundaries.
7. Multiple continuation entries may be specified for additional entity identification numbers, IDi.

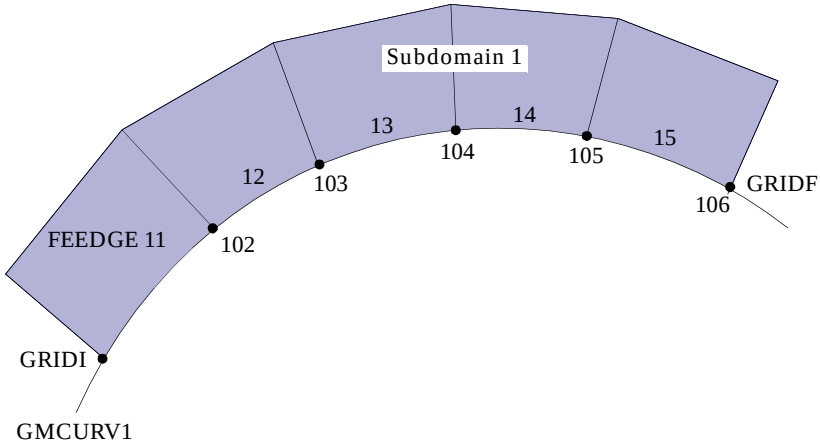


Figure 8-89 Geometric Boundary Definition

8. Interface elements may generate high or negative matrix/factor diagonal ratios. If there are no other modelling errors, these messages may be ignored and PARAM,BAILOUT,-1 may be used to continue the run.
9. When it is referenced by Bulk Data entry, CINTC, which defines an h-interface element, the value of field ENTITY must be GRID.

GMBNDS Geometric Boundary - Surface

Defines a geometric boundary consisting of p-element faces along a surface interface. The boundary may consist of faces of p-solid or p-shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
GMBNDS	BID								
	ENTITY	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	-etc.-							

Examples:

GMBNDS	1								
	GMSURF	1							

GMBNDS	1								
	FEFACE	11	12	13	14	15	16		

GMBNDS	1								
	GRID	101	102	103	104	105	106	107	
	108	109	110	111	112				

Field	Contents	Type	Default
BID	Boundary identification number.	Integer > 0	Required
ENTITY	Entity type for defining boundary.	Character	Required
IDi	Entity ID i for boundary	Integer > 0	Optional

Remarks:

1. All BIDs must be unique.
2. For each boundary, one of the entity types GMSURF, FEFACE, or GRID is required.
3. For the GMSURF entity type, all the faces referencing the GMSURF will be included in the boundary.

4. If more than one boundary references the same GMSURF, then the FEFACE or GRID entity type must be used instead for each to uniquely identify the boundaries.
5. Multiple continuation entries may be used without repeating the ENTITY field.

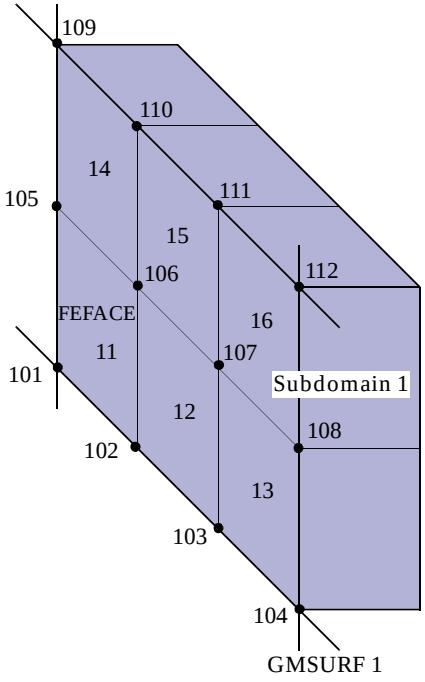


Figure 8-90 Surface Boundary Definition

6. Interface elements may generate high or negative matrix/factor diagonal ratios. If there are no other modelling errors, these messages may be ignored and PARAM,BAILOUT,-1 may be used to continue the run.

GMCONV Define convection boundary conditions.

Defines free convection boundary conditions for GRID, FEEDGE, GMCURV, FEFACE, and GMSURF entries.

Format:

1	2	3	4	5	6	7	8	9	10
GMCONV	LID	ENTITY	ID	METHOD	FIELD1	FIELD2			

Example:

GMCONV	15	FEFACE	3	10	1001	20.0			

Field	Contents	Type	Default
LID	Load set identification number.	Integer > 0	Required
ENTITY	Entity that the convection boundary condition is applied to (specify GRID, FEEDGE, GMCURV, FEFACE, or GMSURF). See Remarks.	Character	Required
ID	ID of the entity selected above. See Remarks.	Character	Required
METHOD	Method used to specify the data. See Remark 2.	Integer $\geq$ 0	0
FIELD1	Convection heat transfer coefficient data. See Remark 2.	Integer or Real	Required
FIELD2	Ambient temperature data. See Remark 2.	Integer or Real	Required

Remarks:

1. For steady-state analysis, the load set is selected in the Case Control Section (LOAD=LID).

2. METHOD specifies the data types of FIELD1 and FIELD2 to be constants, equation IDs, or table IDs. Values in FIELD1 and FIELD2 are:

METHOD	FIELD1	FIELD2
0	Value of heat transfer coefficient (Real > 0.0).	Value of ambient temperature (Real).
1	Value of heat transfer coefficient (Real > 0.0).	ID of a DEQATN entry defining the ambient temperature as a function of location (Integer > 0).
2	Value of heat transfer coefficient (Real > 0.0).	ID of a TABLE3D entry defining the ambient temperature as a function of location (Integer > 0).
10	ID of a DEQATN entry defining the heat transfer coefficient as a function of location (Integer > 0).	Value of ambient temperature (Real).
11	ID of a DEQATN entry defining the heat transfer coefficient as a function of location (Integer > 0).	ID of a DEQATN entry defining the ambient temperature as a function of location (Integer > 0).
12	ID of a DEQATN entry defining the heat transfer coefficient as a function of location (Integer > 0).	ID of a TABLE3D entry defining the ambient temperature as a function of location (Integer > 0).
20	ID of a TABLE3D entry defining the heat transfer coefficient a location (Integer > 0).	Value of ambient temperature (Real).
21	ID of a TABLED3 entry defining the heat transfer coefficient as a function of location (Integer > 0).	ID of a DEQATN entry defining the ambient temperature as a function of location (Integer > 0).
22	ID of a TABLE3D entry defining the heat transfer coefficient as a function of location (Integer > 0).	ID of a TABLE3D entry defining the ambient temperature as a function of location (Integer > 0).

3. The proper units must be specified for the value of FIELD1.

Units of FIELDi for Different ENTITY Fields	
ENTITY	**Units**
GRID	Power / Degree
FEEDGE	Power / Length-Degree
GMCURV	Power / Length-Degree
FEFACE	Power / Area-Degree
GMSURF	Power / Area-Degree

4. Multiple values of convection boundary conditions can be applied to the same geometry region. In general, a hierarchy is set to resolve the conflicts arising in the input data:
   a. Information provided on multiple GMSURF and FEFACE entries are added for all FEFACE entries.
   b. Information provided on multiple GMCURVE and FEEDGE entries are added for all FEEDGE entries.

GMCORD Convective/Follower Coordinate System Definition

Defines a convective/follower coordinate system on an FEEDGE, GMCURV, FEFACE, or GMSURF entry.

Format:

1	2	3	4	5	6	7	8	9	10
GMCORD	CID	ENTITY	ID1	ID2					

Example:

GMCORD	101	GMCURV	26	44					

Field	Contents	Type	Default
CID	Coordinate system identification number, unique with respect to all CORDij entries.	Integer > 0	Required
ENTITY	Type of Bulk Data entry that is used to define the coordinate system. See Remark 3.	Character	Required
ID1,ID2	Entity identification numbers. See Remark 3.	Integer > 0	Required

Remarks:

1. GMCORD defines a (convective) coordinate system associated with an entity. This type of coordinate system can be used to apply loads and boundary conditions only.
2. GMCORD can only be specified for p-version elements.
3. The Bulk Data entries referenced by ID1 and ID2 depends on ENTITY.

ENTITY	ID1	ID2
FEEDGE	FEEDGE entry ID	FEFACE entry ID
GMCURV	GMCURV entry ID	GMSURF entry ID
FEFACE	FEFACE entry ID	Blank
GMSURF	GMSURF entry ID	Blank

- For ENTITY = “FEEDGE” normal is defined by the FEFACE.
- For ENTITY = “GMCURV” normal is defined by the GMSURF.

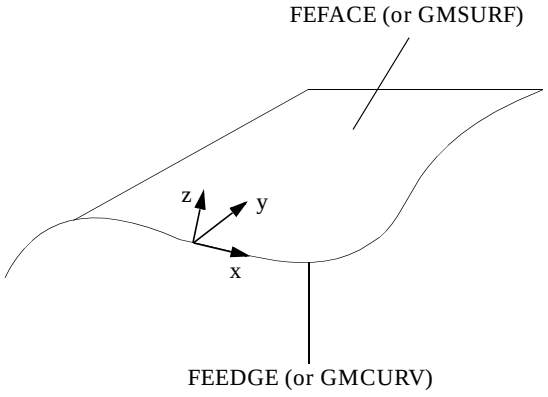


Figure 8-91

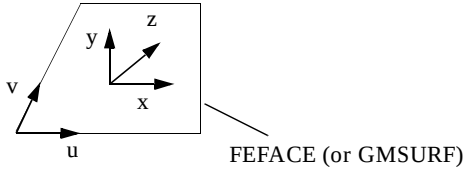


Figure 8-92

GMCURV Curve Definition

Defines geometric curve that will be used in element geometry, load definition, and boundary condition definition.

Format:

1	2	3	4	5	6	7	8	9	10
GMCURV	CURVID	GROUP	CIDIN	CIDBC					
	Evaluator	Specific	Data	and	Format				

Example:

GMCURV	101	FENDER							
	RPC	POINT							
	0.0, 2.0	1.0	1.0		2.0	3.0	4.0,1.0		
	0.0, 2.0	1.0	1.0	1.0	2.0	3.0	4.0		

Field	Contents	Type	Default
CURVID	Unique identification number. See Remarks **1.** and **2.**	Integer > 0	Required
GROUP	Group of curves / surfaces that this curve belongs to. See Remarks **4.** through **8.**	Character	Required
CIDIN	Coordinate system identification number used in defining the geometry of the curve. The coordinate system must be rectangular.	Integer ≥ 0	0
CIDBC	Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined.	Integer ≥ 0	0

Remarks:

1. GMCURV is used to calculate geometric information only. The edges of the finite elements that are connected to the curve will be parametric cubic curves that are calculated from the more complex curve geometry.

2. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
3. The continuation entries are passed directly to the geometry evaluator indicated by the GROUP parameter.
4. The GROUP parameter is initialized by an CONNECT GEOMEVAL statement in the FMS Section. This command specifies the evaluator that will be used for this curve.
5. Two reserved names, MSCGRP0 and MSCGRP1, are provided for the GROUP parameter. These need not be explicitly initialized on the CONNECT FMS statement.
6. If the GROUP parameter is specified as MSCGRP0, the MSC rational parametric cubic (MSCRPC) geometry evaluator is used for this curve. In this case the evaluator specific data in lines 2 through 4 of this Bulk Data entry should be provided as given below. Spaces or a comma character may be used to delimit each value. However, a comma must not be specified in the first field.

1	2	3	4	5	6	7	8	9	10
	RPC	REPRES							
	XW(1)	XW(2)	XW(3)	XW(4)	YW(1)	YW(2)	YW(3)	YW(4)	
	ZW(1)	ZW(2)	ZW(3)	ZW(4)	W(1)	W(2)	W(3)	W(4)	

Field	Contents	Type	Default
RPC	Rational Parametric Cubic Curve.	Character	Required
REPRES	Representation of the curve (“ALGEBRAIC”, “POINT”, “BEZIER”).	Character	Required
XW(1) through W(4)	Data used to define the curve.	Real	Required

- A rational parametric curve (RPC) is defined as

$$x(t) = \frac{xw(t)}{w(t)}$$

$$y(t) = \frac{yw(t)}{w(t)}$$

$$z(t) = \frac{zw(t)}{w(t)}$$

$$0.0 \le t \le 1.0$$

- For REPRES = “ALGEBRAIC”, the parametric curve is defined by the algebraic coefficients (*a, b, c, d*) for a rational cubic equation.

$$P(t) = at^3 + bt^2 + ct + d$$

Expressed in matrix form:

$$P(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} a \\ b \\ c \\ d \end{bmatrix}$$

for the Bulk Data input as algebraic coefficients

$$xw(t) = XW(1)t^3 + XW(2)t^2 + XW(3)t + XW(4)$$

$$yw(t) = YW(1)t^3 + YW(2)t^2 + YW(3)t + YW(4)$$

$$zw(t) = ZW(1)t^3 + ZW(2)t^2 + ZW(3)t + ZW(4)$$

$$w(t) = W(1)t^3 + W(2)t^2 + W(3)t + W(4)$$

where $XW(i)$, $YW(i)$, $ZW(i)$, and $W(i)$ are the algebraic coefficients for each of the independent equations $xw(t)$, $yw(t)$, $zw(t)$, and $w(t)$.

and

$$x(t) = \frac{xw(t)}{w(t)}; y(t) = \frac{yw(t)}{w(t)}; z(t) = \frac{zw(t)}{w(t)}$$

- For REPRES = “BEZIER”, the curve parametric is defined by four rational Bezier control points (V_1, V_2, V_3, and V_4) expressed in matrix form

$$P(t) = V_1(1-t)^3 + V_2 3t(1-n)^2 + V_3 3t^2(1-t) + V_4 t^3$$

$$(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}$$

where Bezier constants are $\begin{bmatrix} -1 & 3 & -3 & 1 \\ 3 & -6 & 3 & 0 \\ -3 & 3 & 0 & 0 \\ 1 & 0 & 0 & 0 \end{bmatrix}$

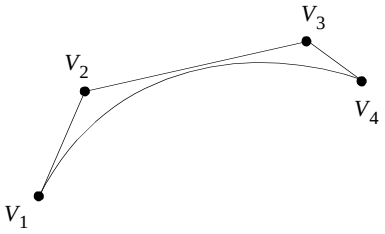


Figure 8-93 REPRES = "BEZIER"

for Bulk Data defined as Bezier control points

$$w(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} XW(1) \\ XW(2) \\ XW(3) \\ XW(4) \end{bmatrix}$$

$$yw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} YW(1) \\ YW(2) \\ YW(3) \\ YW(4) \end{bmatrix}$$

$$w(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} ZW(1) \\ ZW(2) \\ ZW(3) \\ ZW(4) \end{bmatrix}$$

$$(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Bezier} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} W(1) \\ W(2) \\ W(3) \\ W(4) \end{bmatrix}$$

where $XW(i)$, $YW(i)$, $ZW(i)$, and $W(i)$ correspond to V_i, and

$$x(t) = \frac{xw(t)}{w(t)}; y(t) = \frac{yw(t)}{w(t)}; z(t) = \frac{zw(t)}{w(t)}$$

- For REPRES = “POINT”, the parametric curve is defined by four uniformly spaced rational points that are all on the curve similarly expressed in matrix form:

$$(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}$$

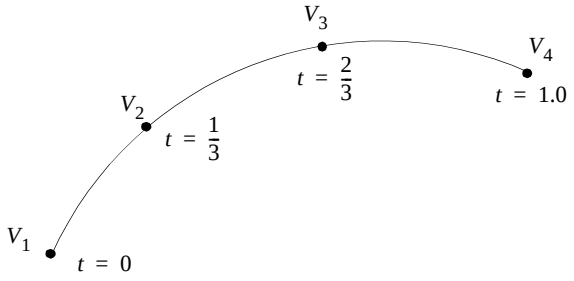


Figure 8-94 REPRES = “POINT”

for the Bulk Data input are uniformly spaced rational points

$$xw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} XW(1) \\ XW(2) \\ XW(3) \\ XW(4) \end{bmatrix}$$

$$yw(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} YW(1) \\ YW(2) \\ YW(3) \\ YW(4) \end{bmatrix}$$

$$w(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} ZW(1) \\ ZW(2) \\ ZW(3) \\ ZW(4) \end{bmatrix}$$

$$(t) = \begin{bmatrix} t^3 & t^2 & t & 1 \end{bmatrix} \begin{bmatrix} \text{Point} \\ \text{Constants} \end{bmatrix} \begin{bmatrix} W(1) \\ W(2) \\ W(3) \\ W(4) \end{bmatrix}$$

$$\text{and } x(t) = \frac{xw(t)}{w(t)}; y(t) = \frac{yw(t)}{w(t)}; z(t) = \frac{zw(t)}{w(t)}$$

7. If the GROUP parameter is specified as MSCGRP1, the MSC generic equation (MSCEQN) geometry evaluator is used for this curve. In this case the evaluator specific data in lines 2 through 3 of this Bulk Data entry should be provided as given below. Spaces or a comma character may be used to delimit each value. However, the comma character should not be used in the first “field”.

1	2	3	4	5	6	7	8	9	10
	EQUATION, MINU	MAXU	IDX	IDY	IDZ	IDDXU	IDDYU		
	IDDZU	IDDXU2	IDDYU2	IDDZU2					

Field	Contents	Type	Default
EQUATION	EQUATION method is to be used	Character	
MINU, MAXU	Range of the curve parameter *u*. If MAXU is found less than MINU, the range is assumed to be [-∞, +∞].	Real	0.0,1.0
IDX, IDY, IDZ	ID of DEQATN entries providing equations for the X,Y,Z coordinates of the curve in terms of the curve parameter *u*.	Integer > 0	Required

Field	Contents	Type	Default
IDDXU, IDDYU, IDDZU	ID of DEQATN entries providing equations for the first derivatives of X,Y,Z functions with respect to the curve parameter *u*. If a value of 0 is used, the derivatives are computed numerically.	Integer ≥ 0	0
IDDXU2, IDDYU2, IDDZU2	ID of DEQATN entries providing equations for the second derivatives of X,Y,Z functions with respect to the curve parameter *u*. If a value of 0 is used, the second derivatives are computed numerically.	Integer > 0	0

8. When a user-supplied geometry evaluator is selected for this curve (through the CONNECT GEOMEVAL FMS command) the continuation entries will not be interpreted**.** In this case an image of this entry is passed on to the evaluator modules provided by the user. Depending on the configuration, these modules could either be linked in with MD Nastran or connected with MD Nastran during execution. If these modules are not accessible, a User Fatal Message will be issued. For example, if in the FMS Section, the following command is given:

   - CONNECT GEOMEVAL FENDER,CATIA,'/u/kiz/data', Version = 68 as of 1/3/94
   - and the GMCURV Bulk Data entry is provided as follows:

1	2	3	4	5	6	7	8	9	10
GMCURV	102	FENDER							
	Sweep	/u/kiz	2.5	arc	2.7	66			

   - In this case, "Sweep /u/kiz 2.5 arc 2.7 66" is passed to the geometry evaluator supplied by the user, and it is expected that the user supplied routines to interpret and use this record.

GMINTC Geometric Interface -- Curve

Defines an interface element along a curve interface between boundaries of multiple subdomains. Typically, the boundaries will consist of edges of p-shell subdomains but also may consist of p-beam subdomains or edges of p-solid subdomains.

Format:

1	2	3	4	5	6	7	8	9	10
GMINTC	EID	PID	ID1	ID2	ID3	ID4	ID5	ID6	

Example:

GMINTC	1001	1	1	2					

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PINTC property entry. (Integer > 0)
IDi	Boundary identification number of a GMBNDC entry. (Integer > 0)

Remarks:

1. All element identification numbers must be unique.
2. For the curve interface it is recommended that only two boundaries be specified.
3. All of the end points for each boundary IDi should be coincident, and may not refer to the same grid point. The two end points of a particular boundary may not refer to the same grid, because there would be multiple directions. The boundaries of each of the subdomains should also be coincident, because no geometrical adjustment is performed.
4. Connecting curve boundaries of solid p-elements is not recommended because of the possibility of stress singularities.

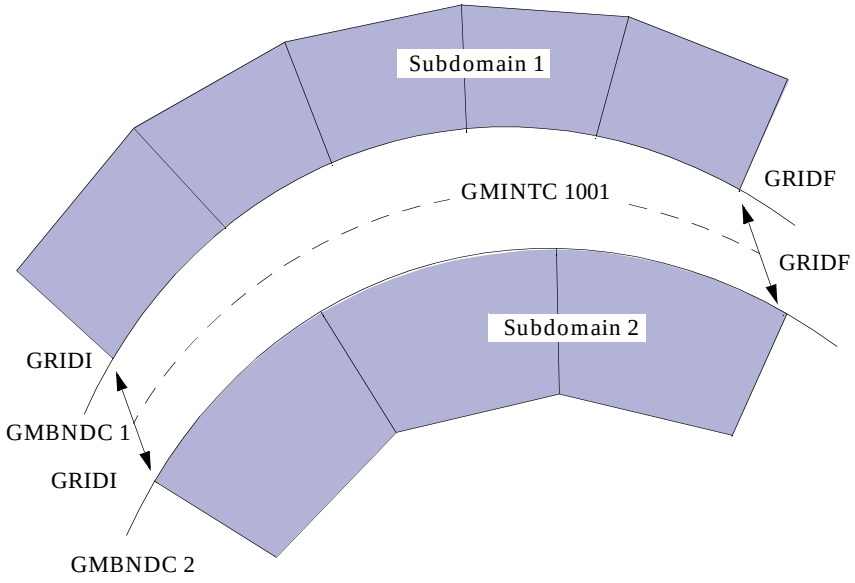


Figure 8-95 Geometric Interface Element Definition (Exploded View)

5. Because of the structure of the interface matrices, the sparse solver (default) should be used for linear statics, and the Lanczos eigensolver should be used for normal modes. In addition, for normal modes, SYSTEM(166) = 4 should be set for models where the shell normal rotations are parallel on the boundaries.

GMINTS Geometric Interface -- Surface

Defines an interface element along a surface interface between boundaries of multiple subdomains. Typically, the boundaries will consist of faces of p-solid subdomains, but also may consist of p-shell subdomains.

Format:

1	2	3	4	5	6	7	8	9	10
GMINTS	EID	PID	ID1	ID2	ID3	ID4			

Example:

GMINTS	1001	1	1	2					

Field	Contents	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number.	Integer > 0	Required
IDi	Boundary IDi of subdomain	Integer > 0	Required

Remarks:

1. All EIDs must be unique.
2. The PID refers to a PINTS Bulk Data entry.
3. The boundary IDi of each subdomain must be defined on a GMBNDS Bulk Data entry.
4. For the surface interface, more than two boundaries are possible, but should be used carefully.
5. The perimeters of each boundary i should be coincident. In addition, the boundaries of each of the subdomains should also be coincident, because no geometrical adjustment is performed.

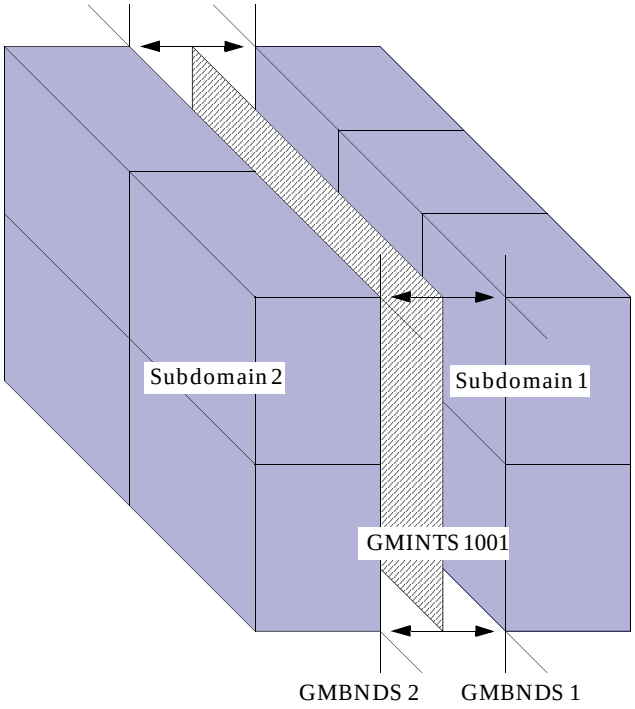


Figure 8-96 Geometric Interface Element Definition (Exploded View)

GMLOAD General Load Definition

Define the forces and moments to be applied to a FEEDGE, GMCURV, FEFACE, or GMSURF entry.

Format:

1	2	3	4	5	6	7	8	9	10
GMLOAD	LID	CID	N1	N2	N3	ENTITY	ID	METHOD	
	FIELD1	FIELD2	FIELD3	FIELD4	-etc.-				

Example:

GMLOAD	105	11	1.2		1.	1.	1		
		7.5	1.9						

Field	Contents	Type	Default
LID	Load set identification number.	Integer > 0	Required
CID	Coordinate system in which the load is supplied. See Remark 2.	Integer ≥ -1	0
Ni	Direction of the force vector or axis of rotation of the moment. See Remark 3.	Real	0., 0., 1.
ENTITY	Entity that is being loaded (FEEDGE, GMCURV, FEFACE, GMSURF).	Character	Required
ID	ID of the entity selected by ENTITY.	Integer ≥ 0	Required
METHOD	Method used to specify forces (EQUATION, TABLE, CONSTANT, LINEAR, QUAD, CUBIC) or moments (MEQUA, MTABLE, MCONST, MLINEAR, MQUAD, MCUBIC). See Remarks 4. through 6.	Character	Required
FIELDi	Load magnitude data. See Remarks 4. through 8.	Real or Integer	

Remarks:

1. GMLOAD is the only method of applying forces and moments to any FEEDGE, FEFACE, GMCURV, or GMSURF in the model.

2. If CID=-1, the coordinate system on the edge or face is a local system based on the FEEDGE or FEFACE definition. (Note that an edge only has the tangent direction uniquely defined.)
3. If N1=N2=N3=0., the normal direction to the face is assumed, with the positive sense dependent on the FEFACE definition. No load will be applied for edges.
4. For both an FEEDGE and FEFACE, the METHOD field can be used to specify equation, table or constant load density. The value of FIELD1 is method-specific:

Applying Equation, Table or Constant Load Density	
METHOD	**FIELD1**
EQUATION, MEQUA	ID of a DEQATN entry defining the load density as a function of location.
TABLE, MTABLE	ID of a TABLE3D entry defining the load density as a function of location.
CONSTANT, MCONST	Value of load density.

5. For an FEEDGE, the METHOD field can be used to specify linear, quadratic or cubic load density. The values of FIELDi are method-specific:

Applying Linear, Quadratic or Cubic Load Density to an FEEDGE					
METHOD	**FIELD1**	**FIELD2**	**FIELD3**	**FIELD4**	**Load Density**
LINEAR, MLINEAR	Value at GRID 1	Value at GRID 2	blank	blank	Linear
QUAD, MQUAD	Value at GRID 1	Value at GRID 2	Value at 1/2 edge length	blank	Quadratic
CUBIC, MCUBIC	Value at GRID 1	Value at GRID 2	Value at 1/3 edge length	Value at 2/3 edge length	Cubic

6. For an FEFACE, the METHOD field can be used to specify linear or quadratic load density. The edges of the face are defined in the order of the grids entered (e.g., edge 1 is between the first and second grid etc.). The values of FIELDi are method-specific:

Applying Linear and Quadratic Load Density to a Quadrilateral FEFACE				
METHOD	**FIELD1 through FIELD4**	**FIELD5 through FIELD8**	**FIELD9**	**Load Density**
LINEAR,MLINEAR	Value at GRID 1, 2, 3, 4	blank	blank	Linear
QUAD, MQUAD	Value at GRID 1, 2, 3, 4	Value at midside of EDGE 1,2,3,4	Value at middle of FEFACE	Quadratic

Applying Linear and Quadratic Load Density to a Triangular FEFACE			
METHOD	**FIELD1 through FIELD3**	**FIELD4 through FIELD6**	**Load Density**
LINEAR, MLINEAR	Value at GRID 1, 2, 3	blank	Linear
QUAD, MQUAD	Value at GRID 1, 2, 3	Value at midside of EDGE 1, 2, 3	Quadratic

7. The proper units must be specified for the value of FIELDi.

Units of FIELDi for Different ENTITY Fields	
ENTITY	**Units**
FEEDGE	Load / Length
GMCURV	Load / Length
FEFACE	Load / Area
GMSURF	Load / Area

8. The load density applied to the edge or face is given by the product of the specified density with the direction vector.

9. The shell p-elements do not have stiffness in the direction of the normal rotation. Any component of moment applied in that direction will be ignored.
10. In general, a hierarchy is set to resolve the conflicts arising in the input data:
    - Information provided on multiple GMSURF and FEFACE entries are added for all GRID, FEEDGE, and FEFACE degrees-of-freedom.
    - Information provided on multiple GMCURV and FEEDGE entries are added for all GRID and FEEDGE degrees-of-freedom.
    - Loads are summed over all entities.

GMNURB (SOL 600) 3D Contact Region Made Up of NURBS

Defines a 3D contact region made up of NURBS using the MSC.Marc style used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
GMNURB	ID	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV	NTRIM	
		G1 or X1	G2 or Y1	G3 or Z1	G4 or X2	G5 or Y2	G6 or Z2	G7	
		G8 or X3	G9 or Y3	G10 or Z3	etc.		[abs(nptu)*nptv vaues]	See Remark 3	
		Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7	
		Homo8	Homo9	Homo10	Homo11	etc	[nptu*nptv values]		
		Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	
		Knot8	Knot9	Knot10	etc.		[(nptu+noru)+(nptv+norv) values]		
		IDtrim	NPTUtrim	NORUtrim	NSUBtrim		(repeat this and all following lines NTRIM times)		
			Xisoparam	Yisoparam			(NPTUtrim entries)		
			Homo1	Homo2	Homo3	etc	(NPTUtrim entries)		
			Knot1	Knot2	Knot3	etc	(NPTUtrim + NORUtrim entries)		

Field	Contents
ID	Identification number of a surface defined by NURBS. ID is called out on a BCBODY entry with a NURBS2 header. (Integer > 0; required)
NPTU	Absolute value of the number of control points. Enter NPTU as a positive number if the control points are to be input using GRID points. Enter NPTU as a negative number if the control points are to be entered using x,y,z. (Integer > 0; required)
NPTV	Number of control points in V direction. (Integer > 0; required)
NORU	Order along U direction. (Integer > 0; required)
NORV	Order along V direction. (Integer > 0; required)
NSUBU	Number of subdivisions in U direction. (Integer > 0; required)

Field	Contents
NSUBV	Number of subdivisions in V direction. (Integer > 0; required)
NTRIM	Number of trimming curves. (Integer ≥ 0 or blank)
G1, G2, G3, etc.	Grid point IDs defining control points (Integer > 0, required). There must be NPTU*NPTV entries.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) in the V direction. There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)
IDtrim	ID of trimming vector. There must NTRIM of these entries and those entries that follow. (Integer > 0)
NPUTtrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
Xisoparam	First coordinate of point in isoparametric space. (Real)
Ysoparam	Second coordinate of point in isoparametric space (Real)
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)

Remarks:

1. GMNURB is recognized only in MD Nastran Implicit Nonlinear (SOL 600).
2. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e. in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will

not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.

3. For NURBS, enter NPTU grid points G1, G2, G3 etc. (set NPTU to a positive value equal to the number of grid points or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for abs(NPTU) points and set NPTU to a negative value).

GMQVOL Define volumetric heat loads.

Defines a rate of volumetric heat generation in a conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
GMQVOL	LID	METHOD	FIELD1		EID1	EID2	EID3	EID4	
	EID5	etc.							

Example:

GMQVOL	100	TABLE	20		18	23	7		

Field	Contents	Type	Default
LID	Load set identification number.	Integer > 0	Required
METHOD	Method used to specify the data (EQUATION, TABLE, CONSTANT). See Remark 2.	Character	Required
FIELD1	Volumetric heat load data. See Remark 2.	Integer or Real	Required
EIDi	A list of heat conduction elements.	Integer > 0	Required

Remarks:

1. For steady-state analysis, the load set is selected in the Case Control Section (LOAD=LID).
2. METHOD specifies the data type of FIELD1 to be constants, equation IDs, or table IDs. Values in FIELD1 and FIELD2 are:

METHOD	FIELD1
EQUATION	ID of a DEQATN entry defining the volumetric heat generation rate as a function of location (Integer > 0).
TABLE	ID of a TABLE3D entry defining the volumetric heat generation rate as a function of location (Integer > 0).
CONSTANT	Value of volumetric heat generation rate (Real).

Note that the fifth field will be reserved for the future development of temperature dependent functions.

GMSPC General Constraint Definition

Defines constraints for entities.

Format:

1	2	3	4	5	6	7	8	9	10
GMSPC	SID	C	ENTITY	ID					

Example:

GMSPC	12	1	FEEDGE	109					

Field	Contents	Type	Default
SID	SPC set identification number.	Integer > 0	Required
C	Component number in the global coordinate system.	$0 \leq$ Integer ≤ 6	0
ENTITY	Entity that the enforced displacement is applied to (Specify GRID, FEEDGE, GMCURV, FEFACE, or GMSURF). See Remark 4.	Character	Required
ID	ID of the entity selected above.	Integer > 0	Required

Remarks:

1. The components of motion specified by C (field 3) of all degrees-of-freedom associated with an entity will be constrained.
2. If C = 0 is specified then the degrees-of-freedom will be taken out of the constraint set.
3. The component C has to be a single integer (1 or 2 or 3, etc.). Use multiple GMSPC entries for constraining multiple components.
4. In general, the hierarchy set to resolve the conflicts arising in the enforced displacement input data is the same as for the constraints. See Remark 10 under “GMBC” on page 1576 for a description of the hierarchy.

GMSURF Surface Definition

Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.

Format:

1	2	3	4	5	6	7	8	9	10
GMSURF	SURFID	GROUP	CIDIN	CIDBC					
	Evaluator Specific Data and Format								

Example:

GMSURF	101	MSCGRP0							
	RPC,	POINT							
	0.0, 2.0	1.0	1.0		2.0	3.0	4.0,1.0		
	0.0, 2.0	1.0	1.0	1.0	2.0	3.0	4.0		

Field	Contents	Type	Default
SURFID	Surface Identification number. See Remark 2.	Integer > 0	Required
GROUP	Group of curves / surfaces that this surface belongs to. See Remarks 5. through 9.	Character	Required
CIDIN	Coordinate system identification number used in defining the geometry of the curve. The coordinate system must be rectangular.	Integer $\geq$ 0	0
CIDBC	Identification number of the coordinate system in which constraints specified on GMBC and GMSPC entries are defined.	Integer $\geq$ 0	0

Remarks:

1. All SURFIDs must be unique.
2. A GMSURF entry is required if:
   - the geometry of the surface defined by this entry is to be used by an element.
   - output (global) coordinate system is assigned to a GMSURF.
   - permanent constraints are specified for a GMSURF.

- loads are applied to a GMSURF.
- enforced boundary conditions are applied to a GMSURF.

3. GMSURF is used to calculate geometric information only. The edges of the finite elements that are connected to the surface will be parametric cubic curves that are calculated from the more complex surface geometry.
4. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
5. The continuation entries are passed directly to the geometry evaluator indicated by the GROUP parameter.
6. The GROUP value is initialized by an CONNECT GEOMEVAL command in the FMS section. This command specifies the evaluator that will be used for this surface.
7. Two reserved names, MSCGRP0 and MSCGRP1, are provided for the GROUP parameter. These need not be explicitly initialized in the FMS Section.
8. If the GROUP parameter is specified as MSCGRP0, the MSC rational parametric cubic (MSCRPC) geometry evaluator is used for this surface. In this case the evaluator specific data in lines 2 through 9 of this Bulk Data entry should be provided as given on the following page. Spaces or a comma character may be used to delimit each value. However, the comma character should not be used in the first field.

1	2	3	4	5	6	7	8	9	10
GMSURF	SURFID	MSCGRP0	CIDIN	CIDOUT					
	RPC	REPRES							
	XW(1)	XW(2)	XW(3)	XW(4)	XW(5)	XW(6)	XW(7)	XW(8)	
	XW(9)	XW(10)	XW(11)	XW(12)	XW(13)	XW(14)	XW(15)	XW(16)	
	YW(1)	YW(2)	YW(3)	YW(4)	YW(5)	YW(6)	YW(7)	YW(8)	
	YW(9)	YW(10)	YW(11)	YW(12)	YW(13)	YW(14)	YW(15)	YW(16)	
	ZW(1)	ZW(2)	ZW(3)	ZW(4)	ZW(5)	ZW(6)	ZW(7)	ZW(8)	
	ZW(9)	ZW(10)	ZW(11)	ZW(12)	ZW(13)	ZW(14)	ZW(15)	ZW(16)	
	W(1)	W(2)	W(3)	W(4)	W(5)	W(6)	W(7)	W(8)	
	W(9)	W(10)	W(11)	W(12)	W(13)	W(14)	W(15)	W(16)	

Field	Contents	Type	Default
RPC	Rational Parametric Cubic Surface.	Character	Required
REPRES	Representation of the curve, (ALGEBRAIC, POINT, BEZIER).	Character	Required
XW(1) through XW(16)	Data used to define the surface.	Real	Required

- A rational parametric surface is defined as

$$x(u, v) = \frac{xw(u, v)}{w(u, v)}$$

$$y(u, v) = \frac{yw(u, v)}{w(u, v)}$$

$$z(u, v) = \frac{zw(u, v)}{w(u, v)}$$

$$0.0 \le u \le 1.0$$

$$0.0 \le v \le 1.0$$

- For REPRES = “ALGEBRAIC”, the rational parametric surface is defined by the algebraic coefficient for rational cubic equations.
- Expressed as a tensor product

$$_{(u, v)} = \begin{bmatrix} u^3 & u^2 & u & 1 \end{bmatrix} [\text{algebraic coefficients}] \begin{Bmatrix} v^3 \\ v^2 \\ v \\ 1 \end{Bmatrix}$$

for the Bulk Data input in algebraic form

$$xw(u, v) = \begin{bmatrix} u^3 & u^2 & u & 1 \end{bmatrix} \begin{bmatrix} XW(1) & XW(2) & XW(3) & XW(4) \\ XW(5) & XW(6) & XW(7) & XW(8) \\ XW(9) & XW(10) & XW(11) & XW(12) \\ XW(13) & XW(14) & XW(15) & XW(16) \end{bmatrix} \begin{Bmatrix} v^3 \\ v^2 \\ v \\ 1 \end{Bmatrix}$$

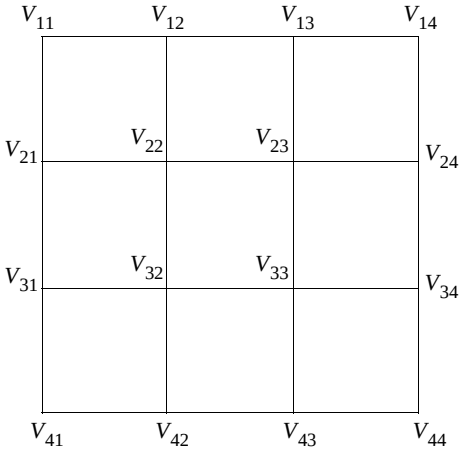
and similarly for $yw(u, v)$, $zw(u, v)$, and $w(u, v)$.

and

$$x(u, v) = \frac{xw(u, v)}{w(u, v)}; y(u, v) = \frac{yw(u, v)}{w(u, v)}; z(u, v) = \frac{zw(u, v)}{w(u, v)}$$

where $XW(i)$, $YW(i)$, $ZW(i)$, and $W(i)$ are the algebraic coefficients for the independent equations $xw(u, v)$, $yw(u, v)$, $zw(u, v)$, and $w(u, v)$.

- For REPRES = “BEZIER”, the surface is defined by 16 rational Bezier control points V_{11} through V_{44}.



and

$$P(u, v) = \sum_{i=1}^{4} \sum_{i=1}^{4} V_{ij} B_{i,4} u \cdot B_{j,4} v$$

where $B_{i,4}u$ and $B_{j,4}v$ are the Bernstein polynomials for curves of degree 3.

For Bulk Data input defined as Bezier control points

$$xw(u, v) = \sum_{i=1}^{4} \sum_{j=1}^{4} XWi, jB_{i,4} \cdot B_{j,4} v$$

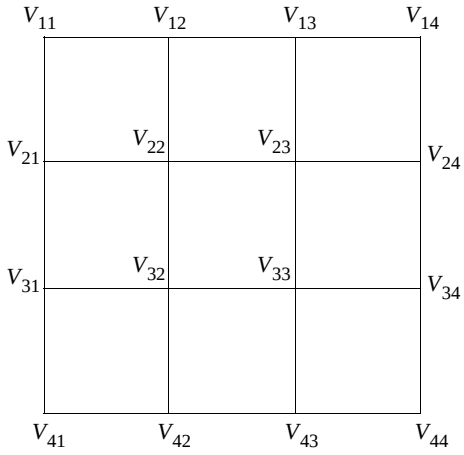
where $XW(1)$ through $XW(16)$ are ordered to conform to the two-dimensional array for V_{ij}; that is, $XW(4 \cdot (i - 1) + j)$ corresponds to $xw(u, v)$ for V_{ij}. For example, $XW(7)$ corresponds to V_{23}.

$yw(u, v)$; $zw(u, v)$; and $w(u, v)$ are solved in a similar fashion.

and

$$x(u, v) = \frac{xw(u, v)}{w(u, v)}; y(u, v) = \frac{yw(u, v)}{w(u, v)}; z(u, v) = \frac{zw(u, v)}{w(u, v)}$$

- When the point representation is used, the surface is defined by 16 uniformly spaced rational points lying on the surface.



where $XW(1)$ through $XW(16)$ are ordered to contain to the two-dimensional mapping above for V_{ij}; that is, $XW(4 \cdot (i - 1) + j)$ corresponds to $xw(u, v)$ for V_{ij}. For example, $XW(7)$ corresponds to V_{23}.

9. If the GROUP parameter is specified as MSCGRP1, the MSC generic equation (MSCEQN) geometry evaluator is used for this surface. In this case the evaluator specific data should be on the continuation entries. Spaces or a comma character may be used to delimit each value. However, a comma must not be specified in the first field.

1	2	3	4	5	6	7	8	9	10
	EQUATION, MINU		MAXU	MINV	MAXV	IDX	IDY	IDZ	
	IDDXU	IDDYU	IDDZU	IDDXV	IDDYV	IDDZV	IDDXU2	IDDYU2	
	IDDZU2	IDDXV2	IDDYV2	IDDZV2	IDDXUV	IDDYUV	IDDZUV		

Field	Contents	Type	Default
EQUATION	EQUATION method is to be used.	Character	
MINU, MAXU	Range of the first parameter describing the surface. If MAXU is found less than MINU, the range for U is assumed to be [-∞,+∞].	Real	0.0,1.0
MINV, MAXV	Range of the second parameter describing the surface. If MAXV is found less than MINV, the range is assumed to be [-∞,+∞].	Real	0.0,1.0
IDX, IDY, IDZ	ID of DEQATN entries providing equations for the X,Y,Z coordinate of the surface in terms of two parameters *u* and *v*.	Integer > 0	Required
IDDXU, IDDYU, IDDZU	ID of DEQATN entries providing equations for the first derivatives of X,Y,Z functions with respect to the first surface parameter *u*. If a value of 0 is used, the derivatives are computed numerically.	Integer $\geq$ 0	0
IDDXV, IDDYV, IDDZV	ID of an DEQATN entry describing the first derivatives of X, Y, Z functions with respect to the first surface parameter *v*. If a value of 0 is used, the derivatives are computed numerically.	Integer $\geq$ 0	0
IDDXU2, IDDYU2, IDDZU2	ID of an DEQATN entry describing the second derivatives of X,Y,Z functions with respect to the first surface parameter *u*. If a value of 0 is used, the derivatives are computed numerically.	Integer $\geq$ 0	0

Field	Contents	Type	Default
IDDXV2, IDDYV2, IDDZV2	ID of an DEQATN entry describing the second derivatives of X,Y,Z functions with respect to the second surface parameter *v*. If a value of 0 is used, the derivatives are computed numerically.	Integer ≥ 0	0
IDDXUV, IDDYUV, IDDZUV	ID of an DEQATN entry describing the mixed second derivatives of X,Y,Z functions with respect to the surface parameters *u*, and *v*. If a value of 0 is used, the derivatives are computed numerically.	Integer ≥ 0	0

10. When an external geometry evaluator class is selected for this group (which is the case when the CONNECT GEOMEVAL statement selects an external geometry evaluator for the specified group), the data in Fields 1 to n will not be interpreted. In this case an image of this entry is passed on to the evaluator modules provided by the user for the specific geometric package that being used. These modules are connected with MD Nastran during execution. If these modules are not provided, a User Fatal Message will be issued. For example, if in the FMS Section, the following command is given:

    - CONNECT GEOMEVAL FENDER,CATIA,'/u/kiz/data', Version=68 as of 1/3/94
    - then the GMSURF entry could use that geometry data base as follows:

1	2	3	4	5	6	7	8	9	10
GMSURF	765	FENDER							
	Extrude	/u/kiz	2.5	arc	2.7	66			

    - In this case, "Extrude u/kiz 2.5 arc 2.7 66" is passed to the geometry evaluator supplied by the user, and it is expected that the user-supplied routines interpret and use this record.

GRAV Acceleration or Gravity Load

Defines acceleration vectors for gravity or other acceleration loading.

Format:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	A	N1	N2	N3	MB		

Example:

GRAV	1	3	32.2	0.0	0.0	-1.0			

Field	Contents
SID	Set identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer $\geq$ 0; Default = 0)
A	Acceleration vector scale factor. (Real)
Ni	Acceleration vector components measured in coordinate system CID. (Real; at least one Ni $\neq$ 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark **10.** (Integer; Default = 0)

Remarks:

1. The acceleration vector is defined by $\vec{a} = A\vec{N}$, where $\vec{N}$ is the vector defined by (N1, N2, N3). The magnitude of $\vec{a}$ is equal to A times the magnitude of $\vec{N}$. The static loads generated by this entry are in the direction of $\vec{a}$.
2. A CID of zero references the basic coordinate system.
3. Acceleration or gravity loads may be combined with “simple loads” (e.g., FORCE, MOMENT) only by specification on a LOAD entry. That is, the SID on a GRAV entry may not be the same as that on a simple load entry.
4. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD = SID. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

5. At most nine GRAV entries can be selected in a given run either by Case Control or the LOAD Bulk Data entry. Multiples or reflections of a given acceleration or gravity load can be economically accomplished by use of the LOAD Bulk Data entry.
6. In cyclic symmetry solution sequences, the T3 axis of the coordinate system referenced in field 3 must be parallel to the axis of symmetry. In dihedral cyclic symmetry (where STYPE = “DIH” on the CYSYM entry), the T1 axis must, in addition, be parallel to Side 1 of segment 1R of the model.
7. For image superelements, the coordinate system must be rotated if the image is rotated relative to its primary superelement.
8. Acceleration or gravity loads do not include effects due to mass on scalar points.
9. The RFORCE entry may be used to specify rotational accelerations.
10. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.

GRDSET GRID Entry Defaults

Defines default options for fields 3, 7, 8, and 9 of all GRID entries.

Format:

1	2	3	4	5	6	7	8	9	10
GRDSET		CP				CD	PS	SEID	

Example:

GRDSET		16				32	3456		

Field	Contents
CP	Identification number of coordinate system in which the location of the grid points are defined. (Integer $\geq$ 0 or blank)
CD	Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors of the grid point are defined. (Integer $\geq$ 0 or blank)
PS	Permanent single-point constraints on the grid point. (Any combination of Integers 1 through 6 with no embedded blanks, or blank.)
SEID	Superelement identification number. (Integer $\geq$ 0 or blank)

Remarks:

1. The contents of fields 3, 7, 8, or 9 of this entry are assumed for the corresponding fields of any GRID entry whose field 3, 7, 8, and 9 are blank. If any of these fields on the GRID entry are blank, the default option defined by this entry occurs for that field. If no permanent single-point constraints are desired, one of the coordinate systems is basic, or the grid is assigned to the residual structure then the default may be overridden on the GRID entry by making one of fields 3, 7, 8, or 9 zero (rather than blank). Only one GRDSET entry may appear in the Bulk Data Section.
2. The primary purpose of this entry is to minimize the burden of preparing data for problems with a large amount of repetition (e.g., two-dimensional pinned-joint problems).
3. At least one of the fields CP, CD, PS, or SEID must be specified.

GRID Grid Point

Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

Example:

GRID	2	3	1.0	-2.0	3.0		316		

Field	Contents
lD	Grid point identification number. (0 < Integer < 100000000)
CP	Identification number of coordinate system in which the location of the grid point is defined. (Integer $\geq$ 0 or blank*)
X1, X2, X3	Location of the grid point in coordinate system CP. (Real; Default = 0.0)
CD	Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors are defined at the grid point. (Integer $\geq$ -1 or blank)*
PS	Permanent single-point constraints associated with the grid point. (Any of the Integers 1 through 6 with no embedded blanks, or blank*.)
SEID	Superelement identification number. (Integer $\geq$ 0; Default = 0)

* See the GRDSET entry for default options for the CP, CD, PS, and SEID fields.

Remarks:

1. All grid point identification numbers must be unique with respect to all other structural, scalar, and fluid points.

2. The meaning of X1, X2, and X3 depends on the type of coordinate system CP as follows (see the CORDij entry descriptions):

Type	X1	X2	X3
Rectangular	X	Y	Z
Cylindrical	R	θ(degrees)	Z
Spherical	R	θ(degrees)	ϕ(degrees)

   See "**Grid Point and Coordinate System Definition**" on page 41 of the *MSC.Nastran Reference Guide*, for a definition of coordinate system terminology.

3. The collection of all CD coordinate systems defined on all GRID entries is called the global coordinate system. All degrees-of-freedom, constraints, and solution vectors are expressed in the global coordinate system.
4. The SEID field can be overridden by use of the SESET entry.
5. If CD = -1, then this defines a fluid grid point in coupled fluid-structural analysis (see "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*). This type of point may only connect the CAABSF, CHACBR, CHACAB, CHEXA, CPENTA, and CTETRA elements to define fluid elements.
6. A zero (or blank if the GRDSET entry is not specified) in the CP and CD fields refers to the basic coordinate system.
7. In p-version analysis, the hierarchy set to resolve the conflicts arising in the global system input data is described under Remark 10 of the GMBC entry description.
8. CID can reference GMCORD type coordinate systems only when the GRID is connected to p-version elements.

GRIDB Axisymmetric Grid Point

Defines the location of a geometric grid point on a fluid point (RINGFL entry) for an axisymmetric fluid model and / or axisymmetric structure. Also defines the boundary of the fluid.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDB	ID			PHI		CD	PS	IDF	

Example:

GRIDB	30			30.0		3	345	20	

Field	Contents
ID	Grid point identification number. (0 < Integer < 1000000)
PHI	Azimuthal position of the fluid in degrees. (Real)
CD	Identification number of the coordinate system in which the displacements are defined at the grid point. (Integer $\geq$ 0 or blank)
PS	Permanent single-point constraints associated with grid point. (Any combination of the Integers 1 through 6 with no embedded blanks, or blank.)
IDF	Identification number of a RINGFL entry. (Integer > 0)

Remarks:

1. GRIDB is allowed only if an AXIF entry is also present. The AXIF entry must define a fluid coordinate system.
2. All GRIDB identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. The referenced RINGFL entry must be present and be included in a boundary list (BDYLIST entry).
4. If no harmonic numbers on the AXIF entry are specified, no fluid elements are necessary.
5. The collection of all CD coordinate systems defined on all GRID and GRIDB entries is called the global coordinate system.

6. Fields 3, 4, and 6 are ignored, which facilitates the conversion of GRID entries to GRIDB entries. Note that the fields are the same except for fields 1 and 9 when a cylindrical coordinate system is used.

GRIDF Fluid Point

Defines a scalar degree-of-freedom for harmonic analysis of a fluid.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDF	ID	R	Z						

Example:

GRIDF	23	2.5	-7.3						

Field	Contents
ID	Identification number of axisymmetric fluid point. (0 < Integer < 1000000)
R	Radial location of point in basic coordinate system. (Real > 0.0)
Z	Axial location of point in basic coordinate system. (Real)

Remarks:

1. This entry is allowed only if an AXSLOT entry is also present.
2. ID must be unique with respect to all other scalar, structural, and fluid points.
3. Grid points on slot boundaries are defined on GRIDS entries. Do not also define them on GRIDF entries.
4. For plotting purposes, the R location corresponds to the basic X coordinate. The Z location corresponds to the basic Y coordinate. Pressures will be plotted as displacements in the basic Z direction.
5. Load and constraint conditions are applied as if GRIDF were a scalar point. Positive loads correspond to inward flow. A single-point constraint causes zero pressure at the point.

GRIDS Slot Surface Point

Defines a scalar degree-of-freedom with a two-dimensional location. Used in defining pressure in slotted acoustic cavities.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDS	ID	R	Z	W	IDF				

Example:

GRIDS	25	2.5	-7.3	0.5					

Field	Contents
ID	Identification number of the slot point. (Integer > 0)
R	Radial location of point in basic coordinate system. (Real ≠ 0.0)
Z	Axial location of point in basic coordinate system. (Real)
W	Slot width or thickness at the GRIDS point. (Real ≥ 0.0 or blank)
IDF	Identification number to define a GRIDF point. (Integer > 0 or blank)

Remarks:

1. This entry is allowed only if an AXSLOT entry is also present.
2. ID (and IDF if present) must be unique with respect to all other scalar, structural, and fluid points.
3. If W is blank, the default value on the AXSLOT entry will be used.
4. The IDF number is referenced on the CAXIFi entry for central cavity fluid elements next to the interface. The IDF number is entered only if the grid point is on an interface. In this case, the IDF should also be defined on a GRIDF entry.
5. If IDF is nonzero, then R must be greater than zero.
6. For plotting purposes, the R location corresponds to the basic X coordinate. The Z location corresponds to the basic Y coordinate. The slot width, W, corresponds to the basic Z coordinate. The pressure will be plotted in the basic Z direction.

7. Load and constraint conditions are applied as if the GRIDS is a scalar point. Positive loads correspond to inward flow. A single-point constraint causes zero pressure at the point.

GUST Aerodynamic Gust Load Description

Defines a stationary vertical gust for use in aeroelastic response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
GUST	SID	DLOAD	WG	X0	V				

Example:

GUST	133	61	1.0	0.	1.+4				

Field	Contents
SID	Gust set identification number. (Integer > 0)
DLOAD	Set identification number of a TLOADi or RLOADi entry that defines the time or frequency dependence. (Integer > 0)
WG	Scale factor (gust velocity / forward velocity) for gust velocity. (Real $\neq$ 0.0)
X0	Streamwise location in the aerodynamic coordinate system of the gust reference point. (Real)
V	Velocity of vehicle. See Remark 5. (Real > 0.0)

Remarks:

1. The GUST entry must be selected with the Case Control command GUST = SID.
2. The gust angle is in the +z direction of the aerodynamic coordinate system. The value is

$$WG \cdot T\left(t - \frac{X - X0}{V}\right)$$

   where T is the tabular function.
3. In random analysis, a unit gust velocity (WG = 1 / velocity) is suggested. The actual rms value is entered on the TABRNDG entry.
4. X0 and V may not change between subcases under one execution.
5. V must be equal to VELOCITY on the AERO Bulk Data entry.

HGSUPPR (SOL 700) Hourglass Suppression Method (SOL 700 only)

Defines the hourglass suppression method, the corresponding hourglass damping coefficients and sets for the bulk viscosity method and coefficients.

Format:

1	2	3	4	5	6	7	8	9	10
HGSUPPR	HID	PROP	PID	HGTYPE	HGCMEM	HGCWRP	HGCBEND	HGCSOL	
	-	-	IBQ	Q1	Q2				

Example:

HGSUPPR	1	SHELL	100	2	0.1	0.1			
				1.5	0.06				

Field	Contents		
HID	Hourglass suppression definition number.	I > 0	Required
PROP	Property type: SHELL For shell elements SOLID For solid Lagrangian elements.	Character	Required
PID	Property number	Integer	Required
HGTYPE	Hourglass suppression method: Hourglass control type. For solid elements six options are available. For quadrilateral shell and membrane elements the hourglass control is based on the formulation of Belytschko and Tsay, i.e., options 1-3 are identical, and options 4-6 are identical: Regardless of IHQ in *control_hourglass, 1: standard LS-DYNA viscous form, 2: Flanagan-Belytschko viscous form, 3: Flanagan-Belytschko viscous form with exact volume integration for solid elements,	Integer	1

Field	Contents		
	4: Flanagan-Belytschko stiffness form, 5: Flanagan-Belytschko stiffness form with exact volume integration for solid elements. 6: Belytschko-Bindeman [1993] assumed strain co-rotational stiffness form for 2D and 3D solid elements only. This form is available for explicit and IMPLICIT solution methods. In fact, type 6 is mandatory for the implicit options. 8: Applicable to the type 16 fully integrated shell element. IHQ=8 activates the warping stiffness for accurate solutions. A speed penalty of 25% is common for this option.		
HGCMEM	Membrane damping coefficient	$0.0 \le R \le 0.15$	0.1
HGCWRP	Warping damping coefficient	$0.0 \le R \le 0.15$	0.1
HGCBEND	Bending damping coefficient	$0.0 \le R \le 0.15$	0.1
HGCSOL	Solid damping coefficient	$0.0 \le R \le 0.15$	0.1
IBQ	Bulk Viscosity Type	Integer	0
Q1	Quadratic Bulk Viscosity	Real	1.5
Q2	Linear Bulk Viscosity	Real	0.06

Remarks:

1. Viscous hourglass control is recommended for problems deforming with high velocities. Stiffness control is often preferable for lower velocities, especially if the number of time steps are large. For solid elements the exact volume integration provides some advantage for highly distorted elements.
2. For automotive crash the stiffness form of the hourglass control with a coefficient of 0.05 is preferred by many users.

3. Bulk viscosity is necessary to propagate shock waves in solid materials and therefore applies only to solid elements. Generally, the default values are okay except in problems where pressures are very high, larger values may be desirable. In low density foams, it may be necessary to reduce the viscosity values since the viscous stress can be significant. It is not advisable to reduce it by more than an order of magnitude.

4. Type 6 hourglass control is for 2D and 3D solid elements only. Based on elastic constants and an assumed strain field, it produces accurate coarse mesh bending results for elastic material when QM=1.0. For plasticity models with a yield stress tangent modulus that is much smaller than the elastic modulus, a smaller value of QM (0.001 to 0.1) may produce better results. For any material, keep in mind that the stiffness is based on the elastic constants, so if the material softens, a QM value smaller than 1.0 may work better. For anisotropic materials, an average of the elastic constants is used. For fluids modeled with null material, type 6 hourglass control is viscous and is scaled to the viscosity coefficient of the material (see *MAT_NULL).

5. In part, the computational efficiency of the Belytschko-Lin-Tsay and the under integrated Hughes-Liu shell elements are derived from their use of one-point quadrature in the plane of the element. To suppress the hourglass deformation modes that accompany one-point quadrature, hourglass viscous or stiffness based stresses are added to the physical stresses at the local element level. The discussion of the hourglass control that follows pertains to all one point quadrilateral shell and membrane elements in LS-DYNA.

   The hourglass shape vector τ_I is defined as

$$\tau_I = h_I - (h_J \hat{x}_{aJ}) B_{aI}$$

   where, $\hat{X}_{aJ}$ are the element coordinates in the local system at the Ith element node, B_{aI} is the strain displacement matrix, and hourglass basis vector is:

$$h = \begin{bmatrix} +1 \\ -1 \\ +1 \\ -1 \end{bmatrix}$$

   is the basis vector that generates the deformation mode that is neglected by one-point quadrature. In the above equations and the reminder of this subsection, the Greek subscripts have a range of 2, e.g., $\hat{x}_{aI} = (\hat{x}_{1I}, \hat{x}_{2I}) = (\hat{x}_I, \hat{y}_I)$

The hourglass shape vector then operates on the generalized displacements to produce the generalized hourglass strain rates

$$\dot{q}_a^M = \tau_I \hat{\upsilon}_{aI}$$

$$\dot{q}_a^B = \tau_I \hat{\theta}_{aI}$$

$$\dot{q}_3^W = \tau_I \hat{\upsilon}_{zI}$$

where the superscripts M, B, and W denote membrane, bending, and warping modes, respectively. The corresponding hourglass stress rates are then given by

$$\dot{Q}_a^M = \frac{QM \cdot EtA}{8} B_{\beta I} B_{\beta I} \dot{q}_a^M$$

$$\dot{Q}_a^B = \frac{QB \cdot Et^3 A}{192} B_{\beta I} B_{\beta I} \dot{q}_3^B$$

$$\dot{Q}_3^W = \frac{QW \cdot kGt^3 A}{12} B_{\beta I} B_{\beta I} \dot{q}_3^B$$

where t is the shell thickness. The hourglass coefficients: QM, QB, and QW are generally assigned values between 0.05 and 0.10.

Finally, the hourglass stresses which are updated using the time step, Δt from the stress rates in the usual way, i.e.,

$$Q^{n+1} = Q^n + \Delta t \dot{Q}$$

and the hourglass resultant forces are then

$$\hat{f}_{aI}^H = \tau_I Q_a^M$$

$$\hat{m}_{aI}^H = \tau_I Q_a^B$$

$$\hat{f}_{3I}^H = \tau_I Q_3^W$$

where the superscript H emphasizes that these are internal force contributions from the hourglass deformations.

HYBDAMP Hybrid Modal Damping for Direct Dynamic Solutions

Specifies the modes and damping for hybrid damping calculations.

Format:

1	2	3	4	5	6	7	8	9	10
HYBDAMP	ID	METHOD	SDAMP	KDAMP					

Example:

HYBDAMP	101	2000	2001	NO					

Field	Contents
ID	Identification number of HYBDMP entry (Integer > 0; Required)
METHOD	Identification number of METHOD entry for modes calculation. (Integer > 0, Required)
SDAMP	Identification number of SDAMP entry for modal damping specification. (Integer > 0; Required)
KDAMP	Selects modal “structural” damping (Character: “Yes” or “NO”, see Remark 1; Default = “NO”)

Remarks:

1. For KDAMP = “YES”, the viscous modal damping is entered into the complex stiffness matrix as structural damping.
2. Hybrid damping is generated using modal damping specified by the user on TABDMP entries.

$$BH = \begin{bmatrix} \phi_1 & \phi_2 & \cdots & \phi_n \end{bmatrix} \begin{bmatrix} b(\omega_1) & & & \\ & b(\omega_2) & & \\ & & \ddots & \\ & & & b(\omega_n) \end{bmatrix} \begin{bmatrix} \phi_1^T \\ \phi_2^T \\ \vdots \\ \phi_n^T \end{bmatrix}$$

For KDAMP = “YES”

$$KH = \begin{bmatrix} \phi_1 & \phi_2 & \cdots & \phi_n \end{bmatrix} \begin{bmatrix} g(\omega_1) & & & \\ & g(\omega_2) & & \\ & & \ddots & \\ & & & g(\omega_n) \end{bmatrix} \begin{bmatrix} \phi_1^T \\ \phi_2^T \\ \vdots \\ \phi_n^T \end{bmatrix}$$

where

ϕ_i = modes of the structure

$b(\omega_i)$ = modal damping values, $b(\omega_i) = g(\omega_i)\omega_i m_i$

$g(\omega_i)$ = twice the critical damping ratio determined from user specified TABDMP entry

ω_i = natural frequency of mode ϕ_i

m_i = generalized mass of mode ϕ_i

INCLUDE Insert External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

Format:

INCLUDE'filename'

Describer:

filename	Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks.

Example:

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
BEGIN BULK
INCLUDE 'MYBULK.DATA'
ENDDATA
```

Remarks:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example the file:

   /dir123/dir456/dir789/filename.dat

   may be included with the following input:

   ```
   INCLUDE '/dir123
                /dir456
           /dir789/filename.dat'
   ```

3. See the *MSC.Nastran 2005 r2 Installation and Operations Guide* for more examples.

IPSTRAIN (SOL 600) Initial Equivalent Plastic Strain Values

Defines initial equivalent plastic strain values. This is the MSC.Marc's initial plastic strain option used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
IPSTRAIN	EID1	EID2	INT1	INTN	LAY1	LAYN	STRAIN		

Example:

IPSTRAIN	2001	2020	1	4	1	5	0.025		

Field	Contents
EID1	First Element ID to which these strains apply. (Integer > 0)
EID2	Last Element ID to which these strains apply. (Integer; Default = EID1)
INT1	First Integration point for which the strain applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the strain applies. (Integer > 0, Default = 4)
LAY1	First element layer for which the strain applies. (Integer >0; no default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the strain applies. (Integer >0; no default. Enter zero or leave blank if the model does not contain beams or shells.)
STRAIN	Equivalent plastic strain value at start of analysis. (Real; Default is 0.0)

Remarks:

1. This entry only applies when MSC.Marc is executed from MD Nastran using MD Nastran Implicit Nonlinear (SOL 600) and is ignored for other solutions.
2. This entry is normally used for metal forming and represents the amount of plastic deformation that the model was previously subjected to. It is used in work (strain) hardening models.

ISTRESS (SOL 600) Initial Stress Values

Defines initial stress values. This is the MSC.Marc's initial stress option used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
ISTRESS	EID1	EID2	INT1	INTN	LAY1	LAYN	STRESS1	STRESS2	
	STRESS3	STRESS4	STRESS5	STRESS6	STRESS7				

Example:

ISTRESS	2001	2020	1	4	1	5	45000.	-2000.	
	0.0	4500.	0.0	2350.					

Field	Contents
EID1	First Element ID to which these strains apply. (Integer > 0)
EID2	Last Element ID to which these strains apply. (Integer; Default = EID1)
INT1	First Integration point for which the strain applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the strain applies. (Integer > 0; Default = 4)
LAY1	First element layer for which the strain applies. (Integer >0; no default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the strain applies. (Integer >0; no default. Enter zero or leave blank if the model does not contain beams or shells.)
STRESS(i)	Up to 7 stress components may be entered. (Real; Default = 0.0)

Remarks:

1. This entry only applies when MSC.Marc is executed from MD Nastran using MD Nastran Implicit Nonlinear (SOL 600) and is ignored for other solutions.
2. Initial stresses must be self-equilibrating and may not produce material nonlinearity.

3. Stress components are as follows:
   Definitions:
   s - normal type stress
   t - shear type stress
   x,y,z in global system
   1,2,3 in element local system

   3D solid elements (for example type 7)
   1 - sxx
   2 - syy
   3 - szz
   4 - txy
   5 - tyz
   6 - tzx
   7 - hydrostatic pressure (Herrmann elements only, otherwise 7 should be blank)

   Thick shells (for example type 75)
   1 - s11
   2 - s22
   3 - t12
   4 - t23
   5 - t31

   Thin shells (for example type 72)
   1 - s11
   2 - s22
   3 - t12

   Beams (for example type 14 or 98)
   1 - s - axial
   2 - t - twist

ITER Iterative Solver Options

Defines options for the iterative solver in SOLs 101, 106, 108, 111 and 153.

Format:

1	2	3	4	5	6	7	8	9	10
ITER	SID								
	OPTION1VALUE1 OPTION2VALUE2 -etc.-								

Example:

ITER	100								
	ITSEPS=1.0E0-7, MSGFLG=YES, PRECOND=BICWELL, IPAD=3								

Field	Contents
SID	Set identification number. (Integer > 0).
PRECOND	Preconditioner option. (Character; Default = “BIC” for real analysis, “BICCMPLX” for complex analysis and “PBDJ” for p-version analysis.) See Remarks **3.** and **4.**

J	Jacobi
JS	Jacobi with diagonal scaling.
C	Incomplete Cholesky.
CS	Incomplete Cholesky with diagonal scaling.
RIC	Reduced incomplete Cholesky.
RICS	Reduced incomplete Cholesky with diagonal scaling.
PBCJ	p-version block Cholesky / Jacobi.
PBDJ	p-version block Direct / Jacobi.
PBDC	p-version block Direct / Cholesky.
BIC	Block incomplete Cholesky for real problems.
BICCMPLX	Block incomplete Cholesky for complex problems.
CASI	Element-based third party iterative solver.

Field	Contents	
	USER	User given preconditioning. For direct frequency response (SOL 108): a decomposition will be done for 1st frequency and the factor will be used for all subsequent frequencies as a preconditioner with the iterative solver. Other solutions require a DMAP alter. Please refer to the ***MSC.Nastran Numerical Methods User's Guide*** description of the SOLVIT module.
CONV	Convergence criterion. (Character; Default = "AREX")	
	AR	$\|\|r\|\| / \|\|b\|\|$ where r is the residual vector of current iteration and b is the initial load vector; internal criterion.
	GE	Alternative convergence criterion using geometric progression and the differences between two consecutive solution updates; internal criterion.
	AREX	Same criterion as AR but with the additional consideration of the external convergence criterion. See Remark **2.** (Default).
	GEEX	Same criterion as GE but with the additional consideration of the external convergence criterion. See Remark **2.**
MSGFLG	Message flag. (Character; Default = "NO")	
	YES	Messages will be printed for each iteration.
	NO	Only minimal messages will be printed from the iterative solver (Default).
ITSEPS	User-given convergence parameter epsilon. (Real > 0.0; Default = 1.E-6)	
ITSMAX	Maximum number of iterations. (Integer > 0; Default = N / 4 where N is the number of rows in the matrix)	
IPAD	Padding value for RIC, RICS, BIC, and BICCMPLX preconditioning. (Integer > 0)	
	Default = 0	for PRECOND = "RIC" or "RICS"
	Default = 2	for PRECOND = "BIC" for purely three-dimensional models and three for two-dimensional and mixed element models. IPAD may be reset automatically by the program to the best value.
	Default = 5	for PRECOND = "BICCMPLX".

Field	Contents
IEXT	Extraction level in reduced incomplete Cholesky preconditioning. Block structuring method in block incomplete Cholesky preconditioning. (Integer = 0 thought 7; Default = 0)
	0 Uses USET / SIL tables (Default).
	1 - 7 The default value of 0 is recommended for all problems. The values 1 - 7 use a heuristic algorithm with a maximum block size equal to IEXT. Although setting IEXT to a value other than 0 could lead to slightly improved performance or reduced disk space use, it should be considered exploratory without the expectation of a benefit.
PREFONLY	Specifies early termination of the iterative solver. (Integer = 0 or -1; Default = 0)
	0 Runs to completion (Default).
	-1 Terminates after preface giving resource estimates.

Remarks:

1. The optional ITER Bulk Data entry is selected by the SMETHOD Case Control command and is only required to override the defaults specified above.
2. The external epsilon is computed as follows:

$$\varepsilon = \frac{(r, x)}{(b, x)}$$

   where r is the final residual vector, x is the final solution vector and b is the initial load vector (r, x) indicates the inner product of r and x and (b, x) indicates the inner product of b and x.
3. See the ***MSC.Nastran Numerical Methods User's Guide*** for more information on these options.
4. Prior to Version 70.5 of MSC.Nastran the user had the burden of choosing BICWELL for well conditioned problems and BICILL for ill-conditioned problems. The enhanced code now determines this automatically. Thus BICWELL and BICILL are now equivalent to BIC.
5. The element-based iterative solver is primarily intended for the solution of very large solid element linear static structural analysis problems. The following restrictions apply:

- SOL 101 only
- No GENEL elements allowed
- No K2GG direct input matrix selection allowed
- No ASET/OMIT reduction allowed
- No SUPORTi allowed
- No SCALAR points (explicitly or implicitly defined) allowed
- No heat transfer allowed
- No p-elements allowed

Only BAR, BEAM, BUSH, ROD, CONROD, ELAS1, ELAS2, SHEAR, QUAD4, QUAD8, QUADR, TRIA3, TRIA6, TRIAR, TETRA, PENTA and HEXA elements are allowed.

LOAD Static Load Combination (Superposition)

Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, DAREA (if these entries have been converted), PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, SLOAD, RFORCE, and GRAV entries.

Format:

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-						

Example:

LOAD	101	-0.5	1.0	3	6.2	4			

Field	Contents
SID	Load set identification number. (Integer > 0)
S	Overall scale factor. (Real)
Si	Scale factor on Li. (Real)
Li	Load set identification numbers defined on entry types listed above. (Integer > 0)

Remarks:

1. The load vector $\{P\}$ is defined by

$$\{P\} = S\sum_{i} Si\{P_{Li}\}$$

2. Load set IDs (Li) must be unique.
3. This entry must be used if acceleration loads (GRAV entry) are to be used with any of the other types.
4. In the static solution sequences, the load set ID must be selected by the Case Control command LOAD=SID. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
5. A LOAD entry may not reference a set identification number defined by another LOAD entry.

LOADCYH Harmonic Load Input for Cyclic Symmetry

Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCYH	SID	S	HID	HTYPE	S1	L1	S2	L2	

Example:

LOADCYH	10	1.0	7	C	0.5	15			

Field	Contents
SID	Load set identification number. (Integer > 0)
S	Scale Factor. (Real)
HID	Harmonic. See Remark **6.** (Integer or blank)
HTYPE	Harmonic type. (Character: “C”, “S”, “CSTAR” “SSTAR”, “GRAV”, “RFORCE”, or blank).
Si	Scale factor on Li. (Real)
Li	Load set identification number. See Remark **10.** (Integer > 0)

Remarks:

1. The LOADCYH entry is selected with the Case Control command LOAD = SID.
2. If HTYPE is blank, the load will be applied to all applicable types in the problem.
3. If HTYPE is “GRAV” or “RFORCE”, GRAV or RFORCE entry loading will be used. Harmonic loads for appropriate available harmonics will be generated automatically in these cases.
4. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and / or GRAV entries.

5. If L1 refers to a set ID defined by an SPCD entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, DAREA (if these entries have been converted), MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX, and SLOAD or converted DAREA entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.
6. If HTYPE is “GRAV” or “RFORCE”, the entry in HID will be ignored and therefore may be blank. S2 and L2 must be blank for this case.
7. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYN entries.
8. If HTYPE = “C”, “S”, “CSTAR”, or “SSTAR”, the load on component (HTYPE) of harmonic (HID) is $L = S(S1 \cdot L1 + S2 \cdot L2)$.
9. S must be nonzero. In addition, either S1 or S2 must be nonzero.
10. L1 and L2 may reference any of the static or dynamic loading entries including GRAV and RFORCE.

LOADCYN Physical Load Input for Cyclic Symmetry

Defines a physical static or dynamic load for use in cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCYN	SID	S	SEGID	SEGTYPE	S1	L1	S2	L2	

Example:

LOADCYN	10	1.0	1	R	0.5	17			

Field	Contents
SID	Load set identification number. (Integer > 0)
S	Scale Factor. (Real)
SEGID	Segment identification number. (Integer)
SEGTYPE	Segment type. (Character: “R”, “L”, or blank)
Si	Scale Factors. (Real)
Li	Load set ID numbers. See Remark **8.** (Integer > 0)

Remarks:

1. The LOADCYN entry is selected by the LOAD Case Control command.
2. If SEGTYPE is blank, both R and L segments will be used in DIH-type symmetry.
3. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and/or GRAV entries.
4. If L1 refers to a set ID defined by SPCD loading entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, DAREA (if these entries have been converted), MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, and SLOAD entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.

5. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYH entries.
6. The load on the segment (or half-segment) is $L = S(S1 \cdot L1 + S2 \cdot L2)$.
7. S must be nonzero. In addition, either S1 or S2 must be nonzero.
8. L1 and L2 may reference any of the static or dynamic loading entries except GRAV and RFORCE.
9. For cyclic buckling loads may only be applied to the first segment and only zero harmonic loads may be applied so the LOADCYH entry should be used.

LOADCYT Table Load Input for Cyclic Symmetry

Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = "AXI" on the CYSYM entry.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCYT	SID	TABLEID1	LOADSET1	METHOD1	TABLEID2	LOADSET2	METHOD2		

Example:

LOADCYT	10	19	27		21	26	1		

Field	Contents
SID	Load set identification number. (Integer > 0)
TABLEIDi	Table ID for table load input for load set Li. See Remark 3. (Integer > 0)
LOADSETi	Load set Li. (Integer > 0)
METHODi	Method of interpolation. See Remark 5. (Integer: 0 or 1, Default = 0)
	0 interpolate the load with the Fourier coefficients specified in the table up to the specified number of harmonics. (Default)
	1 interpolate the magnitude of the load at corresponding grid points in all segments.

Remarks:

1. The LOADCYT entry is selected by the LOAD Case Control command.
2. The load set ID given in fields 4 or 7 of this entry may refer to FORCE, MOMENT, PLOAD, PLOAD2, PLOAD4, SPCD, TEMP, or TEMPP1 Bulk Data entries.
3. Either TABLED1 or TABLED2 type tabular data of azimuth angle (Xi) versus scale factors (Yi) may be used. The azimuth angle values must be in degrees.
4. The scale factors given in the tables referenced by TABLEDi entries will be applied only to the magnitudes of the loads defined by LOADSET IDs given in fields 4 or 7.

5. For grid point loading entries, (like FORCE, MOMENT, SPCD, and TEMP Bulk Data entries) METHODi = 1 option should be used. For element loading entries (like PLOAD, PLOAD2, PLOAD4, and TEMPP1 Bulk Data entries) either METHODi = 0 or METHODi = 1 option can be used. In particular, if harmonic output of element stresses under temperature loading via TEMPP1 Bulk Data entry, METHODi = 0 option should be used to specify TEMPP1 load set.

LSEQ Static Load Set Definition

Defines a sequence of static load sets.

Format:

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	EXCITEID	LID	TID					

Example:

LSEQ	100	200	1000	1001					

Field	Contents
SID	Set identification of the set of LSEQ entries. See Remark 5. (Integer > 0)
EXCITEID	The EXCITEID set identification assigned to this static load vector. See Remark 5. (Integer > 0)
LID	Load set identification number of a set of static load entries such as those referenced by the LOAD Case Control command. (Integer > 0 or blank)
TID	Temperature set identification of a set of thermal load entries such as those referenced by the TEMP(LOAD) Case Control command. (Integer > 0 or blank)

Remarks:

1. LSEQ will not be used unless selected in the Case Control Section with the LOADSET command.
2. The number of static load vectors created for each superelement depends upon the type of analysis. In static analysis, the number of vectors created is equal to the number of unique EXCITEID IDs on all LSEQ entries in the Bulk Data; in dynamic analysis, the number of vectors created is equal to the number of unique EXCITEID IDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data.
3. EXCITEID may be referenced by CLOAD, RLOAD1, RLOAD2, TLOAD1, and TLOAD2 entries in order to apply the static load in nonlinear, static and dynamic analysis.
4. Element data recovery for thermal loads is not currently implemented in dynamics.

5. The SID-EXCITEID number pair must be unique with respect to similar pairs on all other LSEQ entries in the Bulk Data.
6. In a nonsuperelement analysis, LID and TID cannot both be blank. In superelement analysis, they may both be blank as long as static loads are prescribed in the upstream superelements.
7. It is no longer necessary to employ LOADSET/LSEQ data to specify static loading data for use in dynamic analysis. In the absence of LSEQ Bulk Data entries, all static loads whose load set IDs match the EXCITEID IDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data are automatically processed.

MARCIN (SOL 600) Inserts a Text String in MSC.Marc

Inserts a text string directly in the MSC.Marc input file used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MARCIN	ILOC	ICONT	String						

Example:

MARCIN	-1	0	feature,2301						

Field	Contents
ILOC	Identification of the location in the MSC.Marc file where the string will be placed. (Required, integer) -1 String is placed at end of MSC.Marc's Parameter Section 0 String is placed at the end of MSC.Marc's Model Definition Section N (N > 0) String is placed in load case N after the AUTO STEP, AUTO INCREMENT, etc. entry.
ICONT	(Required, integer) -1 String is used as the name of an include file which contains all of the direct MSC.Marc input data for that portion of MSC.Marc (parameter, model definition, etc.). Only one include file per MSC.Marc Section is allowed. 0 Entry is not a continuation of previous MARCIN entry. 1 Entry is a continuation of previous MARCIN entry and the strings will be placed one after the other on the same MSC.Marc line.
String	(Required, character) For ICONT=0 or 1: The desired text string. The string is limited to 48 characters per entry. Multiple entries will be placed in the order entered within each MSC.Marc location. For ICONT=-1, String is an include file name limited to 48 characters.

Remarks:

1. Standard MD Nastran fields 4-9 are ignored for this entry. The string may be entered anywhere within fields 4-9 and will be translated directly to MSC.Marc.
2. If a long line for the MSC.Marc data is required, enter as many MARCIN entries as necessary to describe the entire MSC.Marc string using ICONT=1 for each except the first.
3. The total string length including continuation lines is limited to 160 characters.
4. As many MARCIN entries as necessary may be entered to define all desired input.
5. Each entry in the MD Nastran data file must start with the MARCIN header. Each line in an include file will be translated directly to MSC.Marc (there should be no MARCIN, ILOC or ICONT information in the include file(s).
6. If the direct MSC.Marc input is placed in include file(s), separate files are necessary for each portion of MSC.Marc (parameter, model definition, etc.) requiring direct input.
7. As part of the MD Nastran input process, all strings are converted to uppercase. The internal MSC.Marc translator will convert them to lower case. For input entered without include files, this will normally make any difference. For include file names, file names must be entirely lower case for computer systems that are case sensitive.

MARCOUT (SOL 600) Selects Data Recovery Output

Selects output to be transferred from MSC.Marc to the MD Nastran database used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MARCOUT		IO1	IO2	IO3	IO4	IO5	IO6	IO7	
	IO8	IO9	etc.						

Example:

MARCOUT	125	E11	E21	E41	N1	N2	N35		

Field	Contents
WHERE	Indicates the files that will store this output according to the codes shown below. More than one code may be entered as shown in the example. (Integer, Default = 1 if not entered) 1 = MD Nastran Database 2 = OP2 file 3 = f06 file 4 = Punch file 5 = xdb file
IO(i)	Indicates the type of MSC.Marc output requested according to the following table:

Element Output	Description
E1	strain components, VM strain
E9	Total Temperature (combined heat / structural analysis)
E10	Incremental Temperature (combined heat / structural analysis)
E11	stress components, VM stress, Mean stress
E20	Element Thickness (thickness can change vs time)
E21	plastic strain components, VM plastic strain

Element Output	Description
E29	Value of second state variable if applicable
E38	Total swelling strain
E39	Value of third state variable if applicable
E41	Cauchy stress components, VM Cauchy stress
E48	Strain Energy Density
E68	Plastic strain energy density
E69	Current Volume
E71	Components of thermal strain (separated from total strain)
E78	Original Volume
E79	Grain size if applicable
E81	Components of cracking strain if applicable
E91	Failure indices
E108	Interlaminar shear for thick composite elements
E175	Equivalent viscoplastic strain rate (powder material)
E176	Relative density (powder material)
E241	Gasket pressure if applicable
E242	Gasket closure if applicable
E243	Plastic Gasket closure if applicable
E301	Total strains tensor
E311	Stress tensor
E321	Plastic strain tensor
E331	Creep strain tensor
E341	Cauchy stress tensor
E371	Thermal strain tensor
E381	Cracking strain tensor
E401	Elastic strain tensor
E451	Velocity strains (for fluids)

Element Output	Description
E471	Global components of rebar stresses in undeformed config if applicable
E481	Global components of rebar stresses in deformed config if applicable

Nodal Output	Description
N1,N2	displacements, rotations
N5,N6	reaction forces & moments
N7	Fluid velocity if applicable
N8	Fluid pressure if applicable
N9	External fluid force if applicable
N10	Reaction fluid force if applicable
N11	Sound pressure if applicable
N12	External sound pressure if applicable
N13	Reaction sound pressure if applicable
N23	Pore pressure if applicable (soil analysis)
N24	External mass flux if applicable
N25	Reaction mass flux if applicable
N26	Bearing pressure if applicable
N27	Bearing force if applicable
N28,N29	Velocity
N30,N31	Acceleration
N34,N35	Contact normal stress / force
N36,N37	Contact friction stress / force
N38,N39	Contact status, Contact touched body
N40	Herrmann variable

Remarks:

1. MARCOUT is only available when MSC.Marc is executed from within MD Nastran and controls what results are available in the MSC.Marc t16 file. All elements or nodes of each type selected will be placed on the t16 file (in other words, it is not possible to control this output by selecting various sets). The results in the t16 file may be used to obtain op2, xdb, punch or f06 results output by specifying OUTR options on the SOL 600 Executive Control statement. Thus, op2, xdb, punch and f06 results can not be controlled using sets. See Remarks **9.** to **11.** for other related information.
2. Values such as E1, E21 correspond to MSC.Marc's postcodes 1 and 21, respectively.
3. Outputs produced by MARCOUT are the same for all subcases, load steps, iterations, etc.
4. The MARCOUT entry may be repeated as many times as desired, or all entries may be placed on continuation lines.
5. For entries E1, E11 and E21 corresponding entries E2-E7, E12-E17 and E22-E27 will be generated automatically. These terms correspond to 3 normal stress (or strain) and 3 shear stress / strain values plus the equivalent von Mises value. See MSC.Marc volume C POST description for more details.
6. If this entry is not used, the following defaults are entered automatically: E1, E11, E18, E27, E301, E321, E341, E401, E47, E48, N1, N2, N5, N35, N37, N39. When creep or heat transfer is present, additional items are added appropriately. If the MARCOUT entry is entered, only those items specified will be output.
7. At present, only displacements, rotations, Cauchy stresses and one type of strains (total, plastic or elastic) may be transferred to the MD Nastran database. In subsequent releases, it is anticipated that the MD Nastran database will be expanded to allow all of the output listed previously.
8. Field 2 of each parent entry should be left blank.
9. Displacements, at least one stress tensor and one strain tensor must be selected if any OUTR options are to be used.
10. For SOL 600, MD Nastran Case Control commands such as DISP= STRESS= STRAIN= only control what output will be printed in the Marc .out file. Results for all other output media are controlled by the MARCOUT and SOL 600 OUTR entries. Normally, results in the Marc .out file are not desired and if requested will generate a very large output file. Therefore, for most cases, MD Nastran Case Control requests should be:

DISP(PLOT)=ALL

STRESS(PLOT)=ALL

STRAIN(PLOT)=ALL

Together with desired OUTR and MARCOUT options.

11. Default MARCOUT options are sufficient for most needs and it is recommended that the MARCOUT entry only be employed by advanced users.

MAT1 Isotropic Material Property Definition

Defines the material properties for linear isotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS	MCSID					

Example:

MAT1	17	3.+7		0.33	4.28	6.5-6	5.37+2	0.23	
	20.+4	15.+4	12.+4	1003					

Field	Contents
MID	Material identification number. (Integer > 0)
E	Young's modulus. (Real $\geq$ 0.0 or blank)
G	Shear modulus. (Real $\geq$ 0.0 or blank)
NU	Poisson's ratio. (-1.0 < Real $\leq$ 0.5 or blank)
RHO	Mass density. See Remark **5.** (Real)
A	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks **9.** and **10.** (Real; Default = 0.0 if A is specified.)
GE	Structural element damping coefficient. See Remarks **8.**, **9.**, and **4.** (Real)
ST, SC, SS	Stress limits for tension, compression, and shear are optionally supplied, used only to compute margins of safety in certain elements; and have no effect on the computational procedures. See "**Beam Element (CBEAM)**" in Chapter 3 of the *MSC.Nastran Reference Guide*. (Real $\geq$ 0.0 or blank)
MCSID	Material coordinate system identification number. Used only for PARAM,CURV processing. See "**Parameters**" on page 659. (Integer $\geq$ 0 or blank)

Remarks:

1. The material identification number must be unique for all MAT1, MAT2, MAT3, MAT8, and MAT9 entries.
2. The following rules apply when E, G, or NU are blank:
   - E and G may not both be blank.
   - If NU and E, or NU and G, are both blank, then both are set to 0.0.
   - If only one E, G, or NU is blank, then it will be computed from the equation: $= 2 \cdot (1 + NU) \cdot G$. If this is not desired, then the MAT2 entry is recommended. If E, G, or NU are made temperature dependent by the MATT1 entry, then the equation is applied to the initial values only.
3. If values are specified for all of the properties E, G, and NU, then it is recommended that the following relationship be satisfied:

$$\left| 1 - \frac{E}{2 \cdot (1 + NU) \cdot G} \right| < 0.01$$

   If this relationship is not desired, then the MAT2 entry is recommended.

   It should also be noted that some of the properties are not applied in the stiffness formulation of certain elements as indicated in **Table 8-23**. Therefore, it is recommended that only the applicable properties be specified for a given element.

Table 8-23 Material Property Usage Versus Element Types

Element Entry	E	NU	G
CROD CBEAM CBAR	Extension and Bending	Not Used	Torsion Transverse Shear
CQUADi CTRIAi CCONEAX	Membrane, including In-plane Shear, and Bending		Transverse Shear
CSHEAR	Not Used		Shear
CRAC2D	All Terms		Not Used

Table 8-23 Material Property Usage Versus Element Types (continued)

Element Entry	E	NU	G
CHEXA CPENTA CTETRA CRAC3D	All Terms		Not Used
CTRIAX6	Radial, Axial, Circumferential	All Coupled Ratios	Shear

4. MAT1 materials may be made temperature-dependent by use of the MATT1 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
5. The mass density RHO will be used to compute mass for all structural elements automatically.
6. Weight density may be used in field 6 if the value 1/g is entered on the PARAM,WTMASS entry, where g is the acceleration of gravity (see "**Parameters**" on page 659").
7. MCSID must be nonzero if PARAM,CURV is specified to calculate stresses or strains at grid points on plate and shell elements only.
8. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
9. TREF and GE are ignored if the MAT1 entry is referenced by a PCOMP entry.
10. TREF is used in two different ways:
    - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection.
    - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.

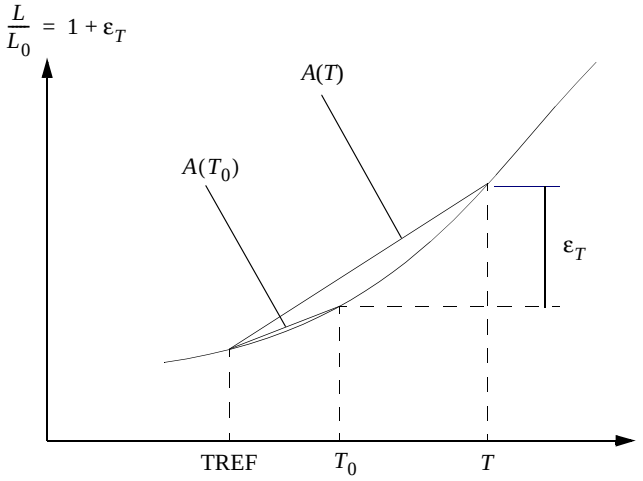


Figure 8-97 Use of TREF in Calculation of Thermal Loads

$$\varepsilon_T = A(T) \cdot (T - TREF) - A(T_0) \cdot (T_0 - TREF)$$

where T is requested by the TEMPERATURE(LOAD) command and T_0 is requested by the TEMPERATURE(INITIAL) command.

Note:

1. A is a secant quantity.
2. TREF is obtained from the same source as the other material properties; e.g., ASTM, etc.
3. If $A(T)$ constant, then $\varepsilon_T = A \cdot (T - T_0)$
4. If PARAM,W4 is not specified, GE is ignored in transient analysis. See "**Parameters**" on page 659.

11. In nonlinear static analysis (SOL 106) the QUAD4 and TRIA3 thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPSILONT,INTEGRAL' in bulk data. See "**Parameters**" on page 659.
12. For SOL 600, E must not be blank or zero.
13. Negative values for ST, SC, and SS lead to no margins of safety being computed.

MAT2 Shell Element Anisotropic Material Property Definition

Defines the material properties for linear anisotropic materials for two-dimensional elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	
	A1	A2	A3	TREF	GE	ST	SC	SS	
	MCSID								

Example:

MAT2	13	6.2+3			6.2+3		5.1+3	0.056	
	6.5-6	6.5-6		-500.0	0.002	20.+5			
	1003								

Field	Contents
MID	Material identification number. See Remark **13.** (Integer > 0)
Gij	The material property matrix. (Real)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient vector. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks **10.** and **11.** (Real or blank)
GE	Structural element damping coefficient. See Remarks **7.**, **10.**, and **12.** (Real)
ST, SC, SS	Stress limits for tension, compression, and shear are optionally supplied (these are used only to compute margins of safety in certain elements) and have no effect on the computational procedures. (Real or blank)
MCSID	Material coordinate system identification number. Used only for PARAM, CURV processing. See "**Parameters**" on page 659. (Integer $\geq$ 0 or blank)

Remarks:

1. The material identification numbers must be unique for all MAT1, MAT2, MAT3, MAT8, and MAT9 entries.
2. MAT2 materials may be made temperature dependent by use of the MATT2 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
3. The mass density, RHO, will be used to automatically compute mass for all structural elements.
4. The convention for the Gij in fields 3 through 8 are represented by the matrix relationship

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} G11 & G12 & G13 \\ G12 & G22 & G23 \\ G13 & G23 & G33 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} - (T - T_0) \begin{Bmatrix} A1 \\ A2 \\ A3 \end{Bmatrix} \right)$$

5. If this entry is referenced by the MID3 field (transverse shear) on the PSHELL, then G13, G23, and G33 must be blank. This may lead to user warning message 6134 which may be ignored. See *The NASTRAN Theoretical Manual*, Section 4.2.
6. MCSID must be nonzero if PARAM,CURV is specified to extrapolate element centroid stresses or strains to grid points on plate and shell elements only. CQUAD4 element corner stresses are not supported by PARAM,CURV.
7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
8. If the MAT2 entry is referenced by the PCOMP entry, the transverse shear flexibility for the referenced lamina is zero.
9. Unlike the MAT1 entry, data from the MAT2 entry is used directly without adjustment of equivalent E, G, or NU values.
10. TREF and GE are ignored if this entry is referenced by a PCOMP entry.

11. TREF is used in two different ways:
    - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark 10 in the MAT1 description.
    - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.
12. If PARAM,W4 is not specified, GE is ignored in transient analysis. See "**Parameters**" on page 659.
13. PCOMP entries generate MAT2 entries equal to 100,000,000 plus the PCOMP PID. Explicitly specified MAT2 IDs must not conflict with internally generated MAT2 IDs. Furthermore, if MID is greater than 400,000,000 then A1, A2, and A3 are a special format. They are $[G4] \cdot [\alpha 4]$ not $[\alpha 4]$. If MIDs larger than 99999999 are used, PARAM,NOCOMPS,-1 must be specified to obtain stress output.
14. In nonlinear static analysis (SOL 106) the QUAD4, TRIA3, QUADR, and TRIAR thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPSILONT,INTEGRAL' in bulk data. See "**Parameters**" on page 659.
15. Negative values for ST, SC, and SS lead to no margins of safety being computed.

MAT3 CTRIAX6 Material Property Definition

Defines the material properties for linear orthotropic materials used by the CTRIAX6 element entry.

Format:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	EX	ETH	EZ	NUXTH	NUTHZ	NUZX	RHO	
			GZX	AX	ATH	AZ	TREF	GE	

Example:

MAT3	23	1.0+7	1.1+7	1.2+7	.3	.25	.27	1.0-5	
			2.5+6	1.0-4	1.0-4	1.1-4	68.5	.23	

Field	Contents
MID	Material identification number. (Integer > 0)
EX, ETH, EZ	Young's moduli in the x, θ, and z directions, respectively. (Real > 0.0)
NUXTH, NUTHZ NUZX	Poisson's ratios (coupled strain ratios in the $x\theta$, $z\theta$, and zx directions, respectively). (Real)
RHO	Mass density. (Real)
GZX	Shear modulus. (Real > 0.0)
AX, ATH, AZ	Thermal expansion coefficients. (Real)
TREF	Reference temperature for the calculation of thermal loads or a temperature-dependent thermal expansion coefficient. See Remark **10.** (Real or blank)
GE	Structural element damping coefficient. See Remarks **9.** and **11.** (Real)

Remarks:

1. The material identification number must be unique with respect to the collection of all MAT1, MAT2, MAT3, and MAT9 entries.

2. MAT3 materials may be made temperature dependent by use of the MATT3 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE (INIT) Case Control command.
3. All seven of the numbers EX, ETH, EZ, NUXTH, NUTHZ, NUZX, and GZX must be present.
4. A warning message will be issued if any value of NUXTH or NUTHZ has an absolute value greater than 1.0.
5. MAT3 materials may only be referenced by the CTRIAX6 entry.
6. The mass density RHO will be used to automatically compute mass for the CTRIAX6 element.
7. The x-axis lies along the material axis (see **Figure 8-74** in the CTRIAX6 entry). The θ-axis lies in the azimuthal direction. The z-axis is normal to both.
8. The strain-stress relationship is

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_\theta \\ \varepsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{EX} & -\frac{NUTHX}{ETH} & -\frac{NUZX}{EZ} & 0 \\ -\frac{NUXTH}{EX} & \frac{1}{ETH} & -\frac{NUZTH}{EZ} & 0 \\ -\frac{NUXZ}{EX} & -\frac{NUTHZ}{ETH} & \frac{1}{EZ} & 0 \\ 0 & 0 & 0 & \frac{1}{GZX} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_\theta \\ \sigma_z \\ \sigma_{zx} \end{Bmatrix} + (T - \mathrm{TREF}) \begin{Bmatrix} AX \\ ATH \\ AZ \\ 0 \end{Bmatrix}$$

Note that:

$$\frac{NUXTH}{EX} = \frac{NUTHX}{E\theta}$$

$$\frac{NUZX}{EZ} = \frac{NUXZ}{EX}$$

$$\frac{UTHZ}{ETH} = \frac{UZTH}{EZ}$$

9. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.

10. TREF is used for two different purposes:
    - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Remark **10.** under the MAT1 description.
    - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must be blank.
11. If PARAM,W4 is not specified, GE is ignored in transient analysis. See “**Parameters**” on page 659.

MAT4 Heat Transfer Material Properties, Isotropic

Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.

Format:

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K	CP	ρ	H	μ	HGEN	REFENTH	
	TCH	TDELTA	QLAT						

Example:

MAT4	1	204.	.900	2700.					

Field	Contents
MID	Material identification number. (Integer > 0)
K	Thermal conductivity. (Blank or Real > 0.0)
CP	Heat capacity per unit mass at constant pressure (specific heat). (Blank or Real ≥ 0.0)
ρ	Density. (Real > 0.0; Default = 1.0)
H	Free convection heat transfer coefficient. (Real or blank)
μ	Dynamic viscosity. See Remark 2. (Real > 0.0 or blank)
HGEN	Heat generation capability used with QVOL entries. (Real ≥ 0.0; Default = 1.0)
REFENTH	Reference enthalpy. (Real or blank)
TCH	Lower temperature limit at which phase change region is to occur. (Real or blank)
TDELTA	Total temperature change range within which a phase change is to occur. (Real ≥ 0.0 or blank)
QLAT	Latent heat of fusion per unit mass associated with the phase change. (Real > 0.0 or blank)

Remarks:

1. The MID must be unique with respect to all other MAT4 and MAT5 entries. MAT4 may specify material properties for any conduction elements as well as properties for a forced convection fluid (see CONVM). MAT4 also provides the heat transfer coefficient for free convection (see CONV).
2. For a forced convection fluid, μ must be specified.
3. REFENTH is the enthalpy corresponding to zero temperature if the heat capacity CP is a constant. If CP is obtained through a TABLEM lookup, REFENTH is the enthalpy at the first temperature in the table.
4. Properties specified on the MAT4 entry may be defined as temperature dependent by use of the MATT4 entry.

MAT5 Thermal Material Property Definition

Defines the thermal material properties for anisotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT5	MID	KXX	KXY	KXZ	KYY	KYZ	KZZ	CP	
	RHO	HGEN							

Example:

MAT5	24	.092			.083		0.20	0.2	
	2.00								

Field	Contents
MID	Material identification number. (Integer > 0)
Kij	Thermal conductivity. (Real)
CP	Heat capacity per unit mass. (Real $\geq$ 0.0 or blank)
RHO	Density. (Real > 0.0; Default = 1.0)
HGEN	Heat generation capability used with QVOL entries. (Real $\geq$ 0.0; Default = 1.0)

Remarks:

1. The thermal conductivity matrix has the following form:

$$K = \begin{bmatrix} KXX & KXY & KXZ \\ KXY & KYY & KYZ \\ KXZ & KYZ & KZZ \end{bmatrix}$$

2. The material identification number may be the same as a MAT1, MAT2, or MAT3 entry but must be unique with respect to other MAT4 or MAT5 entries.
3. MAT5 materials may be made temperature-dependent by use of the MATT5 entry.

4. When used for axisymmetric analysis (CTRIAX6), material properties are represented where:

   KXX = radial conductivity component
   KYY = axial conductivity component

MAT8 Shell Element Orthotropic Material Property Definition

Defines the material property for an orthotropic material for isoparametric shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	GE	F12	STRN						

Example:

MAT8	171	30.+6	1.+6	0.3	2.+6	3.+6	1.5+6	0.056	
	28.-6	1.5-6	155.0	1.+4	1.5+4	2.+2	8.+2	1.+3	
	1.-4		1.0						

Field	Contents
MID	Material identification number. Referenced on a PSHELL or PCOMP entry only. (0 < Integer < 100,000,000)
E1	Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction. (Real $\neq$ 0.0)
E2	Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction. (Real $\neq$ 0.0)
NU12	Poisson's ratio ($\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1-direction). Note that $\upsilon_{21} = \varepsilon_1/\varepsilon_2$ for uniaxial loading in 2-direction is related to υ_{12}, E_1, and E_2 by the relation $\upsilon_{12}E_2 = \upsilon_{21}E_1$. (Real)
G12	In-plane shear modulus. (Real $\geq$ 0.0; Default = 0.0)
G1Z	Transverse shear modulus for shear in 1-Z plane. (Real > 0.0; Default implies infinite shear modulus.)
G2Z	Transverse shear modulus for shear in 2-Z plane. (Real > 0.0; Default implies infinite shear modulus.)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient in i-direction. (Real)

Field	Contents
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks **4.** and **5.** (Real or blank)
Xt, Xc	Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. Required if failure index is desired. See the FT field on the PCOMP entry. (Real > 0.0; Default value for Xc is Xt.)
Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the lateral direction. Required if failure index is desired. (Real > 0.0; Default value for Yc is Yt.)
S	Allowable stress or strain for in-plane shear. See the FT field on the PCOMP entry. (Real > 0.0)
GE	Structural damping coefficient. See Remarks **4.** and **6.** (Real)
F12	Interaction term in the tensor polynomial theory of Tsai-Wu. Required if failure index by Tsai-Wu theory is desired and if value of F12 is different from 0.0. See the FT field on the PCOMP entry. (Real)
STRN	For the maximum strain theory only (see STRN in PCOMP entry). Indicates whether Xt, Xc, Yt, Yc, and S are stress or strain allowables. [Real = 1.0 for strain allowables; blank (Default) for stress allowables.]

Remarks:

1. The material identification numbers must be unique for all MAT1, MAT2, MAT3, and MAT8 entries.
2. If G1Z and G2Z values are specified as zero or blank, then transverse shear flexibility calculations will not be performed, which is equivalent to zero shear flexibility (i.e., infinite shear stiffness).
3. An approximate value for G1Z and G2Z is the in-plane shear modulus G12. If test data are not available to accurately determine G1Z and G2Z for the material and transverse shear calculations are deemed essential; the value of G12 may be supplied for G1Z and G2Z. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed in the TEMPERATURE Case Control command.
4. Xt, Yt, and S are required for composite element failure calculations when requested in the FT field of the PCOMP entry. Xc and Yc are also used but not required.
5. TREF and GE are ignored if this entry is referenced by a PCOMP entry.

6. TREF is used in two different ways:
   - In nonlinear static analysis (SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See **Figure 8-97** in Remark **10.** in the MAT1 description.
   - In all SOLs except 106, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must then be blank.
7. If PARAM,W4 is not specified, GE is ignored in transient analysis. See "**Parameters**" on page 659.
8. In nonlinear static analysis (SOL 106) the QUAD4 and TRIA3 thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPSILONT,INTEGRAL' in bulk data. See "**Parameters**" on page 659.

MAT9 Solid Element Anisotropic Material Property Definition

Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see PSOLID entry description).

Format:

1	2	3	4	5	6	7	8	9	10
MAT9	MID	G11	G12	G13	G14	G15	G16	G22	
	G23	G24	G25	G26	G33	G34	G35	G36	
	G44	G45	G46	G55	G56	G66	RHO	A1	
	A2	A3	A4	A5	A6	TREF	GE		

Example:

MAT9	17	6.2+3						6.2+3	
					6.2+3				
	5.1+3			5.1+3		5.1+3	3.2	6.5-6	
	6.5-6					125.	.003		

Field	Contents
MID	Material identification number. (Integer > 0)
Gij	Elements of the 6×6 symmetric material property matrix in the material coordinate system. (Real)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation thermal loads, or a temperature-dependent thermal expansion coefficient. See Remark **7.** (Real or blank)
GE	Structural element damping coefficient. See Remarks **6.** and **8.** (Real)

Remarks:

1. The material identification numbers must be unique for all MAT1, MAT2, MAT3, and MAT9 entries.

2. MAT9 materials may be made temperature-dependent by use of the MATT9 entry. In nonlinear static analysis (e.g., SOL 106), linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
3. The mass density RHO will be used to compute mass in a structural dynamics problem automatically.
4. The third continuation entry is optional.
5. The subscripts 1 through 6 refer to x, y, z, xy, yz, and zx of the material coordinate system (see the CORDM field on the PSOLID entry description). The stress-strain relationship is

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} G11 & G12 & G13 & G14 & G15 & G16 \\ & G22 & G23 & G24 & G25 & G26 \\ & & G33 & G34 & G35 & G36 \\ & & & G44 & G45 & G46 \\ & \text{symmetric} & & & G55 & G56 \\ & & & & & G66 \end{bmatrix} \left[\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - \begin{Bmatrix} A1 \\ A2 \\ A3 \\ A4 \\ A5 \\ A6 \end{Bmatrix} (T - \text{TREF}) \right]$$

6. The damping coefficient GE is given by

$$\text{GE} = \frac{2.0 \cdot C}{C_0}$$

7. TREF is used in two different ways:
   - In nonlinear static analysis (e.g., SOL 106), TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See **Figure 5-91** in Remark **10.** in the MAT1 description.
   - In all solutions except nonlinear static analysis, TREF is used only as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, but TREF must then be blank.
8. If PARAM,W4 is not specified, GE is ignored in transient analysis. See "**Parameters**" on page 659.

MAT10 Fluid Material Property Definition

Defines material properties for fluid elements in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
MAT10	MID	BULK	RHO	C	GE				

Example:

MAT10	103	0.656	0.011						

Field	Contents
MID	Material identification number. (Integer > 0)
BULK	Bulk modulus. (Real > 0.0)
RHO	Mass density. (Real > 0.0)
C	Speed of sound. (Real > 0.0)
GE	Fluid element damping coefficient. (Real)

Remarks:

1. MAT10 is referenced, with MID, by the PSOLID entry only.
2. The material identification numbers must be unique for all MAT1, MAT2, MAT3, MAT9, and MAT10 entries.
3. The mass density RHO will be used to compute the mass automatically.
4. BULK, RHO, and C are related by

$$\text{BULK} = C^2 \cdot \text{RHO}$$

   Two out of the three must be specified, and the other will be calculated according to this equation.
5. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
6. If PARAM,W4FL is not specified, GE is ignored in transient analysis. See “**Parameters**” on page 659.

MATD001 (SOL 700) LS-DYNA Material #1 -- Isotropic Elastic

Isotropic elastic material available for beam, shell and solid elements. A specialization of this material allows the modeling for fluids. The fluid option is valid for solid elements only. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD001	MID	RO	E	PR	DA	DB	K	Option	

Define the following extra entry for OPTION=FLUID only:

Example:

1	2	33	44	5	6	7	8	9	10
VC	CP								

Field	Contents
MID	Material identification. A unique number has to be chosen. (Real)
RO	Mass density. (Real)
E	Young's modulus. (Real)
PR	Poisson's ratio. (Real)
DA	Axial damping factor (used for Belytschko-Schwer beam only). (Real, Default = 0.0)
DB	Bending damping factor (used for Belytschko-Schwer beam only). (Real, Default = 0.0)
K	Bulk Modulus (define for fluid option only). (Real, Default = 0.0)
Option	Blank or FLUID. (Character)
VC	Tensor viscosity coefficient, values between .1 and .5 should be okay. (Real)
CP	Cavitation pressure (Real, Default = 1.0e+20).

Remarks:

The axial and bending damping factors are used to damp down numerical noise. The update of the force resultants, F_i, and moment resultants, M_i, includes the damping factors:

$$F_i^{n+1} = F_i^n + \left(1 + \frac{DA}{\Delta t}\right)\Delta F_i^{n+1/2}$$

$$M_i^{n+1} = M_i^n + \left(1 + \frac{DB}{\Delta t}\right)\Delta M_i^{n+1/2}$$

For the fluid option the bulk modulus (K) has to be defined as Young's modulus, and Poission's ratio are ignored. With the fluid option fluid-like behavior is obtained where the bulk modulus, K, and pressure rate, p, are given by:

$$K = \frac{E}{3(1-2\nu)}$$

$$\dot{p} = -K\dot{\varepsilon}_{ii}$$

and the shear modulus is set to zero. A tensor viscosity is used which acts only the deviatoric stresses, S_{ij}^{n+1}, given in terms of the damping coefficient as:

$$S_{ij}^{n+1} = VC \cdot \Delta L \cdot \alpha \cdot \rho\dot{\varepsilon}_{ij}^{'}$$

where ρ, is a characteristic element length, α is the fluid bulk sound speed, ρ is the fluid density, and $\dot{\varepsilon}_{ij}^{'}$ is the deviatoric strain rate.

MATD2AN (SOL 700) LS-DYNA Material #2 -- Anisotropic

LS-DYNA style material # 2 for modeling the elastic-anisotropic behavior of solids, shells, and thick shells. Defines material properties for anisotropic materials in the LS-DYNA style. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Card Format of Cards 1-5

1	2	3	4	5	6	7	8	9	10
MATD2AN	MID	RO	C11	C12	C22	C13	C23	C33	
	C14	C24	C34	C44	C15	C25	C35	C45	
	C55	C16	C26	C36	C46	C56	C66	AOPT	
	XP	YP	ZP	A1	A2	A3			
	V1	V2	V3	D1	D2	D3	BETA	REF	

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
Due to symmetry only define the upper triangular Cij's	
C11	The 1,1 term in the 6×6 anisotropic constitutive matrix. Note that 1 corresponds to the *a* material direction. (Real)
C12	The 1,2 term in the 6×6 anisotropic constitutive matrix. Note that 2 corresponds to the *b* material direction. (Real)
.	. (Real)
.	. (Real)
.	. (Real)
C66	The 6,6 term in the 6×6 anisotropic constitutive matrix. (Real)
AOPT	Material axes option (Integer) EQ. 0: locally orthotropic with material axes determined by element nodes. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system. EQ. 1: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only.

Field	Contents
	EQ. 2: globally orthotropic with material axes determined by vectors defined below.
	EQ. 3: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.
	EQ. 4.0: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector v, and an originating point, P, which define the centerline axis. This option is for solid elements only.
G	Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF. This option applies only to solid elements.
SIGF	Limit stress for frequency independent, frictional, damping.
xp yp zp	Define coordinates of point p for AOPT = 1 and 4. (Real)
a1 a2 a3	Define components of vector a for AOPT = 2. (Real)
v1 v2 v3	Define components of vector v for AOPT = 3 and 4. (Real)
d1 d2 d3	Define components of vector d for AOPT = 2: (Real)
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element entry. (Real)
REF	Use reference geometry to initialize the stress tensor. This option is currently restricted to 8-noded solid elements with one point integration. (Integer)
	EQ.0: off,
	EQ.1: on.

Remarks:

The material law that relates stresses to strains is defined as:

$$\underset{\sim}{C} = \underset{\sim}{T}^T \underset{\sim}{C}_L \underset{\sim}{T}$$

where T is a transformation matrix, and $\underset{\sim}{C}_L$ is the constitutive matrix defined in terms of the material constants of the orthogonal material axes, a, b, and c. The inverse of $\underset{\sim}{C}_L$ for the orthotropic case is defined as:

$$\underset{\sim}{C}_L^{-1} = \begin{bmatrix} \frac{1}{E_a} & -\frac{\nu_{ba}}{E_b} & -\frac{\nu_{ca}}{E_c} & 0 & 0 & 0 \\ -\frac{\nu_{ab}}{E_a} & \frac{1}{E_b} & -\frac{\nu_{cb}}{E_c} & 0 & 0 & 0 \\ -\frac{\nu_{ac}}{E_a} & -\frac{\nu_{bc}}{E_b} & \frac{1}{E_c} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ab}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{bc}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}} \end{bmatrix}$$

Note that $\frac{\nu_{ab}}{E_a} = \frac{\nu_{ba}}{E_b}, \frac{\nu_{ca}}{E_c} = \frac{\nu_{ac}}{E_a}, \frac{\nu_{cb}}{E_c} = \frac{\nu_{bc}}{E_b}$

MATD2OR (SOL 700) LS-DYNA Material #2 -- Orthotropic

LS-DYNA material #2 for modeling the elastic-orthotropic behavior of solids, shells, and thick shells. For orthotropic solids and isotropic frictional damping is available. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format of entries 1-4 for the ORTHO option:

1	2	3	4	5	6	7	8	9	10
MATD2OR	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB	
	GAB	GBC	GCA	AOPT	G	SIGF			
	XP	YP	ZP	A1	A2	A3			
	V1	V2	V3	D1	D2	D3	BETA	REF	

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
EA	E_a, Young's modulus in a-direction. (Real)
EB	E_b, Young's modulus in b-direction. (Real)
EC	E_c, Young's modulus in c-direction (nonzero value required but not used for shells). (Real)
PRBA	ν_{ba}, Poisson's ratio ba. (Real)
PRCA	ν_{ca}, Poisson's ratio ca (solids only). (Real)
PRCB	ν_{cb}, Poisson's ratio cb (solids only).
GAB	G_{ab}, shear modulus ab. (Real)
GBC	G_{bc}, shear modulus bc. (Real)
GCA	G_{ca}, shear modulus ca. (Real)
AOPT	Material axes option: (Integer)
	EQ. 0: locally orthotropic with material axes determined by element nodes. Nodes 1, 2, and 4 of an element are identical to the nodes used for the definition of a coordinate system.
	EQ. 1: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only.

Field	Contents
	EQ. 2: globally orthotropic with material axes determined by vectors defined below.
	EQ. 3: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal. The plane of a solid element is the midsurface between the inner surface and outer surface defined by the first four nodes and the last four nodes of the connectivity of the element, respectively.
	EQ. 4: locally orthotropic in cylindrical coordinate system with the material axes determined by a vector *v*, and an originating point, P, which define the centerline axis. This option is for solid elements only.
G	Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF. This option applies only to solid elements. (Real)
SIGF	Limit stress for frequency independent, frictional, damping. (Real)
XP YP ZP	Define coordinates of point p for AOPT = 1 and 4. (Real)
A1 A2 A3	Define components of vector a for AOPT = 2. (Real)
V1 V2 V3	Define components of vector v for AOPT = 3 and 4. (Real)
D1 D2 D3	Define components of vector d for AOPT = 2: (Real)
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element entry. (Real)
REF	Use reference geometry to initialize the stress tensor. This option is currently restricted to 8-noded solid elements with one point integration. (Real)
	EQ.0: off. (Real)
	EQ.1: on. (Integer)

Remarks:

The material law that relates stresses to strains is defined as:

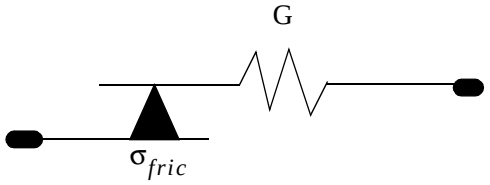
$$\underset{\sim}{C} = \underset{\sim}{T}^T \underset{\sim}{C}_L \underset{\sim}{T}$$

where T is a transformation matrix, and $\underset{\sim}{C}_L$ is the constitutive matrix defined in terms of the material constants of the orthogonal material axes, a, b, and c. The inverse of $\underset{\sim}{C}_L$ for the orthotropic case is defined as:

$$\underset{\sim}{C}_L^{-1} = \begin{bmatrix} \frac{1}{E_a} & -\frac{\nu_{ba}}{E_b} & -\frac{\nu_{ca}}{E_c} & 0 & 0 & 0 \\ -\frac{\nu_{ab}}{E_a} & \frac{1}{E_b} & -\frac{\nu_{cb}}{E_c} & 0 & 0 & 0 \\ -\frac{\nu_{ac}}{E_a} & -\frac{\nu_{bc}}{E_b} & \frac{1}{E_c} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{ab}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{bc}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{ca}} \end{bmatrix}$$

Note that $\frac{\nu_{ab}}{E_a} = \frac{\nu_{ba}}{E_b}, \frac{\nu_{ca}}{E_c} = \frac{\nu_{ac}}{E_a}, \frac{\nu_{cb}}{E_c} = \frac{\nu_{bc}}{E_b}$

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



This option applies only to orthotropic solid elements and affects only the deviatoric stresses.

MATD003 (SOL 700) LS-DYNA Material #3 -- Isotropic with Kinematic Hardening

Used to model isotropic and kinematic hardening plasticity with the option of including rate effects. It is a very cost effective model and is available for beam (Hughes-Liu), shell, and solid elements. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD003	MID	RO	E	PR	SIGY	ETAN	BETA		
	SRC	SRP	FS	VP					

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
E	Young's modulus. (Real)
PR	Poisson's ratio. (Real)
SIGY	Yield stress. (Real)
ETAN	Tangent modulus, see **Figure 8-98**. (Real, Default = 0.0)
BETA	Hardening parameter, $0 < \beta' < 1$ See comments below. (Real, Default = 0.0)
SRC	Strain rate parameter, C, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered. (Real)
SRP	Strain rate parameter, P, for Cowper Symonds strain rate model, see below. If zero, rate effects are not considered. (Real)
FS	Failure strain for eroding elements. (Real)
VP	Formulation for rate effects: (Integer, Default = 0.0)
	EQ.0: Scale yield stress (default),
	EQ.1: Viscoplastic formulation

Remarks:

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. A fully viscoplastic formulation is optional which incorporates the Cowper and Symonds formulation within the yield surface. An additional cost is incurred but the improvement is results can be dramatic. To ignore strain rate effects set both SRC and SRP to zero.

Kinematic, isotropic, or a combination of kinematic and isotropic hardening may be specified by varying β' between 0 and 1. For β' equal to 0 and 1, respectively, kinematic and isotropic hardening are obtained as shown in **Figure 8-98**. For isotropic hardening, $\beta = 1$, Material Model MATD012, requires less storage and is more efficient. Whenever possible, Material 12 is recommended for solid elements, but for shell elements it is less accurate and thus material 3 should be used.

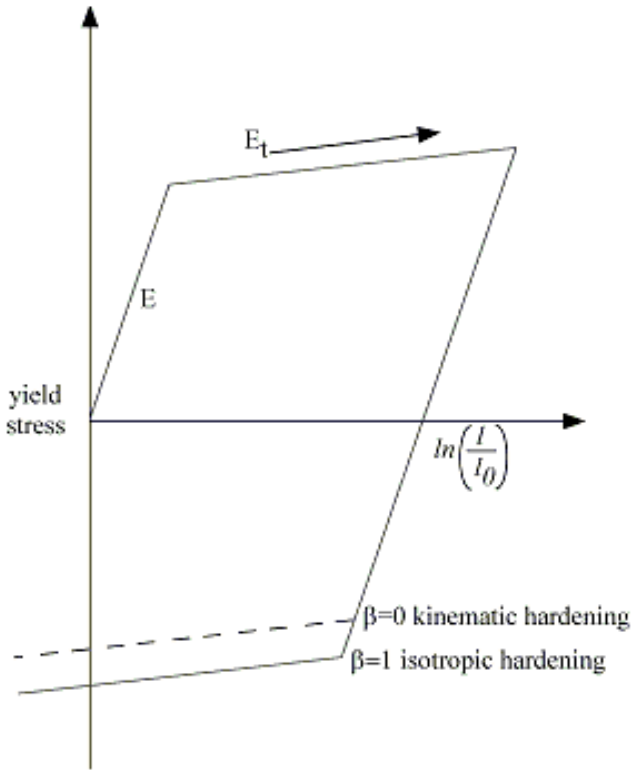


Figure 8-98

Elastic-plastic behavior with kinematic and isotropic hardening where l_0 and l are undeformed and deformed lengths of uniaxial tension specimen. E_t is the slope of the bilinear stress strain curve.

MATD005 (SOL 700) LS-DYNA Material #5 -- Soil and Foam

LS-DYNA Material #5 -- Used to model soil and foam. This is a very simple model and works in some ways like a fluid. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD005	MID	RO	G	BULK	A0	A1	A2	PC	
	VCR	REF							
	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8	
	EPS9	EPS10							
	P1	P2	P3	P4	P5	P6	P7	P8	
	P9	P10							

Example:

MATD005	22	4.562	4.1e9						
	.4	.6							
	4.3	7.8							

Field	Contents
MID	Material identification. A unique number has to be chosen. (Required, Integer)
RO	Mass density. (Required, Real > 0)
G	Shear modulus. (Required, Real > 0)
K	Bulk modulus for unloading used for VCR=0.0. (Real > 0)
A0	Yield function constant for plastic yield function shown below. (Real > 0)

Field	Contents
A1	Yield function constant for plastic yield function shown below. (Real > 0)
A2	Yield function constant for plastic yield function shown below. (Real > 0)
PC	Pressure cutoff for tensile fracture. (Real > 0)
VCR	Volumetric crushing option: (Real > 0) EQ.0.0: on, EQ.1.0: loading and unloading paths are the same.
EPS1,.....	Volumetric strain values (natural logarithmic values), see comments below. A maximum of 10 values are allowed and a minimum of 2 values are necessary. The tabulated values must completely cover the expected values in the analysis. If the first value is not for a volumetric strain value of zero then the point (0.0,0.0) will be automatically generated and up to a further nine additional values may be defined. (Real)
p1, p2,..pn	Pressures corresponding to volumetric strain values. (Real > 0)

Remarks:

Pressure is positive in compression. Volumetric strain is given by the natural log of the relative volume and is negative in compression. Relative volume is ratio of the current volume to the initial volume at the start of the calculation. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value. For a detailed description we refer to Kreig [1972].

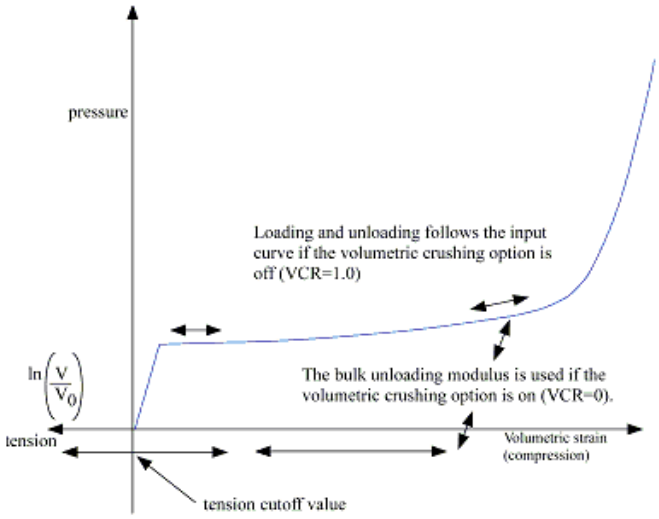


Figure 8-99 Pressure versus volumetric strain curve for soil and crushable foam model. The volumetric strain is given by the natural logarithm of the relative volume, V.

The deviatoric perfectly plastic yield function, ϕ, is described in terms of the second invariant J_2,

$$J_2 = \frac{1}{2}s_{ij}s_{ij},$$

pressure, p, and constants a_0, a_1, and a_2 as:

$$\phi = J_2 - [a_0 + a_1 p + a_2 p^2]$$

On the yield surface $J_2 = 1/3\sigma_y^2$ where σ_y is the uniaxial yield stress, i.e.,

$$\sigma_y = [3(a_0 + a_1 p + a_2 p^2)]^{1/2}$$

There is no strain hardening on this surface.

To eliminate the pressure dependence of the yield strength, set:

$$a_1 = a_2 = 0 \qquad a_0 = \frac{1}{3}\sigma_y^2$$

This approach is useful when a von Mises type elastic-plastic model is desired for use with the tabulated volumetric data.

MATD006 (SOL 700) LS-DYNA Material #6 -- Viscoelastic

LS-DYNA Material #6 -- Used to model the viscoelastic behavior of beams (Hughes-Liu), shells, and solids. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD006	MID	RO	BULK	G0	GI	BETAS			

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
BULK	Elastic bulk modulus. (Real)
G0	Short-time shear modulus, see equations below. (Real)
GI	Long-time (infinite) shear modulus, G_∞. (Real)
BETA	Decay constant. (Real)

Remarks:

The shear relaxation behavior is described by [Herrmann and Peterson, 1968]:

$$G(t) = G_\infty + (G_0 - G_\infty)e^{-\beta t}$$

A Jaumann rate formulation is used

$$\sigma^{\nabla'}_{ij} = 2\int_0^t G(t-\tau)D'_{ij}(\tau)d\tau$$

where the prime denotes the deviatoric part of the stress rate, σ^{∇}_{ij}, and the strain rate D_{ij}.

MATD007 (SOL 700) LS-DYNA Material #7 -- Nearly Incompressible Rubber

LS-DYNA Material #7 -- Used to model nearly incompressible continuum rubber. The Poisson's ratio is fixed to 0.463. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD007	MID	RO	G	REF					

Field	Contents
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
G	Shear modulus.
REF	Use reference geometry to initialize the stress tensor. This option is currently restricted to 8-noded solid elements with one point integration. EQ.0: off, EQ.1: on.

Remarks:

The second Piola-Kirchhoff stress is computed as

$$S_{ij} = G\left[\frac{1}{V}C_{ij} - V^{\left(\frac{1}{1-2\nu}\right)}\delta_{ij}\right]$$

where V is the relative volume defined as being the ratio of the current volume to the initial volume, C_{ij} is the right Cauchy-Green strain tensor, and ν is Poisson's ratio, which is set to .463 internally. This stress measure is transformed to the Cauchy stress, σ_{ij}, according to the relationship

$$\sigma_{ij} = V^{-1}F_{ik}F_{jl}S_{lk}$$

where F_{ij} is the deformation gradient tensor. Also see Blatz and Ko [1962].

MATD009 (SOL 700)

This material allows equations of state to be considered without computing deviatoric stresses. Optionally, a viscosity can be defined. Also, erosion in tension and compression is possible.

Sometimes it is advantageous to model contact surfaces via shell elements which are not part of the structure, but are necessary to define areas of contact within nodal rigid bodies or between nodal rigid bodies.

Beams and shells that use this material type are completely bypassed in the element processing; however, the mass of the null shell elements is computed and added to the nodal points which define the connectivity, but the mass of null beams is ignored. The Young's modulus and Poisson's ratio are used only for setting the contact interface stiffnesses, and it is recommended that reasonable values be input.

Format:

1	2	3	4	5	6	7	8	9	10
MATD009	MID	RO	PC	MU	TEROD	CEROD	YM	PR	

Example:

MATD009	1	4.65E-5							

Field	Contents	Type	Default
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique.	I > 0	Required
RO	Mass density	R > 0	Required
PC	Pressure cutoff.	R 0.0	0.0
MU	Dynamic viscosity coefficient μ (optional)	R > 0.0	0.0
TEROD	Relative volume. V/V_0, for erosion in tension. Typically, use values greater than unity. If zero, erosion in tension is inactive.	R > 0.0	0.0
CEROD	Relative volume, , for erosion in compression.Typically, use values less than unity. If zero, erosion in compression is inactive	R > 0.0	0.0

Field	Contents	Type	Default
YM	Young's modulus (used for null beams and shells only)	R > 0.0	0.0
PR	Poisson's ratio (used for null beams and shells only)	R > 0.0	0.0

Remarks:

1. The null material must be used with an equation of-state. Pressure cutoff is negative in tension. A (deviatoric) viscous stress of the form

$$\sigma_{ij} = \mu\dot{\varepsilon}_{ij}$$

$$\left|\frac{N}{m^2}\right| \approx \left|\frac{N}{m^2}s\right|\left|\frac{1}{S}\right|$$

   is computed for nonzero μ where $\dot{\varepsilon}_{ij}$ is the deviatoric strain rate. μ is the dynamic viscosity with unit of [Pascal*second].

2. The null material has no shear stiffness and hourglass control must be used with great care. In some applications, the default hourglass coefficient might lead to significant energy losses. In general, for fluid(s), the hourglass coefficient QM should be small (in the range 1.0E-4 to 1.0E-6 in the SI unit system for the standard default IHQ choice).
3. The Null material has no yield strength and behaves in a fluid-like manner.
4. The pressure cut-off, PC, must be defined to allow for a material to "numerically" cavitate. In other words, when a material undergoes dilatation above certain magnitude, it should no longer be able to resist this dilatation. Since dilatation stress or pressure is negative, setting PC limit to a very small negative number would allow for the material to cavitate once the pressure in the material goes below this negative value.

MATD010 (SOL 700) LS-DYNA Material #10 -- Elastic-Plastic-Hydrodynamic

LS-Dyna Material #10 – This material allows the modeling of an elastic-plastic hydrodynamic material with or without spall.

Merges MATD020 rigid bodies into one assembly.

Format:

1	2	3	4	5	6	7	8	9	10
MATD010	MID	RHO	G	SIGY	EH	PC	FS		
	A1	A2	SPALL						
	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8	
	EPS9	EPS10	EPS11	EPS12	EPS13	EPS14	EPS15	EPS16	
	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8	
	ES9	ES10	E11	ES12	ES13	ES14	ES15	ES16	

Example:

1	2	3	4	5	6	7	8	9	10
MATD010	101	7.4E-4	10.5E6	30000.	15000.	0.0	.25		
	1.0	0.4	2.0						
	0.0	.05	.1	.15	.2	.25	.3	.35	
	.4	.5	1.0						
	30000.	33000.	34000.	35000.	36000.	37000.	38000.	39000.	
	40000.	40500.	40600.						

Field	Contents
MID	Material identification. A unique number must be selected.
RHO	Mass density (Real > 0.0, No Default)
G	Shear Modulus (Real > 0.0, No Default)
SIGY	Yield stress, see comments (Real > 0.0, No Default)
EH	Plastic hardening modulus, see comments (Real > 0.0, No Default)
PC	Pressure cutoff (Real $\leq$ 0.0, No Default)
FS	Failure strain for erosion (Real > 0.0, No Default)

Field	Contents
A1	Linear pressure hardening coefficient (Real, No Default)
A2	Quadratic pressure hardening coefficient (Real, No Default)
SPALL	Spall type (Real, Default = 1.0)
	– Same as 1.0
	– P ≥ Pcut
	– If max stress ≥ -Pcur, element spalls and tension, P < 0 is not allowed
	- If P, -Pcut, element spalls and tension, P < 0 is not allowed
EPSi	Effective (true) plastic strains. (Real, No Default) Up to 16 values in increasing order may be defined. Linear extrapolation is used if strain exceeds max input value.
ESi	Effective stress values corresponding to EPSi (Real, No Default). As many ESi values as EPSi values entered must be defined.

Remarks:

If ES and EPS are undefined, the yield stress and plastic hardening modulus are taken from SIGY and EH. In this case, the bilinear stress-strain curve shown in **Figure 8-100**. is obtained with hardening parameter, $\beta = 1$. The yield strength is calculated as

$$\sigma_y = \sigma_0 + E_h\bar{\varepsilon}^p + (a_1 + pa_2)\max[p, 0]$$

The quantity Eh is the plastic hardening modulus defined in terms of Young's modulus, E, and the tangent modulus, E_t, as follows

and p is the pressure taken as positive in compression.

If ES and EPS are specified, a curve like that shown in **Figure 8-100**. may be defined. Effective stress is defined in terms of the deviatoric stress tensor, s_{ij}, as:

$$\bar{\sigma} = \left(\frac{3}{2}s_{ij}s_{ij}\right)^{1/2}$$

and effective plastic strain by:

$$\bar{\varepsilon}^p = \int_0^t \left(\frac{2}{3}D_{ij}^p D_{ij}^p\right)^{1/2} dt$$

where t denotes time and is the plastic component of the rate of deformation tensor. In this case the plastic hardening modulus on Card 1 is ignored and the yield stress is given as

$$\sigma_y = f(\bar{\varepsilon}^p)$$

where the value for $f(\bar{\varepsilon}^p)$ is found by interpolation from the data curve.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model, OPT=1, limits the hydrostatic tension to the specified value, p_{cut}. If pressures more tensile than this limit are calculated, the pressure is reset to p_{cut}. This option is not strictly a spall model, since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff, and the pressure cutoff value, p_{cut}, remains unchanged throughout the analysis. The maximum principal stress spall model, OPT=2, detects spall if the maximum principal stress, σ_{max}, exceeds the limiting value $-p_{cut}$. Note that the negative sign is required because p_{cut} is measured positive in compression, while σ_{max} is positive in tension. Once spall is detected with this model, the deviatoric stresses are reset to zero, and no hydrostatic tension ($p < 0$) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material. The hydrostatic tension spall model, OPT=3, detects spall if the pressure becomes more tensile than the specified limit, p_{cut}. Once spall is detected the deviatoric stresses are reset to zero, and nonzero values of pressure are required to be compressive (positive). If hydrostatic tension ($p < 0$) is subsequently calculated, the pressure is reset to 0 for that element.

This model is applicable to a wide range of materials, including those with pressure-dependent yield behavior. The use of 16 points in the yield stress versus effective plastic strain curve allows complex post-yield hardening behavior to be accurately represented. In addition, the incorporation of an equation of state permits accurate modeling of a variety of different materials. The spall model options permit incorporation of material failure, fracture, and disintegration effects under tensile loads.

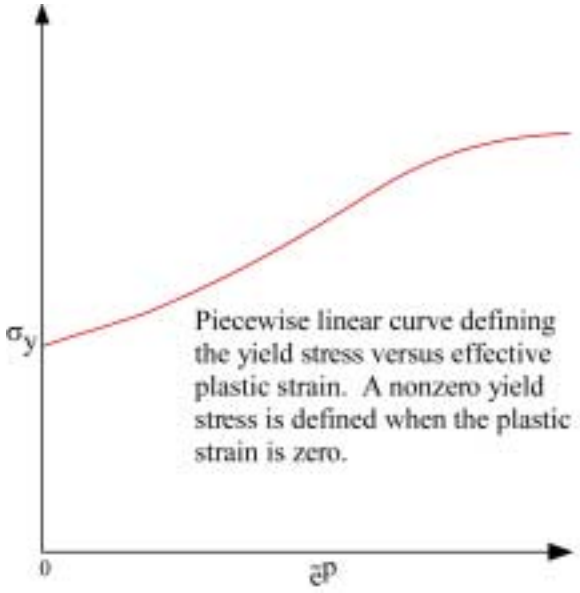


Figure 8-100 Effective Stress Versus Effect Plastic Strain Curve

MATD012 (SOL 700)

LS-DYNA Material #12 -- Low Cost Isotropic Plasticity Model for Solids

LS-DYNA Material #12 – This is a very low cost isotropic plasticity model for three-dimensional solids. In the plane stress implementation for shell elements, a one-step radial return approach is used to scale the Cauchy stress tensor if the state of stress exceeds the yield surface. This approach to plasticity leads to inaccurate shell thickness updates and stresses after yielding. This is the only model in LS-DYNA for plane stress that does not default to an iterative approach. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD012	MID	RO	G	SIGY	ETAN	BULK			

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
G	Shear modulus. (Real)
SIGY	Yield stress. (Real)
ETAN	Plastic hardening modulus. (Real)
BULK	Bulk modulus, K. (Real)

Remarks:

Here the pressure is integrated in time

$$\dot{p} = -K\dot{\varepsilon}_{ii}$$

where $\dot{\varepsilon}_{ii}$ is the volumetric strain rate.

MATD013 (SOL 700)

LS-DYNA Material #13 -- Non-Iterative Plasticity Model with Failure

LS-DYNA Material #13 -- This is a non-iterative plasticity with simple plastic strain failure model. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD013	MID	RO	G	SIGY	ETAN	BULK			
	EPF	PRF	REM	TREM					

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
G	Shear modulus. (Real)
SIGY	Yield stress. (Real)
ETAN	Plastic hardening modulus. (Real, Default = 0.0)
BULK	Bulk modulus. (Real)
EPF	Plastic failure strain. (Real)
PRF	Failure pressure (Real, Default = 0.0).
REM	Element erosion option: (Integer, Default = 0.0)
	EQ.0: failed element eroded after failure,
	NE.0: element is kept, no removal except by Δt below.
TREM	Δt for element removal: (Real, Default = 0.0)
	EQ.0.0: Δt is not considered (default),
	GT.0.0: element eroded if element time step size falls below Δt.

Remarks:

When the effective plastic strain reaches the failure strain or when the pressure reaches the failure pressure, the element loses its ability to carry tension and the deviatoric stresses are set to zero, i.e., the material behaves like a fluid. If Δt for element removal is defined the element removal option is ignored.

The element erosion option based on Δt must be used cautiously with the contact options. Nodes to surface contact is recommended with all nodes of the eroded brick elements included in the node list. As the elements are eroded the mass remains and continues to interact with the master surface.

MATD014 (SOL 700) LS-DYNA Material #14 -- Soil and Foam with Failure

LS-DYNA Material #14 -- The input for this model is the same as for MATD005; however, when the pressure reaches the failure pressure, the element loses its ability to carry tension. It should be used only in situations when soils and foams are confined within a structure or when geometric boundaries are present. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD014	MID	RO	G	BULK	A0	A1	A2	PC	
	VCR	REF							
	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8	
	EPS9	EPS10							
	P1	P2	P3	P4	P5	P6	P7	P8	
	P9	P10							

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
G	Shear modulus. (Real)
K	Bulk modulus for unloading used for VCR=0.0. (Real)
A0	Yield function constant for pressure dependent yield function shown below. (Real)
A1	Yield function constant for pressure dependent yield function shown below. (Real)
A2	Yield function constant for pressure dependent yield function shown below. (Real)
PC	Pressure cutoff for tensile fracture. (Real)
VCR	Volumetric crushing option: (Integer)
	EQ.0: on,
	EQ.1: loading and unloading paths are the same.

Field	Contents
REF	Use reference geometry to initialize the pressure. This option does not initialize the deviatoric stress state. (Integer) EQ.0: off, EQ.1: on.
EPS1,.....	Volumetric strain values (natural logarithmic values), see comments below. A maximum of 10 values are allowed and a minimum of 2 values are necessary. The tabulated values must completely cover the expected values in the analysis. If the first value is not for a volumetric strain value of zero then the point (0.0,0.0) will be automatically generated and up to a further nine additional values may be defined. (Real)
P1, P2,..PN	Pressures corresponding to volumetric strain values. (Real)

Remarks:

1. All continuation lines are required even if blank.
2. Pressure is positive in compression. Volumetric strain is given by the natural log of the relative volume and is negative in compression. Relative volume is ratio of the current volume to the initial volume at the start of the calculation. The tabulated data should be given in order of increasing compression. If the pressure drops below the cutoff value specified, it is reset to that value. For a detailed description we refer to Kreig [1972].

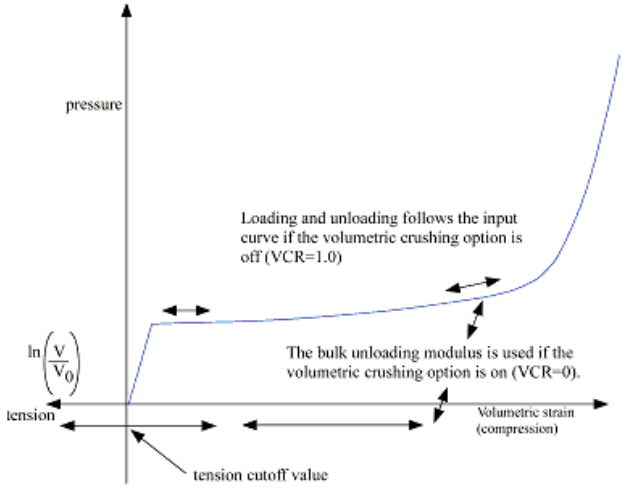


Figure 8-101 Pressure versus volumetric strain curve for soil and crushable foam model. The volumetric strain is given by the natural logarithm of the relative volume, V.

The deviatoric perfectly plastic yield function, ϕ, is described in terms of the second invariant J_2

$$J_2 = \frac{1}{2}s_{ij}s_{ij},$$

pressure, p, and constants a_0, a_1, and a_2 as:

$$\phi = J_2 - [a_0 + a_1 p + a_2 p^2]$$

On the yield surface $J_2 = 1/3\sigma_y^2$ where σ_y is the uniaxial yield stress, i.e.,

$$\sigma_y = [3(a_0 + a_1 p + a_2 p^2)]^{1/2}$$

There is no strain hardening on this surface.

To eliminate the pressure dependence of the yield strength, set:

$$a_1 = a_2 = 0 \qquad a_0 = \frac{1}{3}\sigma_y^2$$

This approach is useful when a von Mises type elastic-plastic model is desired for use with the tabulated volumetric data.

MATD015 (SOL 700)

LS-DYNA Material #15 -- Johnson-Cook Strain and Temperature-Sensitive Plasticity

LS-DYNA Material #15 -- The Johnson / Cook strain and temperature sensitive plasticity is sometimes used for problems where the strain rates vary over a large range and adiabatic temperature increases due to plastic heating cause material softening. When used with solid elements this model requires an equation-of-state. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD015	MID	RO	G	E	PR	DTF	VP	MEID	
	A	B	N	C	M	TM	TR	EPSO	
	CP	CP	SPALL	IT	D1	D2	D3	D4	
	D5								

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
G	Shear modulus. (Real)
E	Young's Modulus (shell elements only) (Real)
PR	Poisson's ratio (shell elements only) (Real)
DTF	Minimum time step size for automatic element deletion (shell elements) (Real, Default = 0.0)
VP	Formulation for rate effects: (Real, Default = 0.0)
	EQ.0.0: Scale yield stress (default),
	EQ.1.0: Viscoplastic formulation.
MEID	Identification number of EOSPOL entry defining the equation of state when used with solid elements. This option may also be specified on the PSOLID entry.
A	See equations below. (Real)
B	See equations below. (Real, Default = 0.0)
N	See equations below. (Real, Default = 0.0)

Field	Contents
C	See equations below. (Real, Default = 0.0)
M	See equations below. (Real)
TM	Melt temperature. (Real)
TR	Room temperature. (Real)
EPSO	Strain rate normalization factor. This value depends on the time units. Input 1.0 for units of seconds, 0.001 for units of milliseconds, 0.000001 for microseconds, etc. (Real)
CP	Specific heat. (Real)
PC	Failure stress or pressure cutoff ($p_{min} < 0.0$) (Real, Default = 0.0)
SPALL	Spall type: (Integer, Default = 2.0)
	EQ. 0: default set to "2",
	EQ. 1: $p \geq p_{min}$,
	EQ. 2: if $\sigma_{max} = -p_{min}$ element spalls and tension, $p < 0$, is never allowed,
	EQ. 3: $p < -p_{min}$ element spalls and tension, $p < 0$, is never allowed.
IT	Plastic strain iteration option. This input applies to solid elements only since it is always necessary to iterate for the shell element plane stress condition. (Integer, Default = 0.0)
	EQ. 0: no iterations (Default),
	EQ. 1: accurate iterative solution for plastic strain. Much more expensive than default.
D1-D5	Failure parameters, see equations below. (Real, Default = 0.0)

Remarks:

Johnson and Cook express the flow stress as

$$\sigma_y = \left(A + B\bar{\varepsilon}^{p^n}\right)(1 + c\ln\dot{\varepsilon}^*)(1 - T^{*m})$$

where:

, B, C, n, and m = input constants

$\bar{\varepsilon}^p$ = effective plastic strain

$$\dot{\varepsilon}^* = \frac{\dot{\varepsilon}^p}{\dot{\varepsilon}_0} = \text{effective plastic strain rate for } \dot{\varepsilon}_0 = 1s^{-1}$$

$$T^* = \text{homologous temperature} = \frac{T - T_{room}}{T_{melt} - T_{room}}$$

Constants for a variety of materials are provided in [Johnson and Cook 1983]. A fully viscoplastic formulation is optional (VP) which incorporates the rate equations within the yield surface. An additional cost is incurred but the improvement is results can be dramatic.

Due to nonlinearity in the dependence of flow stress on plastic strain, an accurate value of the flow stress requires iteration for the increment in plastic strain. However, by using a Taylor series expansion with linearization about the current time, we can solve for σy with sufficient accuracy to avoid iteration.

The strain at fracture is given by

$$\varepsilon^f = [D_1 + D_2 \exp D_3 \sigma^*][1 + D_4 \ln \dot{\varepsilon}^*][1 + D_5 T^*]$$

where σ^* is the ratio of pressure divided by effective stress

$$\sigma^* = \frac{p}{\sigma_{eff}}$$

Fracture occurs when the damage parameter

$$D = \sum \frac{\Delta \dot{\varepsilon}^p}{\varepsilon^f}$$

reaches the value of 1.

A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads. The pressure limit model limits the minimum hydrostatic pressure to the specified value, $p \geq p_{min}$. If pressures more tensile than this limit are calculated, the pressure is reset to p_{min}. This option is not strictly a spall model since the deviatoric stresses are unaffected by the pressure reaching the tensile cutoff and the pressure cutoff value p_{min} remains unchanged throughout the analysis. The maximum principal stress spall model detects spall if the maximum principal stress, σ_{max}, exceeds the limiting value σ_p. Once spall is detected with this model, the deviatoric stresses are reset to zero and no hydrostatic tension is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled

material behaves as rubble. The hydrostatic tension spall model detects spall if the pressure becomes more tensile than the specified limit, p_{min}. Once spall is detected, the deviatoric stresses are set to zero and the pressure is required to be compressive. If hydrostatic tension is calculated then the pressure is reset to 0 for that element.

In addition to the above failure criterion, this material model also supports a shell element deletion criterion based on the maximum stable time step size for the element, Δt_{max}. Generally Δt_{max}, goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the Δt_{max} values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step has fallen below the specified minimum time step, Δt_{crit}. Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and, therefore, the analysis can proceed at a larger time step, and, thus, at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

Material type 15 is applicable to the high rate deformation of many materials including most metals. Unlike the Steinberg-Guinan model, the Johnson-Cook model remains valid down to lower strain rates and even into the quasistatic regime. Typical applications include explosive metal forming, ballistic penetration, and impact.

MATD018 (SOL 700)

LS-DYNA Material #18 -- Isotropic Plasticity with Rate Effects

LS-DYNA Material #18 -- This is an isotropic plasticity model with rate effects that uses a power law hardening rule. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD018	MID	RO	E	PR	K	N	SRC	SRP	
	SIGY	VP							

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
E	Young's modulus. (Real)
PR	Poisson's ratio. (Real)
K	Strength coefficient. (Real)
N	Hardening exponent. (Real)
SRC	Strain rate parameter, C, if zero, rate effects are ignored. (Real, Default = 0.0)
SRP	Strain rate parameter, P, if zero, rate effects are ignored. (Real, Default = 0.0)
SIGY	Optional input parameter for defining the initial yield stress, σ_y. Generally, this parameter is not necessary and the strain to yield is calculated as described below. (Real, Default = 0.0) LT.0.02: $\varepsilon_{yp} = SIGY$ GE.0.02: See below.
VP	Formulation for rate effects: (Integer, Default = 0) EQ.0: Scale yield stress (Default), EQ.1: Viscoplastic formulation.

Remarks:

Elastoplastic behavior with isotropic hardening is provided by this model. The yield stress, σ_y, is a function of plastic strain and obeys the equation:

$$\sigma_y = k\varepsilon^n = k(\varepsilon_{yp} + \bar{\varepsilon}^p)^n$$

where ε_{yp} is the elastic strain to yield and $\bar{\varepsilon}^p$ is the effective plastic strain (logrithmic). If SIGY is set to zero, the strain to yield if found by solving for the intersection of the linearly elastic loading equation with the strain hardening equation:

$$\sigma = E\varepsilon$$
$$\sigma = k\varepsilon^n$$

which gives the elastic strain at yield as:

$$\varepsilon_{yp} = \left(\frac{E}{k}\right)^{\left[\frac{1}{n-1}\right]}$$

If SIGY yield is nonzero and greater than 0.02 then:

$$\varepsilon_{yp} = \left(\frac{\sigma_y}{k}\right)^{\left[\frac{1}{n}\right]}$$

Strain rate is accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate. A fully viscoplastic formulation is optional which incorporates the Cowper and Symonds formulation within the yield surface. An additional cost is incurred but the improvement is results can be dramatic.

MATD019 (SOL 700)

LS-DYNA Material #19 -- Elastic Plastic Material Model with Strain Rate Dependent Yield

LS-DYNA Material #19 -- Used to model strain rate dependent material. For an alternative, see MATD024. Required is a curve for the yield stress versus the effective strain rate. Optionally, Young's modulus and the tangent modulus can also be defined versus the effective strain rate. Also, optional failure of the material can be defined either by defining a von Mises stress at failure as a function of the effective strain rate (valid for solids / shells / thick shells) or by defining a minimum time step size (only for shells). Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD019	MID	RO	E	PR	VP				
	LC1	ETAN	LC2	LC3	LC4	TDEL	RDEF		

Field	Contents
MID	Material identification. A unique number has to be chosen.
RO	Mass density.
E	Young's modulus.
PR	Poisson's ratio.
VP	Formulation for rate effects: EQ.0: Scale yield stress (default), EQ.1: Viscoplastic formulation.
LC1	Load curve ID defining the yield stress σ_0 as a function of the effective strain rate.
ETAN	Tangent modulus, E_t
LC2	Load curve ID defining Young's modulus as a function of the effective strain rate (optional).
LC3	Load curve ID defining tangent modulus as a function of the effective strain rate (optional).
LC4	Load curve ID defining von Mises stress at failure as a function of the effective strain rate (optional).

Field	Contents
TDEL	Minimum time step size for automatic element deletion. Use for shells only.
RDEF	Redefinition of failure curve: EQ.1.0: Effective plastic strain, EQ.2.0: Maximum principal stress.

Remarks:

In this model, a load curve is used to describe the yield strength σ_0 as a function of effective strain rate $\dot{\bar{\varepsilon}}$ where

$$\dot{\bar{\varepsilon}} = \left(\frac{2}{3}\dot{\varepsilon}'_{ij}\dot{\varepsilon}'_{ij}\right)^{1/2}$$

and the prime denotes the deviatoric component. The yield stress is defined as

$$\sigma_y = \sigma_0(\dot{\bar{\varepsilon}}) + E_p\dot{\bar{\varepsilon}}^p$$

where $\bar{\varepsilon}^p$ is the effective plastic strain and E_p is given in terms of Young's modulus and the tangent modulus by

$$E_p = \frac{EE}{E - E_t}$$

Both Young's modulus and the tangent modulus may optionally be made functions of strain rate by specifying a load curve ID giving their values as a function of strain rate. If these load curve ID's are input as 0, then the constant values specified in the input are used.

Note: *All load curves used to define quantities as a function of strain rate must have the same number of points at the same strain rate values.* This requirement is used to allow vectorized interpolation to enhance the execution speed of this constitutive model.

This model also contains a simple mechanism for modeling material failure. This option is activated by specifying a load curve ID defining the effective stress at failure as a function of strain rate. For solid elements, once the effective stress exceeds the failure stress the element is deemed to have failed and is removed from the solution.

For shell elements the entire shell element is deemed to have failed if all integration points through the thickness have an effective stress that exceeds the failure stress. After failure the shell element is removed from the solution.

In addition to the above failure criterion, this material model also supports a shell element deletion criterion based on the maximum stable time step size for the element, Δt_{max}. Generally, Δt_{max} goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the Δt_{max} values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step Δt_{max} has fallen below the specified minimum time step Δt_{crit}. Elements which are severely distorted often indicate that material has failed and supports little load, but these same elements may have very small time steps and therefore control the cost of the analysis. This option allows these highly distorted elements to be deleted from the calculation, and, therefore, the analysis can proceed at a larger time step, and, thus, at a reduced cost. Deleted elements do not carry any load, and are deleted from all applicable slide surface definitions. Clearly, this option must be judiciously used to obtain accurate results at a minimum cost.

A fully viscoplastic formulation is optional which incorporates the rate formulation within the yield surface. An additional cost is incurred but the improvement is results can be dramatic.

MATD020 (SOL 700) LS-DYNA Material #20 -- Rigid Material

Used to model rigid materials. Alternatively, a VDA surface can be attached as surface to model the geometry, e.g., for the tooling in metal-forming applications. Also, global and local constraints on the mass center can be optionally defined. Optionally, a local consideration for output and user-defined airbag sensors can be chosen. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD020	MID	RO	E	PR					
	CMO	CON1	CON2						
	LCO or A1	A2	A3	V1	V2	V3			

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer; Default = none)
RO	Mass density. (Real; Default = none)
E	Young's modulus. Reasonable values have to be chosen for contact analysis (choice of penalty). See Remarks below. (Real; Default = none)
PR	Poisson's ratio. Reasonable values have to be chosen for contact analysis (choice of penalty). See Remarks below. (Real; Default = none)
CMO	Center of mass constraint option, CMO (Integer; Default = 0): EQ.+1 Constraints applied in global directions. EQ.0 No constraints. EQ.-1 Constraints applied in local directions (SPC constraint).
CON1	First constraint parameter (Integer; Default = 0): If CMO=+1.0, then specify global translational constraint: EQ.0 No constraints. EQ.1 Constrained x displacement. EQ.2 Constrained y displacement. EQ.3 Constrained z displacement.

EQ.4 Constrained x and y displacements.

EQ.5 Constrained y and displacements.

EQ.6 Constrained z and x displacements.

EQ.7: Constrain x, y, and z displacements.

If CM0=-1.0, then specify local coordinate system ID. This coordinate system is fixed in time.

CON2 Second constraint parameter (Integer; Default = 0):

If CMO=+1.0, then specify global rotational constraint:

EQ.0 No constraints.

EQ.1 Constrained x rotation.

EQ.2 Constrained y rotation.

EQ.3 Constrained z rotation.

EQ.4 Constrained x and y rotations.

EQ.5 Constrained y and z rotations.

EQ.6 Constrained z and x rotations.

EQ.7 Constrained x, y, and z rotations.

If CM0=-1.0, then specify local (SPC) constraint:

EQ.000000 No constraint.

EQ.100000 Constrained x translation.

EQ.010000 Constrained y translation.

EQ.001000 Constrained z translation.

EQ.000100 Constrained x rotation.

EQ.000010 Constrained y rotation.

EQ.000001 Constrained z rotation.

Any combination of local constraints can be achieved by adding the number 1 into the corresponding column.

LCO Local coordinate system number for output. (Real; Default = 0)

Alternative method for specifying local system below.

A1-V3 Define two vectors a and v, fixed in the rigid body that are used for output. The output parameters are in the directions a, b, and c where the latter are given by the cross products $c = a \times v$ and $b = c \times a$. This input is optional. (Real; Default = 0)

Remarks:

1. A rigid material provides a convenient way of turning one or more parts comprised of beams, shells, or solid elements into a rigid body. Approximating a deformable body as rigid is a preferred modeling technique in many real world SOL 700 applications. For example, an engine block in a car crash simulation can be treated as rigid. Elements belonging to a rigid material are bypassed in the element processing and no storage is allocated for storing history variables. Consequently, using a rigid material is very cost efficient.
2. Elements belonging to a MATD020 are properly treated in the contact calculations, and it is allowed to include them in a BCBODY with BEHAV=DEFORM. The contact calculations will operate as if the material is deformable. The penalty based contact forces applied on the nodes are accumulated for the whole rigid body and applied as an external force and moment to the cg of the rigid body.
3. The inertial properties are calculated from the geometry of the constituent elements and the density RO as specified on the MATD020.
4. The initial velocity of a rigid material is calculated from the initial velocity of the constituent grids.
5. By default, the contact forces in SOL 700 are based on the soft constraint formulation. See the variable SOFT=1 on BCTABLE. In this method, the contact forces as based on the masses of the slave nodes and master segment that are in contact. It is thus important to specify a realistic density since unrealistic values may contribute to numerical problems in contact
6. When the penalty formulation of the contact is used, by setting SOFT=0 on BCTABLE, the Young's modulus, E, and Poission's ratio, ν are used for determining the contact stiffness. In that case, realistic values for E and ν should be defined since unrealistic values may contribute to numerical problems in contact.
7. An error will be given if two rigid bodies share common nodes. It is possible to manually merge multiple rigid materials, using the MATD20M bulk data entry.
8. A rigid body can be made up of disjoint meshes. All elements that are part of a rigid body will move together as one rigid, even if they are disjoint.
9. Motion control for a rigid material can be defined using the SPCD entry. The SPCD must be applied to one gridpoint only.

10. Load control for a rigid material can be defined using the FORCE and MOMENT entries. These loads can be applied to any gridpoint that belongs to the rigid body. The forces and moments acting on the gridpoints will be accumulated and applied to the rigid body.
11. If no constraints are specified for the rigid material (CMO=0) the nodes belonging to the rigid material are scanned to determine constraints of the rigid material in global directions. If constraints are specified for the rigid material (CMO equal to +1 or –1), the nodes belonging to the rigid material are not scanned
12. Constraint directions for rigid materials (CMO equal to +1 or –1) are fixed, that is, not updated, with time.

MATD20M (SOL 700) Merges Two or More Rigid Materials

Merges two or more rigid materials defined using MATD020. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD20M	IDM	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	-Etc.-							

Example:

MATD20M	101	202	203	305					

Field	Contents
IDM	ID of an existing MATD020 rigid material. The other MATD020 rigid materials will be merged into this one. (Integer > 0)
Idi	IDs of an existing MATD020 rigid material. These MATD020 rigid materials will be merged into the one specified by IDM. After merging, these rigid materials will cease to exist. (Integer > 0)

MATD022 (SOL 700)

LS-DYNA Material #22 -- Orthotropic Material with Brittle Failure

Used to model an orthotropic material with optional brittle failure for composites. It can be defined following the suggestion of (Chang and Chang 1987a, 1987b). Three failure criteria are possible. By using the user defined integration rule, the constitutive constants can vary through the shell thickness. For all shells, except the DKT formulation, laminated shell theory can be activated to properly model the transverse shear deformation. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. For sandwich shells where the outer layers are much stiffer than the inner layers, the response will tend to be too stiff unless lamination theory is used. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD022	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB	
	GAB	GBC	GCA	KFAIL	AOPT	MACF			
	XP	YP	ZP	A1	A2	A3			
	SC	XT	YT	YC	ALPH	SN	SYZ	SZX	
	V1	V2	V3	D1	D2	D3	BETA		

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer; Default = none)
RO	Mass density. (Real; Default = none)
EA	E_a, Young's modulus in a-direction.(Real; Default = none)
EB	E_b, Young's modulus in b-direction. (Real; Default = none)
EC	E_c, Young's modulus in c-direction. (Real; Default = none)
PRBA	ν_{ba}, Poisson ratio, ba. (Real; Default = none)
PRCA	ν_{ca}, Poisson ratio, ca. (Real; Default = none)
PRCB	ν_{cb}, Poisson ratio, cb. (Real; Default = none)
GAB	G_{ab}, Shear modulus, ab. (Real; Default = none)

Field	Contents
GBC	G_{bc}, Shear modulus, bc. (Real); Default = none
GCA	G_{ca}, Shear modulus, ca. (Real; Default = none)
KFAIL	Bulk modulus of failed material. Necessary for compressive failure.(Real; Default = 0.0)
AOPT	Material axes option. (Integer; Default = 0):
	EQ.0 Locally orthotropic with material axes determined by element nodes 1, 2, and 4,
	EQ.1 Locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only.
	EQ.2 Globally orthotropic with material axes determined by vectors defined below.
	EQ.3 Locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal.
	EQ.4 Locally orthotropic in cylindrical coordinate system with the material axes determined by a vector **v**, and an originating point, P, which define the centerline axis. This option is for solid elements only.
MACF	Material axes change flag for brick elements (Integer; Default = 1):
	EQ.1 Default.
	EQ.2 Switch material axes a and b.
	EQ.3 Switch material axes a and c.
XP,YP,ZP	Coordinates of point **p** for AOPT = 1. (Real; Default = 0.0)
A1,A2,A3	Components of vector **a** for AOPT = 2. (Real; Default = 0.0)
V1,V2,V3	Components of vector **v** for AOPT = 3. (Real; Default = 0.0)
D1,D2,D3	Components of vector **d** for AOPT = 2. (Real; Default = 0.0)
BETA	Material angle in degrees for AOPT = 3, may be overridden on the element card. (Real; Default = 0.0)

Field	Contents
SC	Shear strength, ab plane. (Real)
XT	Longitudinal tensile strength, a-axis, see the *MSC.Nastran Theoretical Manual*. (Real)
YT	Transverse tensile strength, b-axis, see the *MSC.Nastran Theoretical Manual*. (Real)
YC	Transverse compressive strength, b-axis. See the *MSC.Nastran Theoretical Manual*. (Real)
ALPH	Shear stress parameter for the nonlinear term. Suggested range 0 - 0.5. See the *MSC.Nastran Theoretical Manual*. (Real)
SN	Normal tensile strength (solid elements only). (Real)
SYZ	Transverse shear strength (solid elements only). (Real)
SZX	Transverse shear strength (solid elements only). (Real)

Remarks:

The number of additional integration point variables for shells written to the LS-DYNA database is input by the optional PARAM,DYNEIPS. These additional variables are tabulated below (*ip* = shell integration point):

History Variable	Description	Value	LS-TAURUS Component
ef(i)	Tensile fiber mode	1 - elastic 0 - failed	81
cm(i)	Tensile matrix mode		82
ed(i)	Compressive matrix mode		83

The following components are stored as element component 7 instead of the effective plastic strain:

Description	Integration Point
$\frac{1}{\text{nip}} \sum_{i=1}^{\text{nip}} ef(i)$	1
$\frac{1}{\text{nip}} \sum_{i=1}^{\text{nip}} cm(i)$	2
$\frac{1}{\text{nip}} \sum_{i=1}^{\text{nip}} ed(i)$	3

MATD024 (SOL 700) LS-DYNA Material #24 -- Elasto-Plastic Material

Used to model an elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency. See also Remarks below. Also, failure based on a plastic strain or a minimum time step size can be defined. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD024	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL	
	C	P	LCSS	LCSR	VP				
	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8	
	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8	

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer; Default = none)
RO	Mass density. (Real; Default = none)
E	Young's modulus. (Real; Default = none)
PR	Poisson's ratio. (Real; Default = none)
SIGY	Yield stress. (Real; Default = none)
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined.(Real; Default = 0.0)
FAIL	Failure flag. (Real; Default = 10E+20) LT.0.0: User defined failure subroutine is called to determine failure EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Plastic strain to failure. When the plastic strain reaches this value, the element is deleted from the calculation.
TDEL	Minimum time step size for automatic element deletion. (Real; Default = 0)
C	Strain rate parameter, C, see formula below. (Real)

Field	Contents
P	Strain rate parameter, P, see formula below. (Real)
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, See **Figure 8-102**. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P; the curve ID, LCSR; EPS1-EPS8 and ES1-ES8 are ignored if a Table ID is defined. (Real; Default = 0)
Note:	The strain rate values defined in the table may be given as the natural logarithm of the strain rate. If the *first* stress-strain curve in the table corresponds to a negative strain rate, LS-DYNA assumes that the natural logarithm of the strain rate value is used. Since the tables are internally discretized to equal space the points, natural logarithms are necessary, for example, if the curves correspond to rates from 10.e-04 to 10.e+04. Computing the natural logarithm of the strain rate does slow the stress update down significantly on some computers.
LCSR	Load curve ID defining strain rate scaling effect on yield stress. (Real; Default = 0)
VP	Formulation for rate effects. (Integer; Default = 0): EQ.-1 Cowper-Symonds with deviatoric strain rate rather than total. EQ. 0 Scale yield stress. (Default) EQ. 1 Viscoplastic formulation.
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined. The first point must be zero corresponding to the initial yield stress. (Real; Default = 0)
Note:	If the first point is nonzero, the yield stress is extrapolated to determine the initial yield. If this option is used, SIGY and ETAN are ignored and may be input as zero.
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8. (Real; Default = 0)

Remarks:

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in **Figure 8-102** is expected to be defined by (EPS1,ES1) -- (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Three options to account for strain rate effects are possible.

1. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/p}$$

where $\dot{\varepsilon}$ is the strain rate

$$\dot{\varepsilon} = \sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$$

If VP=-1, the deviatoric strain rates are used instead.

If the viscoplastic option is active, VP=1.0, and if SIGY is > 0 then the dynamic yield stress is computed from the sum of the static stress,

$$\sigma_y^s(\varepsilon_{eff}^p)$$

which is typically given by a load curve ID, and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

$$\sigma_y(\varepsilon_{eff}^p, \dot{\varepsilon}_{eff}^p) = \sigma_y^s(\varepsilon_{eff}^p) + SIGY \cdot \left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}$$

where the plastic strain rate is used. If SIGY=0, the following equation is used instead where the static stress

$$\sigma_y^s(\varepsilon_{eff}^p)$$

must be defined by a load curve:

$$\sigma_y(\varepsilon_{eff}^p, \dot{\varepsilon}_{eff}^p) = \sigma_y^s(\varepsilon_{eff}^p)\left[1 + \left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}\right]$$

This latter equation is always used if the viscoplastic option is off.

2. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.
3. If different stress versus strain curves can be provided for various strain rates, the option using the reference to a table (LCSS) can be used.

   A fully viscoplastic formulation is optional (variable VP) which incorporates the different options above within the yield surface. An additional cost is incurred over the simple scaling but the improvement in results can be dramatic.

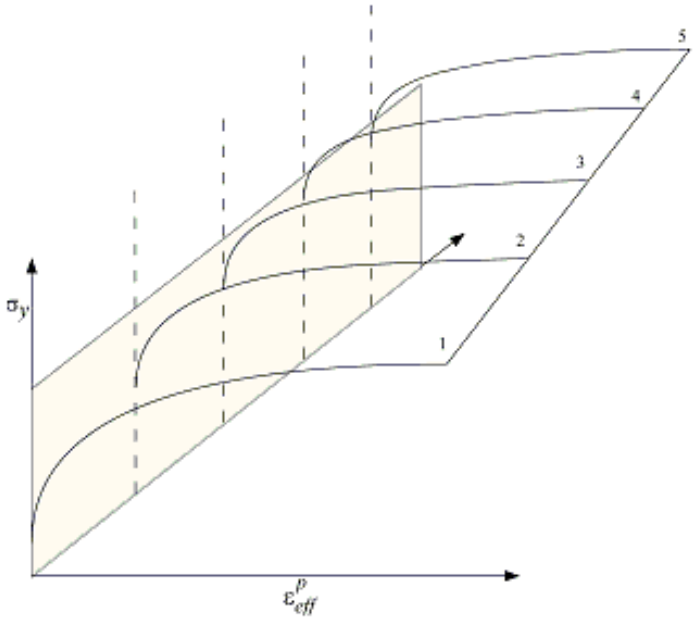


Figure 8-102

Rate effects may be accounted for by defining a table of curves. If a table ID is specified a curve ID is given for each strain rate. Intermediate values are found by interpolating between curves. Effective plastic strain versus yield stress is expected. If the strain rate values fall out of range, extrapolation is not used; rather, either the first or last curve determines the yield stress depending on whether the rate is low or high, respectively.

MATD026 (SOL 700) LS-DYNA Material #26 -- Anisotropic Honeycomb and Foam

Used to model honeycomb and foam materials with anisotropic behavior. A nonlinear elastoplastic material behavior can be defined separately for all normal and shear stresses. These are considered to be uncoupled. See Remarks below. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD026	MID	RO	E	PR	SIGY	VF	MU	BULK	
	LCA	LCB	LCC	LCS	LCAB	LCBC	LCCA	LCSR	
	EAAU	EBBU	ECCU	GABU	GBCU	GCAU	AOPT		
	XP	YP	ZP	A1	A2	A3			
	D1	D2	D3	TSEF	SSEF				

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
E	Young's modulus for compacted honeycomb material. (Real)
PR	Poisson's ratio for compacted honeycomb material. (Real)
SIGY	Yield stress for fully compacted honeycomb. (Real)
VF	Relative volume at which the honeycomb is fully compacted. (Real)
MU	μ, material viscosity coefficient. (Default=.05) Recommended. (Real; Default = .05)
BULK	Bulk viscosity flag (Integer; Default = 0): EQ.0 Bulk viscosity is not used. This is recommended. EQ.1 Bulk viscosity is active and $\mu = 0$. This will give results identical to previous versions of LS-DYNA.
LCA	Load curve identification number for sigma-aa versus either relative volume or volumetric strain. See Remarks below. (Real; Default = none)
LCB	Load curve identification number for sigma-bb versus either relative volume or volumetric strain. See Remarks below. (Real; Default LCB=LCA)

Field	Contents
LCC	Load curve identification number for sigma-cc versus either relative volume or volumetric strain. See Remarks below. (Real; Default LCC=LCA)
LCS	Load curve identification number for shear stress versus either relative volume or volumetric strain. Each component of shear stress may have its own load curve. See Remarks below. (Real; Default LCS=LCA)
LCAB	Load curve identification number for sigma-ab versus either relative volume or volumetric strain. See Remarks below. (Real; Default LCAB=LCS)
LCBC	Load curve identification number for sigma-bc versus either relative volume or volumetric strain. See Remarks below. (Real; Default LCBC=LCS)
LCCA	Load curve identification number for sigma-ca versus either relative volume or volumetric strain. See Remarks below. (Real; Default LCCA=LCS)
LCSR	Load curve identification number for strain-rate effects defining the scale factor versus strain rate. This is optional. The curves defined above are scaled using this curve. (Real)
EAAU	Elastic modulus E_{aau} in uncompressed configuration. (Real)
EBBU	Elastic modulus E_{bbu} in uncompressed configuration. (Real)
ECCU	Elastic modulus E_{ccu} in uncompressed configuration. (Real)
GABU	Shear modulus G_{abu} in uncompressed configuration. (Real)
GBCU	Shear modulus G_{bcu} in uncompressed configuration. (Real)
GCAU	Shear modulus G_{cau} in uncompressed configuration. (Real)
AOPT	Material axes option (Integer): EQ.0 Locally orthotropic with material axes determined by element nodes 1, 2, and 4. EQ.1 Locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. EQ.2 Globally orthotropic with material axes determined by vecors defined below.

Field	Contents
XP YP ZP	Coordinates of point p for AOPT = 1. (Real)
A1 A2 A3	Components of vector a for AOPT = 2. (Real)
D1 D2 D3	Components of vector d for AOPT = 2. (Real)
TSEF	Tensile strain at element failure (element will erode). (Real)
SSEF	Shear strain at element failure (element will erode). (Real)

Remarks:

For efficiency, we recommend that the load curve identification numbers (LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA) contain exactly the same number of points with corresponding strain values on the abscissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abscissa strain values are not consistent between load curves.

The behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, i.e., an *a* component of strain will generate resistance in the local *a*-direction with no coupling to the local *b* and *c* directions. The elastic moduli vary, from their initial values to the fully compacted values at v_f, linearly with the relative volume *V*:

$$E_{aa} = E_{aau} + \beta(E - E_{aau})$$

$$E_{bb} = E_{bbu} + \beta(E - E_{bbu})$$

$$E_{cc} = E_{ccu} + \beta(E - E_{ccu})$$

$$G_{ab} = G_{abu} + \beta(G - G_{abu})$$

$$G_{bc} = E_{bcu} + \beta(G - G_{bcu})$$

$$G_{ca} = E_{cau} + \beta(G - G_{cau})$$

where:

$$\beta = \text{ax}\left[\min\left(\frac{1 - V}{1 - V_f}, 1\right), 0\right]$$

G = the elastic shear modulus for the fully compacted honeycomb material

$$G = \frac{E}{2(1+\nu)}$$

The relative volume, V, is defined as the ratio of the current volume to the initial volume. Typically, $V = 1$ at the beginning of a calculation. The viscosity coefficient μ (MU) should be set to a small number (usually in the range of .02-.10). Alternatively, the two bulk viscosity coefficients on the control entries should be set to very small numbers to prevent the development of spurious pressures that may lead to undesirable and confusing results. The latter is not recommended since spurious numerical noise may develop.

The load curves define the magnitude of the average stress as the material changes density (relative volume), see **Figure 8-103**. Each curve related to this model must have the same number of points and the same abscissa values. There are two ways to define these curves:

1. as a function of relative volume (V).
2. as a function of volumetric strain defined as:

$$\varepsilon_V = (1 - V)$$

In the former, the first value in the curve should correspond to a value of relative volume slightly less than the fully compacted value. In the latter, the first value in the curve should be less than or equal to zero, corresponding to tension, and increase to full compaction.

Note: Care should be taken when defining the curves so that extrapolated values do not lead to negative yield stresses.

At the beginning of the stress update each element's stresses and strain rates are transformed into the local element coordinate system. For the uncompacted material, the trial stress components are updated using the elastic interpolated moduli according to:

$$\sigma_{aa}^{n+1^{\text{trial}}} = \sigma_{aa}^{n} + E_{aa}\Delta\varepsilon_{aa}$$

$$\sigma_{bb}^{n+1^{\text{trial}}} = \sigma_{bb}^{n} + E_{bb}\Delta\varepsilon_{bb}$$

$$\sigma_{cc}^{n+1^{\text{trial}}} = \sigma_{cc}^{n} + E_{cc}\Delta\varepsilon_{cc}$$

$$\sigma_{ab}^{n+1\,\text{trial}} = \sigma_{ab}^{n} + 2G_{ab}\Delta\varepsilon_{ab}$$

$$\sigma_{bc}^{n+1\,\text{trial}} = \sigma_{bc}^{n} + 2G_{bc}\Delta\varepsilon_{bc}$$

$$\sigma_{ca}^{n+1\,\text{trial}} = \sigma_{ca}^{n} + 2G_{ca}\Delta\varepsilon_{ca}$$

Each component of the updated stresses is then independently checked to ensure that they do not exceed the permissible values determined from the load curves; e.g., if

$$\left|\sigma_{ij}^{n+1\,\text{trial}}\right| > \lambda\sigma_{ij}(V)$$

then

$$\sigma_{ij}^{n+1} = \sigma_{ij}(V)\frac{\lambda\sigma_{ij}^{n+1\,\text{trial}}}{\left|\lambda\sigma_{ij}^{n+1\,\text{trial}}\right|}$$

On Format 2, $\sigma_{ij}(V)$ is defined by LCA for the *aa* stress component, LCB for the *bb* component, LCC for the *cc* component, and LCS for the *ab*, *bc*, *cb* shear stress components. The parameter λ is either unity or a value taken from the load curve number, LCSR, that defines λ as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

For fully compacted material it is assumed that the material behavior is elastic-perfectly plastic and the stress components updated according to:

$$_{ij}^{\text{trial}} = s_{ij}^{n} + 2G\Delta\varepsilon_{ij}^{\text{dev}\,n+1/2}$$

where the deviatoric strain increment is defined as

$$\Delta\varepsilon_{ij}^{\text{dev}} = \Delta\varepsilon_{ij} - \frac{1}{3}\Delta\varepsilon_{kk}\delta_{ij}$$

Now a check is made to see if the yield stress for the fully compacted material is exceeded by comparing

$$_{eff}^{\text{trial}} = \left(\frac{3}{2}s_{ij}^{\text{trial}}s_{ij}^{\text{trial}}\right)^{1/2}$$

the effective trial stress to the defined yield stress, SIGY. If the effective trial stress exceeds the yield stress the stress components are simply scaled back to the yield surface

$$s_{ij}^{n+1} = \frac{\sigma_y}{s_{eff}^{\text{trial}}} s_{ij}^{\text{trial}}$$

Now the pressure is updated using the elastic bulk modulus, K

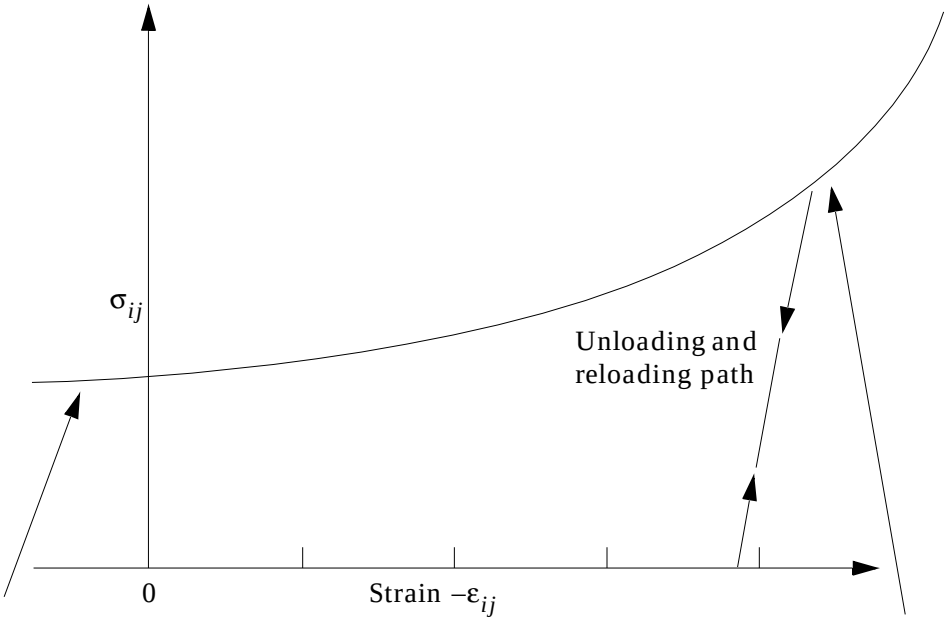
$$p^{n+1} = p^n - K\Delta\varepsilon_{kk}^{n+1/2}$$

$$K = \frac{E}{3(1-2\nu)}$$

to obtain the final value for the Cauchy stress

$$\sigma_{ij}^{n+1} = s_{ij}^{n+1} - p^{n+1}\delta_{ij}$$

After completing the stress update transform the stresses back to the global configuration.



Curve extends into negative strain quadrant since LS-DYNA will extrapolate using the two end points. It is important that the extrapolation does not extend into the negative stress region.

Unloading is based on the interpolated Young's moduli which must provide an unloading tangent that exceeds the loading tangent.

Figure 8-103

Stress quantity versus volumetric strain. Note that the "yield stress" at a volumetric strain of zero is non-zero.

MATD027 (SOL 700) LS-DYNA Material #27 -- Two-Variable Rubber Model

Used to model rubber using two variables. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD027	MID	RO	PR	A	B	REF			
	SGL	SW	ST	LCID					

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
PR	Poisson's ratio (value between 0.49 and 0.5 is recommended, smaller values may not work). (Real)
A	Constant, see literature and equations defined below. (Real)
B	Constant, see literature and equations defined below. (Real)
REF	Use reference geometry to initialize the stress tensor. This option is currently restricted to 8-noded solid elements with one point integration. (Integer) EQ.0 Off EQ.1 On

If A=B=0.0, then a least square fit is computed from tabulated uniaxial data via a load curve. The following information should be defined.

SGL	Specimen gauge length l_0. See **Figure 8-104**. (Real)
SW	Specimen width. See **Figure 8-104**. (Real)
ST	Specimen thickness. See **Figure 8-104**. (Real)
LCID	Load curve ID giving the force versus actual change ΔL in the gauge length. (Integer)

Remarks:

The strain energy density function is defined as:

$$W = A(I-3) + B(II-3) + C(III^{-2}-1) + D(III-1)^2$$

where:

$C = 0.5A + B$

$D = \frac{A(5\upsilon - 2) + B(11\nu - 5)}{2(1-2\nu)}$

ν = Poisson's ratio

$2(A + B)$ = Shear modulus of linear elasticity.

I, II, III = Invariants of right Cauchy-Green Tensor C

The load curve definition that provides the uniaxial data should give the change in gauge length, Δ_L, versus the corresponding force. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

$$\lambda_1 = \frac{L_0 + \Delta L}{L_0}$$

with L_0 being the initial length and L being the actual length.

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force (see **Figure 8-104**).

The least square fit to the experimental data is performed during the initialization phase and is a comparison between the fit and the actual input is provided in the printed file. It is a good idea to visually check to make sure it is acceptable. The coefficients A and B are also printed in the output file. Use the material driver to check out the material model.

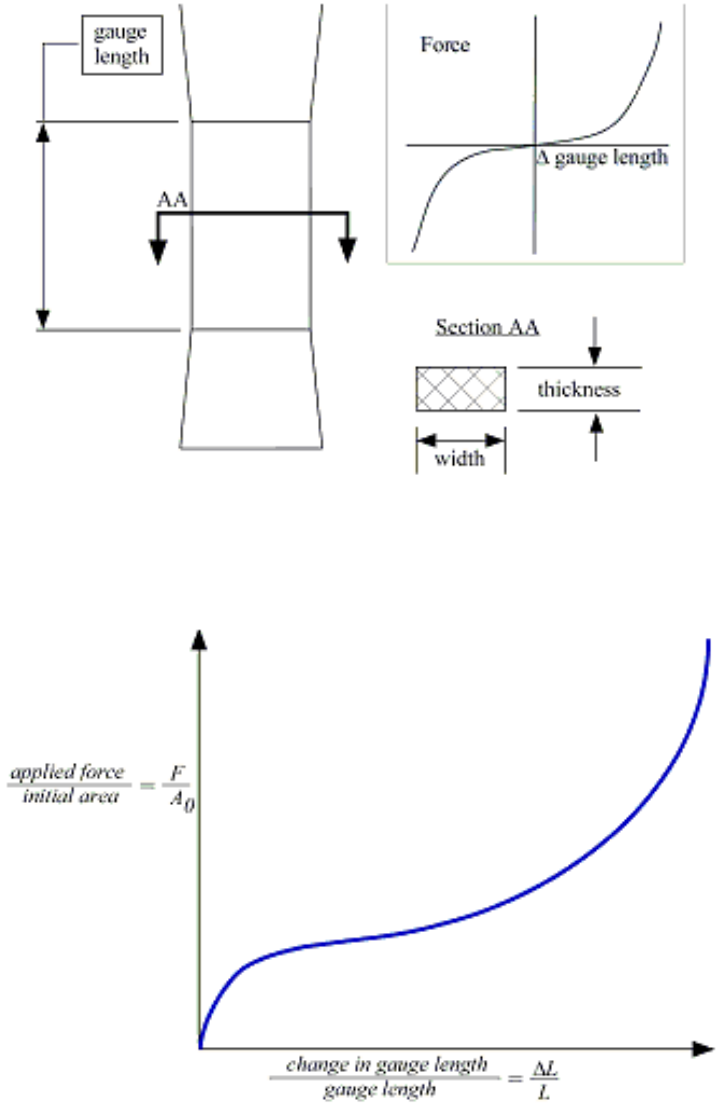


Figure 8-104 Uniaxial Specimen for Experimental Data

The stress versus strain curve can used instead of the force versus the change in the gauge length by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force.

MATD028 (SOL 700) LS-DYNA Material #28 -- Elasto-Plastic Resultant Formulation

A resultant formulation for beam and shell elements including elasto-plastic behavior can be defined. This model is available for the Belytschko-Schwer beam, the C^0 triangular shell, the Belytschko-Tsay shell, and the fully integrated type 16 shell. For beams, the treatment is elastic-perfectly plastic, but for shell elements isotropic hardening is approximately modeled. Since the stresses are not computed in the resultant formulation, the stresses output to the binary databases for the resultant elements are zero. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD028	MID	RO	E	PR	SIGY	ETAN			

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer; Default = none)
RO	Mass density. (Real; Default = none)
E	Young's modulus. (Real; Default = none)
PR	Poisson's ratio. (Real; Default = none)
SIGY	Yield stress. (Real; Default = none)
ETAN	Plastic hardening modulus (for shells only). (Real; Default = 0.0)

MATD030 (SOL 700) LS-DYNA Material #30 -- Shape-Memory Superelastic Material

This material model describes the superelastic response present in shape-memory alloys (SMA), that is the peculiar material ability to undergo large deformations with a full recovery in loading-unloading cycles (See **Figure 8-105**). The material response is always characterized by a hysteresis loop. See references by Auricchio, Taylor, and Lubliner (1997) and Auricchio and Taylor (1997). Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD030	MID	RO	E	PR					
	SIG_ASS	SIG_ASF	SIG_SAS	SIG_SAF	EPSL	ALPHA	YMRT		

Field	Contents
MID	Material identification. (Integer; Default = none)
RO	Density. (Real; Default = none)
E	Young's modulus. (Real; Default = none)
PR	Poisson's ratio. (Real; Default = none)
SIG_ASS	Starting value for the forward phase transformation (conversion of austenite into martensite) in the case of a uniaxial tensile state of stress. A load curve for SIG_ASS as a function of temperature is specified by using the negative of the load curve ID number. (Real; Default = none)
SIG_ASF	Final value for the forward phase transformation (conversion of austenite into martensite) in the case of a uniaxial tensile state of stress. SIG_ASF as a function of temperature is specified by using the negative of the load curve ID number. (Real; Default = none)
SIG_SAS	Starting value for the reverse phase transformation (conversion of martensite into austenite) in the case of a uniaxial tensile state of stress. SIG_SAS as a function of temperature is specified by using the negative of the load curve ID number. (Real; Default = none)

Field	Contents
SIG_SAF	Final value for the reverse phase transformation (conversion of martensite into austenite) in the case of a uniaxial tensile state of stress. SIG_SAF as a function of temperature is specified by using the negative of the load curve ID number. (Real; Default = none)
EPSL	Recoverable strain or maximum residual strain. It is a measure of the maximum deformation obtainable all the martensite in one direction. (Real; Default = 0.0)
ALPHA	Parameter measuring the difference between material responses in tension and compression (set alpha = 0 for no difference). Also, see the following Remark. (Real; Default = 0.0)
YMRT	Young's modulus for the martensite if it is different from the modulus for the austenite. Defaults to the austenite modulus if it is set to zero. (Real; Default = 0.0)

Remarks:

The material parameter alpha, α, measures the difference between material responses in tension and compression. In particular, it is possible to relate the parameter α to the initial stress value of the austenite into martensite conversion, indicated respectively as

$$\sigma_s^{AS,+}$$

and

$$\sigma_s^{AS,-}$$

according to the following expression:

$$\alpha = \frac{\sigma_s^{AS,-} - \sigma_s^{AS,+}}{\sigma_s^{AS,-} + \sigma_s^{AS,+}}$$

In the following, the results obtained from a simple test problem is reported. The material properties are set as:

E	60000 MPa
Nu	0.3

sig_AS_s	520 MPa
sig_AS_f	600 MPa
sig_SA_s	300 MPa
sig_SA_f	200 MPa
epsL	0.07
alpha	0.12
ymrt	50000 MPa

The investigated problem is the complete loading-unloading test in tension and compression. The uniaxial Cauchy stress versus the logarithmic strain is plotted below.

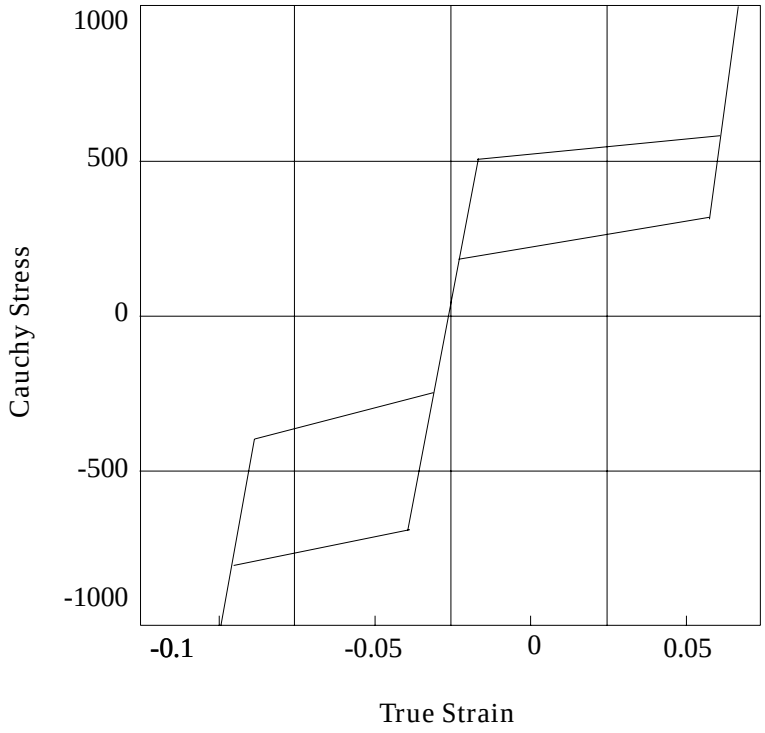


Figure 8-105 Complete Loading-Unloading Test in Tension and Compression

MATD031 (SOL 700) LS-DYNA Material #31 -- Frazer-Nash Rubber

Used to model rubber using the Frazer-Nash formulation. This model defines rubber from uniaxial test data. It is a modified form of the hyperelastic constitutive law first described in (Kendington 1988). See also the Remarks below. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD031	MID	RO	PR	C100	C200	C300	C400		
	C110	C210	C010	C020	EXIT	EMAX	EMIN	REF	
	SGL	SW	ST	LCID					

Field	Contents
MID	Material identification. A unique number has to be defined. (Integer)
RO	Mass density. (Real)
PR	Poisson's ratio. Values between .49 and .50 are suggested. (Real)
C100	C_{100} (EQ.1.0 if term is in the least squares fit.) (Real)
C200	C_{200} (EQ.1.0 if term is in the least squares fit.) (Real)
C300	C_{300} (EQ.1.0 if term is in the least squares fit.) (Real)
C400	C_{400} (EQ.1.0 if term is in the least squares fit.) (Real)
C110	C_{110} (EQ.1.0 if term is in the least squares fit.) (Real)
C210	C_{210} (EQ.1.0 if term is in the least squares fit.) (Real)
C010	C_{010} (EQ.1.0 if term is in the least squares fit.) (Real)
C020	C_{020} (EQ.1.0 if term is in the least squares fit.) (Real)
EXIT	Exit option (Real): EQ.0.0 Stop if strain limits are exceeded (recommended). NE.0.0 Continue if strain limits are exceeded. The curve is then extrapolated.
EMAX	Maximum strain limit, (Green-St, Venant Strain). (Real)
EMIN	Minimum strain limit, (Green-St, Venant Strain). (Real)

Field	Contents
REF	Use reference geometry to initialize the stress tensor. This option is currently restricted to 8-noded solid elements with one point integration. (Real) EQ.0.0 Off. EQ.1.0 On.
SGL	Specimen gauge length. (Real)
SW	Specimen width. (Real)
ST	Specimen thickness. (Real)
LCID	Load curve ID giving the force versus actual change in gauge length. (Integer)

Remarks:

The constants can be defined directly or a least squares fit can be performed if the uniaxial data (SGL, SW, ST and LCID) is available. If a least squares fit is chosen, then the terms to be included in the energy functional are flagged by setting their corresponding coefficients to unity. If all coefficients are zero the default is to use only the terms involving I_1 and I_2. C_{100} defaults to unity if the least square fit is used.

The strain energy functional, U, is defined in terms of the input constants as:

$$U = C_{100}\, I_1 + C_{200}\, I_1^2 + C_{300}\, I_1^3 + C_{400}\, I_1^4 + C_{110}\, I_1 I_2 + C_{210}\, I_1^2\, I_2 + C_{010}\, I_2 + C_{020}\, I_2^2 + f(j)$$

where the invariants can be expressed in terms of the deformation gradient matrix, F_{ij}, and the Green-St. Venant strain tensor, E_{ij}:

$$J = |F_{ij}|$$
$$I_1 = E_{ii}$$
$$I_2 = \frac{1}{2!}\delta^{ij}_{pq} E_{pi} E_{qj}$$

The derivative of U with respect to a component of strain gives the corresponding component of stress

$$S_{ij} = \frac{\partial U}{\partial E_{ij}}$$

here, S_{ij}, is the second Piola-Kirchhoff stress tensor.

The load curve definition that provides the uniaxial data should give the change in gauge length, ΔL, and the corresponding force. In compression both the force and the change in gauge length must be specified as negative values. In tension the force and change in gauge length should be input as positive values. The principal stretch ratio in the uniaxial direction, λ_1, is then given by

$$\lambda = \frac{L_o + \Delta L}{L_o}$$

Alternatively, the stress versus strain curve can also be input by setting the gauge length, thickness, and width to unity and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force.

The least square fit to the experimental data is performed during the initialization phase and is a comparison between the fit and the actual input is provided in the printed file. It is a good idea to visually check the fit to make sure it is acceptable. The coefficients C_{100} - C_{020} are also printed in the output file.

MATD032 (SOL 700) LS-DYNA Material #32 -- Used for Automotive Glass

With this material model, a layered glass including polymeric layers can be modeled. Failure of the glass part is possible. See notes below.

Format:

1	2	3	4	5	6	7	8	9	10
MATD032	MID	RO	EG	PRG	SYG	ETG	EFG	EP	
	PRP	SYP	ETP						

Format for integration point material: Up to 4 additional entries must account for NINTS values (minumum of 2 and maximum of 32 values)

	F1	F2	F3	F4	F5	F6	F7	F8	

Field	Contents
MID	Material identification. A unique number has to be defined. (Required, Integer, No default)
RO	Mass density. (Required, Real, No default)
Eg	Young's modulus for glass. (Required, Real, No default)
PRg	Poisson's ratio for glass. (Required, Real, No default)
SYg	Yield stress for glass. (Real, Default = 1.0E16)
ETg	Plastic hardening modulus for glass. (Real, Default = 1.0E16)
EFg	Plastic strain at failure for glass. (Real, Default = 1.0E16)
Ep	Young's modulus for polymer. (Real)
PRp	Poisson's ratio for polymer. (Real)
SYp	Yield stress for polymer. (Real, Default = 1.0E16)
ETp	Plastic hardening modulus for polymer. (Real, Default=1.0E16)
f1,..fn	Integration point material (Real, At least 2 points are required): f_n = 0.0: glass, f_n = 1.0: polymer. A user-defined integration rule must be specified.

Remark:

1. Isotropic hardening for both materials is assumed. The material to which the glass is bonded is assumed to stretch plastically without failure. A user defined integration rule specifies the thickness of the layers making up the glass. F*i* defines whether the integration point is glass (0.0) or polymer (1.0). The material definition, F*i*, has to be given for the same number of integration points (NIPTS) as specified in the rule. A maximum of 32 layers is allowed.

MATD054 (SOL 700) LS-DYNA Material #54 -- Enhanced Composite Material Model

Either header, MATD054 or MATD055 may be used. They are the same. This material is an enhanced version of the composite model material type 22. Arbitrary orthothropic materials, e.g., unidirectional layers in composite shell structures can be defined. Optionally, various types of failure can be specified following either the suggestions of (Chang and Chang, 1984) or (Tsai and Wu, 1981). In addition special measures are taken for failure under compression. See (Matzenmiller and Schweizerhof, 1990). This model is only valid for thin shell elements. The parameters in parentheses below apply only to solid elements and are therefore always ignored in this material model. They are included for consistency with materials MATD022 and MATD059. By using the user defined integration rule, the constitutive constants can vary through the shell thickness. For all shells, except the DKT formulation, laminated shell theory can be activated to properly model the transverse shear deformation. Lamination theory is applied to correct for the assumption of a uniform constant shear strain through the thickness of the shell. For sandwich shells where the outer layers are much stiffer than the inner layers, the response will tend to be too stiff unless lamination theory is used. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD054	MID	RO	EA	EB	(EC)	PRBA	(PRCA)	(PRCB)	
	GAB	GBC	GCA	(KF)	AOPT				
				A1	A2	A3	MANGLE		
	V1	V2	V3	D1	D2	D3	DFAILM	DFAILS	
	TFAIL	ALPH	SOFT	FBRT	YCFAC	DFAILT	DFAILC	EFS	
	XC	XT	YC	YT	SC	CRIT	BETA		

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
EA	E_a, Young's modulus -- longitudinal direction. (Real)
EB	E_b, Young's modulus -- transverse direction. (Real)

Field	Contents
(EC)	E_c, Young's modulus -- normal direction (not used - internally set to $0.5 \cdot (E_a + E_b)$). (Real)
PRBA	ν_{ba}, Poisson's ratio ba. (Real)
(PRCA)	ν_{ca}, Poisson's ratio ca (not used - internally set to PRBA). (Real)
(PRCB)	ν_{cb}, Poisson's ratio cb (not used - internally set to PRBA). (Real)
GAB	G_{ab}, shear modulus ab. (Real)
GBC	G_{bc}, shear modulus bc. (Real)
GCA	G_{ca}, shear modulus ca. (Real)
(KF)	Bulk modulus of failed material (not used). (Real)
AOPT	Material axes option (Integer): EQ.0 Locally orthotropic with material axes determined by element nodes 1, 2, and 4. EQ.2 Globally orthotropic with material axes determined by vectors defined below. EQ.3 Locally orthotropic material axes determined by rotating the material axes about the element normal by an angle (MANGLE) from a line in the plane of the element defined by the cross product of the vector **v** with the element normal.
A1 A2 A3	Define components of vector a for AOPT = 2. (Real)
V1 V2 V3	Define components of vector v for AOPT = 3. (Real)
MANGLE	Material angle in degrees for AOPT = 3. (Real)
D1 D2 D3	Define components of vector d for AOPT = 2. (Real)
DFAILM	Maximum strain for matrix straining in tension or compression. The layer in the element is completely removed after the maximum strain in the matrix direction is reached. The input value is always positive. (Real)
DFAILS	Maximum shear strain. The layer in the element is completely removed after the maximum shear strain is reached. The input value is always positive. (Real)
TFAIL	Time step size criteria for element deletion (Real):

Field	Contents
	≤ 0 No element deletion by time step size. The crashfront algorithm only works if t_{fail} is set to a value above zero.
	$0 < t_{fail} \leq 0.1$ Element is deleted when its time step is smaller than the given value.
	$> .1$ Element is deleted when the quotient of the actual time step and the original time step drops below the given value.
ALPH	Shear stress parameter for the nonlinear term; see Material 22. (Real)
SOFT	Softening reduction factor for material strength in crashfront elements (Default = 1.0). TFAIL must be greater than zero to activate this option. (Real)
FBRT	Softening for fiber tensile strength. (Real):
	EQ.0.0 Tensile strength = X_t
	GT:0.0 Tensile strength = X_t, reduced to $X_1 \cdot \text{FBRT}$ after failure has occured in compressive matrix mode.
YCFAC	Reduction factor for compressive fiber strength after matrix failure. The compressive strength in the fiber direction after compressive matrix failure is reduced to $X_c = \text{YCFAC} \cdot Y_c$ (default: $\text{YCFAC} = 2.0$). (Real)
DFAILT	Maximum strain for fiber tension (maximum 1 = 100% strain). The layer in the element is completely removed after the maximum tensile strain in the fiber direction is reached. (Real)
DFAILC	Maximum strain for fiber compression (maximum -1 = 100% compression). The layer in the element is completely removed after the maximum tensile strain in the fiber direction is reached. (Real)
EFS	Effective strain failure. (Real)
XC	Longitudinal compressive strength. (Real)
XT	Longitudinal tensile strength (see below). (Real)
YC	Transverse comprehensive strength, b-axis (see below). (Real)
YT	Transverse tensile strength, b-axis (see below). (Real)
SC	Shear strength, ab plane (see below). (Real)
CRIT	Failure criterion (material number) (Real):

Field	Contents
	EQ.54.0 Chang matrix failure criterion (as Material 22) (default).
	EQ.55.0 Tsai-Wu criterion for matrix failure.
BETA	Weighting factor for shear term in tensile fiber mode ($0.0 \le$ BETA ≤ 1.0). (Real)

Remarks:

The Chang / Chang (mat_54) criteria is given as follows:

For the tensile fiber mode:

$$\sigma_{aa} > 0 \text{ then } e_f^2 = \left(\frac{\sigma_{aa}}{X_1}\right)^2 + \beta\left(\frac{\sigma_{ab}}{S_c}\right) - 1 \begin{cases} \ge 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases}$$

$$E_a = E_b = G_{ab} = \nu_{ba} = \nu_{ab} = 0$$

For the compressive fiber mode:

$$\sigma_{aa} < 0 \text{ then } e_c^2 = \left(\frac{\sigma_{aa}}{X_c}\right)^2 - 1 \begin{cases} \ge 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases}$$

$$E_a = \nu_{ba} = \nu_{ab} = 0$$

For the tensile matrix mode:

$$\sigma_{bb} > 0 \text{ then } e_m^2 = \left(\frac{\sigma_{bb}}{X_t}\right)^2 + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \ge 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases}$$

$$E_b = \nu_{ba} = 0. \rightarrow G_{ab} = 0$$

For the compressive matrix mode:

$$\sigma_{bb} < 0 \text{ then } e_d^2 = \left(\frac{\sigma_{bb}}{2S_c}\right)^2 + \left[\left(\frac{Y_c}{2S_c}\right)^2 - 1\right]\frac{\sigma_{bb}}{Y_c} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1 \begin{cases} \ge 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases}$$

$$_b = \nu_{ba} = \nu_{ab} = 0. \rightarrow G_{ab} = 0$$

$$X_c = 2Y_c \text{ for 50\% fiber volume}$$

In the Tsai-Wu (MATD055) criteria, the tensile and compressive fiber modes are treated as in the Chang-Chang criteria. The failure criterion for the tensile and compressive matrix mode is given as:

$$f_{md}^2 < \frac{\sigma_{bb}^2}{Y_c Y_t} + \left(\frac{\sigma_{ab}}{2_c}\right)^2 + \frac{(Y_c - Y_t)\sigma_{bb}}{Y_c Y_t} - 1 \begin{cases} \geq 0 & \text{failed} \\ < 0 & \text{elastic} \end{cases}$$

For $\beta = 1$, we get the original criterion of Hashin (1980) in the tensile fiber mode. For $\beta = 0$, we get the maximum stress criterion which is found to compare better to experiments.

Failure can occur in any of four different ways:

1. If DFAILT is zero, failure occurs if the Chang-Chang failure criterion is satisfied in the tensile fiber mode.
2. If DFAILT is greater than zero, failure occurs if the tensile fiber strain is greater than DFAILT or less than DFAILC.
3. If EFS is greater than zero, failure occurs if the effective strain is greater than EFS.
4. If TFAIL is greater than zero, failure occurs according to the element timestep as described in the definition of TFAIL above.

When failure has occurred in all the composite layers (through-thickness integration points), the element is deleted. Elements which share nodes with the deleted element become “crashfront” elements and can have their strengths reduced by using the SOFT parameter with TFAIL greater than zero.

Information about the status in each layer (integration point) and element can be plotted using additional integration point variables. The number of additional integration point variables for shells written to the LS-DYNA database is input by PARAM,DYNEIPS. For MATD054 and MATD055, these additional variables are tabulated below (*i* = shell integration point):

History Variable	Description	Value	Component
1. *ef (i)*	tensile fiber mode	1 - elastic 0 - failed	81
2. *ec (i)*	compressive fiber model		82
3. *em (i)*	tensile matrix mode		83
4. *ed (i)*	compressive matrix mode		84
5. *efail*	max[ef(ip)]		85
6. *dam*	damage parameter	-1 -- element intact 10^{-8} -- element in crashfront +1 -- element failed	86

These variables can be plotted in some postprocessors as element components 81, 82, ..., 80+ NEIPS. The following components, defined by the sum of failure indicators over all through-thickness integration points, are stored as element component 7 instead of the effective plastic strain:

Description	Integration Point
$\frac{1}{nip}\sum_{i=1}^{nip} ef(i)$	1
$\frac{1}{nip}\sum_{i=1}^{nip} ec(i)$	2
$\frac{1}{nip}\sum_{i=1}^{nip} em(i)$	3

MATD057 (SOL 700) LS-DYNA Material #57 -- Highly Compressible Low Density Foams

This material is used to model highly compressible low density foams. Its main applications are for seat cushions and padding on the Side Impact Dummies (SID). Optionally, a tension cut-off failure can be defined. Also, see the Remarks below. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD057	MID	RO	E	LCID	TC	HU	BETA	DAMP	
	SHAPE	FAIL	BVFLAG	ED	BETA1	KCON	REF		

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
E	Young's modulus. (Real)
LCID	Load curve ID for nominal stress versus strain. (Integer)
TC	Tension cut-off stress. (Real; Default = 1.E+20)
HU	Hysteretic unloading factor between 0 and 1 (Default = 1, i.e., no energy dissipation). See Remark **3.** (Real; Default = 1.)
BETA	β, decay constant to model creep in unloading. See Remark **1.** (Real)
DAMP	Viscous coefficient (.05< recommended value <.50) to model damping effects. (Real; Default = 0.05)
	LT.0.0 |DAMP| is the load curve ID, which defines the damping constant as a function of the maximum strain in compression defined as: $\varepsilon_{max} = \max(1 - \lambda_1, 1 - \lambda_2, 1. - \lambda_3)$ In tension, the damping constant is set to the value corresponding to the strain at 0. The abscissa should be defined from 0 to 1.

Field	Contents
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation. See Remark **3.** See also **Figure 8-106**. (Real; Default = 1.0)
FAIL	Failure option after cutoff stress is reached (Integer; Default = 0): EQ.0 Tensile stress remains at cut-off value. EQ.1 Tensile stress is reset to zero.
BVFLAG	Bulk viscosity activation flag. See Remark **2.** (Integer; Default = 0): EQ.0 No bulk viscosity (recommended). EQ.1 Bulk viscosity active.
ED	Optional Young's relaxation modulus, E_d, for rate effects. See Remark **5.** (Real; Default = 0.0)
BETA1	Optional decay constant, β_1. See Remark **5.** (Real; Default = 0.0)
KCON	Stiffness coefficient for contact interface stiffness. If undefined the maximum slope in stress versus strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended. See Remark **6.** (Real; Default = 0.0)
REF	Use reference geometry to initialize the stress tensor. This option is currently restricted to 8-noded solid elements with one point integration. (Integer; Default = 0) EQ.0 Off. EQ.1 On.

Remarks:

The compressive behavior is illustrated in **Figure 8-106** where hysteresis on unloading is shown. This behavior under uniaxial loading is assumed not to significantly couple in the transverse directions. In tension the material behaves in a linear fashion until tearing occurs. Although the implementation may be somewhat unusual, it was motivated by Storakers (1986).

The model uses tabulated input data for the loading curve where the nominal stresses are defined as a function of the elongations, ε_i, which are defined in terms of the principal stretches, λ_i, as:

$$\varepsilon_i = \lambda_i - 1$$

The stretch ratios are found by solving for the eigenvalues of the left stretch tensor, V_{ij}, which is obtained via a polar decomposition of the deformation gradient matrix, F_{ij}. Recall that,

$$F_{ij} = R_{ik}U_{kj} = V_{ik}R_{kj}$$

The update of V_{ij} follows the numerically stable approach of (Taylor and Flanagan 1989). After solving for the principal stretches, we compute the elongations and, if the elongations are compressive, the corresponding values of the nominal stresses, τ_i are interpolated. If the elongations are tensile, the nominal stresses are given by

$$\tau_i = E\varepsilon_i$$

and the Cauchy stresses in the principal system become

$$\sigma_i = \frac{\tau_i}{\lambda_i\lambda_k}$$

The stresses can now be transformed back into the global system for the nodal force calculations.

Additional Remarks:

1. When hysteretic unloading is used the reloading will follow the unloading curve if the decay constant, β, is set to zero. If β is nonzero the decay to the original loading curve is governed by the expression:

$$1. - e^{-\beta t}$$

2. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.
3. The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown below. This unloading provide energy dissipation which is reasonable in certain kinds of foam.
4. Note that since this material has no effective plastic strain, the internal energy per iniitial volume is written into the output databases.

5. Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^{r} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function. The stress tensor

$$\sigma_{ij}^{r}$$

augments the stresses determined from the foam,

$$\sigma_{ij}^{f}$$

consequently, the final stress, σ_{ij} is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^{f} + \sigma_{ij}^{r}$$

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} a_m e^{-\beta t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a Young's modulus, E_d, and decay constant, β_1.The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided. Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates twelve additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to “remember” the local system of principal stretches.

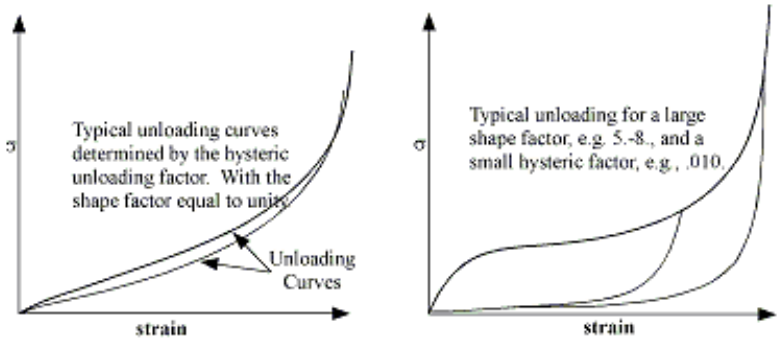


Figure 8-106 Behavior of the Low Density Urethane Foam Model

6. The time step size is based on the current density and the maximum of the instantaneous loading slope, E, and ECON. If ECON is undefined the maximum slope in the loading curve is used instead.

MATD058 (SOL 700) LS-DYNA Material #58 -- Composite and Fabrics

Depending on the type of failure surface, this model may be used to model composite materials with unidirectional layers, complete laminates, and woven fabrics. This model is implemented for shell and thick shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
MATD058	MID	RO	EA	EB	(EC)	PRBA	TAU1	GAMMA1	
	GAB	GBC	GCA	SLIMT1	SLMIC1	SLIMT2	SLIMC2	SLIMS	
	AOPT	TSIZE	ERODS	SOFT	FS				
	XP	YP	ZP	A1	A2	A3			
	V1	V2	V3	D1	D2	D3	BETA		
	E11C	E11T	E22C	E22T	GMS				
	XC	XT	YC	YT	SC				

Field	Contents
MID	Material identification. A unique number has to be chosen. (Required, Integer, no Default)
RO	Mass density. (Required, Real, no Default)
EA	E_a, Young's modulus - longitudinal direction. (Required, Real, no Default)
EB	E_b, Young's modulus - transverse direction. (Required, Real, no Default)
(EC)	E_c, Young's modulus - normal direction (not used)
PRBA	ν_{ba}, Poisson's ratio ba. (Required, Real, no Default)
TAU1	$\tau 1$, stress limit of the first slightly nonlinear part of the shear stress versus shear strain curve. The values $\tau 1$ and $\gamma 1$ are used to define a curve of shear stress versus shear strain. These values are input if FS, defined below, is set to a value of -1. (Real)
GAMMA 1	$\gamma 1$, strain limit of the first slightly nonlinear part of the shear stress versus shear strain curve. (Real)
GAB	G_{ab}, shear modulus ab. (Required, Real, no Default)
GBC	G_{bc}, shear modulus bc. (Real, Default = GAB)
GCA	G_{ca}, shear modulus ca. (Real, Default = GCA)

Field	Contents
SLIMT1	Factor to determine the minimum stress limit after stress maximum (fiber tension). (Real)
SLIMC1	Factor to determine the minimum stress limit after stress maximum (fiber compression). (Real)
SLIMT2	Factor to determine the minimum stress limit after stress maximum (matrix tension). (Real)
SLIMC2	Factor to determine the minimum stress limit after stress maximum (matrix compression). (Real)
SLIMS	Factor to determine the minimum stress limit after stress maximum (shear). (Real)
aopt	Material axes option (Real): EQ. 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4. EQ. 2.0: globally orthotropic with material axes determined by vectors defined below. EQ. 3.0: locally orthotropic material axes determined by rotating the material axes about the element normal by an angle (BETA) from a line in the plane of the element defined by the cross product of the vector **v** with the element normal.
TSIZE	Time step for automatic element deletion. (Real)
ERODS	Maximum effective strain for element layer failure. A value of unity would equal 100% strain. (Real)
SOFT	Softening reduction factor for strength in the crashfront. (Real)
FS	Failure surface type (Real): EQ.1.0: smooth failure surface with a quadratic criterion for both the fiber (a) and transverse (b) directions. This option can be used with complete laminates and fabrics. EQ.0.0: smooth failure surface in the transverse (b) direction with a limiting value in the fiber (a) direction. This model is appropriate for unidirectional (UD) layered composites only. EQ.-1.: faceted failure surface. When the strength values are reached then damage evolves in tension and compression for both the fiber and transverse direction. Shear behavior is also considered. This option can be used with complete laminates and fabrics.

Field	Contents
xp yp zp	Define coordinates of point p for AOPT = 1. (Real)
a1 a2 a3	Define components of vector a for AOPT = 2. (Real)
v1 v2 v3	Define components of vector v for AOPT = 3. (Real)
d1 d2 d3	Define components of vector d for AOPT = 2. (Real)
BETA	Material angle in degrees for AOPT = 3. (Real)
E11C	Strain at longitudinal compressive strength, a-axis. (Real)
E11T	Strain at longitudinal tensile strength, a-axis. (Real)
E22C	Strain at transverse compressive strength, b-axis. (Real)
E22T	Strain at transverse tensile strength, b-axis. (Real)
GMS	Strain at shear strength, ab plane. (Real)
Xc	Longitudinal compressive strength. (Real)
XT	Longitudinal tensile strength, see below. (Real)
YC	Transverse compressive strength, b-axis, see below. (Real)
YT	Transverse tensile strength, b-axis, see below. (Real)
SC	Shear strength, ab plane, see below. (Real)

Remark:

Parameters to control failure of an element layer are: ERODS, the maximum effective strain, i.e., maximum 1 = 100 % straining. The layer in the element is completely removed after the maximum effective strain (compression / tension including shear) is reached.

The stress limits are factors used to limit the stress in the softening part to a given value,

$$\sigma_{min} = SLIMxx \cdot strength$$

thus, the damage value is slightly modified such that elastoplastic like behavior is achieved with the threshold stress. As a factor for *SLIMxx* a number between 0.0 and 1.0 is possible. With a factor of 1.0, the stress remains at a maximum value identical to the strength, which is similar to ideal elastoplastic behavior. For tensile failure a small value for *SLIMTx* is often reasonable; however, for compression *SLIMCx = 1.0* is preferred. This is also valid for the corresponding shear value. If *SLIMxx* is smaller than 1.0 then localization can be observed depending on the total behavior of the lay-up. If the user is intentionally using *SLIMxx < 1.0*, it is generally recommended to

avoid a drop to zero and set the value to something in between 0.05 and 0.10. Then elastoplastic behavior is achieved in the limit which often leads to less numerical problems. Defaults for SLIMXX = 1.0E-8.

The crashfront-algorithm is started if and only if a value for TSIZE (time step size, with element elimination after the actual time step becomes smaller than TSIZE) is input.

The damage parameters can be written to the postprocessing database for each integration point as the first three additional element variables and can be visualized.

Material models with FS=1 or FS=-1 are favorable for complete laminates and fabrics, as all directions are treated in a similar fashion.

For material model FS=1 an interaction between normal stresses and the shear stresses is assumed for the evolution of damage in the a and b-directions. For the shear damage is always the maximum value of the damage from the criterion in a or b-direction is taken.

For material model FS=-1 it is assumed that the damage evolution is independent of any of the other stresses. A coupling is only present via the elastic material parameters and the complete structure.

In tensile and compression directions and in a as well as in b- direction different failure surfaces can be assumed. The damage values, however, increase only also when the loading direction changes.

Special control of shear behavior of fabrics

For fabric materials a nonlinear stress strain curve for the shear part for failure surface FS=-1 can be assumed as given below. This is not possible for other values of FS.

The curve, shown in **Figure 8-107** is defined by three points:

1. the origin (0,0) is assumed,
2. the limit of the first slightly nonlinear part (must be input), stress (TAU1) and strain (GAMMA1), see below.
3. the shear strength at failure and shear strain at failure.

In addition a stress limiter can be used to keep the stress constant via the *SLIMS* parameter. This value must be less or equal 1.0 but positive, and leads to an elastoplastic behavior for the shear part. The default is 1.0E-08, assuming almost brittle failure once the strength limit SC is reached.

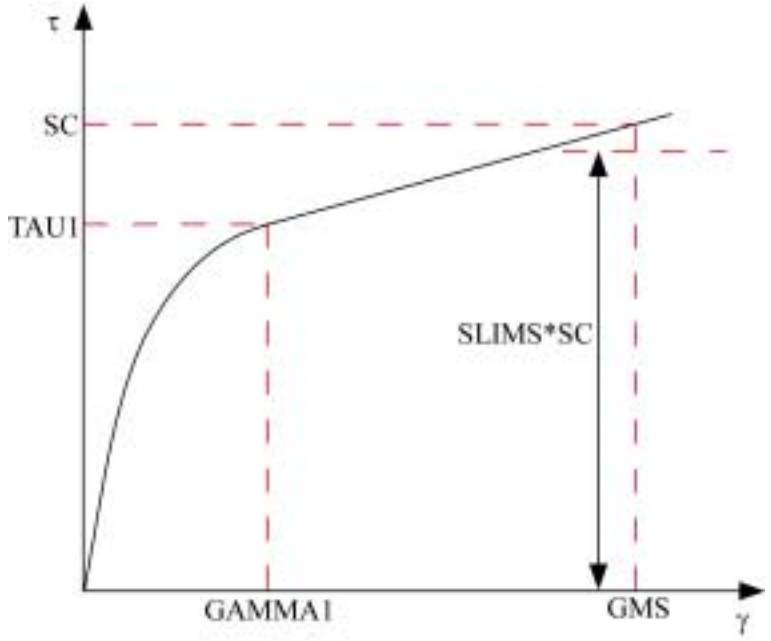


Figure 8-107 Stress-strain diagram for shear.

MATD059 (SOL 700) LS-DYNA Material #59 -- Shell or Solid Composite Models

This material is used to model shells or solid composite structures. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD059	MID	RO	EA	EB	EC	PRBA	PRCA	Option	
	PRCB	GAB	GBC	GCA	KF	AOPT	MAFLAG		
	XP	YP	ZP	A1	A2	A3			
	V1	V2	V3	D1	D2	D3	BETA		

Add Lines 5 and 6 for Option=SHELL:

	TSIZE	ALP	SOFT	FBRT	SR	SF			
	XC	XT	YC	YT	SC				

Add Lines 5 and 6 for Option=SOLID:

	SBA	SCA	SCB	XXC	YYC	ZZC			
	XXT	YYT	ZZT						

Field	Contents
MID	Material identification. (Integer)
RO	Density. (Real)
EA	E_a, Young's modulus - longitudinal direction. (Real)
EB	E_b, Young's modulus - transverse direction. (Real)
EC	E_a, Young's modulus - normal direction. (Real)
PRBA	ν_{ba}, Poisson's ratio ba. (Real)
PRCA	ν_{ca}, Poisson's ratio ca. (Real)
Option	Character value -- must be SHELL or SOLID. (Character)
PRCB	ν_{cb} Poisson's ratio cb. (Real)

Field	Contents
GAB	G_{ab} Shear modulus. (Real)
GBC	G_{bc} Shear modulus. (Real)
GCA	G_{ca} Shear modulus. (Real)
KF	Bulk modulus of failed material. (Real)
AOPT	Material axes option (Integer):
	EQ.0 Locally orthotropic with material axes determined by element nodes 1, 2, and 4.
	EQ.1 Locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. This option is for solid elements only.
	EQ.2 Globally orthotropic with material axes determined by vectors defined below.
	EQ.3 Locally orthotropic material axes determined by rotating the material axes about the element normal by an angle, BETA, from a line in the plane of the element defined by the cross product of the vector v with the element normal.
	EQ.4 Locally orthotropic in cylindrical coordinate system with the material axes determined by a vector v, and an originating point, P, which define the centerline axis. This option is for solid elements only.
MAFLAG	Material axes change flag for brick elements (Integer):
	EQ.1 Default.
	EQ.2 Switch material axes a and b.
	EQ.3 Switch material axes a and c.
XP YP ZP	Define coordinates of point p for AOPT = 1 and 4. (Real)
A1 A2 A3	Define components of vector a for AOPT = 2. (Real)
V1 V2 V3	Define components of vector v for AOPT = 3 and 4. (Real)
D1 D2 D3	Define components of vector d for AOPT = 2. (Real)
BETA	Material angle in degrees for AOPT = 3; may be overridden on the element card.

Field	Contents
TSIZE	Time step for automatic element deletion. (Real)
ALP	Nonlinear shear stress parameter. (Real)
SOFT	Softening reduction factor for strength in crush. (Real)
FBRT	Softening of fiber tensile strength. (Real)
SR	s_r, reduction factor (Default=0.447). (Real)
SF	s_f, softening factor (Default=0.0). (Real)
XC	Longitudinal compressive strength, a-axis. (Real)
XT	Longitudinal tensile strength, a-axis. (Real)
YC	Transverse compressive strength, b-axis. (Real)
YT	Transverse tensile strength, b-axi. (Real)
SC	Shear strength, ab plane (Real): GT:0.0 Faceted failure surface theory. LT:0.0 Ellipsoidal failure surface theory.
SBA	In plane shear strength. (Real)
SCA	Transverse shear strength. (Real)
SCB	Transverse shear strength. (Real)
XXC	Longitudinal compressive strength a-axis. (Real)
YYC	Transverse compressive strength b-axis. (Real)
ZZC	Normal compressive strength c-axis. (Real)
XXT	Longitudinal tensile strength a-axis. (Real)
YYT	Transverse tensile strength b-axis. (Real)
ZZT	Normal tensile strength c-axis. (Real)

MATD062 (SOL 700) LS-DYNA Material #62 -- Confor Viscous Foam Model

Used to model viscous foams. It was written to represent the Confor Foam on the ribs of EuroSID side impact dummy. It is only valid for solid elements, mainly under compressive loading. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD062	MID	RO	E1	N1	V2	E2	N2	PR	

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
E1	Initial Young's modulus (E_1). (Real)
N1	Exponent in power law for Young's modulus (n_1). (Real)
V2	Viscous coefficient (V_1). (Real)
E2	Elastic modulus for viscosity (E_2). See Remarks below. (Real)
N2	Exponent in power law for viscosity (n_s). (Real)
PR	Poisson's ratio, ν. (Real)

Remarks:

The model consists of a nonlinear elastic stiffness in parallel with a viscous damper. The elastic stiffness is intended to limit total crush while the viscosity absorbs energy. The stiffness E_2 exists to prevent timestep problems. It is used for time step calculations a long as E_1^t is smaller than E_2. It has to be carefully chosen to take into account the stiffening effects of the viscosity. Both E1 and V2 are nonlinear with crush as follows:

$$E_1^t = E_1\left(V^{-n_1}\right)$$

$$V_2^t = V_2(abs(1-V))^{1/2}$$

where viscosity generates a shear stress given by

$$\tau = V_2\dot{\gamma}$$

$\dot{\gamma}$ is the engineering shear strain rate, and V is the relative volume defined by the ratio of the current to initial volume. Typical values are (units of N, mm, s)

$E_1 = 0.0035$	$n_1 = 4.0$	$V_2 = 0.0015$
$E_2 = 100.0$	$n_2 = 0.2$	$\nu = 0.05$

MATD063 (SOL 700) LS-DYNA Material #63 -- Crushable Foam with Damping

Used to model crushable foams. It is dedicated to modeling crushable foam with optional damping and tension cutoff. Unloading is fully elastic. Tension is treated as elastic-perfectly-plastic at the tension cut-off value. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD063	MID	RO	E	PR	LCID	TSC	DAMP		

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer; Default = none)
RO	Mass density. (Real; Default = none)
E	Young's modulus. (Real; Default = none)
PR	Poisson's ratio. (Real; Default = none)
LCID	Load curve ID defining yield stress versus volumetric strain, γ. See **Figure 8-108**. (Real; Default = none)
TSC	Tensile stress cutoff. A nonzero, positive value is strongly recommended for realistic behavior. (Real; Default = 0.0)
DAMP	Rate sensitivity via damping coefficient (.05 < recommended value < .50). (Real; Default = 0.10)

Remarks:

The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = 1.-V$$

The relative volume is defined as the ratio of the current to the initial volume. In place of the effective plastic strain in the D3PLOT database, the integrated volumetric strain is output.

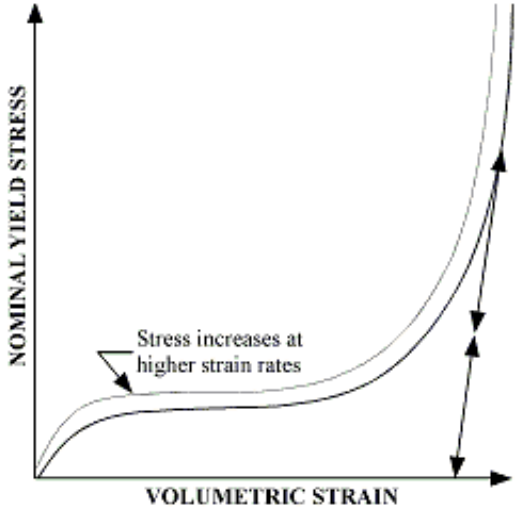


Figure 8-108 Behavior of Strain Rate Sensitive Crushable Foam

Unloading is elastic to the tension cutoff. Subsequent reloading follows the unloading curve.

MATD064 (SOL 700) LS-DYNA Material #64 -- Strain-Rate Dependent Plasticity

Used to model strain rate sensitive elasto-plastic material with a power law hardening. Optionally, the coefficients can be defined as functions of the effective plastic strain. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD064	MID	RO	E	PR	K	M	N	E0	
	VP	EPS0							

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
E	Young's modulus of elasticity. (Real)
PR	Poisson's ratio. (Real)
K	Material constant, k. If k < 0, the absolute value of k is taken as the load curve number that defines k as a function of effective plastic strain. (Real)
M	Strain hardening coefficient, m. If m < 0 the absolute value of m is taken as the load curve number that defines m as a function of effective plastic strain. (Real; Default = 0.0001)
N	Strain rate sensitivity coefficient, n. If n < 0 the absolute value of n is taken as the load curve number that defines n as a function of effective plastic strain. (Real; No default)
E0	Initial strain rate. (Real; Default = 0.0002)
VP	Formulation for rate effects (Integer; Default = 0): EQ.0 Scale yield stress. (Default) EQ.1 Viscoplastic formulation.
EPS0	Factor to normalize strain rate. (Real; Default = 1.0) EQ.1.0 Time units of seconds. (Default) EQ.1.E-3 Time units of milliseconds. EQ.1.E-6 Time units of microseconds.

Remarks:

This material model follows a constitutive relationship of the form:

$$\sigma = k\varepsilon^{m}\dot{\varepsilon}^{n}$$

where σ is the yield stress, ε is the effective plastic strain, $\dot{\varepsilon}$ is the normalized effective plastic strain rate, and the constants k, m, and n can be expressed as functions of effective plastic strain or can be constant with respect to the plastic strain. The case of no strain hardening can be obtained by setting the exponent of the plastic strain equal to a very small positive value, i.e. 0.0001.

This model can be combined with the superplastic forming input to control the magnitude of the pressure in the pressure boundary conditions in order to limit the effective plastic strain rate so that it does not exceed a maximum value at any integration point within the model.

A fully viscoplastic formulation is optional. An additional cost is incurred but the improvement is results can be dramatic.

MATD066 (SOL 700) Linear Discrete Beam Material

This material model is defined for simulating the effects of a linear elastic beam by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the PBDISCR input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this model. A triad is used to orient the beam for the directional springs. Translational / rotational stiffness and viscous damping effects are considered for a local cartesian system, see notes below. Applications for this element include the modeling of joint stiffnesses.

Format:

1	2	3	4	5	6	7	8	9	10
MATD066	MID	R0	TKR	TKS	TKT	RKR	RKS	RKT	
	TDR	TDS	TDT	RDR	RDS	RDT			

Example:

MATD066	21	2.3	23.5	42.4	42.4	0.0	0.0	0.0	

Field	Contents
MID	Material identification. A unique number has to be chosen. (I > 0, Default = Required)
RO	Mass density, see also “volume” in the PBDISCR definition. (R > 0.0 – Default = Required)
TKR	Translational stiffness about local r-axis. (R $\geq$ 0.0, Default = 0.0)
TKS	Translational stiffness about local s-axis. R $\geq$ 0.0, Default = 0.0
TKT	Translational stiffness about local t-axis. (R $\geq$ 0.0, Default = 0.0)
RKR	Rotational stiffness about the local r-axis. (R $\geq$ 0.0, Default = 0.0)
RKS	Rotational stiffness about the local s-axis. (R $\geq$ 0.0, Default = 0.0)
RKT	Rotational stiffness about the local t-axis. (R $\geq$ 0.0, Default = 0.0)
TDR	Translational viscous damper about local r-axis. (R $\geq$ 0.0, Default = 0.0)
TDS	Translational viscous damper about local s-axis. (R $\geq$ 0.0, Default = 0.0)

Field	Contents
TDT	Translational viscous damper about local t-axis. ($R \geq 0.0$, Default=0.0)
RDR	Rotational viscous damper about the local r-axis .($R \geq 0.0$, Default=0.0)
RDS	Rotational viscous damper about the local s-axis. ($R \geq 0.0$, Default=0.0)
RDT	Rotational viscous damper about the local t-axis. ($R \geq 0.0$, Default=0.0)

Remarks:

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r, s, t) is given by the coordinate ID, in the cross sectional input, see PBDISCR, where the global system is the default. The local coordinate system axes can rotate with either node of the beam or an average rotation of both nodes (see SCOOR variable in PBDISCR).

For null stiffness coefficients, no forces corresponding to these null values will develop. The viscous damping coefficients are optional.

This material corresponds to LS-Dyna *MAT_LINEAR_DISCRETE_BEAM - MAT066

PBDISCR - Discrete Beam Property (SOL 700)

MATD067 (SOL 700) Nonlinear Elastic Discrete Beam

This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the PBDISCR input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs. Arbitrary curves to model transitional / rotational stiffness and damping effects are allowed.

Format:

1	2	3	4	5	6	7	8	9	10
MATD067	MID	RO	LCIDTR	LCIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT	
	LCIDTDR	LCIDDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCCIDRDT			

Example:

MATD067	21	3.4				12	=	14	
		16							

Field	Contents
MID	Material identification. (Integer, Required)
RO	Mass density, see also volume in PBDISCR definition. (Real, Required)
LCIDTR	TABLED ID defining translational force resultant along local r-axis versus relative translational displacement, see Remarks and **Figure 8-109**. (Integer, Default = 0)
LCIDTS	TABLED ID defining translational force resultant along local s-axis versus relative translational displacement. (Integer, Default = 0)
LCIDTT	TABLED ID defining translational force resultant along local t-axis versus relative translational displacement. (Integer, Default = 0)
LCIDRR	TABLED ID defining rotational moment resultant about local r-axis versus relative rotational displacement. (Integer, Default = 0)

Field	Contents
LCIDRS	TABLED ID defining rotational moment resultant about local s-axis versus relative rotational displacement. (Integer, Default = 0)
LCIDRT	TABLED ID defining rotational moment resultant about local t-axis versus relative rotational displacement. (Integer, Default = 0)
LCIDTDR	TABLED ID defining translational damping force resultant along local r-axis versus relative translational velocity. (Integer, Default = 0)
LCIDTDS	TABLED ID defining translational damping force resultant along local s-axis versus relative translational velocity. (Integer, Default = 0)
LCIDTDT	TABLED ID defining translational damping force resultant along local t-axis versus relative translational velocity. (Integer, Default = 0)
LCIDRDR	TABLED ID defining rotational damping moment resultant about local r-axis versus relative rotational velocity. (Integer, Default = 0)
LCIDRDS	TABLED ID defining rotational damping moment resultant about local s-axis versus relative rotational velocity. (Integer, Default = 0)
LCIDRDT	TABLED ID defining rotational damping moment resultant about local t-axis versus relative rotational velocity. (Integer, Default = 0)

Remarks:

Corresponds to Ls-Dyna entry *MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM - MAT067

For null load curve ID's, no forces are computed.

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r, s, t) is given by the coordinate ID, see *DEFINE_ COORDINATE_*OPTION*, in the cross sectional input, see *SECTION_BEAM, where the global system is the default. The local coordinate system axes can rotate with either node of the beam or an average rotation of both nodes (see SCOOR variable in *SECTION_BEAM).

If different behavior in tension and compression is desired in the calculation of the force resultants, the load curve(s) must be defined in the negative quadrant starting with the most negative displacement then increasing monotonically to the most positive. If the load curve behaves similarly in tension and compression, define only the positive quadrant. Whenever displacement values fall outside of the defined range, the resultant forces will be extrapolated. **Figure 8-109**. depicts a typical load

curve for a force resultant. Load curves used for determining the damping forces and moment resultants always act identically in tension and compression, since only the positive quadrant values are considered, i.e., start the load curve at the origin [0,0].

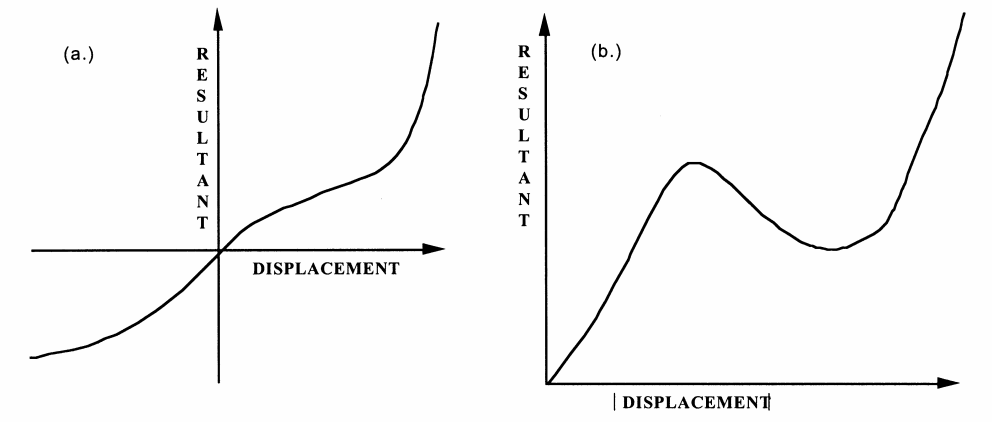


Figure 8-109 The resultant forces and moments are determined by a table lookup. If the origin of the load curve is at [0,0] as in (b.) and tension and compression responses are symmetric.

MATD068 (SOL 700) Nonlinear Plastic Discrete Beam

This material model is defined for simulating the effects of nonlinear elastoplastic, linear viscous behavior of beams by using six springs, each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the PBDISCR input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs. Translational / rotational stiffness and damping effects can be considered. The plastic behavior is modeled using force / moment curves versus displacements / rotation. Optionally, failure can be specified based on a force / moment criterion and a displacement / rotation criterion.

Format:

1	2	3	4	5	6	7	8	9	10
MATD068	MID	RO	TKR	TKS	TKT	RKR	RKS	RKT	
	TDR	TDS	TDT	RDR	RDS	RDT			
	LCPDR	LCPDS	LCPDDT	LCPMR	LCPMS	LCPMT			
	FFAILR	FFAILS	FFAILT	MFAILR	MFAILS	MFAILT			
	UFAILR	UFAILS	UFAIILT	TFAILR	TFAILS	TFAILT	BETA		

Example:

MATD068	21	34.7			23.1				
			51.9						
	12					16			

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer, Required)
RO	Mass density, see also volume on PBEAM definition. (Real, positive, Required)
TKR	Translational stiffness about local r-axis. (Real, $\geq$ 0., Default = 0.0)
TKS	Translational stiffness about local s-axis. (Real, $\geq$ 0., Default = 0.0)
TKT	Translational stiffness about local t-axis. (Real, $\geq$ 0., Default = 0.0)

Field	Contents
RKR	Rotational stiffness about the local r-axis. (Real, $\geq$ 0., Default = 0.0)
RKS	Rotational stiffness about the local s-axis. (Real, $\geq$ 0., Default = 0.0)
RKT	Rotational stiffness about the local t-axis. (Real, $\geq$ 0., Default = 0.0)
TDR	Translational viscous damper about local r-axis. (Real, $\geq$ 0., Default = 0.0)
TDS	Translational viscous damper about local s-axis. (Real, $\geq$ 0., Default = 0.0)
TDT	Translational viscous damper about local t-axis. (Real, $\geq$ 0., Default = 0.0)
RDR	Rotational viscous damper about the local r-axis. (Real, $\geq$ 0., Default = 0.0)
RDS	Rotational viscous damper about the local s-axis. (Real, $\geq$ 0., Default = 0.0)
RDT	Rotational viscous damper about the local t-axis. (Real, $\geq$ 0., Default = 0.0)
LCPDR	TABLED ID-yield force versus plastic displacement r-axis. If the TABLED ID zero, and if TKR is nonzero, then nonlinear elastic behavior is obtained for this component. (Integer $\geq$ 0, Default = 0)
LCPDS	TABLED ID-yield force versus plastic displacement s-axis. If the TABLED ID zero, and if TKS is nonzero, then nonlinear elastic behavior is obtained for this component. (Integer $\geq$ 0, Default = 0)
LCPDT	TABLED ID-yield force versus plastic displacement t-axis. If the TABLED ID zero, and if TKT is nonzero, then nonlinear elastic behavior is obtained for this component. (Integer $\geq$ 0, Default = 0)
LCPMR	TABLED ID-yield moment versus plastic rotation r-axis. If the TABLED ID zero, and if RKR is nonzero, then nonlinear elastic behavior is obtained for this component. (Integer $\geq$ 0, Default = 0)
LCPMS	TABLED ID-yield moment versus plastic rotation s-axis. If the TABLED ID zero, and if RKS is nonzero, then nonlinear elastic behavior is obtained for this component. (Integer $\geq$ 0, Default = 0)
LCPMT	TABLED ID-yield moment versus plastic rotation t-axis. If the TABLED ID zero, and if RKT is nonzero, then nonlinear elastic behavior is obtained for this component. (Integer $\geq$ 0, Default = 0)
FFAILR	Optional failure parameter. If zero, the corresponding force, F_r, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
FFAILS	Optional failure parameter. If zero, the corresponding force, F_s, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)

Field	Contents
FFAILT	Optional failure parameter. If zero, the corresponding force, F_t, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
MFAILR	Optional failure parameter. If zero, the corresponding moment, M_r, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
MFAILS	Optional failure parameter. If zero, the corresponding moment, M_s, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
MFAILT	Optional failure parameter. If zero, the corresponding moment, M_t, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
UFAILR	Optional failure parameter. If zero, the corresponding displacement, u_r, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
UFAILS	Optional failure parameter. If zero, the corresponding displacement, u_s, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
UFAILT	Optional failure parameter. If zero, the corresponding displacement, u_t, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
TFAILR	Optional failure parameter. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
TFAILS	Optional failure parameter. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)
TFAILT	Optional failure parameter. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation. (Real, $\geq$ 0., Default = 0.0)

Remarks:

Corresponds to Ls-Dyna entry *MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM-MAT068.

For the translational and rotational degrees of freedom where elastic behavior is desired, set the load curve ID to zero.

The formulation of the discrete beam (type 6) assumes that the beam is of zero length and requires no orientation node. A small distance between the nodes joined by the beam is permitted. The local coordinate system which determines (r,s,t) is given by the coordinate ID (see *DEFINE_COORDINATE_*OPTION*) in the cross sectional

input, see *SECTION_BEAM, where the global system is the default. The local coordinate system axes can rotate with either node of the beam or an average rotation of both nodes (see SCOOR variable in *SECTION_BEAM).

Catastrophic failure based on force resultants occurs if the following inequality is satisfied.

$$\left(\frac{F_r}{F_r^{fail}}\right)^2 + \left(\frac{F_s}{F_s^{fail}}\right)^2 + \left(\frac{F_t}{F_t^{fail}}\right)^2 + \left(\frac{M_r}{M_r^{fail}}\right)^2 + \left(\frac{M_s}{M_s^{fail}}\right)^2 + \left(\frac{M_t}{M_t^{fail}}\right)^2 - 1. \geq 0.$$

After failure the discrete element is deleted. Likewise, catastrophic failure based on displacement resultants occurs if the following inequality is satisfied:

$$\left(\frac{u_r}{u_r^{fail}}\right)^2 + \left(\frac{u_s}{u_s^{fail}}\right)^2 + \left(\frac{u_t}{u_t^{fail}}\right)^2 + \left(\frac{\theta_r}{\theta_r^{fail}}\right)^2 + \left(\frac{\theta_s}{\theta_s^{fail}}\right)^2 + \left(\frac{\theta_t}{\theta_t^{fail}}\right)^2 - 1. \geq 0.$$

After failure the discrete element is deleted. If failure is included either one or both of the criteria may be used.

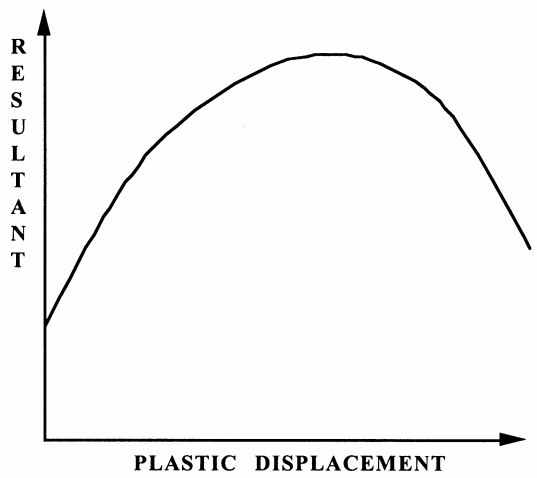


Figure 8-110 The resultant forces and moments are limited by the yield definition. The initial yield point corresponds to a plastic displacement of zero.

MATD069 (SOL 700) SID Damper Discrete Beam

The side impact dummy uses a damper that is not adequately treated by the nonlinear force versus relative velocity curves since the force characteristics are dependent on the displacement of the piston.

Format:

1	2	3	4	5	6	7	8	9	10
MATD069	MID	RO	ST	D	R	H	K	C	
	C3	STF	RHOF	C1	CC2	LCIDF	LCIDD	S0	
	ORFLOC	ORFRAD	SF	DC					

Example:

MATD069	45	4.5E-4	3.5	1.25	.6	.25	.05	1.5	
	1.1	9000.	1.0E-5	35.	2.7	101	102	0.0	
	0.0	0.0	1.0	.02					
	1.0	1.1	1.5	.025					

Field	Contents
MID	Material identification. (Integer, Required)
RO	Mass density. (Real, Required)
ST	S_t, piston stroke. S_t must equal or exceed the length of the beam element, see **Figure 8-111**. (Real > 0.0, Required)
D	d, piston diameter. (Real> 0.0, Required)
R	R, default orifice radius. (Real > 0.0, Required)
H	h, orifice controller position. (Real > 0.0, Required)
K	K, damping constant (Real $\geq$ 0. or Integer < 0, Required)
	If < 0: |K| is the TABLED ID, defining the damping coefficient as a function of the absolute value of the relative velocity.
C	C, discharge coefficient. (Real > 0.0, Required)
C3	Coefficient for fluid inertia term. (Real > 0.0, Required)
STF	k, stiffness coefficient if piston bottoms out. (Real > 0.0, Required)
RHOF	ρ_{fail}, fluid density. (Real > 0.0, Required)

Field	Contents
C1	C_1, coefficient for linear velocity term. (Real > 0.0, Required)
C2	C_2, coefficient for quadratic velocity term. (Real > 0.0, Required)
LCIDF	TABLED ID defining force versus piston displacement, s, i.e., term $f(s+s_0)$. Compressive behavior is defined in the positive quadrant of the force displacement curve. Displacements falling outside of the defined force displacement curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable. (Integer $\geq$ 0, Required)
LCIDD	TABLED ID defining damping coefficient versus piston displacement, s, i.e., $g(s+s_0)$. Displacements falling outside the defined curve are extrapolated. Care must be taken to ensure that extrapolated values are reasonable. (Integer $\geq$ 0, Required)
S0	Initial displacement s_0, typically set to zero. A positive displacement corresponds to compressive behavior. (Real $\geq$ 0.0, Default = 0.0
ORFLOC	d_i, orifice location of ith orifice relative to the fixed end. (Real $\geq$ 0.0, Required, see Remark 2.)
ORFRAD	r_i, orifice radius of ith orifice, if zero the default radius is used. (Real $\geq$ 0.0, Default = 0.0, see Remark 2.)
SF	Scale factor on calculated force. (Real $\geq$ 0.0, Default = 1.0, see Remark 2.)
DC	c, linear viscous damping coefficient used after damper bottoms out either in tension or compression. (Real $\geq$ 0.0, Required, see Remark 2.)

Remarks:

1. Corresponds to Ls-Dyna entry *MAT_SID_DAMPER_DISCRETE_BEAM - MAT069.
2. The third entry (second continuation entry) may be repeated up to 14 times for a total of 15 orifice locations.
3. As the damper moves, the fluid flows through the open orifices to provide the necessary damping resistance. While moving as shown in **Figure 8-111** the piston gradually blocks off and effectively closes the orifices. The number of orifices and the size of their opening control the damper resistance and performance. The damping force is computed from,

$$F = SF\left\{KA_pV_p\left\{\frac{C_1}{A_0^t}+C_2|V_p|\rho_{fluid}\left[\left(\frac{A_p}{CA_0^t}\right)-1\right]\right\}-f(s+s_0)+V_pg(s+s_0)\right\}$$

where K is a user-defined constant or a tabulated function of the absolute value of the relative velocity, V_p is the piston velocity, C is the discharge coefficient, A_p is the piston area, A_0^t is the total open areas of orifices at time t, ρ_{fluid} is the fluid density, C_1 is the coefficient for the linear term, and C_2 is the coefficient for the quadratic term.

In the implementation, the orifices are assumed to be circular with partial covering by the orifice controller. As the piston closes, the closure of the orifice is gradual. This gradual closure is properly taken into account to insure a smooth response. If the piston stroke is exceeded, the stiffness value, k, limits further movement, i.e., if the damper bottoms out in tension or compression the damper forces are calculated by replacing the damper by a bottoming out spring and damper, k and c, respectively. The piston stroke must exceed the initial length of the beam element. The time step calculation is based in part on the stiffness value of the bottoming out spring. A typical force versus displacement curve at constant relative velocity is shown in **Figure 8-112**.

The factor, *SF*, which scales the force defaults to 1.0 and is analogous to the adjusting ring on the damper.

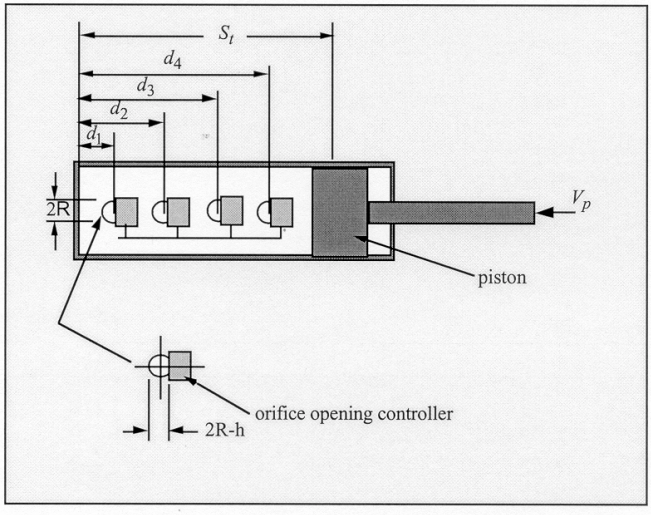


Figure 8-111 Mathematical model for the Side Impact Dummy damper.

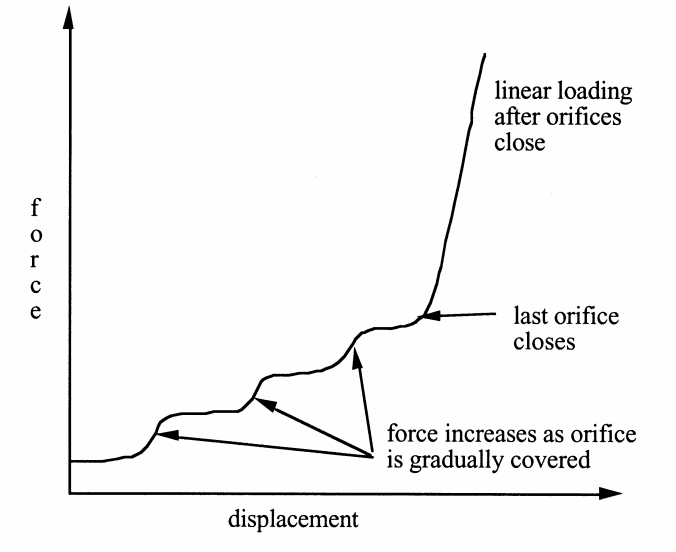


Figure 8-112 Force versus displacement as orifices are covered at a constant relative velocity. Only the linear velocity term is active.

MATD070 (SOL 700) Hydraulic Gas Discrete Beam

This special purpose element represents a combined hydraulic and gas-filled damper which has a variable orifice coefficient. A schematic of the damper is shown in **Figure 8-113**. Dampers of this type are sometimes used on buffers at the end of railroad tracks and as aircraft undercarriage shock absorbers. This material can be used only with discrete beam elements.

Format:

1	2	3	4	5	6	7	8	9	10
MATD070	MID	RO	CO	N	P0	PA	AP	KH	
	LCID	FR	SCLF	CLEAR					

Example:

MATD070	151	4.4E-4	2.0	87683.	200.	14.5	6.457	33.62	
	33	1.0	1.0	0.0					

Field	Contents
MID	Material identification. (Integer > 0, Required)
RO	Mass density. (Real > 0.0, Required)
CO	Length of gas column, Co. (Real > 0.0, Required)
N	Adiabatic constant. (Real $\geq$ 0.0, Required)
P0	Initial gas pressure, P0. (Real $\geq$ 0.0, Required)
PA	Atmospheric pressure, Pa. (Real $\geq$ 0.0, Required)
AP	Piston cross sectional area, Ap. (Real $\geq$ 0.0, Required)
KH	Hydraulic constant, K. (Real $\geq$ 0.0, Required)
LCID	TABLED ID, defining the orifice area, a_0, versus element deflection. (Integer, Required)
FR	Return factor on orifice force. This acts as a factor on the hydraulic force only and is applied when unloading. It is intended to represent a valve that opens when the piston unloads to relieve hydraulic pressure. Set it to 1.0 for no such relief. (Real $\geq$ 0.0, Required)

Field	Contents
SCLF	Scale factor on force. (Real ≥ 0.0, Default = 1.0)
CLEAR	Clearance (if nonzero, no tensile force develops for positive displacements and negative forces develop only after the clearance is closed. (Real ≥ 0.0, Default = 0.0)

Remarks:

1. Corresponds to Ls-Dyna entry
*MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM - MAT070

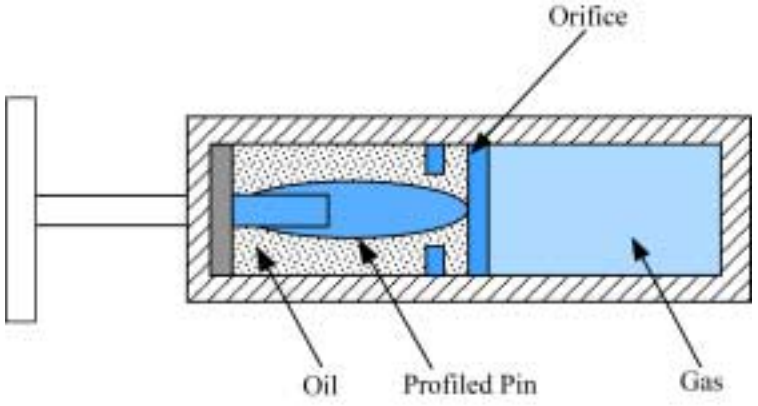


Figure 8-113 Schematic of Hydraulic/Gas damper.

2. As the damper is compressed two actions contribute to the force which develops. First, the gas is adiabatically compressed into a smaller volume. Secondly, oil is forced through an orifice. A profiled pin may occupy some of the cross-sectional area of the orifice; thus, the orifice area available for the oil varies with the stroke. The force is assumed proportional to the square of the velocity and inversely proportional to the available area.

   The equation for this element is:

$$F = SCLF \cdot \left\{ K_h \left(\frac{V}{a_0} \right)^2 + \left[P_0 \left(\frac{C_0}{C_0 - S} \right)^n - P_a \right] \cdot A_p \right\}$$

   where S is the element deflection and V is the relative velocity across the element.

MATD071 (SOL 700) Cable Discrete Beam

This model permits elastic cables to be realistically modeled; thus, no force will develop in compression.

Format:

1	2	3	4	5	6	7	8	9	10
MATD071	MID	RO	E	LCID	F0				

Example:

1	2	3	4	5	6	7	8	9	10
MATD071	21	4.5e-4	30.0E5	15	20000.				

Field	Contents
MID	Material identification. (Integer > 0, Required)
RO	Mass density. (Real > 0.0, Required)
E	> 0.0: Young's modulus < 0.0: Stiffness (Real, Required)
LCID	TABLED ID, defining the stress versus engineering strain. (Integer $\geq$ 0, Default = 0)
F0	Initial tensile force. If F0 is defined, an offset is not needed for an initial tensile force. (Real $\geq$ 0.0, Default = 0.0)

Remarks:

1. Corresponds to Ls-Dyna entry *MAT_CABLE_DISCRETE_BEAM - MAT071.
2. The force, *F*, generated by the cable is nonzero if and only if the cable is tension. The force is given by:

$$F = \max(F_0 + K\Delta l, 0.)$$

where ΔL is the change in length

$$\Delta L = \text{current length} - (\text{initial length} - \text{offset})$$

and the stiffness (E > 0.0 only) is defined as:

$$K = \frac{E \cdot area}{(initial\ length - offset)}$$

Note that a constant force element can be obtained by setting:

$$F_0 > 0 \text{ and } K = 0$$

although the application of such an element is unknown.

The area and offset are defined on either the cross section or element cards. For a slack cable the offset should be input as a negative length. For an initial tensile force the offset should be positive.

If a load curve is specified the Young's modulus will be ignored and the load curve will be used instead. The points on the load curve are defined as engineering stress versus engineering strain, i.e., the change in length over the initial length. The unloading behavior follows the loading.

MATD073 (SOL 700)

This material is for Modeling Low Density Urethane Foam with high compressibility and with rate sensitivity which can be characterized by a relaxation curve. Its main applications are for seat cushions, padding on Side Impact Dummies (SID), bumpers, and interior foams. Optionally, a tension cut-off failure can be defined. Also, see the notes below and the description of MATD057.

Format:

1	2	3	4	5	6	7	8	9	10
MATD073	MID	RO	E	LCID	TC	HU	BETA	DAMP	
	SHAPE	FAIL	BVFLAG	KCON	LCID2	BSTART	TRAMP	VN	

If LCID2 = 0, optional Format for viscoelastic constants: Up to 6 entries must be input. If LCID2 is nonzero skip this input. The variable REF is taken from the first entry of this sequence.

	GI	BETAI	REF						

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer, No Default)
RO	Mass density. (Real, No Default)
E	Young's modulus. (Real, No Default)
LCID	Load curve ID for nominal stress versus strain. (Integer, No Default)
TC	Tension cut-off stress. (Real, Default = 1.E20)
HU	Hysteretic unloading factor between 0 and 1 (Default = 1, i.e., no energy dissipation), see also **Figure 8-114**. (Real)
BETA	β, decay constant to model creep in unloading. (Real, No Default) EQ:0 No relaxation.
DAMP	Viscous coefficient (.05 < recommended value < .50) to model damping effects. (Real, No Default) LT.0.0: |DAMP| is the load cuvre ID, which defines the damping constant as a function of the maximum strain in compression defined as: $\varepsilon_{max} = \max(1 - \lambda_1, 1 - \lambda_2, 1. - \lambda_3)$

Field	Contents
	In tension, the damping constant is set to the value corresponding to the strain at 0. The abscissa should be defined from 0 to 1.
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor. Values less than one reduces the energy dissipation and greater than one increases dissipation, see also **Figure 8-114**. (Real, Default = 1.)
FAIL	Failure option after cutoff stress is reached (Real, No Default): EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.
BETAI	Optional decay constant if ith term. (Real, No Default)
BVFLAG	Bulk viscosity activation flag, see remark below. (Real, No Default): EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
KCON	Stiffness coefficient for contact interface stiffness. Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended. (Real, no default)
LCID2	Load curve ID of relaxation curve. If constants βt are determined via a least squares fit. This relaxation curve is shown in **Figure 8-114**. This model ignores the constant stress. (Integer, no default)
BSTART	Fit parameter. In the fit, $\beta 1$ is set to zero, $\beta 2$ is set to BSTART, $\beta 3$ is 10 times $\beta K2$, $\beta 4$ is 100 times greater than $\beta 3$, and so on. If zero, BSTART= .01. (Real, no default)
TRAMP	Optional ramp time for loading. (Real, No Default)
NV	Number of terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values of 2 are 3 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs. (Integer, No Default)

Field	Contents
GI	Optional shear relaxation modulus for the ith term. (Real, No Default)
NV	Number of terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values of 2 are 3 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs. (Integer, no default)
GI	Optional shear relaxation modulus for the ith term. (Real, No Default)
BETAI	Optional decay constant if ith term. (Real, No Default)

Remarks:

This viscoelastic foam model is available to model highly compressible viscous foams. The hyperelastic formulation of this model follows that of MATD057.

Rate effects are accounted for through linear viscoelasticity by a convolution integral of the form

$$\sigma_{ij}^{r} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ is the relaxation function. The stress tensor, σ_{ij}^{y}, augments the stresses determined from the foam, σ_{ij}^{j}; consequently, the final stress, σ_{ij}, is taken as the summation of the two contributions:

$$\sigma_{ij} = \sigma_{ij}^{f} + \sigma_{ij}^{y}$$

Since we wish to include only simple rate effects, the relaxation function is represented by up to six terms of the Prony series:

$$g(t) = \alpha_0 + \sum \alpha_m e^{-\beta t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. The formulation is performed in the local system of principal stretches where only the principal values of stress are computed and triaxial coupling is avoided.

Consequently, the one-dimensional nature of this foam material is unaffected by this addition of rate effects. The addition of rate effects necessitates 42 additional history variables per integration point. The cost and memory overhead of this model comes primarily from the need to “remember” the local system of principal stretches and the evaluation of the viscous stress components.

Additional Remarks:

1. When hysteretic unloading is used the reloading will follow the unloading curve if the decay constant, β, is set to zero. If β is nonzero the decay to the original loading curve is governed by the expression:

$$1 - e^{-\beta t}$$

2. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.
3. The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in **Figure 8-114**. This unloading provide energy dissipation which is reasonable in certains kinds of foam.

MATD074 (SOL 700) Elastic Spring Discrete Beam

This model permits elastic springs with damping to be combined and represented with a discrete beam element type 6. Linear stiffness and damping coefficients can be defined, and, for nonlinear behavior, a force versus deflection and force versus rate curves can be used. Displacement based failure and an initial force are optional

Format:

1	2	3	4	5	6	7	8	9	10
MATD074	MID	RO	K	F0	D	CDF	TDF		
	FLCID	HLCID	C1	C2	DLE	GLCID			

Example:

1	2	3	4	5	6	7	8	9	10
MATD074	21	23.4	2000.	0.0		0.0	5000.		
	12	14							

Field	Contents
MID	Material identification. (Integer > 0, Required)
RO	Mass density, (Real > 0.0, Required)
K	Stiffness coefficient. (Real > 0.0, Required)
F0	Optional initial force. This option is inactive if this material is referenced in a part referenced by material type MATD093. (Real $\geq$ 0.0, Default = 0.0)
D	Viscous damping coefficient. (Real $\geq$ 0.0, Default = 0.0)
CDF	Compressive displacement at failure. After failure, no forces are carried. This option does not apply to zero length springs. (Real $\geq$ 0.0, Default = 0.0)
TDF	Tensile displacement at failure. After failure, no forces are carried. (Real $\geq$ 0.0, Default = 0.0)
FLCID	TABLED ID defining force versus deflection for nonlinear behavior. (Integer > 0, Required)
HLCID	TABLED ID defining force versus relative velocity for nonlinear behavior. If the origin of the curve is at (0,0) the force magnitude is identical for a given magnitude of the relative velocity, i.e., only the sign changes. (Integer $\geq$ 0, Default = 0)

Field	Contents
C1	Damping coefficient for nonlinear behavior (Real $\geq$ 0.0, Default = 0.0)
C2	Damping coefficient for nonlinear behavior (Real $\geq$ 0.0, Default = 0.0)
DLE	Factor to scale time units. (Real $\geq$ 0.0, Default = 1.0)
GLCID	TABLED ID, defining a scale factor versus deflection for load curve ID, HLCID. (Integer $\geq$ 0, Default = 0)

Remarks:

Corresponds to Ls-Dyna entry *MAT_ELASTIC_SPRING_DISCRETE_BEAM - MAT074.

If the linear spring stiffness is used, the force, F, is given by:

$$F = F_0 + K\Delta L + D\Delta\dot{L}$$

but if the load curve ID is specified, the force is then given by:

$$F = F_0 + Kf(\Delta L)\left[1 + C1 \cdot \Delta\dot{L} + C2 \cdot \mathrm{sgn}(\Delta\dot{L})\ln\left(\max\left\{1., \frac{\Delta\dot{L}}{DLE}\right\}\right)\right] + D\Delta\dot{L} + g(\Delta L)h(\Delta\dot{L})$$

In these equations, ΔL is the change in length

$$\Delta L = \text{current length} - \text{initial length}$$

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

MATD076 (SOL 700)

This material model provides a general viscoelastic Maxwell model having up to 6 terms in the prony series expansion and is useful for modeling dense continuum rubbers and solid explosives. Either the coefficients of the prony series expansion or a relaxation curve may be specified to define the viscoelastic deviatoric and bulk behavior.

The material model can also be used with laminated shells. Either an elastic or viscoelastic layer can be defined with the laminated formulation. With the laminated option a user-defined integration rule is needed.

Format:

1	2	3	4	5	6	7	8	9	10
MATD076	MID	RO	BULK	PCF	EF				
	LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK	

Optional Format for viscoelastic constants. Up to 6 entries may be input. These cards are not needed if relaxation data is defined. The number of terms for the shear behavior may differ from that for the bulk behavior: simply insert zero if a term is not included.

If an elastic layer is defined you only need to define GI and KI (note in an elastic layer only one card is needed)

	GI	BETAI	KI	BETAKI					

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer, No Default)
RO	Mass density. (Real, No Default)
BULK	Elastic bulk modulus. (Real, No Default)
PCF	Tensile pressure elimination flag for solid elements only. If set to unity tensile pressures are set to zero. (Real, No Default)
EF	Elastic flag (if equal 1, the layer is elastic. If 0 the layer is viscoelastic). (Real, No Default)

Field	Contents
LCID	Load curve ID for deviatoric behavior if constants, Gi, and βi are determined via a least squares fit. This relaxation curve is shown below. (Integer, No Default)
NT	Number of terms in shear fit. If zero the default is 6. Fewer than NT terms will be used if the fit produces one or more negative shear moduli. Currently, the maximum number is set to 6. (Integer, no default)
BSTART	In the fit, $\beta 1$ is set to zero, $\beta 2$ is set to BSTART, $\beta 3$ is 10 times $\beta 2$, $\beta 4$ is 100 times greater than $\beta 3$, and so on. If zero, BSTART is determined by an iterative trial and error scheme. (Real, No Default)
TRAMP	Optional ramp time for loading. (Real, No Default)
LCIDK	Load curve ID for bulk behavior if constants, Ki, and βKi are determined via a least squares fit. This relaxation curve is shown below. (Integer, no default)
NTK	Number of terms desired in bulk fit. If zero the default is 6. Currently, the maximum number is set to 6. (Integer, No Default)
BSTARTK	In the fit, βKi is set to zero, $\beta K2$ is set to BSTARTK, $\beta K3$ is 10 times $\beta K2$, $\beta K4$ is 100 times greater than $\beta K3$, and so on. If zero, BSTARTK is determined by an iterative trial and error scheme. (Real, No Default)
TRAMPK	Optional ramp time for bulk loading. (Real, No Default)
GI	Optional shear relaxation modulus for the ith term. (Real, No Default)
BETAI	Optional shear decay constant for the ith term. (Real, No Default)
KI	Optional bulk relaxation modulus for the ith term. (Real, No Default)
BETAKI	Optional bulk decay constant for the ith term. (Real, No Default)

Remarks:

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl(t-\tau)}$ is the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \sum_{m=1}^{N} G_m e^{-\beta_m t}$$

We characterize this in the input by shear moduli, G_i, and decay constants, β_i. An arbitrary number of terms, up to 6, may be used when applying the viscoelastic model.

For volumetric relaxation, the relaxation function is also represented by the Prony series in terms of bulk moduli:

$$(t) = \sum_{m=1}^{N} K_m e^{-\beta_{k_m}}$$

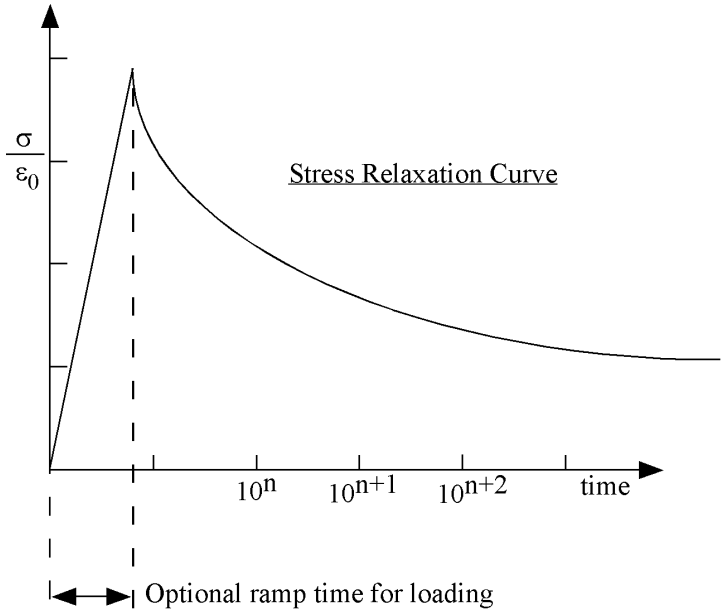


Figure 8-114 Relaxation curve. This curve defines stress versus time where time is defined on a logarithmic scale. For best results, the points defined in the load curve should be equally spaced on the logarithmic scale. Furthermore, the load curve should be smooth and defined in the positive quadrant. If nonphysical values are determined by least squares fit, LS-DYNA will terminate with an error message after the initialization phase is completed. If the ramp time for loading is included, then the relaxation which occurs during the loading phase is taken into account. This effect may or may not be important.

MATD077 (SOL 700)

LS-DYNA Material #77 -- General Christensen Rubber Model for Solid Elements

Used to model a general hyperelastic rubber model combined optionally with linear viscoelasticity as outlined by Christensen (1980). Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD077	MID	RO	PR	N	NV	G	SIGF	OPTION	

Format 2: if N > 0, a least squares fit is computed from unixial data.

	SGL	SW	ST	LCID1	DATA	LCID2	BSTART	TRAMP	

Format 2: if N = 0, define the following constants.

	C10	C01	C11	C20	C02	C30			

Optional Format for Viscoelastic Constants: Enter NV entries.

	GI	BETAI							

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
PR	Poissons ratio (>.49 is recommended, smaller values may not work and should not be used). (Real)
N	Number of constants to solve for (Integer): EQ.1 Solve for C10 and C01. EQ.2 Solve for C10, C01, C11, C20, and C02. EQ.3 Solve for C10, C01, C11, C20, C02, and C30.

Field	Contents
NV	Number of Prony series terms in fit. If zero, the default is 6. Currently, the maximum number is set to 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs. (Integer)
G	Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF. (Real)
SIGF	Limit stress for frequency independent, frictional, damping. (Real)
If N>0, test information from a uniaxial test is used :	
OPTION	OPTION must be HYPER or OGDEN. (Character)
SGL	Specimen gauge length. (Real)
SW	Specimen width. (Real)
ST	Specimen thickness. (Real)
LCID1	Load curve ID giving the force versus actual change in the gauge length. (Real)
DATA	Type of experimental data. (Real) EQ.0.0 Uniaxial data (Only option for this model). (Real)
LCID2	Load curve ID of relaxation curve. (Real) If constants β_t are determined via a least squares fit. This model ignores the constant stress.
BSTART	In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2, β_4 is 100 times greater than β_3, and so on. If zero, BSTART is determined by an iterative trial and error scheme. (Real)
TRAMP	Optional ramp time for loading. (Real)
If N=0, the following constants have to be defined:	
C10	C_{10}. (Real)

Field	Contents
C01	C_{01}. (Real)
C11	C_{11}. (Real)
C20	C_{20}. (Real)
C02	C_{02}. (Real)
C30	C_{30}. (Real)
GI	Optional shear relaxation modulus for the i-th term. (Real)
BETAI	Optional decay constant if i-th term. (Real)

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material, a hydrostatic work term, $W_H(J)$, is included in the strain energy functional that is function of the relative volume, J, (Ogden, 1984):

$$W(J_1, J_2, J) = \sum_{p,\, q\, =\, 0}^{n} C_{pq}(J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 = I_1 J^{-1/3}$$

$$J_2 = I_2 J^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invarients are modified as shown. This procedure is described in more detail by Sussman and Bathe (1987).

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t - \tau)\frac{\partial \varepsilon_{kl}}{\partial \tau} d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij} and Green's strain tensor E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t - \tau)\frac{\partial E_{kl}}{\partial \tau} d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{n} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

The Mooney-Rivlin rubber model (MATD027) is obtained by specifying *n*=1. In spite of the differences in formulations with MATD027, we find that the results obtained with this model are nearly identical with those of MATD027 as long as large values of Poisson's ratio are used.

The frequency independent damping is obtained by the having a spring and slider in series.

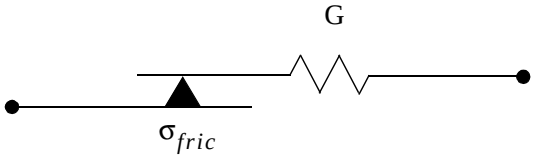


Figure 8-115

This material model may only be used with solid elements.

MATD080 (SOL 700) LS-DYNA Material #80 -- Ramberg-Osgood Plasticity

Used to model Ramberg-Osgood plasticity. This model is intended as a simple model of shear behavior and can be used in seismic analysis. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD080	MID	RO	GAMY	TAUY	ALPHA	R	BULK		

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer; Default = none)
RO	Mass density. (Real; Default = none)
GAMY	Reference shear strain (γ_y). (Real; Default = none)
TAUY	Reference shear stress (τ_y). (Real; Default = none)
ALPHA	Stress coefficient (α). (Real; Default = none)
R	Stress exponent (r). (Real; Default = none)
BULK	Elastic bulk modulus. (Real; Default = none)

Remarks:

The Ramberg-Osgood equation is an empirical constitutive relation to represent the one-dimensional elastic-plastic behavior of many materials, including soils. This model allows a simple rate independent representation of the hysteretic energy dissipation observed in soils subjected to cyclic shear deformation. For monotonic loading, the stress-strain relationship is given by:

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} + \alpha\left|\frac{\tau}{\tau_y}\right|^r \text{ if } \gamma \geq 0$$

$$y\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} - \alpha\left|\frac{\tau}{\tau_y}\right|^r \text{ if } \gamma < 0$$

where γ is the shear and τ is the stress. The model approaches perfect plasticity as the stress exponent $r \to \infty$. These equations must be augmented to correctly model unloading and reloading material behavior. The first load reversal is detected by $\gamma\dot{\gamma} < 0$. After the first reversal, the stress-strain relationship is modified to

$$\frac{(\gamma - \gamma_0)}{2\gamma_y} = \frac{(\tau - \tau_0)}{2\tau_y} + \alpha \left| \frac{(\tau - \tau_0)}{2\tau_y} \right|' \text{ if } \gamma \geq 0$$

$$\frac{(\gamma - \gamma_0)}{2\gamma_y} = \frac{(\tau - \tau_0)}{2\tau_y} - \alpha \left| \frac{(\tau - \tau_0)}{2\tau_y} \right|' \text{ if } \gamma < 0$$

where γ_0 and τ_0 represent the values of strain and stress at the point of load reversal. Subsequent load reversals are detected by $(\gamma - \gamma_0)\dot{\gamma} < 0$.

The Ramberg-Osgood equations are inherently one-dimensional and are assumed to apply to shear components. To generalize this theory to the multidimensional case, it is assumed that each component of the deviatoric stress and deviatoric tensorial strain is independently related by the one-dimensional stress-strain equations. A projection is used to map the result back into deviatoric stress space if required. The volumetric behavior is elastic, and, therefore, the pressure p is found by

$$p = -K\varepsilon_v$$

where ε_v is the volumetric strain.

MATD081 (SOL 700)

LS-DYNA Material #81 -- Elasto-Visco-Plastic with Arbitrary Stress-Strain Curve

Used to model elasto-visco-plastic materials with arbitrary stress versus strain curves and arbitrary strain rate dependency. Damage is considered before rupture occurs. Also, failure based on a plastic strain or a minimum time step size can be defined. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Options include: **BLANK, ORTHO, or RCDC**.

Including ORTHO invokes an orthotropic damage model. It is implemented only for shell elements with multiple through thickness integration points and is an extension to include orthotropic damage as a means of treating failure in aluminum panels. Directional damage begins after a defined failure strain is reached in tension and continues to evolve until a tensile rupture strain is reached in either one of the two orthogonal directions. After rupture is detected at all integration points, the element is deleted. The option RCDC invokes the damage model developed by Wilkins (Wilkins, et.al., 1977). A nonlocal formulation, which requires additional storage, is used if a characteristic length is defined.

Format:

1	2	3	4	5	6	7	8	9	10
MATD081	MID	RO	E	PR	SIGY	ETAN	EPPF	OPTION	
	TDEL	C	P	LCSS	LCSR	EPPFR	VP	LCDM	
	NUMINT	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	
	EPS8	ES1	ES2	ES3	ES4	ES5	ES6	ES7	
	EPS8								

Define the following entry if the option RCDC is active:

	ALPHA	BETA	GAMMA	D0	B	LAMBDA	DS	L	

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer, Default = none)
RO	Mass density. (Real; Default = none)
E	Young’s modulus. (Real; Default = none)
PR	Poisson’s ratio. (Real; Default = none)

Field	Contents
SIGY	Yield stress. (Real; Default = none)
ETAN	Tangent modulus, ignored if (LCSS.GT.0) is defined. (Real; Default = 0.0)
EPPF	Plastic strain, fs, at which material softening begins (logrithmic). (Real; Default = 1.0E+20)
OPTION	Blank, Ortho, or RCDC (Character)
TDEL	Minimum time step size for automatic element deletion. (Real; Default = 0)
C	Strain rate parameter, C. See equation below. (Real; Default = 0)
P	Strain rate parameter, P. See equation below. (Real; Default = 0)
LCSS	Load curve ID or Table ID. Load curve ID defining effective stress versus effective plastic strain. If defined EPS1-EPS8 and ES1-ES8 are ignored. The table ID defines for each strain rate value a load curve ID giving the stress versus effective plastic strain for that rate, see **Figure 8-116**. The stress versus effective plastic strain curve for the lowest value of strain rate is used if the strain rate falls below the minimum value. Likewise, the stress versus effective plastic strain curve for the highest value of strain rate is used if the strain rate exceeds the maximum value. The strain rate parameters: C and P. (Integer; Default = 0)
LCSR	Load curve ID defining strain rate scaling effect on yield stress. (Integer; Default = 0)
EPPFR	Plastic strain at which material ruptures (logrithmic). (Real; Default = 0)
VP	Formulation for rate effects (Integer; Default = 0): EQ.0 Scale yield stress. (Default). EQ.1 Viscoplastic formulation.
LCDM	Load curve ID defining nonlinear damage curve. (Integer; Default = 0)

Field	Contents
NUMINT	Number of through thickness integration points which must fail before the element is deleted. (If zero, all points must fail.) The default of all integration points is not recommended since elements undergoing large strain are often not deleted due to nodal fiber rotations which limit strains at active integration points after most points have failed. Better results are obtained if NUMINT is set to 1 or a number less than one half of the number of through thickness points. For example, if four through thickness points are used, NUMINT should not exceed 2, even for fully integrated shells which have 16 integration points. (Integer; Default = 0)
EPS1-EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined. (Real; Default = 0)
ES1-ES8	Corresponding yield stress values to EPS1 - EPS8. (Real; Default = 0)
ALPHA	Parameter α. For the RCDC option. (Real; Default = 0)
BETA	Parameter β. For the RCDC option. (Real; Default = 0)
GAMMA	Parameter γ. For the RCDC option. (Real; Default = 0)
D0	Parameter D_0. For the RCDC option. (Real; Default = 0)
B	Parameter b. For the RCDC option. (Real; Default = 0)
LAMBDA	Parameter λ. For the RCDC option. (Real; Default = 0)
DS	Parameter D_s. For the RCDC option. (Real; Default = 0)
L	Characteristic element length for this material. (Real; Default = 0)

Remarks:

The stress strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve similar to that shown in **Figure 8-117** is expected to be defined by (EPS1,ES1) - (EPS8,ES8); however, an effective stress versus effective plastic strain curve (LCSS) may be input instead if eight points are insufficient. The cost is roughly the same for either approach. The most general approach is to use the table definition (LCSS) discussed below.

Two options to account for strain rate effects are possible.

1. Strain rate may be accounted for using the Cowper and Symonds model which scales the yield stress with the factor

$$1 + \left(\frac{\dot{\varepsilon}}{C}\right)^{1/6}$$

where $\dot{\varepsilon}$ is the strain rate, $\dot{\varepsilon} = \sqrt{\dot{\varepsilon}_{ij}\dot{\varepsilon}_{ij}}$.

If the viscoplastic option is active, VP=1.0, and if SIGY is > 0 then the dynamic yield stress is computed from the sum of the static stress,

$$\sigma_y^s(\varepsilon_{eff}^p)$$

which is typically given by a load curve ID, and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

$$\sigma_y(\varepsilon_{eff}^p, \dot{\varepsilon}_{eff}^p) = \sigma_y^s(\varepsilon_{eff}^p) + SIGY \cdot \left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}$$

where the plastic strain rate is used. If SIGY=0, the following equation is used instead where the static stress

$$\sigma_y^s(\varepsilon_{eff}^p)$$

must be defined by a load curve:

$$\sigma_y(\varepsilon_{eff}^p, \dot{\varepsilon}_{eff}^p) = \sigma_y^s(\varepsilon_{eff}^p)\left[1 + \left(\frac{\dot{\varepsilon}_{eff}^p}{C}\right)^{1/p}\right]$$

This latter equation is always used if the viscoplastic option is off.

2. For complete generality a load curve (LCSR) to scale the yield stress may be input instead. In this curve the scale factor versus strain rate is defined.

   The constitutive properties for the damaged material are obtained from the undamaged material properties. The amount of damage evolved is represented by the constant, ω, which varies from zero if no damage has occurred to unity for complete rupture. For uniaxial loading, the nominal stress

$$\sigma_{\text{nominal}} = \frac{P}{A}$$

   where P is the applied load and A is the surface area. The true stress is given by:

$$\sigma_{\text{true}} = \frac{P}{A - A_{\text{loss}}}$$

where A_{loss} is the void area. The damage variable can then be defined:

$$\omega = \frac{A_{\text{loss}}}{A} \qquad 0 \le \omega \le 1$$

In this model damage is defined in terms of plastic strain after the failure strain is exceeded:

$$\text{,} = \frac{\varepsilon^p_{eff} - \varepsilon^p_{\text{failure}}}{\varepsilon^p_{\text{rupture}} - \varepsilon^p_{\text{failure}}} \text{ if } \varepsilon^p_{\text{failure}} \le \varepsilon^p_{eff} \le \varepsilon^p_{\text{ruptur}}$$

After exceeding the failure strain softening begins and continues until the rupture strain is reached.

The RCDC option is defined as the following:

The damage D is given by

$$D = \int \omega_1 \omega_2 d\,\varepsilon^p$$

where ε^p is the equivalent plastic strain,

$$\omega_1 = \left(\frac{1}{1 - \gamma\sigma_m}\right)^{\alpha}$$

is a triaxial stress weighting term and

$$\omega_2 = (2 - A_D)^{\beta}$$

is a asymmetric strain weighting term.

In the above, σ_m is the mean stress and

$$A_D = \left(\left|\frac{S_2}{S_3}\right|, \left|\frac{S_3}{S_2}\right|\right)$$

Fracture is initiated when the accumulation of damage is

$$\frac{D}{D_c} > 1$$

where D_c is the a critical damage given by

$$D_c = D_0(1 + B|\nabla D|^{\lambda})$$

A fracture fraction

$$F = \frac{D - D_c}{D_s}$$

defines the degradations of the material by the RCDC option.

The characteristic element length is used in the calculation of ∇D. Calculation of this factor is only done for element with smaller element length than this value.

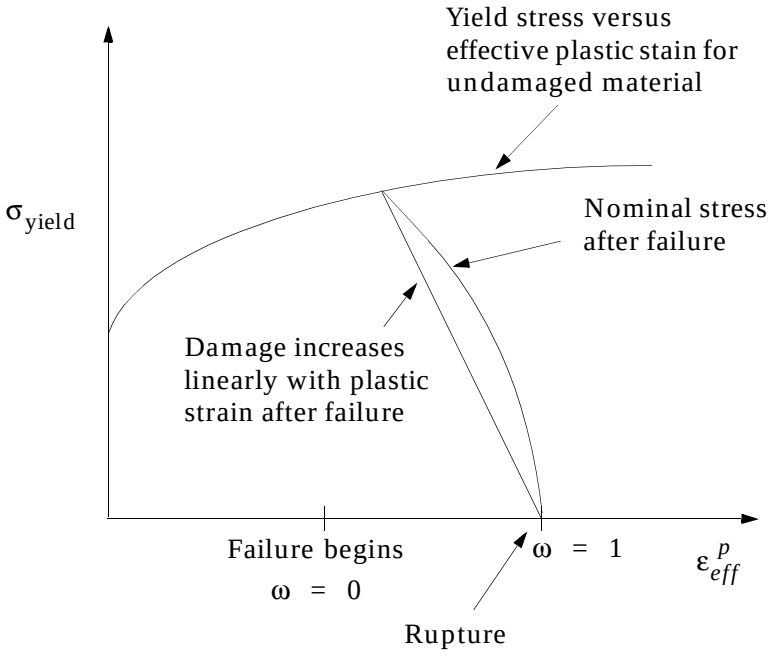


Figure 8-116 Stress Strain Behavior When Damage is Included

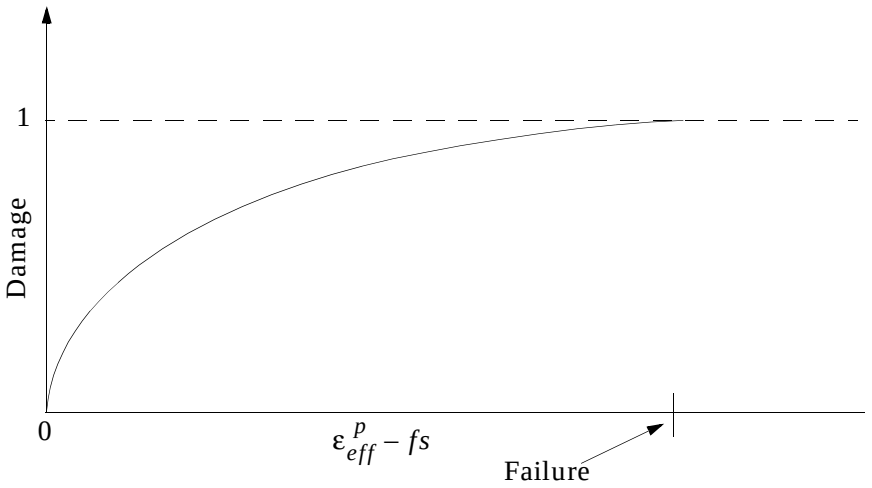


Figure 8-117 A Nonlinear Damage Curve is Optional

Note that the origin of the curve is at (0,0). It is permissible to input the failure strain, fs, as zero for this option. The nonlinear damage curve is useful for controlling the softening behavior after the failure strain is reached.

MATD083 (SOL 700) LS-DYNA Material #80

Rate effects can be modeled in low and medium density foams, see **Figure 8-118**. Hysteretic unloading behavior in this model is a function of the rate sensitivity with the most rate sensitive foams providing the largest hysteretic and visa versa. The unified constitutive equations for foam materials by Fu Chang [1995] provides the basis for this model. The mathematical description given below is excerpted from the reference. Further improvements have been incorporated based on work by Hirth, Du Bois, and Weimar [1998]. Their improvements permit: load curves generated by drop tower test to be directly input, a choice of principal or volumetric strain rates, load curves to be defined in tension, and the volumetric behavior to be specified by a load curve.

Format:

1	2	3	4	5	6	7	8	9	10
MATD083	MID	RO	E	ED	TC	FAIL	DAMP	TBID	
	BVFLAG	SFLAG	RFLAG	TFLAG	PVID	SRAF			
	D0	N0	N1	N2	N3	C0	C1	C2	
	C3	C4	C5	AIJ	SIJ	MINR	MAXR		

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer, no Default)
RO	Mass density. (Real, no Default)
E	Young's modulus. (Real, no Default)
ED	Optional Young's relaxation modulus, E_d, for rate effects. (Real, no Default) See comments below. EQ.0.0: Maximum slope in stress vs. strain curve is used. When the maximum slope is taken for the contact, the time step size for this material is reduced for stability. In some cases Δt may be significantly smaller, and defining a reasonable stiffness is recommended.
TC	Tension cut-off stress. (Real, no Default)
FAIL	Failure option after cutoff stress is reached (Real, no Default): EQ.0.0: tensile stress remains at cut-off value, EQ.1.0: tensile stress is reset to zero.

Field	Contents
DAMP	Viscous coefficient (.05 < recommended value < .50) to model damping effects. (Real, no Default)
TBID	Table ID, see TABLEDR, for nominal stress vs. strain data as a function of strain rate. If the table ID is provided, cards 3 and 4 may be left blank and the fit will be done internally. (Integer, no Default)
BVFLAG	Bulk viscosity activation flag, see Remark below (Real, no Default): EQ.0.0: no bulk viscosity (recommended), EQ.1.0: bulk viscosity active.
SFLAG	Strain rate flag (see Remark **2.** below) (Real, no Default): EQ.0.0: true constant strain rate, EQ.1.0: engineering strain rate.
RFLAG	Strain rate evaluation flag (Real, no Default): EQ.0.0: first principal direction, EQ.1.0: principal strain rates for each principal direction, EQ.2.0: volumetric strain rate.
TFLAG	Tensile stress evaluation (Real, no Default): EQ.0.0: linear in tension. EQ.1.0: input via load curves with the tensile response corresponds to negative values of stress and strain.
PVID	Optional load curve ID defining pressure versus volumetric strain. (Integer, no Default)
SRAF	Strain rate averaging flag. (Real, no Default) EQ.0.0: use weighted running average. EQ.1.0: average the last twelve values.
D0	material constant, see equations below. (Real, no Default)
N0	material constant, see equations below. (Real, no Default)
N1	material constant, see equations below. (Real, no Default)
N2	material constant, see equations below. (Real, no Default)
N3	material constant, see equations below. (Real, no Default)
C0	material constant, see equations below. (Real, no Default)
C1	material constant, see equations below. (Real, no Default)

Field	Contents
C2	material constant, see equations below. (Real, no Default)
C3	material constant, see equations below. (Real, no Default)
C4	material constant, see equations below. (Real, no Default)
C5	material constant, see equations below. (Real, no Default)
AIJ	material constant, see equations below. (Real, no Default)
SIJ	material constant, see equations below. (Real, no Default)
MINR	Ratemin, minimum strain rate of interest. (Real, no Default)
MAXR	Ratemax, maximum strain rate of interest. (Real, no Default)

Remarks:

The strain is divided into two parts: a linear part and a non-linear part of the strain

$$E(t) = E^L(t) + E^N(t)$$

and the strain rate become

$$\dot{E}(t) = \dot{E}^L(t) + \dot{E}^N(t)$$

$\dot{E}^N$ is an expression for the past history of E^N. A postulated constitutive equation may be written as:

$$\sigma(t) = \int_{\tau=0}^{\infty} [E_t^N(\tau), S(t)]d\tau$$

where $S(t)$ is the state variable and $\int_{\tau=0}^{\infty}$ is a functional of all values of τ in $T_\tau: 0 \le \tau \le \infty$ and

$$E_t^N(\tau) = E^N(t - \tau)$$

where τ is the history parameter:

$$E_t^N(\tau = \infty) \Leftrightarrow \text{the virgin material}$$

It is assumed that the material remembers only its immediate past, i.e., a neighborhood about $\tau = 0$. Therefore, an expansion of $E_t^N(\tau)$ in a Taylor series about $\tau = 0$ yields:

$$E_t^N(\tau) = E^N(0) + \frac{\partial E_t^N}{\partial t}(0)dt$$

Hence, the postulated constitutive equation becomes:

$$\sigma(t) = \sigma^*(E^N(t), \bar{E}^N(t), S(t))$$

where we have replaced $\partial E_t^N/(\partial t)$ by $\bar{E}^N$, and σ^* is a function of its arguments.

For a special case,

$$\sigma(t) = \sigma^*(E^N(t), S(t))$$

we may write

$$\bar{E}_t^N = f(S(t), s(t))$$

which states that the nonlinear strain rate is the function of stress and a state variable which represents the history of loading. Therefore, the proposed kinetic equation for foam materials is:

$$\bar{E}_t^N = D_0 \exp\left[-c_0\left(\frac{tr(\sigma S)}{(\|\sigma\|^2)}\right)^{2n_0}\right]$$

where D0, c0, and n0 are material constants, and S is the overall state variable. If either $D_0 = 0$ or $c_0 \to \infty$ then the nonlinear strain rate vanishes.

$$\dot{S}_{ij} = \left[c_1(a_{ij}R - c_2 S_{ij})P + c_3 W^{n_2} I_{ij}\right]R$$

$$R = 1 + c_4\left(\frac{\dot{E}^N}{c_s} - 1\right)^{n_3}$$

$$P = tr(\sigma \dot{E}^N)$$

$$W = \int tr(\sigma(dE))$$

where c_1, c_2, c_3, c_4, c_5, n_1, n_2, n_3, and aija_{ij} are material constants and:

$$\|\sigma\| = (\sigma_{ij}\sigma_{ij})^{\frac{1}{2}}$$

$$\|\dot{E}\| = (\dot{E}_{ij}\dot{E}_{ij})^{\frac{1}{2}}$$

$$\|\dot{E}^N\| = (\dot{E}^N_{ij}\dot{E}^N_{ij})^{\frac{1}{2}}$$

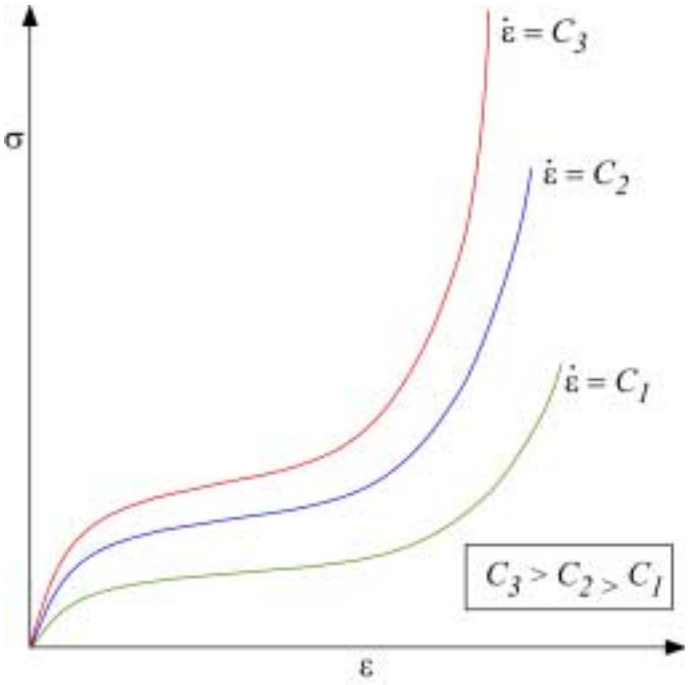


Figure 8-118 Nominal stress versus engineering strain curves, which are used to model rate effects in Fu Chang's foam model.

Remarks:

1. The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and consequently, it is optional with this model.
2. Dynamic compression tests at the strain rates of interest in vehicle crash are usually performed with a drop tower. In this test the loading velocity is nearly constant but the true strain rate, which depends on the instantaneous specimen thickness, is not. Therefore, the engineering strain rate input is optional so that the stress strain curves obtained at constant velocity loading can be used directly.

3. To further improve the response under multiaxial loading, the strain rate parameter can either be based on the principal strain rates or the volumetric strain rate.
4. Correlation under triaxial loading is achieved by directly inputting the results of hydrostatic testing in addition to the uniaxial data. Without this additional information which is fully optional, triaxial response tends to be underestimated.

MATD087 (SOL 700) Cellular Rubber

This material model provides a cellular rubber model with confined air pressure combined with linear viscoelasticity as outlined by Christensen [1980]. See **Figure 8-119**.

Format:

1	2	3	4	5	6	7	8	9	10
MATD087	MID	RO	PR	N					
	SGL	SW	ST	LCID					
	C10	C01	C11	C20	C02				
	P0	PHI	IVS	G	BETA				

Example:

MATD087	21	3.4e3	0.495	2					
	.5	.1	.02	12					

Field	Contents
MID	Material ID. MID must be unique. (Integer, Required)
RO	Mass density (Real, Required)
PR	Poisson's ratio, typical values are between .0 to .2. Due to the large compressibility of air, large values of Poisson's ratio generates physically meaningless results. (Real > 0.0, Required)
N	Order of fit (currently < 3). If $n > 0$ then a least square fit is computed with uniaxial data. The parameters given on card 2 should be specified. Also see MATD027. A Poisson's ratio of .5 is assumed for the void free rubber during the fit. The Poisson's ratio defined on Card 1 is for the cellular rubber. A void fraction formulation is used (Integer, Required)
Define, if N > 0:	
SGL	Specimen gauge length l_0 (R > 0, Default = 0.0)
SW	Specimen width. (R >0, Default = 0.0)
ST	Specimen thickness. (R > 0, Default = 0.0)
LCID	Load curve ID giving the force versus actual change ΔL in the gauge length. (I > 0, Default = 0)

Field	Contents
Define, if N = 0:	
C10	Coefficient, C_{10}. (R > 0, Default = 0.0)
C01	Coefficient, C_{01}. (R > 0, Default = 0.0)
C11	Coefficient, C_{11}. (R > 0, Default = 0.0)
C20	Coefficient, C_{20}. (R > 0, Default = 0.0)
C02	Coefficient, C_{02}. (R > 0, Default = 0.0)
P0	Initial air pressure, P_0. (R > 0, Default = 0.0)
PHI	Ratio of cellular rubber to rubber density, Φ. (R > 0, Default = 0.0)
IVS	Initial volumetric strain, γ_0. (R > 0, Default = 0.0)
G	Optional shear relaxation modulus, G, for rate effects (viscosity). (R > 0, Default = 0.0)
BETA	Optional decay constant, β_1. (R>0, Default = 0.0)

LSDYNA Input:

*MAT_CELLULAR_RUBBER

Remarks:

Corresponds to Ls-Dyna input *MAT_CELLULAR_RUBBER.

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_H(J)$, is included in the strain energy functional which is function of the relative volume, J, [Ogden, 1984]:

$$W(J_1, J_2, J) = \sum_{p,\, q\, =\, 0}^{n} C_{pq}(J_1 - 3)^p (J_2 - 3)^q + W_H(J)$$

$$J_1 + I_1 I_3^{-1/3}$$

$$J_2 + I_2 I_3^{-2/3}$$

In order to prevent volumetric work from contributing to the hydrostatic work the first and second invariants are modified as shown. This procedure is described in more detail by Sussman and Bathe [1987].

The effects of confined air pressure in its overall response characteristics is included by augmenting the stress state within the element by the air pressure.

$$\sigma_{ij} = \sigma_{ij}^{sk} - \delta_{ij}\sigma^{air}$$

where σ_{ij}^{sk} is the bulk skeletal stress and σ^{air} is the air pressure computed from the equation:

$$\sigma^{air} = -\frac{p_0\gamma}{1+\gamma-\phi}$$

where p_0 is the initial foam pressure usually taken as the atmospheric pressure and γ defines the volumetric strain

$$\gamma = V - 1 + \gamma_0$$

where V is the relative volume of the voids and γ_0 is the initial volumetric strain which is typically zero. The rubber skeletal material is assumed to be incompressible.

Rate effects are taken into account through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij}, and Green's strain tensor, E_{ij},

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

Since we wish to include only simple rate effects, the relaxation function is represented by one term from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = E_d e^{-\beta_1 t}$$

This model is effectively a Maxwell fluid which consists of a damper and spring in series. We characterize this in the input by a shear modulus, *G*, and decay constant, β_1.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying *n*=1 without air pressure and viscosity. In spite of the differences in formulations with Model 27, we find that the results obtained with this model are nearly identical with those of material type 27 as long as large values of Poisson's ratio are used.

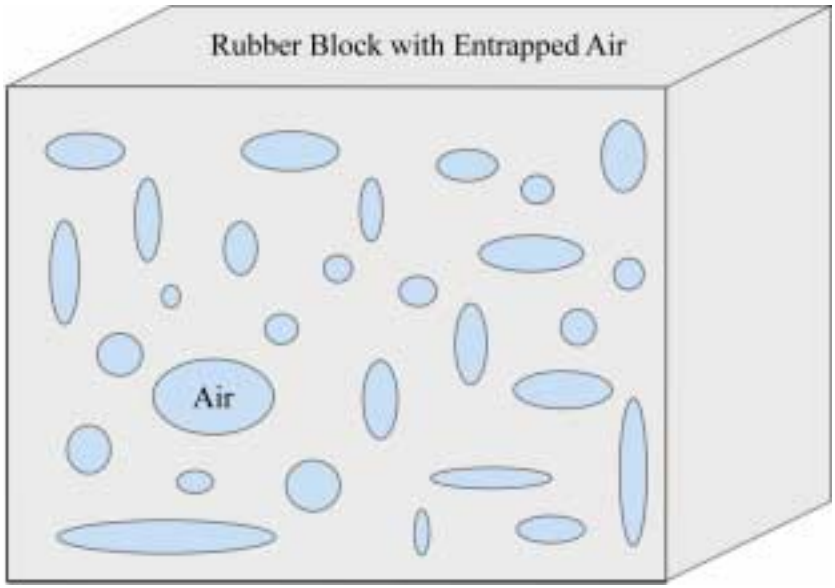


Figure 8-119 Cellular rubber with entrapped air. By setting the initial air pressure to zero, an open cell, cellular rubber can be simulated.

MATD093 (SOL 700) Elastic 6DOF Spring Discrete Beam

This material model is defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The input consists of part ID's that reference material type MATD074. Generally, these referenced parts are used only for the definition of this material model and are not referenced by any elements. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the PBDISCR input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad is used to orient the beam for the directional springs.

Format:

1	2	3	4	5	6	7	8	9	10
MATD093	MID	RO	TPIDR	TPIDS	TPIDT	RPIDR	RPIDS	RPIDT	

Example:

MATD093	21	232.4	30‘	302	303		1		

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer > 0, Required)
RO	Mass density, (Real > 0.0, Required)
TPIDR	Translational motion in the local r-direction is governed by property ID TPIDR. If zero, no force is computed in this direction. (Integer $\geq$ 0, Default = 0)
TPIDS	Translational motion in the local s-direction is governed by property ID TPIDS. If zero, no force is computed in this direction. (Integer $\geq$ 0, Default = 0)
TPIDT	Translational motion in the local t-direction is governed by property ID TPIDT. If zero, no force is computed in this direction. (Integer $\geq$ 0, Default = 0)
RPIDR	Rotational motion about the local r-axis is governed by property ID RPIDR. If zero, no moment is computed about this axis. (Integer $\geq$ 0, Default = 0)

Field	Contents
RPIDS	Rotational motion about the local s-axis is governed by property ID RPIDS. If zero, no moment is computed about this axis. (Integer $\geq$ 0, Default = 0)
RPIDT	Rotational motion about the local t-axis is governed by property ID RPIDT. If zero, no moment is computed about this axis. (Integer $\geq$ 0, Default = 0)

Remark:

Corresponds to Ls-Dyna entry *MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM - MAT093.

MATD094 (SOL 700) Inelastic Spring Discrete Beam

This model permits elastoplastic springs with damping to be represented with a discrete beam element type 6. A yield force versus deflection curve is used which can vary in tension and compression.

Format:

1	2	3	4	5	6	7	8	9	10
MATD094	MID	RO	K	F0	D	CDF	TDF		
	FLCID	HLCID	C1	C2	DLE	GLCID			

Example:

MATD094	21	23.4	2000.	0.0		0.0	5000.		
	12	14							

Field	Contents
MID	Material identification. (Integer > 0, Required)
RO	Mass density. (Real > 0.0, Required)
K	Elastic loading / unloading stiffness. (Real > 0.0, Required)
F0	Optional initial force. This option is inactive if this material is referenced in a part referenced by material type MATD095. (Real $\geq$ 0.0, Default = 0.0)
D	Optional viscous damping coefficient. (Real $\geq$ 0.0, Default = 0.0)
CDF	Compressive displacement at failure. Input as a positive number. After failure, no forces are carried. This option does not apply to zero length springs. (Real $\geq$ 0.0, Default = 0.0)
TDF	Tensile displacement at failure. After failure, no forces are carried. (Real $\geq$ 0.0, Default = 0.0)
FLCID	TABLED ID defining the yield force versus plastic deflection. If the origin of the curve is at (0,0) the force magnitude is identical in tension and compression, i.e., only the sign changes. If not, the yield stress in the compression is used when the spring force is negative. The plastic displacement increases monotonically in this implementation. The load curve is required input. (Integer > 0, Required)

Field	Contents
HLCID	TABLED ID defining force versus relative velocity. If the origin of the curve is at (0,0) the force magnitude is identical for a given magnitude of the relative velocity, i.e., only the sign changes. (Integer > 0, Default = 0)
C1	Damping coefficient. (Real $\geq$ 0.0, Default = 0.0)
C2	Damping coefficient. (Real $\geq$ 0.0, Default = 0.0)
DLE	Factor to scale time units. (Real $\geq$ 0.0, Default = 1.0)
GLCID	TABLED ID defining a scale factor versus deflection for load curve ID, HLCID. (Integer > 0, Default = 0)

Remarks:

1. Corresponds to Ls-Dyna entry *MAT_INELASTIC_SPRING_DISCRETE_BEAM - MAT094.
2. The yield force is taken from the load curve:

$$F^Y = F_y(\Delta L^{plastic})$$

where L^{small} is the plastic deflection. A trial force is computed as:

$$F^T = F^n + K\Delta\dot{L}(\Delta t)$$

and is checked against the yield force to determine:

$$F = \begin{cases} F^Y \text{ if } F^T > F^Y \\ F^T \text{ if } F^T \leq F^Y \end{cases}$$

The final force, which includes rate effects and damping, is given by:

$$F^{n+1} = F \cdot \left[1 + C1 \cdot \Delta\dot{L} + C2 \cdot \text{sgn}(\Delta\dot{L}) \ln\left(max\left\{1., \frac{|\Delta\dot{L}|}{DLE}\right\}\right)\right] + D\Delta\dot{L} + g(\Delta L)h(\Delta\dot{L})$$

Unless the origin of the curve starts at (0,0), the negative part of the curve is used when the spring force is negative where the negative of the plastic displacement is used to interpolate, F_y. The positive part of the curve is used whenever the force is positive. In these equations ΔL, is the change in length

$$\Delta L = \text{current length} - \text{initial length}$$

The cross sectional area is defined on the section card for the discrete beam elements, See *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

MATD095 (SOL 700) Inelastic 6DOF Spring Discrete Beam

This material model is defined for simulating the effects of nonlinear inelastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees-of-freedom. The input consists of part ID's that reference material type, MAT094. Generally, these referenced parts are used only for the definition of this material model and are not referenced by any elements. The two nodes defining a beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the PBDISCR input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this material model. A triad must be used to orient the beam for zero length beams.

Format:

1	2	3	4	5	6	7	8	9	10
MATD095	MID	RO	TPIDR	TPIDS	TPIDT	RPIDR	RPIDS	RPIDT	

Example:

MATD095	21	23.4	301	302	303		1	0	

Field	Contents
MID	Material identification. (Integer > 0, Required)
RO	Mass density, (Real > 0.0, Required)
TPIDR	Translational motion in the local r-direction is governed by Property ID TPIDR. If zero, no force is computed in this direction. (Integer ≥ 0, Default = 0)
TPIDS	Translational motion in the local s-direction is governed by Property ID TPIDS. If zero, no force is computed in this direction. (Integer ≥ 0, Default = 0)
TPIDT	Translational motion in the local t-direction is governed by Property ID TPIDT. If zero, no force is computed in this direction. (Integer ≥ 0, Default = 0)
RPIDR	Rotational motion about the local r-axis is governed by Property ID RPIDR. If zero, no moment is computed about this axis. (Integer ≥ 0, Default = 0)

Field	Contents
RPIDS	Rotational motion about the local s-axis is governed by Property ID RPIDS. If zero, no moment is computed about this axis. (Integer $\geq$ 0, Default = 0)
RPIDT	Rotational motion about the local t-axis is governed by Property ID RPIDT. If zero, no moment is computed about this axis. (Integer $\geq$ 0, Default = 0)

Remark:

Corresponds to Ls-Dyna Input
*MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM - MAT095.

MATD097 (SOL 700) General Joint Discrete Beam

This model is used to define a general joint constraining any combination of degrees of freedom between two nodes. The nodes may belong to rigid or deformable bodies. In most applications the end nodes of the beam are coincident and the local coordinate system (r,s,t axes) is defined by CID (see PBDISCR).

Format:

1	2	3	4	5	6	7	8	9	10
MATD097	MID	RO	TR	TS	TT	RR	RS	RT	
	RPST	RPSR	C1	C2	DLE	GLCID			

Format:

MATD097	21	232.4			1		1		
	10000.	1000.							

Field	Contents
MID	Material identification. (Integer > 0, Required)
RO	Mass density. (Real > 0.0, Required)
TR	Translational constraint code along the r-axis (0 ≥ free, 1 ≥ constrained) (Integer ≥ 0, Default = 0)
TS	Translational constraint code along the s-axis (0 ≥ free, 1 ≥ constrained) (Integer ≥ 0, Default = 0)
TT	Translational constraint code along the y-axis (0 ≥ free, 1 ≥ constrained) (Integer ≥ 0, Default = 0)
RR	Rotational constraint code along the r-axis (0 ≥ free, 1 ≥ constrained) (Integer ≥ 0, Default = 0)
RS	Rotational constraint code along the r-axis (0 ≥ free, 1 ≥ constrained) (Integer ≥ 0, Default = 0)
RT	Rotational constraint code about the t-axis (0 ≥ free, 1 ≥ constrained) (Integer ≥ 0, Default = 0)
RPST	Penalty stiffness scale factor for translational constraints. (Real > 0.0, Required)
RPSR	Penalty stiffness scale factor for rotational constraints. (Real > 0.0, Required)

Remarks:

1. Corresponds to Ls-Dyna entry *MAT_GENERAL_JOINT_DISCRETE_BEAM - MAT097.
2. For explicit calculations, the additional stiffness due to this joint may require addition mass and inertia for stability. Mass and rotary inertia for this beam element is based on the defined mass density, the volume, and the mass moment of inertia defined in the PBDISCR input.

MATD100 (SOL 700) LS-DYNA Material #100 -- Material for Spot Weld

The material model applies to beam elements to solid elements with hourglass control. The failure models apply to both beam and solid elements.

The beam elements, based on the Hughes-Liu beam formulation, may be placed between any two deformable shell surfaces and tied with constraint contact which eliminates the need to have adjacent nodes at spot weld locations. Beam spot welds may be placed between rigid bodies and rigid / deformable bodies by making the node on one end of the spot weld a rigid body node which can be an extra node for the rigid body. In the same way rigid bodies may also be tied together with this spot weld option. This weld option should not be used with rigid body switching.

In flat topologies the shell elements have an unconstrained drilling degree-of-freedom which prevents torsional forces from being transmitted. If the torsional forces are deemed to be important, brick elements should be used to model the spot welds.

Beam and solid element force resultants are written to the spot weld force file, SWFORC, and the file for element stresses and resultants for designated elements in file ELOUT.

Note: It is advisable to include all spot welds, which provide the slave nodes, and spot welded materials, which define the master segments, within a single interface for beam element spot welds or a CONTACT interface for solid element spot welds.

As a constraint method these interfaces are treated independently which can lead to significant problems if such interfaces share common nodal points. An added benefit is that memory usage can be substantially less with a single interface.

Options include: **BLANK and DAMAGE**

The DAMAGE option also includes failure and causes one additional line to be read with the damage parameter and a flag that determines how failure is computed from the resultants. On this line the parameter, RS, if nonzero, invokes damage mechanics combined with the plasticity model to achieve a smooth drop off of the resultant forces prior to the removal of the spotweld. The parameter OPT determines the method used in computing resultant based failure, which is unrelated to damage. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD100	MID	RO	E	PR	SIGY	ET	DT	OPTION	

Entry 2 and 3 when the DAMAGE-FAILURE option is inactive. Entry 4 is not defined.

	TFAIL	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	
	NF								

Entry 2 and 3 for OPT=0.0, Resultant Based Failure:

	TFAIL	EFAIL	NRR	NRS	NRT	MRR	MSS	MTT	
	NF								

Entry 2 and 3 for OPT=1.0, stress based failure if strain rate effects are included.

	TFAIL	EFAIL	-LCAX	-LCTAU					
	NF								

Entry 2 and 3 for OPT=3.0, 4.0, and 5.0.

	TFAIL	EFAIL	ZD	ZT	ZALP1	ZALP2	ZALP3	ZRRAD	
	NF								

Entry 4 is defined only for the DAMAGE option.

	RS	OPT	FVAL						

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
E	Young's modulus. (Real)
PR	Poisson's ratio. (Real)
SIGY	Initial yield stress. (Real)
ET	Hardening modulus, E_t. (Real)

Field	Contents
DT	Time step size for mass scaling, Δt. (Real)
OPTION	Damage option. Allowable items are blank or DAMAGE. (Character)
TFAIL	Failure time if nonzero. If zero this option is ignored. (Real)
EFAIL	Effective plastic strain in weld material at failure. If the damage option is inactive, the spot weld element is deleted when the plastic strain at each integration point exceeds EFAIL. If the damage option is active, the plastic strain must exceed the rupture strain at each integration point before deletion occurs. (Real)
NRR	Axial force resultant N_{rr_F} or maximum axial stress σ_{rr}^F at failure depending on the value of OPT (see below). If zero, failure due to this component is not considered. If negative, |NRR| is the load curve ID defining the maximum axial stress at failure as a function of the effective strain rate. (Real)
NRS	Force resultant N_{rt_F} or maximum shear stress τ^F at failure depending on the value of OPT (see below). If zero, failure due to this component is not considered. If negative, |NRS| is the load curve ID defining the maximum shear stress at failure as a function of the effective strain rate. (Real)
NRT	Force resultant N_{rs_F} at failure. If zero, failure due to this component is not considered. (Real)

Field	Contents
MRR	Torsional moment resultant M_{rr_F} at failure. If zero, failure due to this component is not considered. (Real)
MSS	Moment resultant M_{ss_F} at failure. If zero, failure due to this component is not considered. (Real)
MTT	Moment resultant M_{tt_F} at failure. If zero, failure due to this component is not considered. (Real)
NF	Number of force vectors stored for filtering. The default value is set to zero which is generally recommended unless oscillatory resultant forces are observed in the time history databases. Even though these welds should not oscillate significantly, this option was added for consistency with the other spot weld options. NF affects the storage since it is necessary to store the resultant forces as history variables. When NF is nonzero, the resultants in the output databases are filtered. (Real)
SIGAX	Maximum axial stress σ_{rr}^F at failure. If zero, failure due to this component is not considered.
SIGTAU	Maximum shear stress τ^F at failure. If zero, failure due to this component is not considered.
-LCAX	Load curve ID defining the maximum axial stress at failure as a function of the effective strain rate. Input as a negative number. (Real)
-LCTAU	Load curve ID defining the maximum shear stress at failure as a function of the effective strain rate. Input as a negative number.
USERVn	Failure constants for user failure subroutine, n=1,2,...6.
ZD	Notch diameter. (Real)

Field	Contents
ZT	Sheet thickness. (Real)
ZALP1	Correction factor alpha1. (Real)
ZALP2	Correction factor alpha2. (Real)
ZALP3	Correction factor alpha3. (Real)
ZRRAD	Notch root radius (OPT=3.0 only) (Real)
RS	Rupture strain. Define if and only if damage is active. (Real)
OPT	Failure option (Integer):
	EQ.0 Resultant based failure.
	EQ.1 Stress based failure computed from resultants (Toyota).
	EQ.3 Notch stress based failure (beam weld only).
	EQ.4 Stress intensity factor at failure (beam weld only).
	EQ.5 Structural stress at failure (beam weld only).
	EQ.6 Stress based failure computed from resultants (Toyota). In this option a shell strain rate dependent failure model is used (beam weld only).
	EQ.7 Resultant based failure including rate effects for solid elements based on the transverse and normal stress resultants and part ID pairs.
FVAL	Failure parameter. (Real) If OPT:
	EQ.0 Not used.
	EQ.1 Not used.
	EQ.2 Not used.
	EQ.3 Notch stress value at failure (σ_{KF}).
	EQ.4 Stress intensity factor value at failure (K_{eqF}).
	EQ.5 Structural stress value at failure (σ_{sF}).
	EQ.6 Rupture filter number.

Remarks:

The weld material is modeled with isotropic hardening plasticity coupled to four failure models. The first model specifies a failure strain which fails each integration point in the spot weld independently. The second model fails the entire weld if the resultants are outside of the failure surface defined by:

$$\left(\frac{\max(N_{rr},0)}{N_{rr_F}}\right)^2+\left(\frac{N_{rs}}{N_{rs_F}}\right)^2+\left(\frac{N_{rt}}{N_{rt_F}}\right)^2+\left(\frac{M_{rr}}{M_{rr_F}}\right)^2+\left(\frac{M_{ss}}{M_{ss_F}}\right)^2+\left(\frac{M_{tt}}{M_{tt_F}}\right)^2-1=0$$

where the *numerators* in the equation are the resultants calculated in the local coordinates of the cross section, and the **denominators** are the values specified in the input. If NF is nonzero the resultants are filtered before failure is checked. The stress based failure model, which was developed by Toyota Motor Corporation and is based on the peak axial and transverse shear stresses, fails the entire weld if the stresses are outside of the failure surface defined by

$$\left(\frac{\sigma_{rr}}{\sigma_{rr}^F}\right)^2+\left(\frac{\tau}{\tau^F}\right)^2-1=0$$

If strain rates are considered then the failure criteria becomes:

$$\left(\frac{\sigma_{rr}}{\sigma_{rr}^F(\dot{\varepsilon}_{eff})}\right)^2+\left(\frac{\tau}{\tau^F(\dot{\varepsilon}_{eff})}\right)^2-1=0$$

where

$$\sigma_{rr}^F(\dot{\varepsilon}_{eff})$$

and

$$\tau^F(\dot{\varepsilon}_{eff})$$

are defined by load curves. The peak stresses are calculated from the resultants using simple beam theory.

$$\sigma_{rr}=\frac{N_{rr}}{A}+\frac{\sqrt{M_{rs}^2+M_{rt}^2}}{Z}\qquad \tau=\frac{M_{rr}}{2Z}+\frac{\sqrt{N_{rs}^2+N_{rt}^2}}{A}$$

where the area and section modulus are given by:

$$A = \pi \frac{d^2}{r}$$

$$Z = \pi \frac{d^3}{32}$$

and d is either the diameter of the spotweld beam or the square root of the area divided by pi of the face of a solid element used as a spotweld.

The failure based on notch stress (see Zhang, 1999) occurs when the failure criterion:

$$\sigma_k - \sigma_{kF} \geq 0$$

is satisfied. The notch stress is give by the equation:

$$\sigma_k = \alpha_1 \frac{4F}{\pi dt}\left(1 + \frac{\sqrt{3}+\sqrt{19}}{8\sqrt{\pi}}\sqrt{\frac{t}{\rho}}\right) + \alpha_2 \frac{6M}{\pi dt^2}\left(1 + \frac{2}{\sqrt{3\pi}}\sqrt{\frac{t}{\rho}}\right) + \alpha_3 \frac{4F_{rr}}{\pi d^2}\left(1 + \frac{5}{3\sqrt{2\pi t}}\sqrt{\frac{t}{\rho}}\right)$$

Here,

$$F = \sqrt{F_{rs}^2 + F_{rt}^2}$$

$$M = \sqrt{M_{ss}^2 + M_{tt}^2}$$

and α_i $i = 1, 2, 3$ are input corrections factors with default values of unity. If spot welds are between sheets of unequal thickness, the minimum thickness of the spot welded sheets may be introduced as a crude approximation.

The failure based on structural stress intensity occurs (see Zhang, 1999) when the failure criterion:

$$K_{eq} - K_{eqF} \geq 0$$

is satisfied where

$$K_{eq} - \sqrt{K_I^2 + K_{II}^2}$$

and

$$K_I = \alpha_1 \frac{\sqrt{3F}}{2\pi d\sqrt{t}} + \alpha_2 \frac{2\sqrt{3M}}{\pi d t\sqrt{t}} + \alpha_3 \frac{5\sqrt{2F_{rr}}}{3\pi d\sqrt{t}}$$

$$K_{II} = \alpha_1 \frac{2F}{\pi d\sqrt{t}}$$

Here, F and M are as defined above for the notch stress formulas and again, α_i $i = 1, 2, 3$ are input corrections factors with default values of unity. If spotwelds are between sheets of unequal thickness, the minimum thickness of the spot welded sheets may be used as a crude approximation.

The maximum structural stress at the spot weld was utilized successfully for predicting the fatigue failure of spotwelds; see Rupp, et. al., 1994, and Sheppard, 1993. The corresponding results according to Rupp, et. al. are listed below where it is assumed that they may be suitable for crash conditions.

The failure criterion is given by:

$$\max(\sigma_{v1}, \sigma_{v2}, \sigma_{v3}) - \sigma_{sF} = 0$$

where σ_{sF} is the critical value of structural stress at failure. It is noted that the forces and moments in the equations below are referred to the beam nodes 1, 2, and to the midpoint, respectively. The three stress values, $\sigma_{v1}, \sigma_{v2}, \sigma_{v3}$, are defined by:

$$\sigma_{v1}(\zeta) = \frac{F_{rs1}}{\pi d t_1}\cos\zeta + \frac{F_{rt1}}{\pi d t_1}\sin\zeta - \frac{1.046\beta_1 F_{rr1}}{t_1\sqrt{t_1}} - \frac{1.123 M_{ss1}}{d t_1\sqrt{t_1}}\sin\zeta + \frac{1.123 M_{tt1}}{d t_1\sqrt{t_1}}\cos\zeta$$

with

$$\beta_1 = 0 \text{ if } F_{rr1} \le 0$$

$$\beta_1 = 1 \text{ if } F_{rr1} > 0$$

$$\sigma_{v2}(\zeta) = \frac{F_{rs2}}{\pi d t_2}\cos\zeta + \frac{F_{rt2}}{\pi d t_2}\sin\zeta - \frac{1.046\beta_1 F_{rr2}}{t_2\sqrt{t_2}} - \frac{1.123 M_{ss2}}{d t_2\sqrt{t_2}}\sin\zeta + \frac{1.123 M_{tt2}}{d t_2\sqrt{t_2}}\cos\zeta$$

with

$$\beta_2 = 0 \text{ if } F_{rr2} \le 0$$

$$\beta_2 = 1 \text{ if } F_{rr2} > 0$$

$$\sigma_{v3}(\zeta) = 0.5\sigma(\zeta) + 0.5\sigma(\zeta)\cos(2\alpha) + 0.5\tau(\zeta)\sin(2\alpha)$$

where

$$\sigma(\zeta) = \frac{4\beta_3 F_{rr}}{\pi d^2} + \frac{3M_{ss}}{\pi d_3}\sin\zeta - \frac{32M_{tt}}{\pi d_3}\cos\zeta$$

$$\tau(\zeta) = \frac{16F_{rs}}{3\pi d^2}\sin^2\zeta + \frac{16F_{rt}}{3\pi d^2}\cos^2\zeta \qquad \text{with} \quad \beta_3 = 0 \text{ if } F_{rr} \le 0, \quad \beta_3 = 1 \text{ if } F_{rr} > 0$$

$$\alpha = \frac{1}{2}\tan^{-1}\frac{2\tau(\zeta)}{\sigma(\zeta)}$$

The stresses are calculated for all directions, $0° \le \zeta \le 90°$, in order to find the maximum.

If the failure strain is set to zero, the failure strain model is not used. In a similar manner, when the value of a resultant at failure is set to zero, the corresponding term in the failure surface is ignored. For example, if only

$$N_{rr_F}$$

is nonzero, the failure surface is reduced to

$$|N_{rr}| = N_{rr_F}$$

None, either, or both of the failure models may be active depending on the specified input values.

The inertias of the spot welds are scaled during the first time step so that their stable time step size is Δt. A strong compressive load on the spot weld at a later time may reduce the length of the spot weld so that stable time step size drops below Δt. If the value of Δt is zero, mass scaling is not performed, and the spotwelds will probably limit the time step size. Under most circumstances, the inertias of the spot welds are small enough that scaling them will have a negligible effect on the structural response and the use of this option is encouraged.

Spotweld force history data is written into the SWFORC ascii file. In this database the resultant moments are not available, but they are in the binary time history database and in the ASCII elout file.

The constitutive properties for the damaged material are obtained from the undamaged material properties. The amount of damage evolved is represented by the constant, ω, which varies from zero if no damage has occurred to unity for complete rupture. For uniaxial loading, the nominal stress in the damaged material is given by

$$\sigma_{nominal} = \frac{P}{A}$$

where P is the applied load and A is the surface area. The true stress is given by:

$$\sigma_{true} = \frac{P}{A - A_{loss}}$$

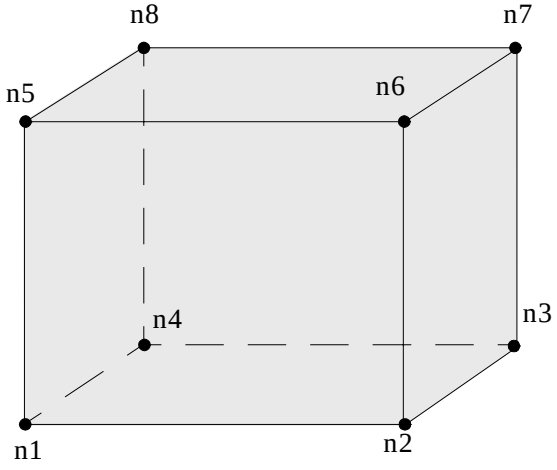
where A_{loss} is the void area. The damage variable can then be defined:

$$\omega = \frac{A_{loss}}{A} \qquad 0 \le \omega \le 1$$

In this model damage is defined in terms of plastic strain after the failure strain is exceeded:

$$\omega = \frac{\varepsilon^p_{eff} - \varepsilon^p_{failure}}{\varepsilon^p_{rupture} - \varepsilon^p_{failure}} \text{ if } \varepsilon^p_{failure} \le \varepsilon^p_{eff} \le \varepsilon^p_{ruptur}$$

After exceeding the failure strain softening begins and continues until the rupture strain is reached.



A solid element used as spotweld is shown. When resultant based failure is used orientation is very important. Nodes n1-n4 attach to the lower shell mid-surface and nodes n5-n8 attach to the upper shell mid-surface. The resultant forces and moments are computed based on the assumption that the brick element is properly oriented.

MATD119 (SOL 700) General Nonlinear 6DOF Discrete Beam

This is a very general spring and damper model. This beam is based on the MATDS06 material model. Additional unloading options have been included. The two nodes defining the beam may be coincident to give a zero length beam, or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the PBDISCR input should be set to a value of 2.0 or 3.0 to give physically correct behavior. A triad is used to orient the beam for the directional springs.

Format:

1	2	3	4	5	6	7	8	9	10
MATD119	MID	RO	KT	KR	UNLDOPT	OFFSET	DAMPF		
	LCIDTR	CLIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT			
	LCIDTUR	LCIDTUS	LCIDTUT	LCIDRUR	LCIDRUS	LCIDRUT			
	LCIDTDR	LCIDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCCIDRDT			
	LCIDTER	LCIDTES	LCIDTET	LCIDRER	LCIDRES	LCIDRET			
	UTFAILR	UTFAILS	UTFAILT	WTFAILR	WTFAILS	WTFAILT			
	UCFAILR	UCFAILS	UCFAILT	WCFAILR	WCFAILS	WCFAILT			
	IUR	IUS	IUT	IWR	IWWS	IWT			

Example:

MATD119	21	5.43	4000.	54000.	2		1.0		
	12	13	14	15	16	17			
	112	113	114	115	116	117			
	212	213	214	215	216	217			
	212	213	214	215	216	217			

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer > 0, Required)
RO	Mass density. Type (Real > 0.0, Required)
KT	Translational stiffness for unloading option UNLDOPT=2. (Real > 0.0, Required)
KR	Rotational stiffness for unloading option UNLDOPT=2. (Real > 0.0, Required)

Field	Contents
UNLDOPT	Unloading option (Also see **Figure 8-120**): (Integer ≥ 0 Default = 0) = 0: Loading and unloading follow loading curve = 1: Loading follows loading curve, unloading follows unloading curve. The unloading curve ID if undefined is taken as the loading curve. = 2: Loading follows loading curve, unloading follows unloading stiffness, KT or KR, to the unloading curve. The loading and unloading curves may only intersect at the origin of the axes. = 3: Quadratic unloading from peak displacement value to a permanent offset.
OFFSET	Offset factor between 0 and 1.0 to determine permanent set upon unloading if the UNLDOPT=3. The permanent sets in compression and tension are equal to the product of this offset value and the maximum compressive and tensile displacements, respectively. (Real 0.0 ≤ OFFSET ≤ 1.0, Default = 1.0)
DAMPF	Damping factor for stability. Values in the neighborhood of unity are recommended. This damping factor is properly scaled to eliminate time step size dependency. Also, it is active if and only if the local stiffness is defined. (Real > 0.0, Default=1.0)
LCIDTR	Load curve ID defining translational force resultant along local r-axis versus relative translational displacement. If undefined, no stiffness related forces are generated for this degree of freedom. The loading curves must be defined from the most negative displacement to the most positive displacement. The force does not need to increase monotonically. The curves in this input are linearly extrapolated when the displacement range falls outside the curve definition. (Integer ≥ 0, Default=0)
LCIDTS	Table ID defining translational force resultant along local s-axis versus relative translational displacement. (Integer ≥ 0, Default = 0)
LCIDTT	Table ID defining translational force resultant along local t-axis versus relative translational displacement. (Integer ≥ 0, Default = 0)
LCIDRR	Table ID defining rotational moment resultant about local r-axis versus relative rotational displacement. (Integer ≥ 0, Default = 0)
LCIDRS	Table ID defining rotational moment resultant about local s-axis versus relative rotational displacement. (Integer ≥ 0, Default = 0)

Field	Contents
LCIDRT	Table ID defining rotational moment resultant about local t-axis versus relative rotational displacement. (Integer $\geq$ 0, Default = 0)
LCIDTUR	Table ID defining translational force resultant along local r-axis versus relative translational displacement during unloading. The force values defined by this curve must increase monotonically from the most negative displacement to the most positive displacement. For UNLDOPT=1, the slope of this curve must equal or exceed the loading curve for stability reasons. This is not the case for UNLDOPT=2. For loading and unloading to follow the same path simply set LCIDTUR=LCIDTR. For options UNLDOPT=0 or 3 the unloading curve is not required. (Integer $\geq$ 0, Default = 0)
LCIDTUS	Table ID defining translational force resultant along local s-axis versus relative translational displacement during unloading. (Integer $\geq$ 0, Default = 0)
LCIDTUT	Table ID defining translational force resultant along local t-axis versus relative translational displacement during unloading. (Integer $\geq$ 0, Default = 0)
LCIDRUR	Table ID defining rotational moment resultant about local r-axis versus relative rotational displacement during unloading. (Integer $\geq$ 0, Default = 0)
LCIDRUS	Table ID defining rotational moment resultant about local s-axis versus relative rotational displacement. during unloading. (Integer $\geq$ 0, Default = 0)
LCIDRUT	Table ID defining rotational moment resultant about local t-axis versus relative rotational displacement during unloading. If zero, no viscous forces are generated for this degree of freedom. (Integer $\geq$ 0, Default = 0)
LCIDTDR	Table ID defining translational damping force resultant along local r-axis versus relative translational velocity. (Integer $\geq$ 0, Default = 0)
LCIDTDS	Table ID defining translational damping force resultant along local s-axis versus relative translational velocity. (Integer $\geq$ 0, Default = 0)
LCIDTDT	Table ID defining translational damping force resultant along local t-axis versus relative translational velocity. (Integer $\geq$ 0, Default = 0)

Field	Contents
LCIDRDR	Table ID defining rotational damping moment resultant about local r-axis versus relative rotational velocity. (Integer $\geq$ 0, Default = 0)
LCIDRDS	Table ID defining rotational damping moment resultant about local s-axis versus relative rotational velocity. (Integer $\geq$ 0, Default = 0)
LCIDRDT	Load curve ID defining rotational damping moment resultant about Table versus relative rotational velocity. (Integer $\geq$ 0, Default = 0)
LCIDTER	Table ID defining translational damping force scale factor versus relative displacement in local r-direction. (Integer $\geq$ 0, Default = 0)
LCIDTES	Table ID defining translational damping force scale factor versus relative displacement in local s-direction. (Integer $\geq$ 0, Default = 0)
LCIDTET	Table ID defining translational damping force scale factor versus relative displacement in local t-direction. (Integer $\geq$ 0, Default = 0)
LCIDRER	Table ID defining rotational damping moment resultant scale factor versus relative displacement in local r-rotation. (Integer $\geq$ 0, Default = 0)
LCIDRES	Table ID defining rotational damping moment resultant scale factor versus relative displacement in local s-rotation. (Integer $\geq$ 0, Default = 0)
LCIDRET	Table ID defining rotational damping moment resultant scale factor versus relative displacement in local t-rotation. (Integer $\geq$ 0, Default = 0)
UTFAILR	Optional, translational displacement at failure in tension. If zero, the corresponding displacement, u_r, is not considered in the failure calculation. (Real $\geq$ 0.0, Default = 0.0)
UTFAILS	Optional, translational displacement at failure in tension. If zero, the corresponding displacement, u_s, is not considered in the failure calculation. (Real $\geq$ 0.0, Default = 0.0)
UTFAILT	Optional, translational displacement at failure in tension. If zero, the corresponding displacement, u_t, is not considered in the failure calculation. (Real $\geq$ 0.0, Default = 0.0)
WTFAILR	Optional, rotational displacement at failure in tension. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation. (Real $\geq$ 0.0, Default = 0.0)

Field	Contents
WTFAILS	Optional, rotational displacement at failure in tension. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation. (Real $\geq$ 0.0, Default = 0.0)
WTFAILT	Optional rotational displacement at failure in tension. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation. (Real $\geq$ 0.0, Default = 0.0)
UCFAILR	Optional, translational displacement at failure in compression. If zero, the corresponding displacement, u_r, is not considered in the failure calculation. Define as a positive number. (Real $\geq$ 0.0, Default = 0.0)
UCFAILS	Optional, translational displacement at failure in compression. If zero, the corresponding displacement, u_s, is not considered in the failure calculation. Define as a positive number. (Real $\geq$ 0.0, Default = 0.0)
UCFAILT	Optional, translational displacement at failure in compression. If zero, the corresponding displacement, u_t, is not considered in the failure calculation. Define as a positive number. (Real $\geq$ 0.0, Default = 0.0)
WCFAILR	Optional, rotational displacement at failure in compression. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation. Define as a positive number. (Real $\geq$ 0.0, Default = 0.0)
WCFAILS	Optional, rotational displacement at failure in compression. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation. Define as a positive number. (Real $\geq$ 0.0, Default = 0.0)
WCFAILT	Optional, rotational displacement at failure in compression. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation. Define as a positive number. (Real $\geq$ 0.0, Default = 0.0)
IUR	Initial translational displacement along local r-axis. (Real $\geq$ 0.0, Default = 0.0)
IUS	Initial translational displacement along local s-axis. (Real $\geq$ 0.0, Default = 0.0)
IUT	Initial translational displacement along local t-axis. (Real $\geq$ 0.0, Default = 0.0)

Field	Contents
IWR	Initial rotational displacement about the local r-axis. (Real $\geq$ 0.0, Default = 0.0)
IWS	Initial rotational displacement about the local s-axis. (Real $\geq$ 0.0, Default = 0.0)
IWT	Initial rotational displacement about the local t-axis. (Real $\geq$ 0.0, Default = 0.0)

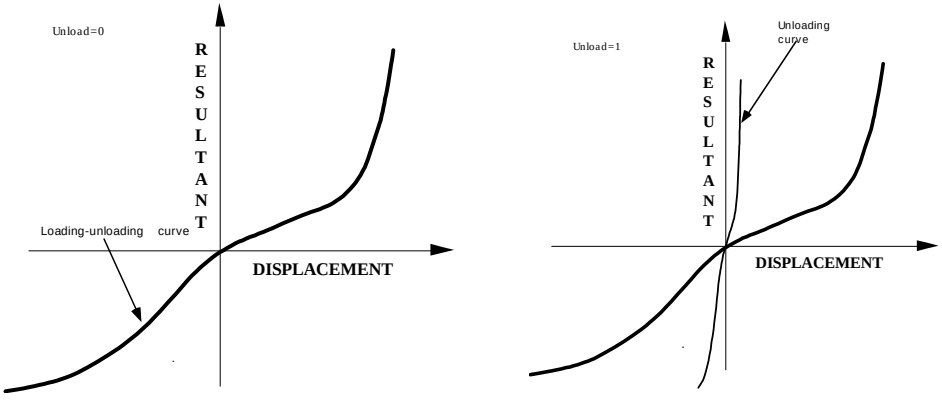
Remarks:

1. Corresponds to Ls-Dyna entry * **MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM** – MAT119
2. Catastrophic failure, which is based on displacement resultants, occurs if either of the following inequalities are satisfied:

$$\left(\frac{u_r}{u_r^{tfail}}\right)^2+\left(\frac{u_s}{u_s^{tfail}}\right)^2+\left(\frac{u_t}{u_t^{tfail}}\right)^2+\left(\frac{\theta_r}{\theta_r^{tfail}}\right)^2+\left(\frac{\theta_s}{\theta_s^{tfail}}\right)^2+\left(\frac{\theta_t}{\theta_t^{tfail}}\right)^2-1.\geq 0$$

$$\left(\frac{u_r}{u_r^{cfail}}\right)^2+\left(\frac{u_s}{u_s^{cfail}}\right)^2+\left(\frac{u_t}{u_t^{cfail}}\right)^2+\left(\frac{\theta_r}{\theta_r^{cfail}}\right)^2+\left(\frac{\theta_s}{\theta_s^{cfail}}\right)^2+\left(\frac{\theta_t}{\theta_t^{cfail}}\right)^2-1.\geq 0$$

After failure the discrete element is deleted. If failure is included either the tension failure or the compression failure or both may be used.



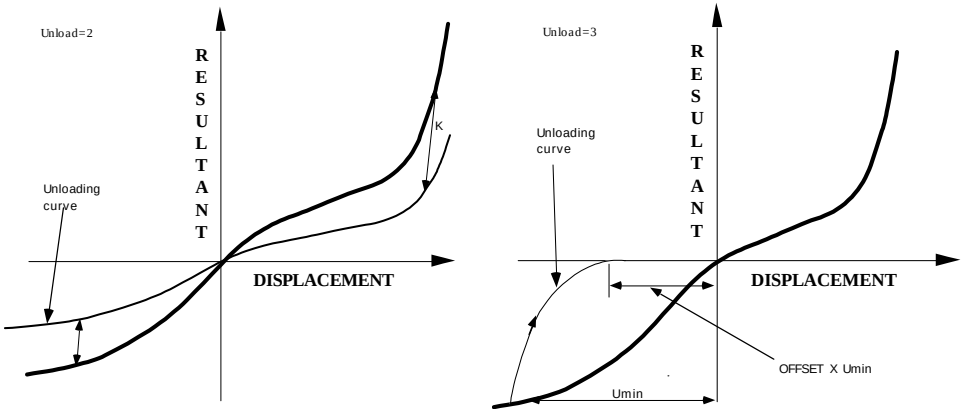


Figure 8-120 Load and unloading behavior.

MATD121 (SOL 700) General Nonlinear 1DOF Discrete Beam

This is Material Type 121. This is a very general spring and damper model. This beam is based on the MATDS06 material model and is a one-dimensional version of MATD119. Additional unloading options have been included.

Format:

1	2	3	4	5	6	7	8	9	10
MATD121	MID	RO	K	UNLDOPT	OFFSET	DAMPF			
	LCIDT	LCIDTU	LCIDTD	LCIDTE					
	UTFAIL	UCFAIL	IU						

Example:

MATD121	21	4.5	3000.0	2					
	12	13	14	15					

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer > 0, Required)
RO	Mass density. (Real > 0.0, Required)
K	Translational stiffness for unloading option UNLDOPT=2. (Real > 0.0, Required)
DAMPF	Damping factor for stability. Values in the neighborhood of unity are recommended. This damping factor is properly scaled to eliminate time step size dependency. Also, it is active if and only if the local stiffness is defined. (Real > 0.0, Default = 1.0)
UNLDOPT	Unloading option (Also see **Figure 8-120**): (Integer $\geq$ 0, Default = 0) =0: Loading and unloading follow loading curve =1: Loading follows loading curve, unloading follows unloading curve. The unloading curve ID if undefined is taken as the loading curve. =2: Loading follows loading curve, unloading follows unloading stiffness, K, to the unloading curve. The loading and unloading curves may only intersect at the origin of the axes.

Field	Contents
	=3: Quadratic unloading from peak displacement value to a permanent offset.
OFFSET	Offset to determine permanent set upon unloading if the UNLDOPT=3. The permanent sets in compression and tension are equal to the product of this offset value and the maximum compressive and tensile displacements, respectively. (Real $\geq$ 0.0, Default = 0.0)
LCIDT	Table ID defining translational force resultant along the axis versus relative translational displacement. If zero, no stiffness related forces are generated for this degree of freedom. The loading curves must be defined from the most negative displacement to the most positive displacement. The force does not need to increase monotonically for the loading curve. The curves in this input are extrapolated when the displacement range falls outside the curve definition. (Integer $\geq$ 0, Default = 0)
LCIDTU	Table ID defining translational force resultant along the axis versus relative translational displacement during unloading. The force values defined by this curve must increase monotonically from the most negative displacement to the most positive displacement. For UNLDOPT=1, the slope of this curve must equal or exceed the loading curve for stability reasons. This is not the case for UNLDOPT=2. For loading and unloading to follow the same path simply set LCIDTU=LCIDT. (Integer $\geq$ 0, Default = 0)
LCIDTD	Table ID defining translational damping force resultant along local the axis versus relative translational velocity. (Integer $\geq$ 0, Default = 0)
LCIDTE	Table ID defining translational damping force scale factor versus relative displacement in along axis. (Integer $\geq$ 0 Default = 0)
UTFAIL	Optional, translational displacement at failure in tension. If zero, failure in tension is not considered. (Real $\geq$ 0.0, Default=0.0)
UCFAIL	Optional, translational displacement at failure in compression. If zero, failure in compression is not considered. (Real $\geq$ 0.0, Default=0.0)
IU	Initial translational displacement along axis. (Real $\geq$ 0.0, Default=0.0)

Remark:

Corresponds to Ls-Dyna Input *
MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM – MAT121.

MATD126 (SOL 700)

The major use of this material model is for aluminum honeycomb crushable foam materials with anisotropic behavior. Two yield surfaces are available. In the first, nonlinear elastoplastic material behavior can be defined separately for all normal and shear stresses, which are considered to be fully uncoupled. In the second a yield surface is defined that considers the effects of off axis loading. The second yield surface is transversely anisotropic. The choice of yield surfaces is flagged by the sign of the first load curve ID, LCA. The development of the second yield surface is based on experimental test results of aluminum honeycomb specimens at Toyota Motor Corporation. The default element for this material is solid type 0, a nonlinear spring type brick element. *The recommended hourglass control is the type 2 viscous formulation for one point integrated solid elements. The stiffness form of the hourglass control when used with this constitutive model can lead to nonphysical results since strain localization in the shear modes can be inhibited.*

Format:

1	2	3	4	5	6	7	8	9	10
MATD126	MID	RO	E	PR	SIGY	VF	MU	BULK	
	LCA	LCB	LCC	LCS	LCAB	LCBC	LCCA	LCSR	
	EAAU	EBBU	ECCU	GABU	GBCU	GCAU	AOPT		
	XP	YP	ZP	A1	A2	A3			
	D1	D2	D3	TSEF	SSEF	VREF	TREF		

Example:

MATD126	20	4.3-5	2.0E4	0.1	150.	0.8	.05	0.0	
	201			211					
	2.0E4	1.5E4	1.4E4	0.7E4	0.4E4	0.3E4	0.0		
	0.	0.	0.	0.	0.	0.			
	0.	0.	0.	.22	.11	.6	.05		

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer, no default, required field)
RO	Mass density. (Real, no default, required field)

Field	Contents
E	Young's modulus for compacted honeycomb material. (Real, no default, required field)
PR	Poisson's ratio for compacted honeycomb material. (Real, no default, required field)
SIGY	Yield stress for fully compacted honeycomb. (Real, no default, required field)
VF	Relative volume at which the honeycomb is fully compacted. This parameter is ignored for corotational solid elements, types 0 and 9. (Real, no default)
MU	μ, material viscosity coefficient. (Real, Default = .05) Recommended.
BULK	Bulk viscosity flag: = 0.0: bulk viscosity is not used. This is recommended. = 1.0: bulk viscosity is active and $\mu = 0$. This will give results identical to previous versions of LS-DYNA. (Real, Default = 0.0)
LCA	Load curve ID: LCA < 0: Yield stress as a function of the angle off the material axis in degrees. LCA > 0: sigma-aa versus normal strain component aa. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. See notes below. (Integer, no default, required field)
LCB	Load curve ID: LCA < 0: strong axis stress as a function of the volumetric strain. The abscissa values must range between 0 to 90 degrees, inclusive. LCA > 0: sigma-bb versus normal strain component bb. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCB=LCA. See notes below. (Integer, Default = LCA)
LCC	Load curve ID: LCA<0: weak axis stress as a function of the volumetric strain.

Field	Contents
	LCA>0: sigma-cc versus normal strain component cc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a logarithmic strain is expected. Default LCC=LCA. See notes below. (Integer, Default = LCA)
LCS	Load curve ID: LCA< 0: damage curve giving shear stress multiplier as a function of the shear strain component. This curve definition is optional and may be used if damage is desired. LCA>0: shear stress versus shear strain. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. Default LCS=LCA. Each component of shear stress may have its own load curve. See notes below. (Integer, Default = LCS)
LCAB	Load curve ID LCA<0: damage curve giving shear ab-stress multiplier as a function of the ab-shear strain component. This curve definition is optional and may be used if damage is desired. LCA>0: sigma-ab versus shear strain-ab. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. See notes below. (Integer, Default = LCS)
LCBC	Load curve ID LCA < 0: damage curve giving bc-shear stress multiplier as a function of the ab-shear strain component. This curve definition is optional and may be used if damage is desired. LCA > .0: sigma-bc versus shear strain-bc. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. See notes below. (Integer, Default = LCS)
LCCA	Load curve ID. LCA<.0: damage curve giving ca-shear stress multiplier as a function of the ca-shear strain component. This curve definition is optional and may be used if damage is desired.

Field	Contents
	LCA>.0: sigma-ca versus shear strain-ca. For the corotational solid elements, types 0 and 9, engineering strain is expected, but for all other solid element formulations a shear strain based on the deformed configuration is used. See notes below. (Integer, Default = LCS)
LCRS	Load curve ID,, for strain-rate effects defining the scale factor versus strain rate $\dot{\varepsilon} = \sqrt{}$. This is optional. The curves defined above are scaled using this curve. (Integer, No Default)
EAAU	Elastic modulus E_{aau} in uncompressed configuration. (Real, No Default, Required field)
EBBU	Elastic modulus E_{bbu} in uncompressed configuration. (Real, No Default, Required field)
ECCU	Elastic modulus E_{ccu} in uncompressed configuration. (Real, No Default, Required field)
GABU	Shear modulus G_{abu} in uncompressed configuration. (Real, No Default, Required field)
GBCU	Shear modulus G_{bcu} in uncompressed configuration. (Real, No Default, Required field)
GCAU	Shear modulus G_{cau} in uncompressed configuration. (Real, No Default, Required field)
aopt	Material axes option (see MAT_*OPTION* TROPIC_ELASTIC for a more complete description): = 0.0: locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_ NODES. = 1.0: locally orthotropic with material axes determined by a point in space and the global location of the element center; this is the a-direction. = 2.0: globally orthotropic with material axes determined by vectors defined below, as with *DEFINE_COORDINATE_ VECTOR. (Real, Default = 0.0)
xp yp zp	Coordinates of point p for AOPT = 1. (Real, Default = 0.0)
a1 a2 a3	Components of vector a for AOPT = 2. (Real, Default = 0.0)
d1 d2 d3	Components of vector d for AOPT = 2. (Real, Default = 0.0)
TSEF	Tensile strain at element failure (element will erode). (Real, Default = 0.0)

Field	Contents
SSEF	Shear strain at element failure (element will erode). (Real, Default = 0.0)
VREF	This is an optional input parameter for solid elements types 1, 2, 3, 4, and 10. Relative volume at which the reference geometry is stored. At this time the element behaves like a nonlinear spring. The TREF, below, is reached first then VREF will have no effect. (Real, Default = 0.0)
TREF	This is an optional input parameter for solid elements types 1, 2, 3, 4, and 10. Element time step size at which the reference geometry is stored. When this time step size is reached the element behaves like a nonlinear spring. If VREF, above, is reached first then TREF will have no effect. (Real, Default = 0.0)

Remarks:

1. For efficiency it is strongly recommended that the load curve ID's: LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA, contain exactly the same number of points with corresponding strain values on the abscissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abscissa strain values are not consistent between load curves.

2. For solid element formulations 1 and 2, the behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, i.e., an a component of strain will generate resistance in the local *a*-direction with no coupling to the local *b* and *c* directions. The elastic moduli vary from their initial values to the fully compacted values linearly with the relative volume:

$$E_{aa} = E_{aau} + \beta(E - E_{aau})G_{ab} = E_{abu} + \beta(G - G_{abu})$$
$$E_{bb} = E_{bbu} + \beta(E - E_{bbu})G_{bc} = G_{bcu} + \beta(G - G_{bcu})$$
$$E_{cc} = E_{ccu} + \beta(E - E_{ccu})G_{ca} = G_{cau} + \beta(G - G_{cau})$$

where

$$\beta = \max\left[\min\left(\frac{1-v}{1-v}, 1\right), 0\right]$$

and G is the elastic shear modulus for the fully compacted honeycomb material

$$G = \frac{E}{2(1+v)}$$

The relative volume, V, is defined as the ratio of the current volume over the initial volume, and typically, V=1 at the beginning of a calculation.

For corotational solid elements, types 0 and 9, the components of the stress tensor remain uncoupled and the uncompressed elastic moduli are used, that is, the fully compacted elastic moduli are ignored.

The load curves define the magnitude of the stress as the material undergoes deformation. The first value in the curve should be less than or equal to zero corresponding to tension and increase to full compaction. **Care should be taken when defining the curves so the extrapolated values do not lead to negative yield stresses.**

At the beginning of the stress update we transform each element's stresses and strain rates into the local element coordinate system. For the uncompacted material, the trial stress components are updated using the elastic interpolated moduli according to:

$$\sigma_{aa}^{n+1^{trial}} = \sigma_{aa}^{n} + E_{aa}\Delta\varepsilon_{aa} \qquad \sigma_{ab}^{n+1^{trial}} = \sigma_{ab}^{n} + 2G_{ab}\Delta\varepsilon_{ab}$$

$$\sigma_{bb}^{n+1^{trial}} = \sigma_{bb}^{n} + E_{bb}\Delta\varepsilon_{bb} \qquad \sigma_{bc}^{n+1^{trial}} = \sigma_{bc}^{n} + 2G_{bc}\Delta\varepsilon_{bc}$$

$$\sigma_{cc}^{n+1^{trial}} = \sigma_{cc}^{n} + E_{cc}\Delta\varepsilon_{cc} \qquad \sigma_{ca}^{n+1^{trial}} = \sigma_{ca}^{n} + 2G_{ca}\Delta\varepsilon_{ca}$$

If LCA>0, each component of the updated stress tensor is checked to ensure that it does not exceed the permissible value determined from the load curves, e.g., if

$$\left|\sigma_{ij}^{n+1^{trial}}\right| > \lambda\sigma_{ij}(\varepsilon_{ij})$$

then

$$\sigma_{ij}^{n+1} = \sigma_{ij}(\varepsilon_{ij})\frac{\lambda\sigma_{ij}^{n+1^{trial}}}{\sigma_{ij}^{n+1^{trial}}}$$

On Card 3 $\sigma(\varepsilon_{ij})$ is defined in the load curve specified in columns 31-40 for the aa stress component, 41-50 for the bb component, 51-60 for the cc component, and 61-70 for the ab, bc, cb shear stress components. The parameter λ is either unity or a value taken from the load curve number, LCSR, that defines λ as a function of strain-rate. Strain-rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor.

If LCA < 0, a transversely anisotropic yield surface is obtained where the uniaxial limit stress, $\sigma^y(\varphi, \varepsilon^{vol})$, can be defined as a function of angle φ with the strong axis and volumetric strain, ε^{vol}. In order to facilitate the input of data to such a limit stress surface, the limit stress is written as:

$$\sigma^y(\varphi, \varepsilon^{vol}) = \sigma^b(\varphi) + (\cos\varphi)^2\sigma^s(\varepsilon^{vol}) + (\sin\varphi)^2\sigma^w(\varepsilon^{vol})$$

where the functions σ^b, σ^s, and σ^w are represented by load curves LCA, LCB, LCC, respectively. The latter two curves can be used to include the stiffening effects that are observed as the foam material crushes to the point where it begins to lock up. To ensure that the limit stress decreases with respect to the off-angle the curves should be defined such that following equations hold:

$$\frac{\partial\sigma^b(\varphi)}{\partial\varphi} \le 0$$

and

$$\sigma^s(\varepsilon^{vol}) - \sigma^w(\varepsilon^{vol}) \ge 0$$

For fully compacted material (element formulations 1 and 2), we assume that the material behavior is elastic-perfectly plastic and updated the stress components according to:

$$s_{ij}^{trial} = s_{ij}^{n} + 2G\Delta\varepsilon_{ij}^{dev}{}^{n+\frac{1}{2}}$$

where the deviatoric strain increment is defined as

$$\Delta\varepsilon_{ij}^{dev} = \Delta\varepsilon_{ij} - \frac{1}{3}\Delta\varepsilon_{kk}\delta_{ij}$$

We now check to see if the yield stress for the fully compacted material is exceeded by comparing

$$s_{eff}^{trial} = \left(\frac{3}{2}s_{ij}^{trial}s_{ij}^{trial}\right)^{\frac{1}{2}}$$

the effective trial stress to the yield stress, σ_y (Card 3, field 21-30). If the effective trial stress exceeds the yield stress we simply scale back the stress components to the yield surface

$$s_{ij}^{n+1} = \frac{\sigma_y}{s_{eff}^{trial}} s_{ij}^{trial}$$

We can now update the pressure using the elastic bulk modulus, K

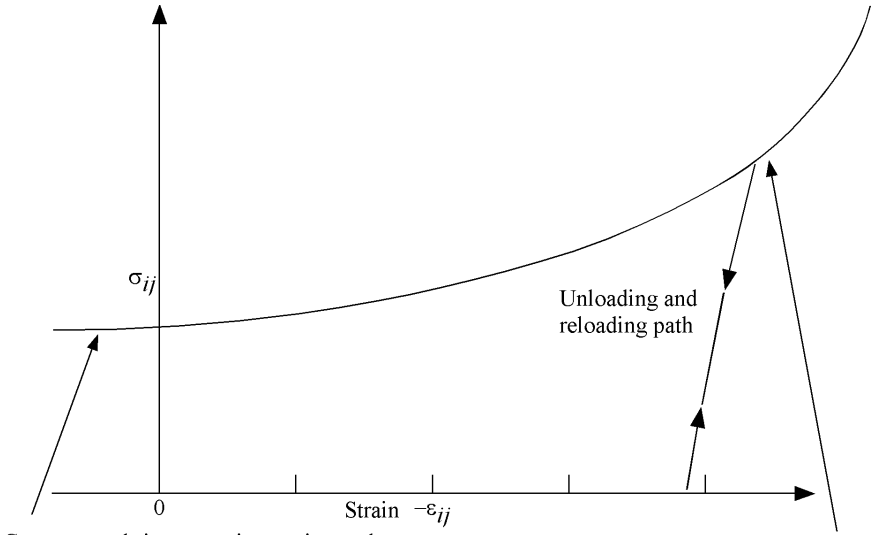
$$p^{n+1} = p^n - K\Delta\varepsilon_{kk}^{n+\frac{1}{2}}$$

$$K = \frac{E}{3(1-2v)}$$

and obtain the final value for the Cauchy stress

$$\sigma_{ij}^{n+1} = s_{ij}^{n+1} - p^{n+1}\delta_{ij}$$

After completing the stress update we transform the stresses back to the global configuration.



Curve extends into negative strain quadrant since LS-DYNA will extrapolate using the two end points. It is important that the extrapolation does not extend into the negative stress region.

Unloading is based on the interpolated Young's moduli which must provide an unloading tangent that exceeds the loading tangent.

Figure 8-121 Stress quantity versus strain. Note that the "yield stress" at a strain of zero is nonzero. In the load curve definition the "time" value is the directional strain and the "function" value is the yield stress. Note that for element types 0 and 9 engineering strains are used, but for all other element types the rates are integrated in time.

MATD127 (SOL 700) LS-DYNA Material #127 -- Arruda-Boyce Rubber

Used to model rubber using the Arruda-Boyce formulation. This material model provides a hyperelastic rubber model, (see Arruda and Boyce, 1993) combined optionally with linear viscoelasticity as outlined by Christensen, 1980. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATD127	MID	RO	K	G	N				
	LCID	TRAMP	NT						

Format for Viscoelastic Constants; up to 6 additional entries.

	GI	BETAI							

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
K	Bulk modulus. (Real)
G	Shear modulus. (Real)
N	Number of statistical links. (Integer)
LCID	Optional load curve ID of relaxation curve
	If constants β_I are determined via a least squares fit. This model ignores the constant stress. (Real)
TRAMP	Optional ramp time for loading. (Real)

NT	Number of Prony series terms in optional fit. If zero, the default is 6. Currently, the maximum number is 6. Values less than 6, possibly 3-5 are recommended, since each term used adds significantly to the cost. Caution should be exercised when taking the results from the fit. Always check the results of the fit in the output file. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once a satisfactory fit has been achieved it is recommended that the coefficients which are written into the output file be input in future runs. (Real)
GI	Optional shear relaxation modulus for the ith term. (Real)
BETAI	Optional decay constant if ith term. (Real)

Remarks:

Rubber is generally considered to be fully incompressible since the bulk modulus greatly exceeds the shear modulus in magnitude. To model the rubber as an unconstrained material a hydrostatic work term, $W_j(J)$, is included in the strain energy functional which is function of the relative volume, J, (Ogden, 1984):

$$W(J_1, J_2, J) = nk\theta\left[\frac{1}{2}(J_1-3) + \frac{1}{20N}(J_1^2-9) + \frac{11}{1050N^2}(J_1^3-27)\right] + nk\theta\left[\frac{19}{7000N^3}(J_1^4-81) + \frac{519}{673750N^4}(J_1^5-243) + W_H(J)\right]$$

where the hydrostatic work term is in terms of the bulk modulus, K, and the third invariant, J, as:

$$W_H(J) = \frac{K}{2}(J-1)^2$$

Rate effects are taken into accounted through linear viscoelasticity by a convolution integral of the form:

$$\sigma_{ij} = \int_0^t g_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

or in terms of the second Piola-Kirchhoff stress, S_{ij},and Green's strain tensor, E_{ij}

$$S_{ij} = \int_0^t G_{ijkl}(t-\tau)\frac{\partial \varepsilon_{kl}}{\partial \tau}d\tau$$

where $g_{ijkl}(t-\tau)$ and $G_{ijkl}(t-\tau)$ are the relaxation functions for the different stress measures. This stress is added to the stress tensor determined from the strain energy functional.

If we wish to include only simple rate effects, the relaxation function is represented by six terms from the Prony series:

$$g(t) = \alpha_0 + \sum_{m=1}^{N} \alpha_m e^{-\beta t}$$

given by,

$$g(t) = \sum_{i=1}^{N} G_i e^{-\beta_i t}$$

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i, and decay constants, β_i. The viscoelastic behavior is optional and an arbitrary number of terms may be used.

MATD181 (SOL 700)

LS-DYNA Material #181 -- Simplified Rubber and Foam Model

Used to model rubber or foam using a simplified formulation. This material model provides a rubber and foam model defined by a single uniaxial load curve or by a family of uniaxial curves at discrete strain rates. The foam formulation is triggered by defining a Poisson's ratio. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

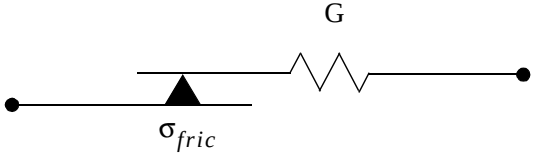
1	2	3	4	5	6	7	8	9	10
MATD181	MID	RO	K	MU	G	SIGF			
MATD181	SGL	SW	ST	LC/TBID	TENSION	RTYPE	AVGOPT	PR	

Field	Contents
MID	Material identification. A unique number has to be chosen. (Integer)
RO	Mass density. (Real)
K	Linear bulk modulus. (Real)
MU	Damping coefficient. (Real)
G	Shear modulus for frequency independent damping. Frequency independent damping is based of a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250-1000 times greater than SIGF. (Real)
SIGF	Limit stress for frequency independent, frictional, damping. (Real)
SGL	Specimen gauge length. (Real)
SW	Specimen width. (Real)
ST	Specimen thickness. (Real)
LC/TBID	Load curve or table ID defining the force versus actual change in the gauge length. If the table definition is used a family of curves are defined for discrete strain rates. The load curves should cover the complete range of expected loading, i.e., the smallest stretch ratio to the largest. (Real)
TENSION	Parameter that controls how the rate effects are treated. Applicable to the table definition. (Integer)

Field	Contents	
	EQ.-1	Rate effects are considered during tension and compression loading, but not during unloading.
	EQ.0	Rate effects are considered for compressive loading only.
	EQ.1	Rate effects are treated identically in tension and compression.
RTYPE	Strain rate type if a table is defined (Integer):	
	EQ.0	True strain rate.
	EQ.1	Engineering strain rate.
AVGOPT	Averaging option determine strain rate to reduce numerical noise. (Integer)	
	EQ.0	Simple average of twelve time steps.
	EQ.1	Running 12 point average.
PR	Optional Poisson's ratio, where a nonzero value triggers the foam formulation. If zero, an incompressible rubber like behavior is assumed and a default value of 0.495 is used internally. If a Poisson's ratio of 0.0 is desired, input a small value for PR such as 0.001. When fully integrated solid elements are used and when a nonzero Poisson's ratio is specified, a foam material is assumed and selective-reduced integration is not used due to the compressibility. This is true even if PR approaches 0.500. (Real)	

Remarks:

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



MATDB01 (SOL 700) Seat Belt Material

Defines a seat belt material.

Format:

1	2	3	4	5	6	7	8	9	10
MATDB01	MID	MPUL	LLCID	ULCID	LMIN				

Example:

MATDB01	21	3.4e2	11	12	0.01				

Field	Contents
MID	Belt material number. A unique number has to be chosen. (Integer, Required)
MPUL	Mass per unit length. (Real > 0.0, Default = 0.0)
LLCID	Load curve identification for loading (force vs. engineering strain). (Integer, Default = 0)
ULCID	Load curve identification for unloading (force vs. engineering strain). (Integer, Default = 0)
LMIN	Minimum length (for elements connected to slip rings and retractors), see Remarks. (Real, Default = 0.0)

Remarks:

Corresponds to Ls-Dyna input *MAT_SEATBELT

Each belt material defines stretch characteristics and mass properties for a set of belt elements. The user enters a load curve for loading, the points of which are (Strain, Force). Strain is defined as engineering strain, i.e.

$$Strain = \frac{currentlength}{initiallength} - 1.$$

Another similar curve is entered to describe the unloading behavior. Both load curves should start at the origin (0,0) and contain positive force and strain values only. The belt material is tension only with zero forces being generated whenever the strain becomes negative. The first non-zero point on the loading curve defines the initial yield point of the material. On unloading, the unloading curve is shifted along the strain axis until it crosses the loading curve at the ‘yield’ point from which unloading

commences. If the initial yield has not yet been exceeded or if the origin of the (shifted) unloading curve is at negative strain, the original loading curves will be used for both loading and unloading. If the strain is less than the strain at the origin of the unloading curve, the belt is slack and no force is generated. Otherwise, forces will then be determined by the unloading curve for unloading and reloading until the strain again exceeds yield after which the loading curves will again be used.

A small amount of damping is automatically included. This reduces high frequency oscillation, but, with realistic force-strain input characteristics and loading rates, does not significantly alter the overall forces-strain performance. The damping forced opposes the relative motion of the nodes and is limited by stability:

$$D = \frac{.1 \times \text{mass} \times \text{relative velocity}}{\text{timestep size}}$$

In addition, the magnitude of the damping force is limited to one-tenth of the force calculated from the force-strain relationship and is zero when the belt is slack. Damping forces are not applied to elements attached to sliprings and retractors.

The user inputs a mass per unit length that is used to calculate nodal masses on initialization.

A ‘minimum length’ is also input. This controls the shortest length allowed in any element and determines when an element passes through sliprings or are absorbed into the retractors. One tenth of a typical initial element length is usually a good choice.

MATDS01 (SOL 700)

Defines a translational or rotational elastic spring located between two nodes. The DRO variable on the PSPRMAT entry defines if the translational or rotational DOFs are connected.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS01	MID	K							

Example:

MATDS01	10	5000.							

Field	Contents	Type	Default
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique.	I > 0	Required
K	Elastic stiffness (force / displacement) or (moment / rotation).	R > 0	Required

MATDS02 (SOL 700)

Defines a translational or rotational linear damperlocated between two nodes. The DRO variable on the PSPRMAT entry defines if the translational or rotational DOFs are connected.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS02	MID	DC							

Example:

MATDS02	10	4000.							

Field	Contents	Type	Default
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique.	I > 0	Required
DC	Damping constant (force / displacement rate) or (moment / rotation rate).	R > 0	Required

MATDS03 (SOL 700) Elastoplastic Spring Material

Defines a translational or rotational elastoplastic spring with isotropic hardening located between two nodes. The DRO variable on the PSPRMAT entry defines if the translational or rotational DOFs are connected.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS03	MID	K	KT	FY					

Example:

MATDS03	22	2000.							

Field	Contents
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique. (Integer, Required)
K	Elastic stiffness (force / displacement) or (moment / rotation). (Real, Required)
KT	Tangent stiffness (force / displacement) or (moment / rotation). (Real, Default = 0.0)
FY	Yield (force) or (moment). (Real, Default = 0.0)

Remark:

This input corresponds to LS-Dyna Input *MAT_S03 = *MAT_SPRING_ELASTOPLASTIC.

MATDS04 (SOL 700) Nonlinear Elastoplastic Spring Material

Defines a translational or rotational nonlinear elastic spring with arbitrary force versus displacement or moment versus rotation, located between two nodes. Optionally, strain rate effects can be considered through a velocity dependent scale factor. The DRO variable on the PSPRMAT entry defines if the translational or rotational DOFs are connected.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS04	MID	LCD	LCR						

Example:

MATDS04	22	12	13						

Field	Contents
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique. (Integer, Required)
LCD	Load curve ID describing force versus displacement or moment versus rotation relationship. (Integer, Required)
LCR	Optional load curve describing scale factor on force or moment as a function of relative velocity or rotational velocity, respectively. The load curve must define the response in the negative and positive quadrants and pass through point (0,0). (Integer, Default = 0)

Remark:

Corresponds to LS-Dyna Input *MAT_S04 = *MAT_SPRING_NONLINEAR_ELASTIC.

MATDS05 (SOL 700) Nonlinear Viscous Damper Material

Defines a translational or rotational viscous damper with arbitrary force versus displacement or moment versus rotation, located between two nodes. The DRO variable on the PSPRMAT entry defines if the translational or rotational DOFs are connected.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS05	MID	LCDR							

Example:

MATDS05	22	12							

Field	Contents
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique. (Integer, Required)
LCDR	Load curve ID describing force versus rate-of-displacement relationship or a moment versus rate-of-rotation relationship. The load curve must define the response in the negative and positive quadrants and pass through point (0,0). (Integer, Required)

Remark:

Corresponds to LS-Dyna Input *MAT_S05 = *MAT_DAMPER_NONLINEAR_VISCOUS.

MATDS06 (SOL 700) General Nonlinear Spring Material

Defines a translational or rotational nonlinear spring with arbitrary loading and unloading definitions, located between two nodes. Optionally, hardening or softening can be defined. The DRO variable on the PSPRMAT entry defines if the translational or rotational DOFs are connected.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS06	MID	LCDL	LCDU	BETA	TYI	CYI			

Example:

MATDS06	22	12	14						

Field	Contents
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique. (Integer, Required)
LCDL	Load curve identification describing force versus displacement resp. moment versus rotation relationship for loading. (Integer, Required)
LCDU	Load curve identification describing force versus displacement resp. moment versus rotation relationship for unloading. (Integer, Required)
BETA	Hardening parameter, β: (Real, Default = 0.0) = 0.0: tensile and compressive yield with strain softening (negative or zero slope allowed in the force versus disp. load curves), > 0.0: kinematic hardening without strain softening. = 1.0: isotropic hardening without strain softening.
TYI	Initial yield force in tension. (Real, Default = 0.0)
CYI	Initial yield force in compression. (Real, Default = 0.0)

Remarks:

Corresponds to LS-Dyna Input *MAT_S06 = *MAT_SPRING_GENERAL_NONLINEAR .

Load curve points are in the format (displacement, force or rotation, moment). The points must be in order starting with the most negative (compressive) displacement resp. rotation and ending with the most positive (tensile) value. The curves need not be symmetrical.

The displacement origin of the “unloading” curve is arbitrary, since it will be shifted as necessary as the element extends and contracts. On reverse yielding the “loading” curve will also be shifted along the displacement resp. rotation axis. The initial tensile and compressive yield forces (TYI and CYI) define a range within which the element remains elastic (i.e. the “loading” curve is used for both loading and unloading). If at any time the force in the element exceeds this range, the element is deemed to have yielded, and at all subsequent times the “unloading” curve is used for unloading.

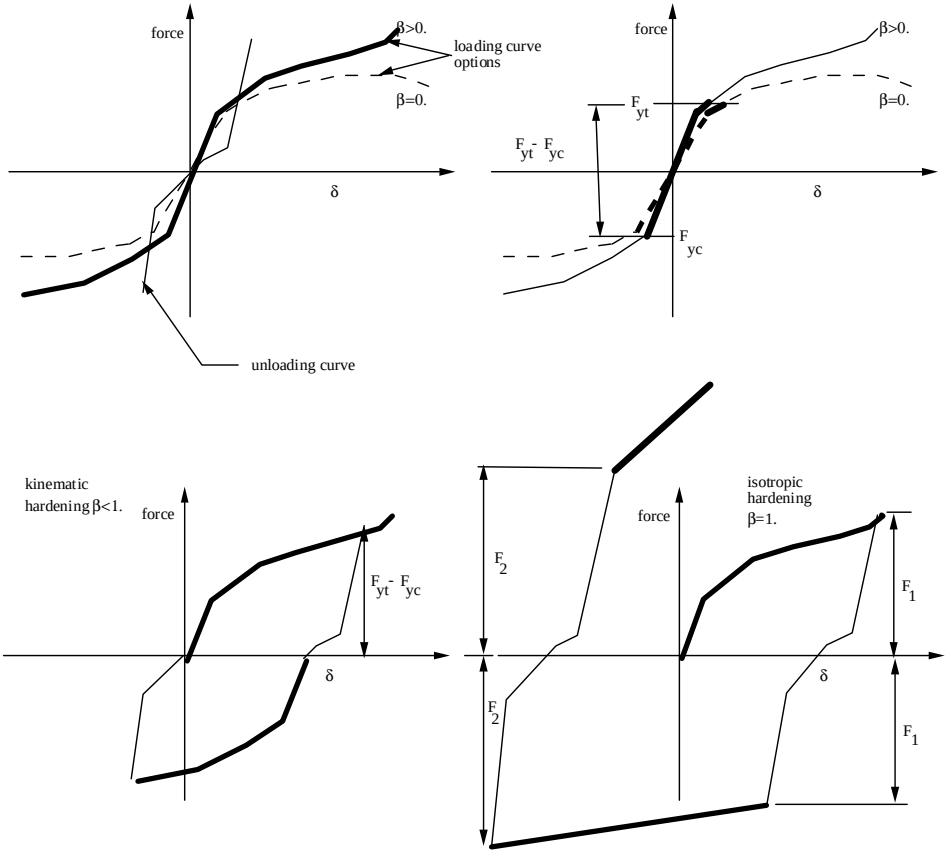


Figure 8-122 General Nonlinear Material for Discrete Elements.

MATDS07 (SOL 700) Maxwell Viscoelastic Spring Material

Defines a translational or rotational three Parameter Maxwell Viscoelastic spring located between two nodes. Optionally, a cutoff time with a remaining constant force / moment can be defined. The DRO variable on the PSPRMAT entry defines if the translational or rotational DOFs are connected.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS07	MID	KO	KI	BETA	TC	FC	COPT		

Example:

MATDS07	22	2000.	3000.	4.					

Field	Contents
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique. (Integer, Required)
K0	K_O, short time stiffness (Real, Required)
KI	K_∞, long time stiffness (Real, Required)
BETA	Decay parameter. (Real, Required)
TC	Cut off time. After this time a constant force / moment is transmitted. (Real, Default = 1.0E20)
FC	Force / moment after cutoff time Type: (Real, Default = 0.0)
COPT	Time implementation option (See Remark 1.) (Integer, Required) = 0: incremental time change, > 0: continuous time change.

Remarks:

Corresponds to LS-Dyna Input *MAT_S07 = *MAT_SPRING_MAXWELL.

1. The time varying stiffness $K(t)$ may be described in terms of the input parameters as

$$K(t) = K_\infty + (K_0 - K_\infty)e^{-\beta t}$$

This equation was implemented by Schwer [60] as either a continuous function of time or incrementally following the approach of Herrmann and Peterson [61]. The continuous function of time implementation has the disadvantage of the energy absorber's resistance decaying with increasing time even without deformation. The advantage of the incremental implementation is that an energy absorber must undergo some deformation before its resistance decays, i.e., there is no decay until impact, even in delayed impacts. The disadvantage of the incremental implementation is that very rapid decreases in resistance cannot be easily matched.

MATDS08 (SOL 700) Inelastic Spring Material

Defines a translational or rotational inelastic tension or compression only spring located between two nodes. Optionally, a user-specified unloading stiffness can be taken instead of the maximum loading stiffness. The DRO variable on the PSPRMAT entry defines if the translational or rotational DOFs are connected.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS08	MID	LCFD	KU	CTF					

Example:

MATDS08	22	12							

Field	Contents
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique. (Integer, Required)
LCFD	Load curve identification describing arbitrary force / torque versus displacement / twist relationship. This curve must be defined in the positive force-displacement quadrant regardless of whether the spring acts in tension or compression. (Integer, Required)
KU	Unloading stiffness (optional). The maximum of KU and the maximum loading stiffness in the force / displacement or the moment / twist curve is used for unloading. (Real, Default = 0.0)
CTF	Flag for compression / tension: (Real, Default = 0.0) = -1.0: tension only, = 0.0: default is set to 1.0, = 1.0: compression only.

Remark:

Corresponds to LS-Dyna Input *MAT_S08 = *MAT_SPRING_INELASTIC.

MATDS13 (SOL 700) Tri-linear Degrading Material

Defines a translational spring located between two nodes. This material allows concrete shear walls to be modeled as discrete elements under applied seismic loading. It represents cracking of the concrete, yield of the reinforcement and overall failure. Under cyclic loading, the stiffness of the spring degrades but the strength does not.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS13	MID	DEFL1	F1	DEFL2	F2	DEFL3	F3	FFLAG	

Example:

MATDS13	12	22.5	5000.						

Field	Contents
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique. (Integer, Required)
DEFL1	Deflection at point where concrete cracking occurs. (Real, Default = 0.0)
F1	Force corresponding to DEFL1. (Real, Default = 0.0)
DEFL2	Deflection at point where reinforcement yields. (Real, Default = 0.0)
F2	Force corresponding to DEFL2. (Real, Default = 0.0)
DEFL3	Deflection at complete failure. (Real, Default = 0.0)
F3	Force corresponding to DEFL3. (Real, Default = 0.0)
FFLAG	Failure flag. (Real, Default = 0.0)

Remark:

Corresponds to LS-Dyna Input *MAT_S13 = *MAT_SPRING_TRILINEAR_DEGRADING .

MATDS14 (SOL 700) Squat Shear Wall Material

Define a translational spring located between two nodes. This material allows squat shear walls to be modeled using discrete elements. The behavior model captures concrete cracking, reinforcement yield, ultimate strength followed by degradation of strength finally leading to collapse.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS14	MID	A14	C14	D14	E14	LCID	FSD		

Example:

MATDS14	12	1.0	2.3						

Field	Contents
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique. (Integer, Required)
A14	Material coefficient A. (Real, Default = 0.0)
B14	Material coefficient B. (Real, Default = 0.0)
C14	Material coefficient C. (Real, Default = 0.0)
D14	Material coefficient D. (Real, Default = 0.0)
E14	Material coefficient E. (Real, Default = 0.0)
LCID	Load curve ID referencing the maximum strength envelope curve. (Integer or blank, Default = 0)
FSD	Sustained strength reduction factor. (Real, Default = 0.0)

Remarks:

Corresponds to LS-Dyna Input *MAT_S14 = *MAT_SPRING_SQUAT_SHEARWALL.

Material coefficients A, B, C and D are empirically defined constants used to define the shape of the polynomial curves which govern the cyclic behavior of the discrete element. A different polynomial relationship is used to define the loading and unloading paths allowing energy absorption through hysteretic. Coefficient E is used in the definition of the path used to ‘jump’ from the loading path to the unloading path (or vice versa) where a full hysteretic loop is not completed. The load curve referenced

is used to define the force displacement characteristics of the shear wall under monotonic loading. This curve is the basis to which the polynomials defining the cyclic behavior refer to. Finally, on the second and subsequent loading / unloading cycles, the shear wall will have reduced strength. The variable FSD is the sustained strength reduction factor

MATDS15 (SOL 700) Muscle Material

Defines a translational spring located between two nodes. This material is a Hill-type muscle model with activation. It is for use with discrete elements. The Dytran LS-DYNA implementation is due to Dr. J.A. Weiss.

Format:

1	2	3	4	5	6	7	8	9	10
MATDS15	MID	LO	VMAX	SV	A	FMAX	TL	TV	
	FPE	LMAX	KSH						

Example:

MATDS15	21	1.0	123	-12	-13	-14	-15	-16	
	-17								

Field	Contents
MID	Material ID. MID is referenced on a PSPRMAT entry and must be unique. (Integer, Required)
L0	Initial muscle length, L_0. (Real, Default = 1.0)
VMAX	Maximum CE shortening velocity, V_{max}. (Real, Required)
SV	Scale factor, S_V, for V_{max} vs. active state. (Real if $\geq$ 0, Integer if < 0, Default = 1.0) < 0: absolute value gives load curve ID $\geq$ 0: constant value of 1.0 is used
A	Activation level vs. time function. (Real if $\geq$ 0, Integer if < 0, Required) < 0: absolute value gives load curve ID $\geq$ 0: constant value of A is used
FMAX	Peak isometric force, F_{max}. (Real, Required)
TL	Active tension vs. length function. (Real if $\geq$ 0, Integer if < 0, Default = 1.0) < 0: absolute value gives load curve ID $\geq$ 0: constant value of 1.0 is used

Field	Contents
TV	Active tension vs. velocity function. (Real if ≥ 0, Integer if < 0, Default = 1.0) < 0: absolute value gives load curve ID ≥ 0: constant value of 1.0 is used
FPE	Force vs. length function, F_{pe}, for parallel elastic element. (Real if ≥ 0, Integer if < 0, Default = 0.0) < 0: absolute value gives load curve ID = 0: exponential function is used (see below) > 0: constant value of 0.0 is used
LMAX	Relative length when F^{pe} reaches F_{MAX}. (Real, Required if F^{pe} = 0 above)
KSH	Constant, K_{sh}, governing the exponential rise of F^{pe}. (Real, Required if F^{pe} = 0 above)

Remarks:

Corresponds to LS-Dyna Input *MAT_S15 = *MAT_SPRING_MUSCLE.

The material behavior of the muscle model is adapted from the original model proposed by Hill (1938). Reviews of this model and extensions can be found in Winters (1990) and Zajac (1989). The most basic Hill-type muscle model consists of a contractile element (CE) and a parallel elastic element (PE) (**Figure 8-123**). An additional series elastic element (SEE) can be added to represent tendon compliance. The main assumptions of the Hill model are that the contractile element is entirely stress free and freely distensible in the resting state, and is described exactly by Hill's equation (or some variation). When the muscle is activated, the series and parallel elements are elastic, and the whole muscle is a simple combination of identical sarcomeres in series and parallel. The main criticism of Hill's model is that the division of forces between the parallel elements and the division of extensions between the series elements is arbitrary, and cannot be made without introducing auxiliary hypotheses. However, these criticisms apply to *any* discrete element model. Despite these limitations, the Hill model has become extremely useful for modeling musculoskeletal dynamics, as illustrated by its widespread use today.

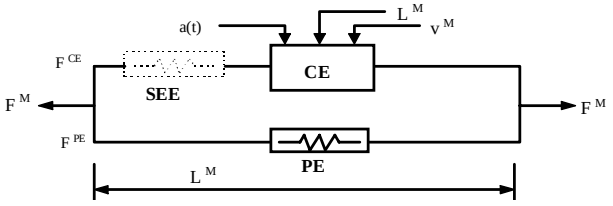


Figure 8-123 Discrete model for muscle contraction dynamics, based on a Hill-type representation. The total force is the sum of passive force F^{PE} and active force F^{CE}. The passive element (PE) represents energy storage from muscle elasticity, while the contractile element (CE) represents force generation by the muscle. The series elastic element (SEE), shown in dashed lines, is often neglected when a series tendon compliance is included. Here, $a(t)$ is the activation level, LM is the length of the muscle, and VM is the shortening velocity of the muscle.

When the contractile element (CE) of the Hill model is inactive, the entire resistance to elongation is provided by the PE element and the tendon load-elongation behavior. As activation is increased, force then passes through the CE side of the parallel Hill model, providing the contractile dynamics. The original Hill model accommodated only full activation - this limitation is circumvented in the present implementation by using the modification suggested by Winters (1990). The main features of his approach were to realize that the CE force-velocity input force equals the CE tension-length output force. This yields a three-dimensional curve to describe the force-velocity-length relationship of the CE. If the force-velocity y-intercept scales with activation, then given the activation, length and velocity, the CE force can be determined.

Without the SEE, the total force in the muscle FM is the sum of the force in the *CE* and the *PE* because they are in parallel:

$$F^M = F^{PE} + F^{CE}$$

The relationships defining the force generated by the CE and PE as a function of L_M, V_M and $a(t)$ are often scaled by F_{max}, the peak isometric force (p. 80, Winters 1990), L_O, the initial length of the muscle (p. 81, Winters 1990), and V_{max}, the maximum unloaded *CE* shortening velocity (p. 80, Winters 1990). From these, dimensionless length and velocity can be defined:

$$L = \frac{L^M}{L_O}$$

$$V = \frac{V^M}{V_{max} \cdot S_V(a(t))}$$

Here, S_v scales the maximum *CE* shortening velocity V_{max} and changes with activation level $a(t)$. This has been suggested by several researchers, i.e. Winters and Stark (1985). The activation level specifies the level of muscle stimulation as a function of time. Both have values between 0 and 1. The functions $S_v(a(t))$ and $a(t)$ are specified via load curves in LS-DYNA, or default values of $S_v = 1$ and $a(t) = 0$ are used. Note that L is always positive and that V is positive for lengthening and negative for shortening.

The relationship between F_{CE}, V and L was proposed by Bahler et al. (1967). A three-dimensional relationship between these quantities is now considered standard for computer implementations of Hill-type muscle models (i.e., eqn 5.16, p. 81, Winters 1990). It can be written in dimensionless form as:

$$F^{CE} = a(t) \cdot F_{max} \cdot f_{TL}(L) \cdot f_{TV}(V)$$

Here, fTL and fTV are the tension-length and tension-velocity functions for active skeletal muscle. Thus, if current values of L_M, V_M, and $a(t)$ are known, then F_{CE} can be determined (Figure 22.36).

The force in the parallel elastic element F_{PE} is determined directly from the current length of the muscle using an exponential relationship (eqn 5.5, p. 73, Winters 1990):

$$f_{PE} = \frac{F^{PE}}{F_{MAX}} = 0, \; L \le 1$$

$$f_{PE} = \frac{F^{PE}}{F_{MAX}} = \frac{1}{\exp(K_{sh}) - 1}\left[\exp\left(\frac{K_{sh}}{L_{max}}(L-1)\right) - 1\right], \; L > 1$$

Here, L_{max} is the relative length at which the force F_{max} occurs, and K_{sh} is a dimensionless shape parameter controlling the rate of rise of the exponential. Alternatively, the user can define a custom f_{PE} curve giving tabular values of normalized force versus dimensionless length as a load curve.

For computation of the total force developed in the muscle F_M, the functions for the tension-length f_{PE} and force-velocity f_{TV} relationships used in the Hill element must be defined. These relationships have been available for over 50 years, but have been

refined to allow for behavior such as active lengthening. The active tension-length curve f_{TL} describes the fact that isometric muscle force development is a function of length, with the maximum force occurring at an optimal length. According to Winters, this optimal length is typically around L=1.05, and the force drops off for shorter or longer lengths, approaching zero force for L=0.4 and L=1.5. Thus the curve has a bell-shape. Because of the variability in this curve between muscles, the user must specify the function f_{TL} via a load curve, specifying pairs of points representing the normalized force (with values between 0 and 1) and normalized length L (**Figure 8-124**).

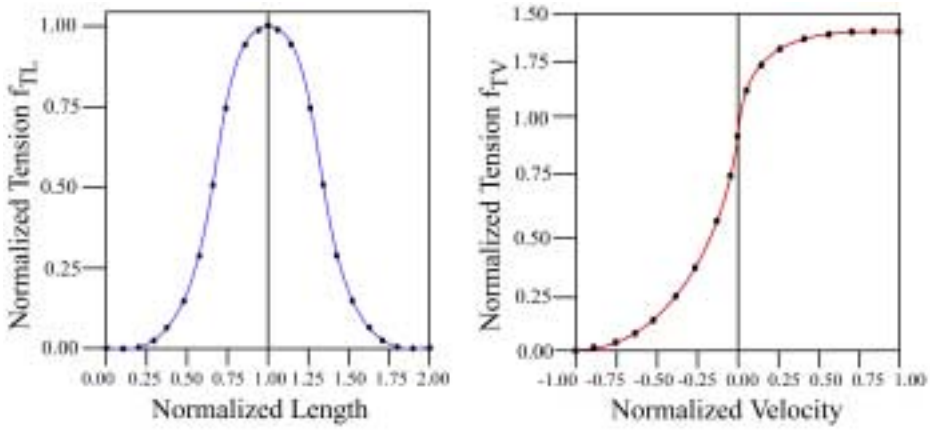


Figure 8-124 Typical Normalized Tension-Length (TL) and Tension-Velocity (TV) Curves for Skeletal Muscle.

The active tension-velocity relationship f_{TV} used in the muscle model is mainly due to the original work of Hill. Note that the dimensionless velocity V is used. When V=0, the normalized tension is typically chosen to have a value of 1.0. When V is greater than or equal to 0, muscle lengthening occurs. As V increases, the function is typically designed so that the force increases from a value of 1.0 and asymptotes towards a value near 1.4. When V is less than zero, muscle shortening occurs and the classic Hill equation hyperbola is used to drop the normalized tension to 0 (Figure 22.37). The user must specify the function f_{TV} via a load curve, specifying pairs of points representing the normalized tension (with values between 0 and 1) and normalized velocity V.

MATEP (SOL 600) Elasto-Plastic Material Properties

Specifies elasto-plastic material properties to be used for large deformation analysis. Used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATEP	MID	Form	Y0	FID		Wkhard	Method	H	
	“Reffect”	option	RTID	C	P				
	“Aniso”	N / A	R11	R22	R33	R12	R23	R31	
	“ORNL”	option	Yc10	TID	N / A				
	“Press”	option	alpha	beta	cracks	soften	crushs	srfac	
	“Gurson”	q1	q2	initial	critical	failure	nucl	Mean	
		Sdev	Nfrac						

Example:

MATEP	100	table		20	cauchy	isotrop	addm		
	aniso		1.1	0.9		1.02			

Field	Contents
MID	Identification number of MAT1, MAT2, MATORT or MAT9 entry. (Integer > 0).
Form	Selects a form of stress-plastic strain function to be specified (Character): **Table** for defining the function in TABLES1. (Default) **Perfect** for defining perfectly plastic material. See Remarks **1.** and **3.**
Y0	Initial yield stress Y_0 or hydrostatic stress for Mohr-Coulomb materials. See Remark **2.** (Real > 0 or blank)
FID	Identification number of TABLES1 entry. (Integer > 0 or blank)

Field	Contents
Wkhard	Selects a hardening rule defined by various work-hardening rules (Character): **Isotrop** for isotropic hardening. (Default) **Kinem** for kinematic hardening. **Combine** for combination between kinematic and isotropic hardening. See Remark 4.
Method	Selects a material processing method (Character): **Addmean** for additive decomposition using the mean normal process. (Default) **Multrad** for multiplicative decomposition using the radial return process.
H	plasticity modulus, ignored if FID field is specified. (Real $\geq$ 0; Default = 0)
“Reffect”	A keyword signifying that the following data pertains to the rate-dependent material properties.
Option	selects an option for strain-rate dependent yield stress (Character): **Table** for Tables1 input. (Default) **Cowper** for Cowper and Symonds model. See Remark 5.
RTID	TABLES1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate.
C	Specifies the constant C for Cowper and Symonds model. (Real; Default = 1)
P	Specifies the constant P for Cowper and Symonds model. (Real; Default = 1)
“Aniso”	A keyword signifying that the following data (Rij) pertain to the Hill’s an isotropic material option. See Remark 6.
Rij	Stress ratios of initial yield stresses in various material directions to the reference yield stress from FID / Y0 field. (Real > 0; Default = 1.0 for R11, R22, R33, R12, R23, R31, respectively)
“ORNL”	A keyword signifying that the following data pertains to the ORNL’s yield criteria. See Remark 7.

Field	Contents
Option	Selects an option for ORNL yield criteria (Character): **Norm** for normal ORNL model for stainless steel. (Default) **CRMO** for ORNL 2-1/4 Cr-Mo steel model. **REVP** for ORNL reversed plasticity model. **ARST** for ORNL full alpha reset model.
Yc10	Equivalent 10th cycle tensile yield stress. (Real > 0)
TID	Identification number of TABLES1 entry for normalized 10th cycle stress-plastic strain curve. (Integer > 0)
“Press”	A keyword signifying that the following data pertain to the pressure-dependent yield criteria. See Remarks **8.** and **9.**
Option	Selects an option for pressure-dependent yield criteria. (Character): **Lin** for linear Mohr-Coulomb model. (Default) **Pbl** for parabolic Mohr-Coulomb model. **Conc** for Buyukozturk concrete model.
Alpha	Specifies a parameter alpha for linear Mohr-Coulomb model or concrete model. (Real > 0 or blank)
Beta	Specifies a parameter beta for parabolic Mohr-Coulomb model or Buyukozturk concrete model (not used for linear Mohr-Coulomb model). (Real > 0 or blank)
Cracks	Critical cracking stress for Buyukozturk concrete model. (Real > 0 or blank)
Soften	Tension-softening modulus for Buyukozturk concrete model. (Real $\geq$ 0; Default = 0)
Crushs	Strain at which material crushes. (Real > 0; Default = 1.E10)
Srfac	Shear retention factor defining shear stress carrying capacity when crack closes for Buyukozturk concrete model. (0 $\leq$ Real < 1; Default = 0)
“Gurson”	A keyword signifying that the following data pertain to the modified Gurson model for porous metal plasticity with damage effects. See Remark **10.**
q1	First coefficient for the Gurson yield function. (Real $\geq$ 0; Default = 1.5).
q2	Second coefficient for the Gurson yield function. (Real $\geq$ 0; Default = 1)

Field	Contents
initial	Initial void volume fraction. (0 ≤ Real < 1; Default = 0)
critical	Critical void volume fraction at which void coalescence starts. (0 < Real < 1; Default = 0.2)
failure	Failure void volume fraction at which the material loses strength. (1./q1 < Real < 1; Default = 0.733)
nucl	Select a method of void nucleation (character, Default = strain): **none** for no nucleation. **strain** for plastic strain controlled nucleation. **stress** for stress controlled nucleation. See Remark **11.**
Mean	Mean strain or stress for void nucleation. (Real > 0; Default = 0.3)
Sdev	Standard deviation in the assumed normal distribution of nucleation strain or stress. (Real > 0; Default = 0.01)
Nfrac	Volume fraction of nucleating particles for void nucleation. (0 < Real < 0.5; Default = 0.04)

Remarks:

1. Unless continuation entry is present specifying various material models, von Mises yield criterion is used as default.
2. If Y_0 is not specified, FID field referring to a stress-plastic strain curve must be provided. If Y_0 is specified without FID field, the material is assumed perfectly plastic. If both Y_0 and FID fields are specified, FID supersedes Y0 field and the first data point in TABLES1 represents Y_0.

   The initial yield point corresponds to the first data point in the function specified on the FID field.

$$Y(\bar{\varepsilon}^p) = Y_0(1 + b\,\bar{\varepsilon}^p)^n$$

   where Y_0 is an initial yield stress, b and n are parameters characterizing the stress-strain relationship. In case of an anisotropic material, the initial yield point corresponds to the reference yield stress (Y_a in Remark **6.**)
3. The STRESS field controls only the input data. The stress-strain pair for output quantities are determined by PARAM, STRESS (i.e., Cauchy, Nominal or Kirch), where:

   Cauchy for the Cauchy stress and logarithmic strain pair (default)

Nominal for the nominal stress and engineering strain pair

Kirch for the second Piola-Kirchhoff stress and Lagrangian strain pair.

Notice that the input data is not allowed in the Kirchhoff stress-Lagrange strain pair.

The STRESS field is ignored for SOL 600

4. The plastic deformation starts when the effective stress ($\bar{\sigma}$) exceeds the yield stress.

   The yield stress is initially defined by the initial yield point, which is subsequently modified by the hardening rule to account for strain hardening. Under the isotropic hardening rule, the size of the yield surface expands as a function of effective plastic strain ($\bar{\varepsilon}^p$). Under the kinematic hardening rule, the center of the yield surface moves in stress space while keeping the same size and shape. Ziegler's law is used to define the translation of the yield surface. Under the combined hardening, the initial hardening is assumed to be entirely isotropic, but the elastic range attains a constant value (i.e., behaving like kinematic hardening) after some plastic straining. The effective stress for von Mises is expressed as

$$\bar{\sigma} = \sqrt{\frac{1}{2}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

   where the stress components are measured from the center of yield surface.

5. The Cowper and Symonds model scales the initial yield stress as a function of strain-rate, i.e.,

$$Y(\dot{\bar{\varepsilon}}) = Y_0\left[1 + \left(\frac{\dot{\bar{\varepsilon}}}{C}\right)^{1/P}\right]$$

6. Hill's anisotropic model introduces orthotropic plastic material. This option can only be combined with orthotropic or anisotropic elastic material (i.e., with MAT2, MATORT or MAT9). The plastic anisotropy proposed by Hill introduces six parameters to the von Mises yield function, from which an effective stress may be derived as

$$\bar{\sigma} = \sqrt{F(\sigma_2 - \sigma_3)^2 + G(\sigma_3 - \sigma_1)^2 + H(\sigma_1 - \sigma_2)^2 + 2L\tau_{23}^2 + 2M\tau_{31}^2 + 2N\tau_{12}^2}$$

   in which the material parameters can be related to the yield stress ratios by

$$F = \frac{1}{2}\left(\frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2}\right)$$

$$G = \frac{1}{2}\left(\frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2}\right)$$

$$H = \frac{1}{2}\left(\frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2}\right)$$

$$L = \frac{3}{2R_{23}^2},\ M = \frac{3}{2R_{31}^2},\ N = \frac{3}{2R_{12}^2}$$

with

$$R_{11} = \frac{Y_1}{Y_a},\ R_{22} = \frac{Y_2}{Y_a},\ R_{33} = \frac{Y_3}{Y_a}$$

$$R_{12} = \frac{\sqrt{3}T_{12}}{Y_a},\ R_{23} = \frac{\sqrt{3}T_{23}}{Y_a},\ R_{31} = \frac{\sqrt{3}T_{31}}{Y_a}$$

where Y_1, Y_2, and Y_3 are the initial tensile yield stresses measured in material directions 1, 2 and 3, respectively; T_{12}, T_{23}, and T_{31} are the shear yield stresses in pure shear; and Y_a is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by r-values defined by strain ratio measured in the uniaxial tension, i.e.,

$$a = \frac{\varepsilon_w}{\varepsilon_t} = \frac{l_n(w_o/w)}{l_n(t_o/t)} = \frac{H + (2N - F - G - 4H)\sin^2\alpha\cos^2\alpha}{F\sin^2\alpha + G\cos^2\alpha}$$

where t and w denote thickness and width, respectively; and α denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation, F, G, H and N can be determined by r-values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0 \quad , \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left(r_{45} + \frac{1}{2}\right)\left(1 + \frac{r_0}{r_{90}}\right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction (Y_0 or Y_{90}) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0\sqrt{\frac{r_{90}(1 + r_0)}{r_0 + r_{90}}} = Y_{90}\sqrt{\frac{r_0(1 + r_{90})}{r_0 + r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th}\sqrt{\frac{1}{2r_{45} + 1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

7. The elasticity constants must be isotropic for ORNL plasticity except for normal ORNL model for stainless steel. The 10th cycle stress-plastic strain curve in TID field should be a normalized function so that the yield stress at zero plastic strain is unity.

8. The pressure-dependent yielding, based on Drucker-Prager yield criterion, contains three options for frictional materials such as rocks and concrete. The generalized Mohr-Coulomb criterion introduces linear and parabolic models, developed by Drucker and Prager. The linear Mohr-Coulomb model assumes a linear function of hydrostatic stress for a yield function, i.e.,

$$aI_1 + \sqrt{J_2} - \frac{\sigma}{\sqrt{3}} = 0 \text{ or } \bar{\sigma} = \sqrt{3}\alpha\, I_1 + \sqrt{3J_2}$$

where

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

and

$$J_2 = \frac{1}{6}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]$$

The parameters α and $\bar{\sigma}$ (effective stress coinciding with the yield stress) can be related to material constants c (cohesion) and ϕ (frictional angle) by

$$c = \frac{\sigma}{3\sqrt{(1 - 12\alpha^2)}}$$

and

$$\sin\phi = \frac{3\alpha}{\sqrt{1 - 3\alpha^2}}$$

The parabolic Mohr-Coulomb model allows a yield envelope to be parabolic in the plane strain case, for which the yield function is expressed as

$$\sqrt{3J_2 + \sqrt{3}\beta\bar{\sigma}I_1} - \bar{\sigma} = 0$$

in which parameters are related to the material constants by

$$\bar{\sigma}^2 = 3\left(c^2 - \frac{\alpha^2}{3}\right)$$

and

$$\beta = \frac{\alpha}{\sqrt{3(3c^2 - \alpha^2)}}$$

9. The Drucker-Prager plasticity models can only be combined with isotropic elasticity. The Buyukozturk concrete plasticity model is a particular form of the generalized Drucker-Prager plasticity model, which is developed specifically for plane stress cases by Buyukozturk. The Buyukozturk yield function is expressed as

$$\beta\sqrt{3}Y\ I_1 + \gamma I_1^2 + 3J_2 - Y^2 = 0$$

where β is a user-specified constant, γ is an internal parameter (set to 0.2) with no user's access, and Y is the yield stress.

The Buyukozturk concrete plasticity model is coupled with crack and crush capability, which is designed for a low-tension material. The low-tension material develops a crack in the perpendicular direction to the maximum principal stress when it exceeds a critical value. The tension softening

modulus can be specified (in absolute value) by the user to process the cracking process gradually. The default value (0.) is intended for a sudden cracking with a complete loss of the stiffness upon cracking. After the initial crack, a second crack can initiate in the perpendicular direction to the first crack. Likewise, the third crack can be formed in 3D solid elements. The loading may reverse the direction after the crack is formed. In this case, the crack will close and some load carrying capacity is resumed. The compression capability is fully resumed and the shear stresses may be transmitted over the crack surface with a reduced stiffness by a factor specified as shear retention factor. The material may fail in compression by crushing. The input value for the crush strain is positive, which implies an absolute value of a compressive strain. The material loses its integrity for good upon crushing. The reinforcement bars may be simulated by adding REBAR elements.

10. The Gurson model for porous metal plasticity may be used only with isotropic hardening rule. All other hardening rules will be ignored if Gurson model is selected. The Gurson model modified by Tvergaard and Needleman is designed for porous metal plasticity with damage effects in the ductile material. The material is assume to form voids under loading, which grow, coalesce, then leads to crack formation and eventually failure. This process is a function of hydrostatic stress and the void volume fraction f_v. The yield function is established as follows:

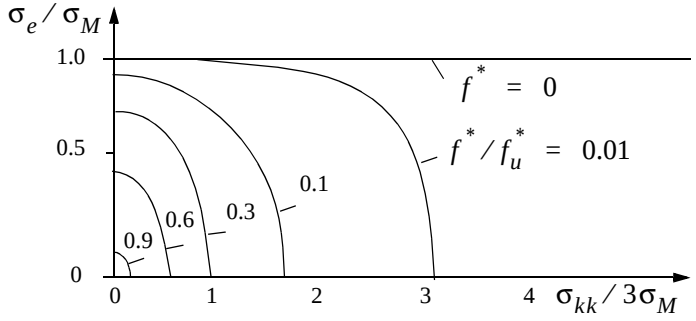
$$\frac{3J_2}{Y^2} + 2q_1 f_v^* \cosh\left(\frac{q_2 I_1}{2Y}\right) - \left[1 - (q_1 f_v^*)^2\right] = 0$$

in which Y denotes a yield stress of the fully dense matrix material, I_1 denotes the first invariant of stresses, and the modified void volume fraction f_v^* is computed by

$$f_v^* = f_v \qquad \text{if } f_v \le f_v^c$$

$$f_v^* = f_v^c + \left(\frac{f_v^u - f_v^c}{f_v^f - f_v^c}\right)(f_v - f_v^c) \qquad \text{if } f_v > f_v^c$$

where f_v^c is the critical void volume fraction, f_v^f is the void volume at failure and $f_v^u = 1/q_1$. The solid loses all stress carrying capability when the void volume fraction reaches f_v^f.



11. The evolution of damage as measured by void volume fraction is due to void nucleation and void growth. Void nucleation occurs by debonding of the second phase particles. The strain for nucleation depends on the particle sizes. Assuming a normal distribution of particle sizes, the void nucleation itself is modeled as a normal distribution in strains if nucleation is strain-controlled. If the void nucleation is assumed to be stress controlled in the matrix, a normal distribution is assumed in stresses. The void volume fraction changes due to the growth of existing voids and nucleation of new voids, i.e.,

$$\dot{f}_v = \dot{f}_{growth} + \dot{f}_{nucleation}$$

in which the void growth can be determined based on the compressibility of the material

$$\dot{f}_{growth} = (1 - f_v)\dot{\varepsilon}_{kk}^{p}$$

and the nucleation can be defined either as strain or stress-controlled with a normal distribution about the mean value. In case of strain-controlled nucleation, the rate is expressed as

$$\dot{f}_{nucleation} = \frac{f_v^n}{S\sqrt{2\pi}} Exp\left[-\frac{1}{2}\left(\frac{\bar{\varepsilon}_m^p - \varepsilon_n}{S}\right)^2\right] \dot{\bar{\varepsilon}}_m^p$$

where f_v^n is the volume fraction of void forming particles, $\bar{\varepsilon}_m^p$ denotes the effective plastic strain in the matrix material, and the void nucleation strain is assumed normally distributed with a mean value of ε_n and a standard deviation of S. In case of stress-controlled nucleation, the rate is expressed as

$$\dot{f}_{nucleation} = \frac{f_v^n}{S\sqrt{2\pi}} Exp\left[-\frac{1}{2}\left(\frac{\bar{\sigma} + \frac{1}{3}\sigma_{kk} - \sigma_n}{S}\right)^2\right] \left(\dot{\bar{\sigma}} + \frac{1}{3}\dot{\sigma}_{kk}\right)$$

If the size of the second phase particles are widely dispersed, the standard deviation would be larger than more uniform cases. A typical values for an engineering alloy as suggested by numerical experiments are set as default values for ε_n, S, and f_v^n.

12. The keywords may appear in any order. However, aniso, ORNL, press, and Gurson are mutually exclusive, and cannot coexist.
13. All the alphanumeric fields are recognizable by the first four letters.

Notes:

1. The Bulk Data fields denoted by N/A are different from blank fields. Those fields with N/A are not used currently, but the space is reserved in case of future additions. On the other hand, the blank fields which have no specifications are neither used nor reserved (any memory space) for future use. The Method field determines the options under MSC.Marc parameter PLASTICITY.

   The initial yield stress should be extracted from the table provided in the FID field. The yield stress-plastic strain function specified in TABLES1 under FID field is not a normalized function.

   The anisotropic material parameters Rij are equivalent to MSC.Marc input data as follows:

   R11 = YRDIR1
   R22 = YRDIR2
   R33 = YRDIR3
   R12 = YRSHR1
   R23 = YRSHR2
   R31 = YRSHR3

   The crack/crush capability in MSC.Marc may be combined with other isotropic material options.
2. The strain effect on the yield stress (Reffect) is specified under the model definition option STRAIN RATE in MSC.Marc.
3. This Bulk Data entry accommodates MSC.Marc's input data under the model definition options ISOTROPIC, ORTHOTROPIC, ANISOTROPIC, RATE EFFECTS, WORK HARD, DAMAGE and CRACK DATA as well as the parameter PLASTICITY.
4. The fields without default values can be left blank only if they are specifically permitted to be blank.

MATF (SOLs 600/700) Material Failure Model

Specifies failure model properties for linear elastic materials to be used for static, quasi static or transient dynamic analysis in MD Nastran Implicit Nonlinear (SOLs 600 and 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATF	MID	Criteria	Xt	Xc	Yt	Yc	Zt	Zc	
	Sxy	Syz	Szx	Find	Fxy	Fyz	Fzx	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	

Example:

MATF	100	134	2500.	4000.	2500.	4000.	2000.	3000.	
	4500.	4500.	4500.						

Field	Contents
MID	Identification number of a MAT1, MAT2, MAT8, MATORT or MAT9 entry. See Remarks **1.** and **2.** (Integer > 0; no Default)
Criteria	Select up to three failure criteria to be applied (combination of maximum three integers 1 through 5 with no embedded blanks, No Default) from: 1 for maximum stress criterion. See Remark **3.**. 2 for maximum strain criterion. See Remark **4.** 3 for Hill failure criterion. See Remark **5.** 4 for Hoffman failure criterion. See Remark **6.** 5 Tsai-Wu failure criterion. See Remark **7.** 6 Chang-Chang failure criterion. (SOL 700 only)
Xt	Maximum tensile stress in x-direction. (Real > 0 or blank)
Xc	Maximum compressive stress (absolute value) in x-direction (Real > 0; Default = Xt)
Yt	Maximum tensile stress in y-direction. (Real > 0; Default = Xt)
Yc	Maximum compressive stress (absolute value) in y-direction. (Real > 0; Default = Yt)
Zt	Maximum tensile stress in z-direction. (Real > 0; Default = Xt)

Field	Contents
Zc	Maximum compressive stress (absolute value) in z-direction. (Real > 0; Default = Zt)
Sxy	Maximum shear stress in xy-plane. (Real > 0 or blank)
Syz	Maximum shear stress in yz-plane. (Real > 0; Default = Sxy)
Szx	Maximum shear stress in zx-plane (Real > 0; Default = Sxy)
Find	Failure index. See Remarks 5.-7. (Real > 0; Default = 1)
Fxy	Interactive strength constant for xy-plane. (Real < 0; Default = $-\frac{1}{2}\sqrt{\frac{1}{X_t X_c}\frac{1}{Y_t Y_c}}$).
Fyz	Interactive strength constant for yz-plane. (Real < 0; Default = $-\frac{1}{2}\sqrt{\frac{1}{Y_t Y_c}\frac{1}{Z_t Z_c}}$)
Fzx	Interactive strength constant for zx-plane (Real < 0; Default = $-\frac{1}{2}\sqrt{\frac{1}{Z_t Z_c}\frac{1}{X_t X_c}}$)
Ext	Maximum tensile strain in x-direction. (Real > 0 or blank)
Exc	Maximum compressive strain (absolute value) in x-direction. (Real > 0; Default = Ext)
Eyt	Maximum tensile strain in y-direction. (Real > 0; Default = Ext)
Eyc	Maximum compressive strain (absolute value) in y-direction. (Real > 0; Default = Eyt)
zt	Maximum tensile strain in z-direction. (Real > 0; Default = Ext)
Ezc	Maximum compressive strain (absolute value) in z-direction. (Real > 0; Default = Ezt)
Gxy	Maximum shear strain in xy-plane. (Real > 0)
Gyz	Maximum shear strain in yz-plane. (Real > 0; Default = Gxy)
Gzx	Maximum shear strain in zx-plane. (Real > 0; Default = Gxy)

Remarks:

1. The MATF Bulk Data entry contains supplementary data for failure prediction of the elastic materials with the same MID. If this capability is used in nonlinear analysis, MATF will activate progressive failure process.
2. A progressive failure behavior for a linear elastic material is simulated by MATF Bulk Data entry. Failure occurs when any one of the specified failure criteria is satisfied. Upon failure, the elastic modulus reduces to 10% of the original modulus if there is only one value of modulus as in isotropic material or in a beam or truss element. If it pertains to an orthotropic material, all of the material moduli at the integration point are reduced to the lowest modulus specified. The behavior up to the failure point is linear elastic even if an elasto-plastic material is specified, which is followed by a nonlinear behavior for the post-failure analysis. If the initial yield stress is less than the allowable maximum stress, the failure criteria will be ignored. In case of the anisotropic material (MAT2, MATORT or MAT9), the progressive failure cannot be applied because no apparent elastic modulus exists in the material constants.
3. According to the Maximum Stress Criterion, the material fails when any of the stress components (9 components including 6 normal stress components in tension and compression, and three shear stress components) exceeds the maximum allowable stress:

$$|\sigma_{ij}| > Xt,\ Xc,\ Yt,\ \ldots,\ S_{xy},\ \ \ldots \text{etc.}$$

   where the indices (x, y, z or i, j) denote material coordinate direction.
4. According to the Maximum Strain Criterion, the material fails when any of the strain components (9 components including 6 normal strain components in tension and compression, and three shear strain components) exceeds the maximum allowable strain:

$$|\varepsilon_{ij}| > Ext,\ Exc,\ Ext,\ \ldots,\ Gxy,\ \ldots \text{etc.}$$

   where the indices (x, y, z or i, j) denote material coordinate direction.
5. According to the Hill Failure Criterion, there is no distinction between tensile and compressive behavior. The failure is determined based on

$$\frac{\sigma_x^2}{X^2} + \frac{\sigma_y^2}{Y^2} + \frac{\sigma_z^2}{Z^2} - \left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}\right)\sigma_x\sigma_y - \left(\frac{1}{Y^2} + \frac{1}{Z^2} - \frac{1}{X^2}\right)\sigma_y\sigma_z$$

$$-\left(\frac{1}{Z^2} + \frac{1}{X^2} - \frac{1}{Y^2}\right)\sigma_z\sigma_x + \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} > F_{ind}$$

in which $X, Y, Z, S_{xy}, S_{yz}, S_{zx}$ are maximum allowable stresses and F_{ind} is the failure index prescribed by the user.

6. The Hoffman Failure Criterion introduces distinction between tensile and compressive stresses to generalize the Hill Failure Criterion, i.e.,

$$C_x(\sigma_x - \sigma_y)^2 + C_y(\sigma_y - \sigma_z)^2 + C_z(\sigma_z - \sigma_x)^2 + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_x$$

$$+ \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_y + \left(\frac{1}{Z_t} - \frac{1}{Z_c}\right)\sigma_z + \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} > F_{ind}$$

with

$$C_x = \frac{1}{2}\left(\frac{1}{X_tX_c} + \frac{1}{Y_tY_c} - \frac{1}{Z_tZ_c}\right)$$

$$C_y = \frac{1}{2}\left(\frac{1}{Y_tY_c} + \frac{1}{Z_tZ_c} - \frac{1}{X_tX_c}\right)$$

$$C_z = \frac{1}{2}\left(\frac{1}{Z_tZ_c} + \frac{1}{X_tX_c} + \frac{1}{Y_tY_c}\right)$$

in which $X_t, X_c, Y_t, Y_c, Z_t, Z_c, S_{xy}, S_{yz}, S_{zx}$ are maximum allowable stresses and F_{ind} is the failure index, prescribed by the user.

7. The Tsai-Wu Failure Criterion is another generalization of the Hill Failure Criterion:

$$\left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_x + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_y + \left(\frac{1}{Z_t} - \frac{1}{Z_c}\right)\sigma_z + \frac{\sigma_x^2}{X_tX_c} + \frac{\sigma_y^2}{Y_tY_c} + \frac{\sigma_z^2}{Z_tZ_c}$$

$$+ \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} + 2F_{xy}\sigma_x\sigma_y + 2F_{yz}\sigma_y\sigma_z + 2F_{zx}\sigma_x\sigma_z > F_{ind}$$

in which $X_t, X_c, Y_t, Y_c, Z_t, Z_c, S_{xy}, S_{yz}, S_{zx}$ are maximum allowable stresses, F_{xy}, F_{yz}, F_{zx} are interactive strength constants, and F_{ind} is the failure index, prescribed by the user.

Notes:

1. This Bulk Data entry accommodates MSC.Marc's input data under the model definition option FAIL DATA.
2. This entry is also needed to implement element failure in SOL 700 using dytran-ls-dyna.
3. Failure indices are not completed by SOL 600 or 700. The element is actually allowed to fail and when it fails its stiffness is removed from the model during subsequent time on load steps.
4. Stress limits such as ST, SC, SS, X_t, X_c, Y_t, Y_c in the MAT1, MAT2 and MAT8 entries are not used in SOLs 600 or 700.

MATG (SOL 600) Gasket Material Properties

Specifies gasket material properties to be used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATG	MID	IDMEM	BEHAV	TABLD	TABLU1	TABLU2	TABLU3	TABLU4	
	TABLU5	TABLU6	TABLU7	TABLU8	TABLU9	TABLU10	YPRS	EPL	
	GPL	GAP	TABYPRS	TABEPL	TABGPL	TABGAP	N / A	N / A	

Example:

MATG	100	10	0	1001	1002	1003			
							100.	2500.	
	950.	0.0							

Field	Contents
MID	Material ID number. (Integer)
IDMEM	ID of MAT1 providing material behavior for membrane behavior. See Remarks. (Integer)
BEHAV	Behavior type (presently only type 0 is supported). (Integer)
TABLD	ID of a TABLES1 table providing loading path of the gasket (pressure versus displacement). See Remarks 1 and 2. (Integer)
TABLUi	ID of TABLES1 table providing unloading path(s) of the gasket (pressure versus displacement) can range from 1 to 10. If there is no unloading, no unloading tables need be entered. Leave fields blank for tables that are not required. See Remarks. (Integer)
YPRESS	Yield pressure. See Remark 3. (Real)
EPL	Tensile modulus (pressure per unit length). (Real)
GPL	Transverse shear modulus (force per unit area). (Real)
GAP	Initial gap (if present). (Real)
TABYPRS	ID of TABLES1 table associated with yield pressure (not presently used). (Integer)

Field	Contents
TABEPL	ID of TABLES1 table associated with tensile modulus (not presently used). (Integer)
TABGPL	ID of TABLES1 table associated with transverse shear modulus (not presently used). (Integer)
TABGAP	ID of TABLES1 table associated with initial gap (not presently used). (Integer)

Remarks:

1. MATG defines nonlinear properties in the thickness direction for compression only, designed for gasket-like materials. MATG has anisotropy only in the thickness direction, which is called normal anisotropy.
2. The MATG entry defines the compressive behavior in thickness. The thickness direction is the principal direction (3) in 3-dimensional solids and (2) for 2-dimensional solids (plane strain and axisymmetric elements). Since MATG material allows only normal anisotropy, linear properties in MAT1 are required for in-plane behavior.
3. The initial yield pressure should match a point in table TABLD.
4. The loading path for the gasket is always in compression. However, it starts from the origin to initial yield pressure (nonlinear elastic range) and continues with strain hardening slope into the plastic region. All the data points are specified in the first quadrant.
5. As many as 10 unloading paths may be defined in the thickness direction using TABLS1 in pressure vs. gasket closure distance as in the loading path. All the unloading paths must start from zero pressure and end at the loading path in the plastic region. Unloading behavior at undefined paths will be interpolated between two adjacent unloading paths. The last point of the last specified unloading path signifies full compression, which does not allow any further closure beyond the point.

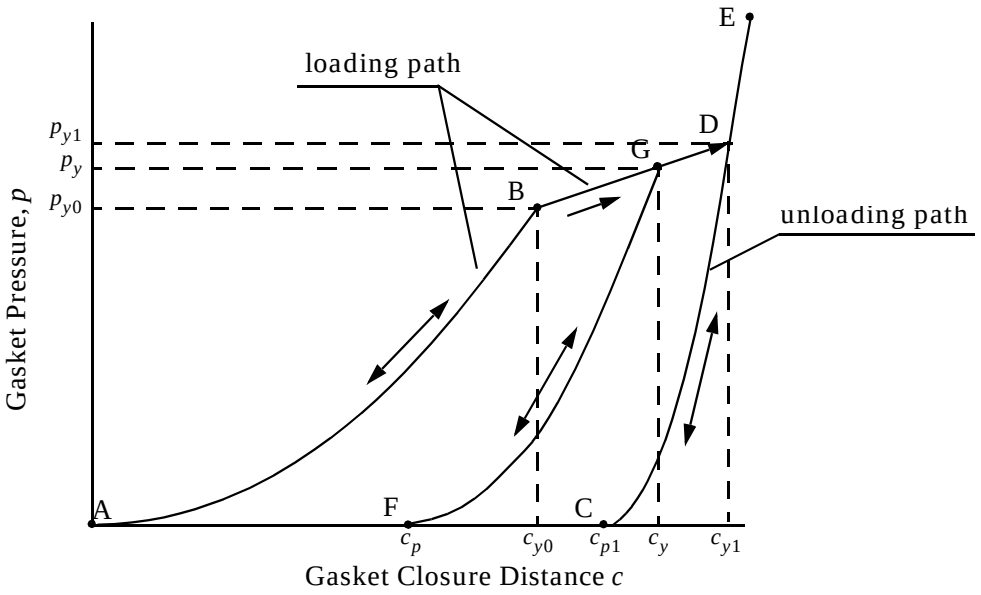


Figure 8-125 Pressure-closure Relation of a Gasket

6. See associated MATTG entry for temperature variation of these properties.
7. All continuation cards must be entered.
8. MID, IDMEM, BEHAV, TABLD, TABLU1, YPRS, EPL and GPL must be non-zero.
9. Each unloading curve must begin with gasket pressure of 0.0. Subsequent unloading curves must start with larger closure distances (when gasket pressure is 0.0) than previous unloading curves.
10. Points on loading and unloading curves must be defined in order of increasing gasket pressure.
11. MATG may be referenced by solid composite elements only.

MATHE (SOL 600) Hyperelastic Material Properties

Specifies hyperelastic (rubber-like) material properties for nonlinear (large strain and large rotation) analysis in MD Nastran Implicit Nonlinear (SOL 600) only.

Format 1 (default): Generalized Mooney-Rivlin model (Model = Mooney)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	N / A	K	RHO	Texp	Tref	GE	
	C10	C01	N / A	TAB1	TAB2	TAB3	TAB4	TABD	
	C20	C11	N / A	N / A	N / A	N / A			
	C30	N / A	N / A	N / A	N / A	N / A			

Format 2: Ogden Model or Hyperfoam model (Model = Ogden or foam)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	NOT	K	RHO	Texp	Tref	GE	
	Mu1	Alpha1	Beta1	TAB1	TAB2	TAB3	TAB4	TABD	
	Mu2	Alpha2	Beta2	Mu3	Alpha3	Beta3			
	Mu4	Alpha4	Beta4	Mu5	Alpha5	Beta5			

Format 3: Arruda-Boyce model or Gent model (Model = Aboyce or Gent)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	N / A	K	RHO	Texp	Tref	GE	
	NKT	N / E	Im	TAB1	TAB2	TAB3	TAB4	TABD	

Field	Contents
MID	Identification number of a MATHE entry. (Integer > 0; no Default)
Model	Select hyperelastic material model from (Character; Default = Mooney):
	Mooney for generalized Mooney-Rivlin hyperelastic model. See Remark 1.
	Ogden for Ogden hyperelastic model. See Remark 2.
	Foam for hyperfoam model. See Remark 3.
	Aboyce for Arruda-Boyce strain energy model. See Remark 4.
	Gent for Gent strain energy model. See Remark 5.

Field	Contents
NOT	Number of terms to be included in the curve fitting with experimental data. See Remark **6.** for Mooney model. (0 < Integer ≤ 5; Default = 2)
K	Specifies a bulk modulus. Not used in hyperfoam material. See Remark **7.** (Real > 0; Default = automatically set for nearly incompressible condition)
RHO	Mass density in original configuration. (Real; Default = 0.0)
Texp	Coefficient of thermal expansion. See Remark **8.** (Real; Default = 0.0)
Tref	Reference temperature at which the thermal expansion coefficient is measured. Tref is used only if the thermal expansion coefficient is temperature-dependent. (Real; Default = 0.0)
GE	Structural damping coefficient. (Real; Default = 0.0).
Cij	Material constants related to distortional deformation for generalized Mooney-Rivlin model. SOL 600 uses only five constants (C10, C01, C20, C11, and C30) and ignores others. (Real; Default = 1 for C10 and C01, 0 for other constants)
TAB1	Table identification number of a TABLES1 entry that contains simple tension-compression data to be used in the estimation of the material constants Cij, μ_k, α_k, and β_k. The x-values in the TABLES1 entry must be stretch ratios l/l_0 and y-values must be values of the engineering stress F/A_0. l_0 is the initial length and A_0 is the initial cross-sectional area. See Remark **9.** (Integer > 0 or blank)
TAB2	Table identification number of a TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Cij, μ_k, α_k, and β_k. The x-values in the TABLES1 entry must be stretch ratios l/l_0 and y-values must be values of the engineering stress F/A_0. l_0 is the initial length and & is the initial cross-sectional area. See Remark **9.** (Integer > 0 or blank)
TAB3	Table identification number of a TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Cij or μ_k, α_k, and β_k. The x-values in the TABLES1 entry must be values of the shear strain and y-values must be values of the engineering shear stress. (Integer > 0 or blank)

Field	Contents
TAB4	Table identification number of a TABLES1 entry that contains pure shear data to be used in the estimation of the material constants Cij, μ_k, α_k, and β_k. The x and y values in the TABLES1 entry must be stretch ratios $\lambda_1 = l/l_0$ and the values of the nominal stress F/A_0. l_0 and A_0 are the initial length and cross-sectional area, respectively, in the l-direction. See Remark **9.** (Integer > 0 or blank)
TABD	Table identification number of a TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constant K. The x-values in the TABLES1 entry must be values of the volume ration $J = \lambda^3$ where $\lambda = l/l_0$ is the stretch ratio in all three directions; y-values must be values of the pressure, assumed positive in compression. See Remark **9.** (Integer > 0 or blank)
Muk	Coefficients μ_k of the strain energy function for Ogden or hyperfoam material. See Remarks **2.** and **3.** (Real; Default = 0)
Alphak	Coefficients α_k of the strain energy function for Ogden or hyperfoam material. See Remarks **2.** and **3.** (Real; Default = 0)
Betak	Coefficients β_k of the strain energy function for hyperfoam material. These fields should be left blank for Ogden mode. See Remarks **2.** and **3.** (Real; Default = 0)
NTK	Material constant for Arruda-Boyce strain energy model. (Real > 0; Default = 1)
N / E	Material constant representing the number (N) of statistical links of the chain for Arruda-Boyce model; or tensile modulus (E) for Gent strain energy model. (Real > 0; Default = 1)
Im	Maximum first invariant for Gent strain energy model. (Real > 0; Default = 0)

Remarks:

1. The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$W(J, \bar{I}_1, \bar{I}_2) = \sum_{i+j=1}^{3} Cij(\bar{I}_1 - 3)^i(\bar{I}_2 - 3)^j + 4.5K(J^{1/3} - 1)^2$$

with

$$\bar{I}_1 = \bar{\lambda}_1^2 + \bar{\lambda}_2^2 + \bar{\lambda}_3^2$$

$$\bar{I}_2 = \bar{\lambda}_1^2\bar{\lambda}_2^2 + \bar{\lambda}_2^2\bar{\lambda}_3^2 + \bar{\lambda}_3^2\bar{\lambda}_1^2$$

where K and J are bulk modulus and volume ratio, respectively.

For small strains, the shear modulus G is related to the Mooney-Rivlin constants by

$$= 2(C_{10} + C_{01}) \text{ with } C_{01} \approx 0.25\ C_{10}$$

The model reduces to a Mooney-Rivlin material with only two constants (C10 and C01), and to a Neo-Hookean material with one constant (C10). The third order Mooney-Rivlin model in SOL 600 uses only five distortional constants (C10, C01, C11, C20, C30) and the bulk modulus K for volumetric deformation. This MATHE Bulk Data entry is provided only for MD Nastran Implicit Nonlinear (SOL 600). The hyperelastic material can be specified using MATHP Bulk Data entry in SOLs 106, 129, and 600.

2. For the Ogden material model, the strain energy function is

$$W = \sum_{k=1}^{5} \frac{\mu_k}{\alpha}\left(\bar{\lambda}_1^{\alpha_k} + \bar{\lambda}_2^{\alpha_k} + \bar{\lambda}_3^{\alpha_k} - 3\right) + 4.5K(J^{1/3} - 1)^2$$

where μ_k represents moduli, $\bar{\lambda}_i$ is the deviatoric stretch ratio defined as

$$\bar{\lambda}_i = J^{-\frac{1}{3}}\lambda_i$$

and J and K are the determinant of the deformation gradient and the bulk modulus, respectively. A two-term Ogden model is equivalent to a simple Mooney-Rivlin model

$$\mu_1 = 2C_{10} \quad \text{and} \quad \mu_2 = 2C_{01}$$

with $\alpha_1 = 2.$ and $\alpha_2 = 2.$

3. For the hyperfoam material model, the strain energy function is

$$W = \sum_{k=1}^{5} \frac{\mu_k}{\alpha_k}\left(\bar{\lambda}_1^{\alpha_k} + \bar{\lambda}_2^{\alpha_k} + \bar{\lambda}_3^{\alpha_k} - 3\right) + \sum_{k=1}^{5} \frac{\mu_k}{\beta_k}\left(1 - J^{\beta_k}\right)$$

where β_k represent additional constants for volumetric deformation.

4. For the Arruda-Boyce model, the strain energy function is

$$W = N_{KT}\left[\frac{1}{2}(\bar{I}_1 - 3) + \frac{1}{20N}\left(\bar{I}_1^2 - 9\right) + \frac{1}{1050N^2}\left(\bar{I}_1^3 - 27\right) + \frac{19}{7000N}\left(\bar{I}_1^4 - 81\right) + \frac{519}{67375N^4}\left(\bar{I}_1^5 - 243\right)\right] + 4.5K(J^{1/3} - 1)^2$$

with $\bar{I}_1 = \bar{\lambda}_1^2 + \bar{\lambda}_2^2 + \bar{\lambda}_3^2$

where N_{KT} is a material constant and N is a material parameter representing the number of statistical links of the material chain.

If the material test data are available from multiple experiments such as uniaxial and equi-biaxial tests, the Ogden model is more accurate in fitting experimental results. If only uniaxial tension data is available, the Arruda-Boyce model provides more accurate data fitting for multiple modes of deformation.

5. For the Gent model, the strain energy function is

$$W = -\frac{1}{6}EI_m \log\left[\frac{I_m}{I_m - \bar{I}_1 + 3}\right]$$

where E and I_m are tensile modulus and maximum first invariant, respectively.

6. The NOT field is used to curve fit the experimental data (not used for Mooney). The curve fitting is activated if TAB1, TAB2, TAB3, TAB4 and / or TABD are specified. The order of the polynomial in the Mooney model for curve fitting purpose; is controlled for each individual material constant by specifying 1. in the Cij and Di fields. Therefore, if TAB1, TAB2, TAB3, TAB4 and / or TABD are specified, then the curve fitting for Mooney will include the terms that have a unity in the fields corresponding to the active material constant.

7. Although the conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible, SOL 600 provides a compressible rubber model. Nearly incompressible material may be simulated with a large value of K. The default value for the Mooney-Rivlin model represents a nearly incompressible condition, which is $K = 10^4(C10 + C01)$. In the Ogden model, the default is $K = 0$. for incompressibility.

8. The thermal expansion coefficient is a secant value measured with respect to a temperature, Tref.

   The thermal strain is computed by $\varepsilon_{th} = \bar{\alpha}(T - T_0)$

   where T_0 is an initial temperature. The secant coefficient of thermal expansion is related to the instantaneous coefficient of thermal expansion by

$$\alpha = \frac{d\varepsilon_{th}}{dT} = \bar{\alpha} + \frac{d\bar{\alpha}}{dT}(T - T_0)$$

9. All the material constants may be obtained from least squares fitting of experimental data. One or more of four experiments (TAB1 to TAB4) may be used to obtain constants for distortional properties. The bulk modulus K may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 fields are blank, the material constants must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supersedes the manual input of the parameters.
10. All the alphanumeric fields are recognizable by the first four letters.
11. Enter NKT and N / E for Aboyce (leave Im blank). Enter N / E and Im for Gent (leave NKT blank).

Notes:

1. The structural damping constant GE is specified in MSC.Marc under the option DAMPING as a numerical damping γ, i.e.,

$$C = \alpha M + \left(\beta + \frac{2\gamma}{\omega}\right) K$$

   in which 2γ is equivalent to GE; α and β are equivalent to parameters ALPHA1 and ALPHA2 in MD Nastran.

2. This Bulk Data entry accommodates MSC.Marc's input data under the model definition options MOONEY, OGDEN, and FOAM as well as the parameter ELASTICITY. It also accommodates MATHP input data in MD Nastran.

MATHED (SOL 600) Damage Model Properties for Hyperelastic Materials

Specifies damage model properties for hyperelastic materials to be used for static, quasi static or transient dynamic analysis in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATHED	MID	method	Scale1	Relax1	Prop1	Scale2	Relax2	Prop2	
	Dinf	N/A	Scalem1	Relaxm1	N/A	Scalem2	Relaxm2		

Example:

MATHED	100		1.	0.3	0.5	0.8	0.4	1.0	

Field	Contents
MID	Identification number of MATHE entry. See Remark **1.** (Integer > 0)
Method	Select a method for damage calculation (Character) from: **Multip** for multiplicative decomposition. (Default) **Additiv** for additive decomposition. See Remarks **2.** and **3.**
Scale1,2	Scaling factor d_n for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark **3.** (Real $\geq$ 0.0, < 1.0; Default = 0)
Relax1,2	Relaxation rate η_n for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark **3.** (Real $\geq$ 0 or blank)
Prop1,2	Proportionality factor δ_n for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark **3.** See Remark **4.** (Real > 0; Default = 1)
Dinf	d^{∞} described in the equations of Remark **3.** If Blank, the program will compute it; however even if set, the program usually calculates it. In most cases d^{∞} =1.0-scale1-scale2. (Real > 0; or blank)
Scalem1,2	Scaling factor d_m for m=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark **3.** (Real $\geq$ 0; Default = 0)

Field	Contents
Relaxm1,2	Relaxation rate λ_m for m=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark **3.** (Real > 0 or blank)
KIND	Damage model type. (Integer; no Default=5) 0 – Gurson Model, with no nucleation. 1 – Gurson Model, with plastic-strain controlled nucleation. 2 – Gurson Model, with stress controlled nucleation 3 – Gurson Model, with nucleation controlled by the UVOIDN user subroutine. (See Remark **5.**) 4 – Elastomeric damage model; additive decomposition of the Kachanov factor. 5 – Elastomeric damage model; multiplicative decomposition of the Kachanov factor. 6 – Elastomeric damage model controlled by the UELDAM user subroutine.(See Remark **5.**) 9 – Simplified damage model, damage applied to yield stress uses UDAMAG user subroutine.(See Remark **5.**) 10 – Simplified model, damage applied to yield stress and Young's modulus uses UDAMAG user subroutine. (See Remark **5.**)

Remarks:

1. The MATHE Bulk Data entry with the same MID must exist for MATHED to be effective.

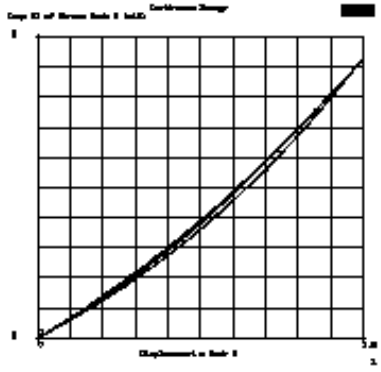
   The damage capability is available for all the elastomeric materials (Mooney-Rivlin, Ogden, Gent, Arruda-Boyce).

2. Under repeated application of loads, elastomers undergo damage by mechanisms involving chain breakage, multi-chain damage, micro-void formation, and micro-structural degradation due to detachment of filler particles from the network entanglement. The damage model for elastomeric materials is based on the undamaged strain energy function W_0, multiplied by a Kachanov damage factor, K, i.e.,

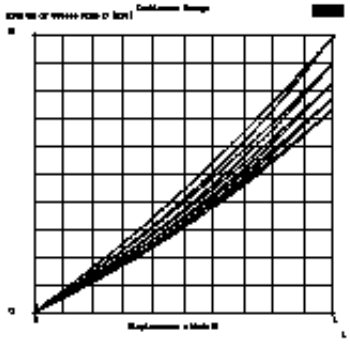
   $$W = K(\alpha, \beta)W_0$$

   where α and β are parameters for discontinuous and continuous damage models, respectively. Two types of phenomenological models, discontinuous and continuous, exist to simulate the damage. The discontinuous damage model simulates the "Mullins' effect," which

involves a loss of stiffness (represented by a parameter α) below the previously attained maximum strain. The higher the maximum attained strain the larger the loss of stiffness is. There is a progressive stiffness loss with increasing maximum strain amplitude. Most of the stiffness loss takes place in the first few cycles provided the maximum strain level is not increased. This phenomenon is observed in both filled as well as natural rubber although the higher level of carbon black particles increases the hysteresis and the loss of stiffness. The continuous damage model (Miehe's formulation) can simulate the damage accumulation for strain cycles for which the values of effective energy is below the maximum attained value of the past history. The evolution of continuous damage parameter is governed by the arc-length of the effective strain energy, represented by a parameter β.



Discontinuous Damage



Continuous Damage

3. Both the continuous damage as well as the discontinuous damage can be modeled by a cumulative Kachanov factor in multiplicative or additive decomposition form.

   For multiplicative decomposition

$$K(\alpha, \beta) = d^{\infty} + \sum_{n=1}^{2} d_n \exp\left(-\frac{\alpha + \delta_n \beta}{\eta_n}\right)$$

   For additive decomposition

$$K(\alpha, \beta) = d^{\infty} + \sum_{n=1}^{2} d_n \exp\left(-\frac{\alpha}{\eta_n}\right) + \sum_{m=1}^{2} d_m \exp\left(-\frac{\beta}{\lambda_m}\right)$$

where d_n, δ_n, η_n, d_m, and λ_m are constants specified by the user, and d^{∞} is calculated by the program such that the Kachanov factor assumes a value of unity at zero damage if left blank.

4. The proportionality factor is not used by additive decomposition which requires the continuation fields to include the continuous damage model.
5. User subroutines must be called out using PARAM,MARCUSUB,CHAR where CHAR is a character variable such as UDAMAG.

Note:

1. This Bulk Data entry accommodates MSC.Marc's input data under the model definition option DAMAGE.

MATHP Hyperelastic Material Properties

Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).

Format:

1	2	3	4	5	6	7	8	9	10
MATHP	MID	A10	A01	D1	RHO	AV	TREF	GE	
		NA	ND						
	A20	A11	A02	D2					
	A30	A21	A12	A03	D3				
	A40	A31	A22	A13	A04	D4			
	A50	A41	A32	A23	A14	A05	D5		
	TAB1	TAB2	TAB3	TAB4				TABD	

Field	Contents
MID	Identification number of a MATHP entry. (Integer > 0; No Default)
Aij	Material constants related to distortional deformation. (Real; Default = 0.0)
Di	Material constants related to volumetric deformation. [Real $\geq$ 0; Default for D1 is $10^3 \cdot (A10 + A01)$; Default for D2 through D5 is 0.0.]
RHO	Mass density in original configuration. (Real; Default = 0.0)
AV	Coefficient of volumetric thermal expansion. (Real; Default = 0.0)
TREF	Reference temperature. See "MAT1" on page 1663. (Real; Default = 0.0)
GE	Structural damping element coefficient. (Real; Default = 0.0)
NA	Order of the distortional strain energy polynomial function. (0 < Integer $\leq$ 5; Default = 1)
ND	Order of the volumetric strain energy polynomial function. (0 < Integer $\leq$ 5; Default = 1)
TAB1	Table identification number of a TABLES1 entry that contains simple tension/compression data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios l/l_0 and yi values must be values of the engineering stress F/A_0. Stresses are negative for compression and positive for tension. (Integer > 0 or blank)

Field	Contents
TAB2	Table identification number of a TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios l/l_0. yi values must be values of the engineering stress F/A_0. l is the current length, F is the current force, l_0 is the initial length and A_0 is the cross-sectional area. In the case of pressure of a spherical membrane, the engineering stress is given by $Pr_0\lambda^2/2t_0$ where P = current value of the pressure and r_0, t_0 = initial radius and thickness. (Integer > 0 or blank)
TAB3	Table identification number of a TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be values of the shear tangent γ and yi values must be values of the engineering shear stress F/A_0. (Integer > 0 or blank)
TAB4	Table identification number of a TABLES1 entry that contains pure shear data to be used in the estimation of the material constants Aij. xi and yi values in the TABLES1 entry must be stretch ratios $\lambda_1 = l/l_0$ and values of the nominal stress F/A_0. l is the current length, F is the current force, l_0 and A_0 are the initial length and cross-sectional area, respectively in the 1-direction. (Integer > 0 or blank)
TABD	Table identification number of a TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constants Di. xi values in the TABLES1 entry must be values of the volume ratio $J = \lambda^3$ where $\lambda = l/l_0$ is the stretch ratio in all three directions; yi values must be values of the pressure, assumed positive in compression. (Integer > 0 or blank)

Remarks:

1. The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$U(J, \dot{I}_1, \dot{I}_2) = \sum_{i+j=1}^{NA} Aij(\dot{I}_1 - 3)^i(\dot{I}_2 - 3)^j + \sum_{i=1}^{ND} Di(J - 1 - AV(T - T_0))^{2i}$$

A00 = 0

where $\bar{I}_1$ and $\bar{I}_2$ are the first and second distortional strain invariants, respectively; $J = \det F$ is the determinant of the deformation gradient; and D1 = K and 2(A10 + A01) = G at small strains, in which K is the bulk modulus and G is the shear modulus. The model reduces to a Mooney-Rivlin material if NA=1 and to a Neo-Hookean material if NA = 1 and A01 = 0.0. See Remark **2.** For Neo-Hookean or Mooney-Rivlin materials no continuation entry is required. T is the current temperature and T_0 is the initial temperature.

2. Conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible. Full incompressibility is not presently available but may be simulated with a large enough value of D1. A value of D1 higher than $10^3 \cdot (A10 + A01)$ is, however, not recommended.
3. Aij and Di are obtained from least squares fitting of experimental data. One or more of four experiments (TAB1 to TAB4) may be used to obtain Aij. Di may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 are blank, Aij must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supercedes the manual input of the parameters.
4. IF ND=1 and a nonzero value of D1 is provided or is obtained from experimental data in TABD, then the parameter estimation of the material constants Aij takes compressibility into account in the cases of simple tension/compression, equibiaxial tension, and general biaxial deformation. Otherwise, full incompressibility is assumed in estimating the material constants.
5. See Chapters "**Hyperelastic Elements**" on page 220, "**Hyperelastic Material**" on page 281 and "**Hyperelastic Material**" on page 586 of the *MSC.Nastran Reference Manual* for further details.

MATORT (SOL 600) Elastic 3D Orthotropic Material Properties

Specifies elastic orthotropic material properties for 3-dimensional and plane strain behavior for linear and nonlinear analyses in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATORT	MID	E1	E2	E3	NU12	NU23	NU31	RHO	
	G12	G23	G31	A1	A2	A3	TREF	GE	
	IYLD	IHARD	SY	Sornl	Y1	Y2	Y3	N/A	
	Yshr1	Yshr2	Yshr3	N/A	N/A	N/A	N/A	N/A	

Example:

MATORT	100	3.e6	2.8e7	1.5e5	0.25				

Field	Contents
MID	Identification number of a MATORT entry, referenced on PSOLID or PLSOLID entry. (Integer > 0; No default, must be entered).
E1	Modulus of elasticity in longitudinal or 1-direction. See Remark 1. (Real > 0)
E2	Modulus of elasticity in lateral direction or 2-direction. (Real > 0; no Default, must be entered)
E3	Modulus of elasticity in thickness direction or 3-direction. (Real > 0; no Default, must be entered)
NU12	Poisson's ratio $\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1-direction. See Remark 2. (Real, no Default, must be entered,)
NU23	Poisson's ratio $\varepsilon_3/\varepsilon_2$ for uniaxial loading in 2-direction. (Real; no Default, must be entered)
NU31	Poisson's ratio $\varepsilon_1/\varepsilon_3$ for uniaxial loading in 3-direction. (Real; Default = NU23)
RHO	Mass density. (Real; Default = 0.0)
G12	Shear modulus in plane 1-2.See Remark 3. (Real > 0, no Default, must be entered,)

Field	Contents
G23	Shear modulus in plane 2-3. (Real > 0., no Default, must be entered)
G31	Shear modulus in plane 3-1. (Real > 0; no Default, must be entered)
Ai	Coefficient of thermal expansion in i-direction (Real; Default = 0.0).
TREF	Reference temperature at which the thermal expansion coefficient is measured. TREF is used only if the thermal expansion coefficient is temperature-dependent. (Real; Default = 0.0)
GE	Structural damping coefficient. (Real; Default = 0.0).
IYLD	Integer pertaining to one of the following yield criteria: (Integer) -1 = Elastic 1 = von Mises (Default) 2 = Normal ORNL 3 = 2 ¼ Cr-Mo ORNL 4 = Reversed plasticity ORNL 5 = Full alpha reset ORNL 6 = Generalized plasticity model 7 = Hill (1948) yield 8 = Barlat (1991) yield 9 = Viscoplasticity through user subroutine UVSCPL
IHARD	Integer pertaining to one of the following work hardening rules: (Integer) 1 = Isotropic (Default) 2 = Kinematic 3 = Combined Isotropic/Kinematic
SY	Equivalent (von Mises) tensile yield stress. (Real > 0.0 or blank; default =0.0)
SORNL	For ORNL only, 10th cycle equivalent yield stress. (real >0.0 or blank; default =0.0)
Y1	Hill's yield stress ratio in direction 1. (Real > 0.0 or blank; default =0.0)
Y2	Hill's yield stress ratio in direction 2. (Real > 0.0 or blank; default =0.0)
Y3	Hill's yield stress ratio in direction 3. (Real > 0.0 or blank; default =0.0)
Yshr1	Hill's yield shear stress ratio in direction 1. (Real > 0.0 or blank; default =0.0)

Field	Contents
Yshr2	Hill's yield shear stress ratio in direction 2. (Real > 0.0 or blank; default =0.0)
Yshr3	Hill's yield shear stress ratio in direction 3. (Real > 0.0 or blank; default =0.0)

Remarks:

1. The MATORT Bulk Data entry is used only in MD Nastran Implicit Nonlinear (SOL 600). All the material constants are specified in the orthotropic material coordinates in 1, 2, and 3 direction.
2. In general, v_{12} is not the same as v_{21}, but they are related by $v_{ij}/E_i = v_{ji}/E_j$. Furthermore, material stability requires that

$$E_i > v_{ij}^2 E_j$$

and $1 - v_{12}v_{21} - v_{23}v_{32} - v_{31}v_{13} - 2v_{21}v_{32}v_{13} > 0$.

3. It may be difficult to find all nine orthotropic constants. In some practical problems, the material properties may be reduced to normal anisotropy in which the material is isotropic in a plane, e.g., in plane 1-2 and has different properties in the direction normal to the plane 1-2. In the plane of isotropy, the properties are reduced to

$$\begin{aligned} E_1 &= E_2 = E_p \\ v_{31} &= v_{32} = v_{np} \\ v_{13} &= v_{23} = v_{pn} \\ G_{13} &= G_{23} = G_n \end{aligned}$$

with $v_{np}/E_n = v_{pn}/E_p$ and $G_p = \frac{E_p}{2(1+v_p)}$.

There are five independent material constants for normal anisotropy (e.g., $E_p, E_n, v_p, v_{np}, G_n$).

In case the material has a planar anisotropy, in which the material is orthotropic only in a plane, the elastic constants are reduced to seven, e.g., $E_1, E_2, E_3, v_{12}, G_{12}, G_{23}, G_{31}$.

4. If Y2 and / or Y3 are blank, then Y1 is assumed if entered. If Yshr2 and / or Yshr3 are blank, Yshr1 is assumed if entered.5.

5. Do not enter values for SY, SORNL, Y1, Y2, Y3, YSHR1, YSHR2 or YSHR3 unless plasticity is to be taken into account.

Notes:

1. The structural damping constant GE is specified in SOL 600 and MSC.Marc under the option DAMPING as a numerical damping γ, i.e.,

$$C = \alpha M + \left(\beta + \frac{2\gamma}{\omega}\right) K$$

   in which 2γ is equivalent to GE.

2. This Bulk Data entry accommodates MSC.Marc's input data under the model definition option ORTHOTROPIC.

MATRIG (SOL 700) Rigid-Body Properties

Defines the properties of a rigid body.

Format:

1	2	3	4	5	6	7	8	9	10
MATRIG	MID	RHO	E	NU	MASS	XC	YC	ZC	
	IXX	IXY	IXZ	IYY	IYZ	IZZ	CID	COG-FL	
	VX	VY	VZ	WX	WY	WZ			
	XC-LOCAL	YC-LOCAL	ZC-LOCAL	CMO	CON1	CON2			

Example:

MATRIG	7	7850.	210.E9	0.3	750	0.0	7.0	-3.0	
	17.0	13.2	14.3	20.9	15.7	10.0	12		
			13.3						

Field	Contents	Type	Default
MID	Unique material number.	I > 0	Required
RHO	Density	R > 0	1.0
E	Young's modulus	R > 0	1.0
NU	Poisson's ratio	0.0 R < 0.5	0.2
MASS	Mass of the rigid body.	R > 0.0	See Remark **2.**
XC, YC, ZC	x, y, and z coordinates of the center of gravity.	R	See Remark **6.**
IXX, IXY, IXZ, IYY, IYZ, IZZ	Inertia tensor of the rigid body about the center of gravity.	R	See Remark **6.**
CID	Number of a coordinate system in which the inertia tensor and the center of gravity are defined.	I > 0	See Remarks **7.** and **8.**
VX, VY, VZ	Initial translational velocity of the center of gravity in the basic coordinate system.	R	0.0

Field	Contents	Type	Default
WX, WY, WZ	Initial rotational velocities of the rigid body about the center of gravity in the basic coordinate system.	R	0.0
XC-LOCAL YC-LOCAL ZC-LOCAL	x, y, and z local coordinates of the center of gravity	R	See Remark **8.**
CMO	Center of mass constraint option, CMO: EQ. + 1.0: constraints applied in global directions, EQ. 0.0: no constraints EQ. -1.0: constraints applied in local directions (SPC constraint)	R	0.0
CON1	First constraint parameter: If CMO = +1.0, then specify global translation constraint: EQ.0: no constraints, EQ.1: constrained x displacement, EQ.2: constrained y displacement, EQ.3: constrained z displacement, EQ.4: constrained x and y displacements, EQ.5: constrained y and z displacements, EQ.6: constrained z and x displacements, EQ.7: constrained x, y, and z displacements. If CMO = -1.0, then specify local coordinate system ID. This coordinate system is fixed in time.	I	0

Field	Contents	Type	Default
CON2	Second constraint parameter: If CMO = +1.0, then specify global rotational constraint: EQ.0: no constraints, EQ.1: constrained x rotation, EQ.2: constrained y rotation, EQ.3: constrained z rotation, EQ.4: constrained x and y rotations, EQ.5: constrained y and z rotations, EQ.6: constrained z and x rotations, EQ.7: constrained x, y, and z rotations. If CMO = -1.0, then specify local (SPC) constraint: EQ.000000 no constraint, EQ.100000 constrained x translation, EQ.010000 constrained y translation, EQ.001000 constrained z translation, EQ.000100 constrained x rotation, EQ.000010 constrained y rotation, EQ.000001 constrained z rotation. Any combination of local constraints can be achieved by adding the number 1 into the corresponding column.	I	0

Remarks:

1. All coordinates are defined in the basic coordinate system.
2. If MASS is blank or zero, the mass will be calculated from the density and the geometry of the mesh defining the rigid body.
3. The continuation lines are not required.
4. The MATRIG definition is used instead of a MATDn definition and is referenced by properties PSOLIDn, PSHELLn, PBAR, and PBEAMn. Different properties can refer to the same MATRIG entry forming one rigid body. The MATRMERG or MATRMRG1 option (see PARAM,MATRM(E)RG(1)) can be used for merging different MATRIG and RBE2-FULLRIG definitions into one single rigid body.

5. If the fields VX, VY, VZ, WX, WY, and WZ are blank, then the initial conditions of the rigid body are calculated from the initial velocities on the TIC and TIC1 entries referring to grid points attached to the rigid body. The net initial conditions are the average of those for all the grid points attached to the rigid body.
   If the initial conditions are set using the VX, VY, VZ, WX, WY, and WZ fields, the TIC and TIC1 entries referring to grid points attached to the rigid body are ignored.
6. If the inertia tensor or the coordinates of the center of gravity are undefined, then they will be computed from the density or mass and the geometry of the mesh defining the rigid body.
7. The inertia tensor can only be defined in a local rectangular coordinate system. If the entry for a local coordinate system is left blank, then the inertia tensor is defined in the global coordinate system.
8. The center of gravity can be defined in a local rectangular coordinate system (CID). However, XC YC ZC (x, y, and z coordinates of the center of gravity in the basic coordinate system) should be left blank when XC-LOCAL YC-LOCAL ZC-LOCAL (x, y, and z coordinates of the center of gravity in a local coordinate system) defined.

MATS1 Material Stress Dependence

Specifies stress-dependent material properties for use in applications involving nonlinear materials. This entry is used if a MAT1, MAT2 or MAT9 entry is specified with the same MID in a nonlinear solution sequence (SOLs 106, 129 and 400).

Format:

1	2	3	4	5	6	7	8	9	10
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1	LIMIT2	

Example:

MATS1	17	28	PLASTIC	0.0	1	1	2.+4		

Field	Contents
MID	Identification number of a MAT1, MAT2, or MAT9 entry. (Integer > 0)
TID	Identification number of a TABLES1 or TABLEST entry. If H is given, then this field must be blank. See Remark **3.** (Integer $\geq$ 0 or blank)
TYPE	Type of material nonlinearity. See Remarks. (Character: "NLELAST" for nonlinear elastic or "PLASTIC" for elastoplastic.)
H	Work hardening slope (slope of stress versus plastic strain) in units of stress. For elastic-perfectly plastic cases, H = 0.0. For more than a single slope in the plastic range, the stress-strain data must be supplied on a TABLES1 entry referenced by TID, and this field must be blank. See Remark **2.** (Real)
YF	Yield function criterion, selected by one of the following values (Integer): 1 = von Mises (Default) 2 = Tresca 3 = Mohr-Coulomb 4 = Drucker-Prager
HR	Hardening Rule, selected by one of the following values (Integer): 1 = Isotropic (Default) 2 = Kinematic 3 = Combined isotropic and kinematic hardening

Field	Contents
LIMIT1	Initial yield point. See **Table 8-1**. (Real)
LIMIT2	Internal friction angle, measured in degrees, for the Mohr-Coulomb and Drucker-Prager yield criteria. See **Table 8-1**. ($0.0 \leq$ Real < 45.0)

Table 8-1 Yield Functions Versus LIMIT1 and LIMIT2

Yield Function (YF)	LIMIT1	LIMIT2
von Mises (1) or Tresca (2)	Initial Yield Stress in Tension, Y_1	Not used
Mohr-Coulomb (3) or Drucker-Prager (4)	2*Cohesion, *2c* (in units of stress)	Angle of Internal Friction ϕ (in degrees)

Remarks:

1. If TYPE = “NLELAST”, then MID may refer to a MAT1 entry only. Also, the stress-strain data given in the TABLES1 entry will be used to determine the stress for a given value of strain. The values H, YF, HR, LIMIT1, and LIMIT2 will not be used in this case.

   Thermoelastic analysis with temperature-dependent material properties is available for linear and nonlinear elastic isotropic materials (TYPE = “NLELAST”) and linear elastic anisotropic materials. Four options of constitutive relations exist. The relations appear in **Table 8-2** along with the required Bulk Data entries.

Table 8-2 Constitutive Relations and Required Material Property Entries

Constitutive Relation	Required Bulk Data Entries
$\{\sigma\} = [G_e(T)]\{\varepsilon\}$	MAT1 and MATT1
$\{\sigma\} = \frac{\bar{E}(\sigma, \varepsilon)}{E}[G_e(T)]\{\varepsilon\}$	MAT1, MATT1, MATS1, and TABLES1
$\{\sigma\} = \frac{\bar{E}(T, \sigma, \varepsilon)}{E}[G_e]\{\varepsilon\}$	MAT1, MATS1, TABLEST, and TABLES1
$\{\sigma\} = \frac{\bar{E}(T, \sigma, \varepsilon)}{E}[G_e(T)]\{\varepsilon\}$	MAT1, MATT1, MATS1, TABLEST, and TABLES1

In **Table 8-2** $\{\sigma\}$ and $\{\varepsilon\}$ are the stress and strain vectors, $[G_e]$ the elasticity matrix, $\bar{E}$ the effective elasticity modulus, and E the reference elasticity modulus.

2. If TYPE = “PLASTIC”, the elastic stress-strain matrix is computed from MAT1, MAT2, or MAT9 entry, and then the isotopic plasticity theory is used to perform the plastic analysis. In this case, either the table identification TID or the work hardening slope H may be specified, but not both. If the TID is omitted, the work hardening slope H must be specified unless the material is perfectly plastic. The plasticity modulus (H) is related to the tangential modulus (E_T) by

$$H = \frac{E_T}{1 - \frac{E_T}{E}}$$

where E is the elastic modulus and $E_T = dY/d\varepsilon$ is the slope of the uniaxial stress-strain curve in the plastic region. See **Figure 8-126**.

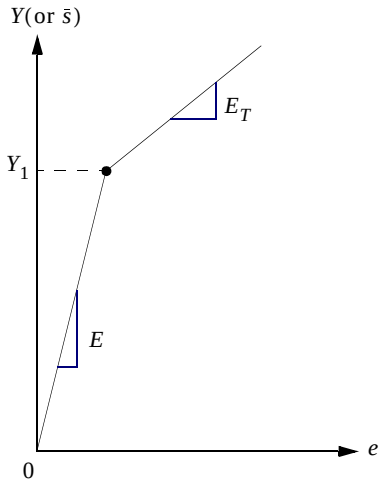


Figure 8-126 Stress-Strain Curve Definition When H Is Specified in Field 5

3. If TID is given, TABLES1 entries (Xi, Yi) of stress-strain data (ε_k, Y_k) must conform to the following rules (see **Figure 8-127**):

- If TYPE = “PLASTIC”, the curve must be defined in the first quadrant. The first point must be at the origin (X1 = 0, Y2 = 0) and the second point (X2, Y2) must be at the initial yield point (Y_1 or 2*c*) specified on the MATS1 entry. The slope of the line joining the origin to the yield stress must be equal to the value of *E*. Also, TID may not reference a TABLEST entry.
- If TYPE = “NLELAST”, the full stress-strain curve ($-\infty < x < \infty$) may be defined in the first and the third quadrant to accommodate different uniaxial compression data. If the curve is defined only in the first quadrant, then the curve must start at the origin (X1 = 0.0, Y = 0.0) and the compression properties will be assumed identical to tension properties.

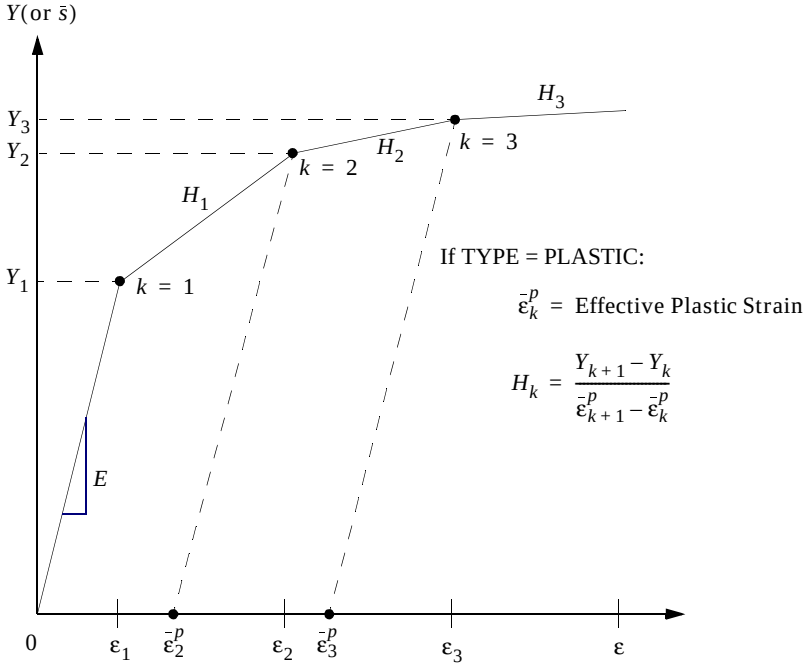


Figure 8-127 Stress-Strain Curve Definition When TID Is Specified in Field 3

MATT1 Isotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT1 entry fields via TABLEMi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT1	MID	T(E)	T(G)	T(NU)	T(RHO)	T(A)		T(GE)	
	T(ST)	T(SC)	T(SS)						

Example:

MATT1	17	32				15			
	52								

Field	Contents
MID	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
T(E)	Identification number of a TABLEMi entry for the Young's modulus. (Integer ≥ 0 or blank)
T(G)	Identification number of a TABLEMi entry for the shear modulus. (Integer ≥ 0 or blank)
T(NU)	Identification number of a TABLEMi entry for the Poisson's ratio. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for the mass density. (Integer ≥ 0 or blank)
T(A)	Identification number of a TABLEMi entry for the thermal expansion coefficient. See Remark 4. (Integer or blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer ≥ 0 or blank)
T(ST)	Identification number of a TABLEMi entry for the tension stress limit. (Integer ≥ 0 or blank)
T(SC)	Identification number of a TABLEMi entry for the compression limit. (Integer ≥ 0 or blank)
T(SS)	Identification number of a TABLEMi entry for the shear limit. (Integer ≥ 0 or blank)

Remarks:

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E is modified by TABLEMi 32, A is modified by TABLEMi 15, and ST is modified by TABLEMi 52. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT1 entry.
2. Any quantity modified by this entry must have a value on the MAT1 entry. Initial values of E, G, or NU may be supplied according to Remark 3 on the MAT1 entry.
3. Table references must be present for each item that is temperature dependent. For example, it is not sufficient to give table references only for fields 3 and 4 (Young's modulus and shear modulus) if Poisson's ratio is temperature dependent.
4. For a nonlinear static analysis of a composite element with the parameter COMPMATT set to ON, if the TABLEMi ID for the coefficient of thermal expansion is negative, the TABLEMi values will be interpreted as the thermal strain $\varepsilon(T)$ rather than the coefficient of thermal expansion $\alpha(T)$.
5. The continuation entry is not used by SOL 600 or SOL 700. For SOL 600, see MATTEP.
6. T(GE) is not used by SOL 600 or SOL 700.

MATT2 Anisotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT2 entry fields via TABLEMj entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT2	MID	T(G11)	T(G12)	T(G13)	T(G22)	T(G23)	T(G33)	T(RHO)	
	T(A1)	T(A2)	T(A3)		T(GE)	T(ST)	T(SC)	T(SS)	

Example:

MATT2	17	32					15		
	62								

Field	Contents
MID	Material property identification number that matches the identification number on a MAT2 entry. (Integer > 0)
T(Gij)	Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer $\geq$ 0 or blank)
T(RHO)	Identification number of a TABLEMk entry for the mass density. (Integer $\geq$ 0 or blank)
T(Ai)	Identification number of a TABLEMk entry for the thermal expansion coefficient. See Remark **3.** (Integer or blank)
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer $\geq$ 0 or blank)
T(ST)	Identification number of a TABLEMk entry for the tension stress limit. (Integer $\geq$ 0 or blank)
T(SC)	Identification number of a TABLEMk entry for the tension compression limit. (Integer $\geq$ 0 or blank)
T(SS)	Identification number of a TABLEMk entry for the tension shear limit. (Integer $\geq$ 0 or blank)

Remarks:

1. Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT2 entry referenced in field 2. The value in a particular field of the MAT2 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, G11 is modified by TABLEMk 32, G33 is modified by TABLEMk 15, and A1 is modified by TABLEMk 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT2 entry.
2. Any quantity modified by this entry must have a value on the MAT2 entry.
3. For a nonlinear static analysis of a composite element with the parameter COMPMATT set to ON, if the TABLEMi ID for the coefficient of thermal expansion is negative, the TABLEMi values will be interpreted as the thermal strain $\varepsilon(T)$ rather than the coefficient of thermal expansion $\alpha(T)$.

MATT3 CTRIAX6 Material Temperature Dependence

Specifies temperature-dependent material properties on MAT3 entry fields via TABLEMi entries that are temperature dependent. Applies to the CTRIAX6 element only.

Format:

1	2	3	4	5	6	7	8	9	10
MATT3	MID	T(EX)	T(ETH)	T(EZ)	T(NUXTH)	T(NUTHZ)	T(NUZX)	T(RHO)	
			T(GZX)	T(AX)	T(ATH)	T(AZ)		T(GE)	

Example:

MATT3	23	32		15					
			62						

Field	Contents
MID	Material property identification number that matches the identification number on MAT3 entry. (Integer > 0)
T(EX) T(ETH) T(EZ)	Identification number of a TABLEMi entry for the Young's modulus in the *x*, θ, and *z* directions. (Integer ≥ 0 or blank)
T(GZX)	Identification number of a TABLEMi entry for the shear modulus. (Integer ≥ 0 or blank)
T(NUXTH) T(NUTHZ) T(NUZX)	Identification number of a TABLEMi entry for the Poisson's ratio in the *x*θ, *z*θ, and *zx* directions. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for the mass density. (Integer ≥ 0 or blank)
T(AX) T(ATH) T(AZ)	Identification number of a TABLEMi entry for the thermal expansion coefficients in the *x*, θ, and *z* directions. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer ≥ 0 or blank)

Remarks:

1. Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT3 entry referenced in field 2. The value recorded in a particular field of the MAT3 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, EX is modified by TABLEMi 32, EZ is modified by TABLEMi 15, and GZX is modified by TABLEMi 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT3 entry.
2. Any quantity modified by this entry must have a value on the MAT3 entry.

MATT4 Thermal Material Temperature Dependence

Specifies table references for temperature-dependent MAT4 material properties.

Format:

1	2	3	4	5	6	7	8	9	10
MATT4	MID	T(K)	T(CP)		T(H)	T(μ)	T(HGEN)		

Example(s):

MATT4	2	10	11						

Field	Contents
MID	Identification number of a MAT4 entry that is temperature dependent. (Integer > 0)
T(K)	Identification number of a TABLEMj entry that gives the temperature dependence of the thermal conductivity. (Integer ≥ 0 or blank)
T(CP)	Identification number of a TABLEMj entry that gives the temperature dependence of the thermal heat capacity. (Integer ≥ 0 or blank)
T(H)	Identification number of a TABLEMj entry that gives the temperature dependence of the free convection heat transfer coefficient. (Integer ≥ 0 or blank)
T(μ)	Identification number of a TABLEMj entry that gives the temperature dependence of the dynamic viscosity. (Integer ≥ 0 or blank)
T(HGEN)	Identification number of a TABLEMj entry that gives the temperature dependence of the internal heat generation property for QVOL. (Integer ≥ 0 or blank)

Remarks:

1. The basic quantities on the MAT4 entry are always multiplied by the corresponding tabular function referenced by the MATT4 entry.
2. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the MAT4 entry.

MATT5 Thermal Anisotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT5 entry fields via TABLEMi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT5	MID	T(KXX)	T(KXY)	T(KXZ)	T(KYY)	T(KYZ)	T(KZZ)	T(CP)	
		T(HGEN)							

Example:

MATT5	24	73							

Field	Contents
MID	Identification number of a MAT5 entry that is to be temperature dependent. (Integer > 0)
T(Kij)	Identification number of a TABLEMi entry. The TABLEMi entry specifies temperature dependence of the matrix term. (Integer $\geq$ 0 or blank)
T(CP)	Identification number of a TABLEMi entry that specifies the temperature dependence of the thermal heat capacity. (Integer $\geq$ 0 or blank)
T(HGEN)	Identification number of a TABLEMi entry that gives the temperature dependence of the internal heat generation property for the QVOL entry. (Integer $\geq$ 0 or blank)

Remarks:

1. The basic quantities on the MAT5 entry are always multiplied by the tabular function referenced by the MATT5 entry.
2. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the basic MAT5 entry.

MATT8 Shell Orthotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT8 entry fields via TABLEMi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT8	MID	T(E1)	T(E2)	T(NU12)	T(G12)	T(G1Z)	T(G2Z)	T(RHO)	
	T(A1)	T(A2)		T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(S)	
	T(GE)	T(F12)							

Example:

MATT1	17	32							
	15			52					

Field	Contents
MAT	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
T(E1)	Identification number of a TABLEMi entry for the Young's modulus 1. (Integer ≥ 0 or blank)
T(E2)	Identification number of a TABLEMi entry for the Young's modulus 2. (Integer ≥ 0 or blank)
T(NU12)	Identification number of a TABLEMi entry for Poisson's ratio 12. (Integer ≥ 0 or blank)
T(G12)	Identification number of a TABLEMi entry for shear modulus 12. (Integer ≥ 0 or blank)
T(G1Z)	Identification number of a TABLEMi entry for transverse shear modulus 1Z. (Integer ≥ 0 or blank)
T(G2Z)	Identification number of a TABLEMi entry for transverse shear modulus 2Z. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for mass density. (Integer ≥ 0 or blank)
T(A1)	Identification number of a TABLEMi entry for thermal expansion coefficient 1. See Remark 3. (Integer or blank)

Field	Contents
T(A2)	Identification number of a TABLEMi entry for thermal expansion coefficient 2. See Remark **3.** (Integer or blank)
T(Xt)	Identification number of a TABLEMi entry for tension stress / strain limit 1. (Integer $\geq$ 0 or blank)
T(Xc)	Identification number of a TABLEMi entry for compression stress / strain limit 1. (Integer $\geq$ 0 or blank)
T(Yt)	Identification number of a TABLEMi entry for tension stress / strain limit 2. (Integer $\geq$ 0 or blank)
T(Yc)	Identification number of a TABLEMi entry for compression stress / strain limit 2. (Integer $\geq$ 0 or blank)
T(S)	Identification number of a TABLEMi entry for shear stress / strain limit. (Integer $\geq$ 0 or blank)
T(GE)	Identification number of a TABLEMi entry for structural damping coefficient. (Integer $\geq$ 0 or blank)
T(F12)	Identification number of a TABLEMi entry for Tsai-Wu interaction term. (Integer $\geq$ 0 or blank)

Remarks:

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT8 entry referenced in field 2. The value in a particular field of the MAT8 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E1 is modified by TABLEMi 32, A1 is modified by TABLEMi 15, and Xt is modified by TABLEMi 52. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT8 entry.
2. Any quantity modified by this entry must have a value on the MAT8 entry.
3. For a nonlinear static analysis of a composite element with the parameter COMPMATT set to ON, if the TABLEMi ID for the coefficient of thermal expansion is negative, the TABLEMi values will be interpreted as the thermal strain $\varepsilon(T)$ rather than the coefficient of thermal expansion $\alpha(T)$.

MATT9 Solid Element Anisotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT9 entry fields via TABLEMk entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT9	MID	T(G11)	T(G12)	T(G13)	T(G14)	T(G15)	T(G16)	T(G22)	
	T(G23)	T(G24)	T(G25)	T(G26)	T(G33)	T(G34)	T(G35)	T(G36)	
	T(G44)	T(G45)	T(G46)	T(G55)	T(G56)	T(G66)	T(RHO)	T(A1)	
	T(A2)	T(A3)	T(A4)	T(A5)	T(A6)		T(GE)		

Example:

MATT9	17	32			18			17	
				12					
				5			10		

Field	Contents
MID	Material property identification number that matches the identification number on MAT9 entry. (Integer > 0)
T(Gij)	Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer $\geq$ 0 or blank)
T(RHO)	Identification number of a TABLEMk entry for the mass density. (Integer $\geq$ 0 or blank)
T(Ai)	Identification number of a TABLEMk entry for the thermal expansion coefficients. (Integer $\geq$ 0 or blank)
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer $\geq$ 0 or blank)

Remarks:

1. Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT9 entry referenced in field 2. The value recorded in a particular field of the MAT9 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, G11 is modified by TABLEMj 32, G14 is modified by TABLEMj 18, etc. If the fields are zero or blank, then there is no temperature dependence of the fields on the MAT9 entry.
2. Any quantity modified by this entry must have a value on the MAT9 entry.
3. The continuation entries are optional.

MATTEP (SOL 600) Thermo-Elastic-Plastic Material Properties

Specifies temperature-dependent elasto-plastic material properties to be used for static, quasi static or transient dynamic analysis in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTEP	MID		T(Y0)	T(FID)				T(H)	
	N/A		T(yc10)	N/A					

Example:

MATTEP	100		20						

Field	Contents
MID	Identification number of MATEP entry. See Remark **1.** (Integer > 0)
T(Y0)	Identification number of TABLEMi entry for thermo-elasto-plastic material. See Remarks **2.** (Integer > 0 or blank)
T(FID)	Identifcation number of TABLEST entry for temperature-dependent stress-strain curves (Integer>0 or blank). See Remark **4.**
T(H)	Identification number of TABLEMi entry for temperature-dependent plasticity moduli in thermo-elasto-plastic material. See Remarks **3.** (Integer > 0 or blank)
T(yc10)	Identification number of TABLEMi entry for equivalent 10th cycle tensile yield stress specified in the Yc10 field of MATEP entry. (Integer > 0 or blank).

Remarks:

1. The MATEP Bulk Data entry with the same MID must exist for MATTEP to be effective. All the fields defined in MATTEP correspond to the same fields of MATEP. The value in a particular field of the MATEP entry is replaced or modified by the table referenced in the corresponding field of this entry.

2. The table represents yield stresses as a function of temperature. Therefore, the curve should comprise the initial stress from Y0 or FID field on MATEP (most likely at room temperature). T(Y0) field accommodates FID field in case FID field defines the initial yield stress instead of Y0 field. In this case, the yield stresses at any plastic strain will be scaled by the same ratio as the initial yield stress at the same temperature.
3. The table represents a normalized plasticity moduli (work hardening slope) as a function of temperature.
4. Temperature dependent stress-strain curves may be entered in a general manner using the T(FID) option. The integer value entered in this field represents the ID of a TABLEST entry which provides IDs of TABLES1 stress-plastic strain curves vs. temperature. All such curves must be entered as stress vs. plastic strain. No curves should be referenced on the MATS1 entry. For this option T(Y0) and T(H) should be left blank and if entered, MD Nastran will re-set them to blank if T(FID) is a positive integer.
5. This entry must be used in conjunction with MAT1, MATEP and MATT1 all with the same MID. The MATT1 entry must have at least one non-blank entry in fields 3-7 of the primary MATT1 entry.

Note:

1. This Bulk Data entry accommodates MSC.Marc's input data under the model definition options TEMPERATURE EFFECTS.

MATTG (SOL 600) Temperature Variation of Interlaminar Materials

Specifies gasket material property temperature variation to be used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTG	MID	IDYM	IDVM	IDDM	IDLD	IDU1	IDU2	IDU3	
	IDU4	IDU5	IDU6	IDU7	IDU8	IDU9	IDU10	IDYPR	
	IDEPL	IDGPL	IDGAP	N/A	N/A	N/A	N/A	N/A	

Example:

MATTG	100	10	20	1001	1002	1003			
								1010	
	1020	1030							

Field	Contents
MID	Material ID number that matches the material ID of a corresponding MATG material. (Integer > 0; required)
IDYM	ID of TABLEMi entry that gives the temperature variation of Young's modulus for the membrane behavior of the material. (Integer $\geq$ 0 or blank)
IDVM	ID of TABLEMi entry that gives the temperature variation of Poisson's ratio for the membrane behavior of the material. (Integer $\geq$ 0 or blank)
IDDM	ID of TABLEMi entry that gives the temperature variation of the mass density for the membrane behavior of the material. (Integer $\geq$ 0 or blank)
IDLD	ID of TABLEMi entry that gives the temperature variation of the loading curve of the material. This table has X and Y values of temperature and "Table ID". The "Table ID" must be entered as a "real" value with a decimal point but is converted internally to an integer value. It gives the table to use at that particular temperature, X. (Integer $\geq$ 0 or blank)

Field	Contents
IDUi	ID of TABLEMi entry that gives the temperature variation of the unloading curve of the material. There can be up to 10 unloading curves and each can have a different temperature variation. If there is no unloading, there does not need to be any unloading curves.This table has X and Y values of temperature and “Table ID”. The “Table ID” must be entered as a “real” value with a decimal point but is converted internally to an integer value. It gives the table to use at that particular temperature, X. (Integer $\geq$ 0 or blank)
IDYPR	ID of TABLEMi entry that gives the temperature variation of the yield pressure for theout-of -plane behavior of the material. (Integer $\geq$ 0 or blank)
IDEPL	ID of TABLEMi entry that gives the temperature variation of the tensile modulus for the out-of -plane behavior of the material. (Integer $\geq$ 0 or blank)
IDGPL	ID of TABLEMi entry that gives the temperature variation of the transverse shear modulus for the out-of -plane behavior of the material. (Integer $\geq$ 0 or blank)
IDGAP	ID of TABLEMi entry that gives the temperature variation of the initial gap for the out-of -plane behavior of the material. (Integer $\geq$ 0 or blank)

MATTHE (SOL 600) Thermo-Hyperelastic Material

Specifies temperature-dependent properties of hyperelastic (rubber-like) materials (elastomers) for nonlinear (large strain and large rotation) analysis in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTHE	MID	N/A	N/A	N/A	N/A	T(Texp)	N/A	N/A	
	T(XX)	T(YY)	T(ZZ)	T(TAB1)	T(TAB2)	T(TAB3)	T (TAB4)	T(TABD)	

Field	Contents
MID	Identification number of a MATHE entry. See Remark 1. (Integer > 0; no Default)
T(Texp)	Identification number of a TABLEMi entry for the coefficient of thermal expansion versus temperature. This field is not presently active. (Integer > 0 or blank)
T(XX)	Identification number of TABLEMi entry for the material constant versus temperature related to the distortional deformation. This field is used for Mooney or Aboyce, or Gent option depending on the “Model” field of MATHE entry. For Mooney, it is c10; for Aboyce, it is nkT; for Gent, it is E. (Integer > 0 or blank)
T(YY)	Identification number of TABLEMi entry for the material constant versus temperature related to the distortional deformation. This field is used for Mooney, Aboyce or Gent option depending on the “Model” field of MATHE entry. For Mooney, it is c01; for Aboyce, it is N; for Gent, it is Im. (Integer > 0 or blank)
T(ZZ)	Identification number of TABLEDMi entry for the material constant versus temperature related to the distoritional deformation. This field is used for ABoyce or Gent option depending on the “Model” field of MATHE entry. (Integer > 0 or blank) This field is not presently active.
T(TABi)	Identification number of a TABLEST entry for the experimental material data for the Mooney-Rivlin material. See Remark 2. (Integer > 0 or blank) This field is not presently active.

Remarks:

1. The MATTHE entry must have the same ID as the corresponding MATHE entry. Each table ID on the MATTHE entry corresponds to a parameter on the MATHE entry.
2. If experimental data is provided, it is expected that the user has the data for multiple tests of the same type at different temperatures. The T(TABi) fields refer to TABLEST entries which in turn refer to TABLES1 entries for each temperature. The TABLES1 entries contain the measured stress-strain curves described under MATHE.
3. For Ogden and Foam materials, no temperature dependent* properties are presently available.

MATTORT (SOL 600) Thermoelastic Orthotropic Material

Specifies temperature-dependent properties of elastic orthotropic materials for linear and nonlinear analyses used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTORT	MID	T(E1)	T(E2)	T(E3)	T(NU12)	T(NU23)	T(NU31)	N/A	
	T(G12)	T(G23)	T(G31)	T(A1)	T(A2)	T(A3)	T(SY)	T(WHS)	

Example:

MATTORT	100	5	6	7					

Field	Contents
MID	Identification number of a MATORT entry. See Remark **1.** (Integer > 0; no Default)
T(Ei)	Identification number of a TABLEMi entry for the Young's modulus as a function of temperature in each respective direction. Remark **2.** (Integer > 0 or blank)
T(Nuij)	Identification number of a TABLEMi entry for the Poisson's ratio as a function of temperature in each respective direction. (Integer > 0 or blank)
T(Gij)	Identification number of a TABLEMi entry for the shear modulus as a function of temperature in each respective direction. (Integer > 0 or blank)
T(Ai)	Identification number of a TABLEMi entry for the coefficient of thermal expansion as a function of temperature. (Integer > 0 or blank)
T(SY)	Identification number of a TABLEMi entry for the yield stress as a function of temperature. (Integer > 0 or blank)
T(WHS)	Identification number of a TABLEMi entry for the work hardening slope as a function of temperature. (Integer > 0 or blank)

Remarks:

1. The MATTORT entry must have the same ID as the corresponding MATORT entry. Each table ID on the MATTORT entry corresponds to a parameter on the MATORT entry. This capability is available only for MD Nastran Implicit Nonlinear (SOL 600).
2. The table represents material constants as a function of temperature. Therefore, the curve should comprise the original value specified in the MATORT entry (most likely at room temperature).

Note:

1. This Bulk Data entry accommodates MSC.Marc's input data under the model definition option ORTHO TEMP.

MATTVE (SOL 600) Thermo-Visco-Elastic Material Properties

Specifies temperature-dependent visco-elastic material properties in terms of Thermo-Rheologically Simple behavior to be used for quasi-static or transient dynamic analysis in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTVE	MID	function	RT	ENER	FRACT	TDIF	TREF	NP	
	C1	C2							
	C0	C1	C2	C3	C4	etc.			
	W0	W1	W2	W3	W4	etc.			
	T0	T1	T2	T3	T4	etc.			

Example:

MATTVE	100	WLF	100.						
	0.8	1.2							
MATTVE	101	POWER	100.					12	
	0.0	0.0							
	1.0	.99	.98	.97	.96	.95	.94	.93	W1-8
	.92	.915	.914						W9-12
MATTVE	102	NARA		4.3E-8	.75	200.	400.	12	
	0.0	0.0							
	1.0	.99	.98	.97	.96	.95	.94	.93	W1-8
	.92	.915	.914	.914					W9-12
	235.	234.	233.	232.	231.	230.	229.	228.	T1-8
	227.	226.	225.	224.					T9-12
MATTVE	103	USER	345.					8	

Fields	Contents
MID	Identification number. See Remark 1. (Integer > 0)
Function	Name of the shift function. See Remarks 2. and 3. (Character)

Fields	Contents
	WLF for Williams-Landell-Ferry form, Requires C1 and C2. (Default) **POWER** for power series form **NARA** for Narayanaswamy model **USER** to specify the shift function with a user subroutine
RT	Used for WLF and Power only, enter the reference or glass transition temperature. (Real; Default = 0.)
ENER	Used for NARA model only, enter the activation energy divided by the gas constant Q/R. (Real, no Default)
FRACT	Used for NARA only, enter the fraction parameter. (Real, no Default)
TDIF	Used for NARA only, enter the temperature shift between your temperature and absolute temperature for calculating fictitive temperatures. (Real, no Default)
TREF	Used for NARA only, enter the reference temperature for structural relaxation. (Real, no Default)
NP	For Power, enter the number of coefficients in the power series representation.
C1, C2	For WLF model enter the constants C1 and C2. For other models, enter 0.0 for C1 and C2. Do not enter a blank line as MD Nastran will strip it out. (Real, no Default)
Ci	Coefficients of the shift function, enter NP values for POWER only (leave the line with C2 and C2 blank). (C0 is included in NP). Do not enter Ci for NARA. (Real; no Defaults)
Wi	For NARA model only, enter the weighting factors in increasing order of subscript. Enter NP values. For WLF and POWER, skip these values. (Real, no Defaults)
Ti	For the NARA model only, enter the relaxation time values in increasing order of subscript. Enter NP values. For WLF and POWER, skip these values. (Real, no Defaults)

Remarks:

1. The MATVE or MATVORT Bulk Data entry with the same MID must exist for MATTVE to be effective.

2. The viscoelastic behavior is especially noticeable in the organic high polymers. There are many different kinds of such materials including various plastics, natural and synthetic rubbers. Their mechanical properties depend strongly on temperature, and these properties change drastically in the vicinity of a critical temperature called the glass-transition temperature T_g. The polymer well below T_g is an organic glass with a relatively high modulus. The viscoelastic behavior predominates in the transition range around T_g. The polymer above the transition region (but below the melting point) becomes a rubbery solid with a low modulus.

   Polymers are broadly classified as amorphous polymers and polycrystalline polymers. Under stress-relaxation at a constant strain in the glass-transition region temperature, the amorphous polymer exhibits a phase change over time from the glassy state to the rubbery state. The response is manifested in the shear modulus as a function of time, in which initially high shear modulus changes into low shear modulus. The relaxation curve of the modulus in a log-log scale plot appears as a flat plateau of glassy modulus G_g shifting down to the equilibrium modulus G_e at the rubbery plateau. Such a relaxation behavior of the amorphous polymer is observed even when the temperature is well below T_g for a prolonged period of time in a very slow process. A similar behavior is found in the rubbery elastic region, but the process is faster.

   Fortunately, the mechanical properties of amorphous polymers obey a time-temperature superposition principle, which allows the use of data obtained at different temperatures to extend the time scale at any given temperature. For such a behavior, the amorphous polymer is characterized as thermo-rheologically simple (TRS). Since the relaxation process extends several decades on the logarithmic time scale at lower temperatures, it is not feasible to determine the whole curve by a constant strain test at one temperature. Instead, the relaxation characteristics are measured at elevated temperatures in reduced time scale. Then the polymers exhibit a translational shift of all the characteristic responses with a change of temperature along the logarithmic time axis. This shift occurs parallel to the time axis without a change in properties: glassy and rubbery moduli. The modulus curve shifts towards shorter time with an increased temperature.

3. The reduced (ξ), or pseudo, time is related to the actual time (t) through a shift function which is a function of temperature, i.e.,

$$\xi(t) = \int_0^t \frac{ds}{A(T(s))}$$

where $A(T)$ is a shift function in terms of temperature T at time t. The shift function is a material property and must be determined experimentally. A shift function approximated by Williams-Landell-Ferry, known as WLF equation, has the form:

$$\log A = h(T) = -\frac{C_1(T - T_0)}{C_2 + (T - T_0)}$$

where T_0 is the reference temperature at which relaxation data are given and C_1, C_2 are calibration constants obtained at this temperature. Notice that $A = 1$ if the reduced time is the same as the actual time. If $T \le T_0 - C_2$, the deformation will be elastic.

Another form of the shift function is available as a power series in $(T - T_0)$, i.e.,

$$\log A = h(T) = \sum_{i=0}^{10} C_i (T - T_0)^i$$

4. The WLF shift function requires C1 and C2. The power series can have a maximum 11 coefficients C0 through C10.

Note:

1. This entry matches the three options for SHIFT FUNCTION in MSC.Marc: William-Landell-Ferry, Power Series and Narayanaswamy models.

MATVE (SOL 600) Visco-Elastic Material Properties

Specifies isotropic visco-elastic material properties to be used for quasi-static or dynamic analysis in MD Nastran Implicit Nonlinear (SOL 600) only.

Format (for types Iso, Mooney and Ogden):

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model	Alphas	Alphal	Wd1	Td1	Wv1	Tv1	
	Wd2	Td2	Wd3	Td3	Wd4	Td4	Wd5	Td5	
	Wv2	Tv2	Wv3	Tv3	Wv4	Tv4	Wv5	Tv5	

Alternate Format (for type Ortho):

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Ortho	N/A	N/A	N/A	N/A	N/A	N/A	
	Td1	Exx1	Eyy1	Ezz1	Vxy1	Vyz1	Vzx1	N/A	
	Gxy1	Gyz1	Gzx1	N/A	N/A	N/A	N/A	N/A	
	Td2	Exx2	Eyy2	Ezz2	Vxy2	Vyz2	Vzx2	N/A	
	Gxy2	Gyz2	Gzx2	N/A	N/A	N/A	N/A	N/A	
	Td3	Exx3	Eyy3	Ezz3	Vxy3	Vyz3	Vzx3	N/A	
	Gxy3	Gzx3	Gzx3	N/A	N/A	N/A	N/A	N/A	
	Td4	Exx4	Eyy4	Ezz4	Vxy4	Vyz4	Vzx4	N/A	
	Gxy4	Gyz4	Gzx4	N/A	N/A	N/A	N/A	N/A	
	Td5	Exx5	Eyy5	Ezz5	Vxy5	Vyz5	Vzx5	N/A	
	Gxy5	Gyz5	Gzx5	N/A	N/A	N/A	N/A	N/A	

(Only enter as many continuation lines as required to describe up to five terms).

Example:

MATVE	100	Iso			0.9	0.002			
	0.8	0.003							

Fields	Contents
MID	Identification number of MAT1 or MATHE entry (Integer > 0). See Remark **1.**
Model	Selects a visco-elastic model defining time-dependent deformation behavior (Character): ISO for isotropic materials referenced by MAT1 (Default) (an alternate name is Linear). Ortho if referenced by MAT2, MAT3, MAT8, MAT9, MATORT, MATVORT, or MATHE. Mooney for Mooney-Rivlin model if referenced by MATHE. Ogden for Ogden model if referenced by MATHE. See Remarks **2.**, **3.**, and **4.**
Alphas	Solid coefficient of thermal expansion (Real, Default = 0). See Remark **5.**
Alphal	Liquid coefficient of thermal expansion (Real, Default = 0).
Wdi	Multiplier (scale factor) for deviatoric behavior in Prony series (Real $\geq$ 0., Default = 0).
Tdi	Defines time constants for deviatoric behavior in Prony series (Real $\geq$ 0., Default = 0). See Remark **6.**
Wvi	Multiplier (scale factor) for volumetric behavior in Prony series (Real $\geq$ 0., Default = 0.). See Remark **6.**
Tvi	Defines time constants for volumetric behavior in Prony series (Real $\geq$ 0., Default = 1000.).

Remarks:

1. The linear isotropic elastic (MAT1), Mooney and Ogden hyperelastic (MATHE) materials may reference this Bulk Data entry. But the linear orthotropic material should be specified in MATORT and MATVORT. The hyperfoam material is not supported for viscous behavior.

2. The time-dependent behavior in the viscoelastic material is modeled by a Prony series expression for both small and large strain problems. The stress relaxation behavior can be modeled by relaxation functions of the shear modulus and bulk modulus in terms of a series of exponential decay terms, which is known as the Prony series. This is equivalent to the generalized Maxwell model, which consists of many Maxwell models connected in parallel along with an elastic spring representing a long-term behavior. The constitutive behavior of viscoelasticity depends not only on the current state of stress and strain, but also on the entire history of the development of these states. Such a behavior is most readily expressed by hereditary or Duhamel integral. The Prony series is a discrete form of this hereditary integral.

3. The Prony series defines the relaxation modulus by a dimensionless function of time

$$g_r(t) = 1 - \sum_i w_i\left(1 - e^{-t/\tau_i}\right)$$

where w_i are weighting factors and τ_i are time constants. If a Prony series is selected, at least one pair of weighting factor and time constant must be provided: Wd1 and Tv1.

The function is introduced to each stress-strain component, i.e.,

$$\tau(t) = G_0\int_0^t g_r(t-s)\dot{\gamma}(s)ds = G_0\int_0^t \dot{g}_r(s)\gamma(t-s)ds$$

where G_0 being an instantaneous shear modulus, and the second term with the integral sign represents the cumulative viscoelastic creep strain with

$$G_r = 1 - \sum_1 \frac{G_i}{G_0}\left(1 - e^{-t/\tau_i}\right)$$

in which $g_r(0) = 1$, $g_r(\infty) = G_\infty/G_0$, and $w_i = G_i/G_0$ denotes a long-term shear modulus, which would have settled the stress-strain relationship when the time approaches infinity. In case of the volumetric deformation, the shear modulus is replaced by a bulk modulus K to define the parameters with

$$K_0 = \frac{E_0}{3(1-2\upsilon_0)}$$

4. In case of a viscous hyperelastic material, the Second Piola-Kirchhoff stress may be expressed as a function of time

$$S = \int_0^t g_r(t-\tau)\frac{\partial^2 W}{\partial E^2}\dot{E}d\tau$$

where W is a strain energy potential and E is Lagrangian strain. For deviatoric deformation, this equation becomes

$$S = \frac{\partial W}{\partial E} + \int_0^t \dot{g}_r(t-\tau)\frac{\partial W(\tau)}{\partial E}d\tau$$

For the duel Prony series, two different Prony series are applied separately to the deviatoric and volumetric strains (g^d and g^v). This will result in relaxation functions for hyperelastic materials as follows:

$$C_{ij}(t) = C_{ij}^0\left[1 - \sum_i w_i^d\left(1 - e^{-t/\tau_i}\right)\right]$$

for Mooney-Rivlin model and

$$\mu_k(t) = \mu_k^0\left[1 - \sum_i w_i^d\left(1 - e^{-t/\tau_i}\right)\right]$$

for Ogden model.

5. If Alphas or Alphal is specified, the thermal expansion coefficient specified in MAT1, MATT1, MATHE, MATTHE, MATORT or MATTORT will be ignored. Use of ALPHAS ro ALPHAL requires the MATTVE Narayanaswami model.
6. If the weighting factor is left blank, the relaxation function corresponding to that coefficient is omitted. Linear and Ogden models supports volumetric viscoelastic behavior by allowing volumetric terms (Wvi and Tvi) in the Prony series, but the Mooney-Rivlin model does not allow volumetric terms in the Prony series.
7. For Mooney-Rivlin materials, enter WDi and TDi, but do not enter WVi or TVi.

Notes:

1. The Prony series constants in MD Nastran Implicit Nonlinear and MSC.Marc are raw values of G and K Caution: Some other programs used weighting functions.

2. This Bulk Data entry accommodates MSC.Marc's input data under the model definition options VISCELPROP, VISCELMOON, VISCELOGDEN, and VISCELEXP as well as the parameter VISCOELAS.

MATVP (SOL 600) Visco-Plastic or Creep Material Properties

Specifies viscoplastic or creep material properties to be used for quasi-static analysis in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MATVP	MID	Form	Coeff	Stress	Strain	Temp	Time	Thresh	

Example:

MATVP	100	Table	3.5e-15	101	102	103	104	5.e3	

Field	Contents
MID	Identification number of MAT1, MAT2, MATORT or MAT9 entry. See Remark 1. (Integer > 0)
Form	Selects a creep data input form defining creep strain rate (Character) among: Power for exponent input in power law form. (Default) Table for piece-wise linear curve input in TABLEM1 entry. See Remark 2.
Coeff	Specifies the coefficient value, A in equation. (Real > 0)
Stress	Identification number of TABLEM1 (Integer > 0) for the function f or exponent m for an effective stress function, depending on the Form field. (Real > 0)
Strain	Identification number of TABLEM1 (Integer > 0) for the function g or exponent n (Real; Default = 0.) for an equivalent creep strain function, depending on the Form field.
Temp	Identification number of TABLEM1 (Integer > 0) for the function h or exponent p (Real; Default = 0) for a temperature function, depending on the Form field.
Time	Identification number of TABLEM1 (Integer > 0) for the function K or exponent q (Real > 0; Default = 1) for a time function, depending on the Form field.
Thresh	Specifies the threshold stress limit, or back stress, for creep process, under which there is no creep deformation. See Remark 3. (Real $\geq$ 0; Default = 0)

Remarks:

1. This Bulk Data entry is activated if a MAT1, MAT2, MATORT, or MAT9 entry with the same MID is specified in a nonlinear analysis using MD Nastran Implicit Nonlinear (SOL 600) only. This creep capability is available for isotropic, orthotropic, and anisotropic elasticity, which can be coupled with plasticity using MATEP entry. Coupling with plasticity is allowed only for selected plasticity models, which include von Mises, Hill's anisotropy (creep stays isotropic), and Mohr-Coulomb models. However, viscoelasticity (MATVE) cannot be combined with viscoplasticity (MATVP).
2. The creep behavior of the material is expressed in terms of creep strain rate as a product of a number of terms (functions of effective stress, equivalent creep strain, temperature, and time) in either piece-wise linear curves or exponential form, i.e.,

$$\dot{\bar{\varepsilon}}^c = A \cdot \bar{\sigma}^m \cdot (\bar{\varepsilon}^c)^n \cdot T^p \cdot (qt^{q-1})$$

or

$$\dot{\bar{\varepsilon}}^c = A \cdot f(\bar{\sigma}) \cdot g(\bar{\varepsilon}^c) \cdot h(T) \cdot \frac{dK(t)}{dt}$$

The functions f, g, h and K are specified as piece-wise linear functions in a tabular form using TABLEM1 entry, if the Table Form is selected. Notice that the last term in time shows function K for the equivalent creep strain in terms of time, instead of creep strain rate.

The creep strain from the creep material is a permanent strain unlike the creep strain for materials using the CREEP Bulk Data entry. As such, this creep material may be classified as viscoplastic material. This creep capability is provided for the primary and the secondary creep behavior, because the tertiary creep involves material instability such as necking.

3. If the threshold stress is the same as the initial yield stress, the creep deformation occurs only in the plastic range. If the threshold stress is specified, an implicit integration scheme is used. Otherwise, an explicit integration is used at the Gauss point level.

Note:

1. This Bulk Data entry accommodates MSC.Marc's input data under the model definition option CREEP as well as the parameter CREEP.

MBOLT (SOL 600) Defines a Bolt For Use in SOL 600 in Countries Outside the USA

Defines a bolt for use in SOL 600 in countries outside the USA. Used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MBOLT	ID	GRIDC	V1	V2	V3				
	GRIDS	G1	G2	G3	G4	G5	G6	G7	
		G8	G9	Etc.					
	ELEMS	E1	E2	E3	E4	E5	E6	E7	
		E8	E9	Etc.					

Example:

1	2	3	4	5	6	7	8	9	10
MBOLT	100	1025	0.0	1.0	0.0				
	GRIDS	101	102	103	104	105			
	ELEMS	1	2	3	4	5	6	7	
		10							

Field	Contents
ID	Element ID of the bolt. (Integer, required, no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer, no Default, required)
V1	First component of vector normal to the bolt cross section in basic coordinate system. (Real, Default = 0.0)
V2	Second component of vector normal to the bolt cross section in basic coordinate system. (Real, Default = 0.0)
V3	Third component of vector normal to the bolt cross section in basic coordinate system. (Real, Default = 0.0)
GRIDS	Enter the character string GRIDS to define the start of the entry that defines all of the grids at the bolt intersection cross section (do not enter the ID for GRIDC). (Integer, no Default)
G1, G2, etc.	Grid IDs of the grid points at the bolt intersection. (Integer, no Default)

Field	Contents
ELEMS	Enter the character string ELEMS to define the start of the entry that defines all of the elements at the bolt intersection cross section lying on the side of the cross section corresponding to the negative normal direction. (Integer, no Default)
E1, E2, etc.	Element IDs of the grid points at the bolt intersection. (Integer, no Default)

Remarks:

1. Enter as many GRIDS and ELEMS lines as necessary to define all the grid and element IDs in the cross section.
2. All GRIDS must proceed all ELEMS.
3. The bolt itself is not actually modeled, just the intersecting surfaces. The nodes and elements where the bolt goes through the intersecting surfaces are described by this entry.
4. Specify a different MBOLT entry for each individual bolt.
5. This entry can only be used with MSC.Marc 2003 or later outside the USA.
6. For more information, please consult the MSC.Marc Theoretical Manual (Volume A of the MSC.Marc documentation).
7. This entry maps to MSC.Marc's CROSS SECTION entry.
8. GRIDC must already exist. It is not generated by the MBOLT entry. It typically would not be used by any other element, MPC, etc.

MBOLTUS (SOL 600) Defines a bolt for use in SOL 600 in the USA

Defines a bolt for use in SOL 600 in the USA in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MBOLTUS	ID	GRIDC							
	TOP	GT1	GT2	GT3	GT4	GT5	GT6	GT7	
		GT8	GT9	Etc.					
	BOTTOM	GB1	BG2	BG3	BG4	GB5	GB6	GB7	
		GB8	GB9	Etc.					

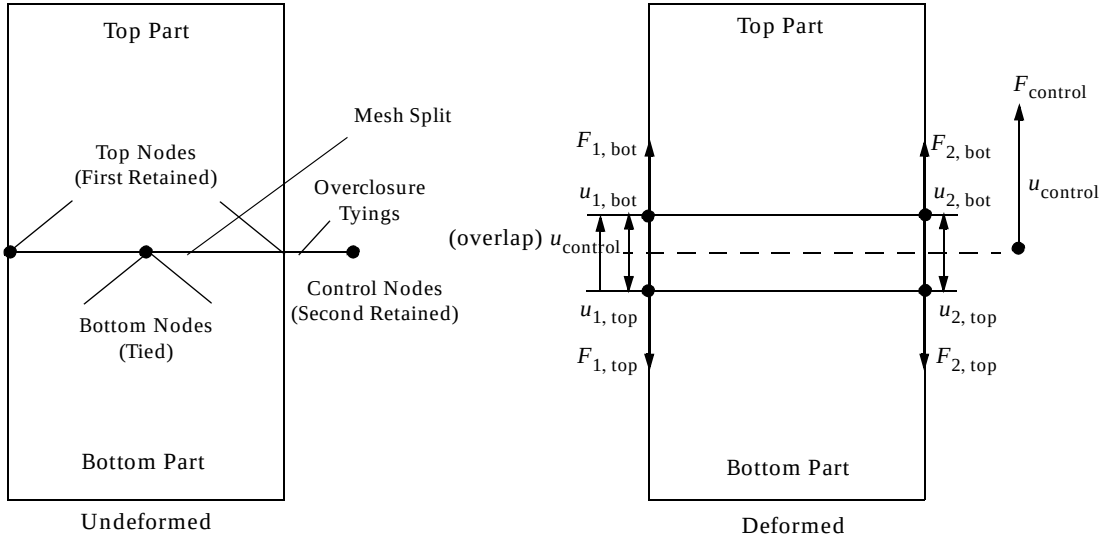
Example:

1	2	3	4	5	6	7	8	9	10
MBOLTUS	100	1025	0.0	1.0	0.0				
	TOP	101	102	103	104	105			
	BOTTOM	1	2	3	4	5			

Field	Contents
ID	Element ID of the bolt. (Integer, required, no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer, no Default, required)
TOP	Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer, no Default)
GT1, GT2, etc.	Grid IDs of the grid points at the bolt intersection. (Integer, no Default)
BOTTOM	Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC) (Integer, no Default)
GB1, GB2, etc.	Element Id’s of the grid points at the bolt intersection (Integer, no Default)

Remarks:

1. Enter as many GRIDS as necessary to define all the grids at the “top” and “bottom” of the bolt intersection with the structure.
2. The bolt itself is not actually modeled, just the intersecting surfaces. The nodes and elements where the bolt goes through the intersection surfaces are described by the entry.
3. Specify a different MBOLD entry for each individual bolt.
4. This entry can only be used with MSC.Marc 2003 or later.
5. For more information, please consult the MSC.Marc Theoretical Manual (Volume A of the MSC.Marc documentation)
6. This entry maps to MSC.Marc’s TYING type 69.
7. GRIDC must already exist. It is not generated by the MBOLTUS entry. It typically would not be used by any other element, MPC, etc.
8. The following figure indicates the required modeling and data input.



MDLPRM Model Parameters

Specifies parameters which affect the solution of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
MDLPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	-etc.-		

Example:

MDLPRM	QR6ROT	2	QRSHEAR	1					

Field	Contents
PARAMi	Name of the parameter. Allowable names are given in **Table 8-24**. (Character)
VALi	Value of the parameter. (Real or Integer, see **Table 8-24**)

Remark:

1. Multiple entries of MDLPRM are allowed in the Bulk Data Section. However, multiple entries of a particular parameter PARAMi are illegal.

Table 8-24 PARAMi Names and Descriptions

Name	Description, Type, and Default Values
QR6ROT	Parameter to determine whether the drilling degrees-of-freedom are to be deactivated for QUADR/TRIAR elements. If the drilling degrees-of-freedom are deactivated, the QUADR/TRIAR become elements similar to QUAD4/TRIA3. QR6ROT has the following values: 0: The drilling degrees-of-freedom are active, default. 1: The drilling degrees-of-freedom are deactivated for all QUADR/TRIAR element in the model. 2: The drilling degrees-of-freedom are deactivated for those QUADR/TRIAR which have membrane stiffness only (MID2 and MID3 are blank on the PSHELL entry)
QRSHEAR	Parameter to select the off-plane shear formulation for the QUADR element. There are two types of off-plane shear formulations: the stiffness method and the flexibility method. The stiffness method is a new method implemented in QUADR. The flexibility method was the method implemented in the QUAD4 element. Therefore, if the flexibility method is selected, the solution results of QUADR are closer to those of QUAD4. QRSHEAR has the following values: 0: Select the stiffness method, default. 1: Select the flexibility method.
TWBRBML	Parameter to select method for computing properties of PBARL/PBEAML. 0: Select Finite Element Method (Default). 1: Select Beam Library Equations.

MDMIOUT (SOL 600) Matrices from MSC.Marc

Defines full or reduced stiffness and mass matrices to be output from the MSC.Marc portion of SOL 600. This entry may be used to generate External Superelements using DMIG Matrices or an MSC.Adams MNF File from the MSC.Marc portion of a SOL 600 analysis. SOL 600 only. (See also the **MNF600** Bulk Data entry.)

Format:

1	2	3	4	5	6	7	8	9	10
MDMIOUT	ID	IDOF	G1	THRU	G2		NAME	ISOL	
	IDOF2	G3	THRU	G4					
	etc.								

Example:

MDMIOUT	100	123456	1	THRU	5456	1	nlkm	103	

Field	Contents
ID	Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example, if the case control contains SUBCASE 20, ID would be 20. (Integer, Default = 1)
IDOF, IDOF2	List of DOF's to be output (any or all of the integers 1-6 are acceptable). (Integer, Default = 123456)
G1, G3	Starting grid ID for reduced matrices. (Integer, Required, no Default)
G2, G4	Ending grid ID for reduced matrices. (Integer, Required, no Default)
NAME	Name of a file (without extension) to which the DMIG matrices will be copied. The extension .dmi is appended automatically. NAME is limited to 8 characters. NAME will be converted to lower case even if entered in upper case. (Character, no Default)
ISOL	Solution sequence to run using the DMIG matrices. To use speed up the solution, use DOMAINSOLVER ACMS (PARTOP=DOF) for eigenvalues and set ISOL to -103, -111 or -112. (For MD Nastran 2004 r3, only -103 is available. (Integer absolute value > 100, Default = 0 which means do not run any solution sequence using the DMIG's created by MSC.Marc in this execution)

Remarks:

1. The continuation line(s) are not required.
2. This entry corresponds to MSC.Marc's entry, SUPERELEM with a value of 1 in the second line 4th field.
3. The reduced stiffness and mass matrices are found in DMIG format in the MSC.Marc t19 file.
4. Bulk Data entry, PARAM,MARCT19,1 is necessary to obtain the t19 file and these matrices. The presence of any MDMIOUT entries in the input will automatically turn on the MARCT19,1 parameter.
5. The reduced matrices may be used in the MD Nastran analysis for eigenvalue extraction or any other purpose by invoking the CONTINUE=5 option on the SOL 600 entry.
6. If the SOL 600 CONTINUE options is invoked, case control commands and a bulk data entry include statements to receive the matrices will be automatically added to the original input data file. A second MD Nastran execution will be spawned from the original MD Nastran execution after completion of the MSC.Marc execution.
7. ID may not be 600 and 700 in the Executive Control statement, SOL 600,ID.
8. Only on MDMIOUT entry should be entered per run. If more are entered, only the first will be used.
9. MNF controls for other solution sequences are ignored for SOL 600.

MESUPER (SOL 600) Defines External Superelement DMIG Input for SOL 600 Residual Analyses

Superelement DMIG matrices are created by MD Nastran and used when MSC.Marc is spawned from MD Nastran, SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MESUPER	ID	Fname							

Example:

MESUPER	1	super1.pch	
MESUPER	2	super2.pch	

Field	Contents
ID	Superelement ID. (Integer, no Default)
Fname	Filename containing external superelement data from the creation run (using Case Control EXTSEOUT (ASMBULK, DMIGPCH,EXTID=ID) (Character, no Default).

Remarks:

1. Enter as many MESUPER lines as necessary to define all external superelements.
2. This entry can only be used with MSC.Nastran 2005 r2 or later.
3. This entry can presently only be used with SOL 600,106, SOL 600,101, SOL 600,nlstatic, or SOL 600,sestatic. External Superelements are not presently available for other types of SOL 600 analyses such as nonlinear (or linear) transient dynamics, eigenvalue analysis or buckling.
4. Fname is limited to 56 characters.
5. Include entries with the same Fname must be specified as include files in the MD Nastran input file. The include specifications must appear at the end of the Bulk Data portion of the file.
6. External superelement creation runs should use the Case Control command:

   EXTSEOUT(ASMBULK,DMIGPCH,EXTID=N)

   where N is the external superelement ID number. All creation runs must have the same number of subcases and use the same subcase IDs.

7. The SOL 600 residual input file must have the same number of subcases and subcase numbers as the creation runs.
8. This entry acts like an element, in other words it is not controlled by a case control command. It is always active if entered.
9. The MESUPER entries should normally be coded in small fixed field format. If coded in small format free field, Fname is limited to 8 characters. If coded in large format free field, Fname is limited to 16 characters. The filename may be extended to a continuation line. All filenames should be entered in lower case. MD Nastran will convert to upper cae, and the SOL 600 translator will reconvert to lower case. The creation runs should also use lower case for all external superelement punch filenames for case-sensitive computer systems.
10. If there are no elements in the residual (that is, all elements are in the external superelements, PARAM,MARCND99,-1 is required to output the displacements in the MSC.Marc .out file regardless of the specified Case Control request.
11. The ASMBULK option in the creation runs is required for SOL 600 when outr options are specified. It is recommended whether or not outr options are specified. All .asm files (created by the ASMBULK option) for all external superelements should be included in the Bulk Data before any punch files (produced using the DMIGPCH option). See the following input file examples.
12. At present, an OP2 with results datablocks only can be produced by a SOL 600 External Superelement residual execution. OP2 files which combine geometry and results datablocks cannot be produced. Other outr options such as xdb, f06 and punch are also not available for SOL 600 External Superelement residual runs.

Typical File Setup for External Superelement Creation Run for SOL 600 (same as for other solution sequences)

```
SOL 101
CEND
TITLE = 2 SUPERELEMENTS AND THE RESIDUAL -- TEST PROBLEM NO. EXTSE2A
SUBTITLE = 8 X 8 MESH OF QUAD4 ELEMENTS; GM-CMS PROJECT
EXTSEOUT(ASMBULK,DMIGPCH,EXTID=100)
SPC = 100
BEGIN BULK
aset1,123456,840,thru,848
CORD2R,1001,1002,,,,,1.0
```

(rest of file same as any other MD Nastran run)

Typical File Setup for External Superelement Residual Run for SOL 600

Note: This input makes reference to two External Superelement Creation run punch and .asm files. All asm files must be included before the punch files. The punch files are included as well as placed on MESUPER entries. Also, the PARAM,MEXTSEE,1 is used.

```
SOL 600,101 outr=op2
CEND
TITLE = 2 SUPERELEMENTS AND THE RESIDUAL -- TEST PROBLEM NO. EXTSE2R
SUBTITLE = 8 X 8 MESH OF QUAD4 ELEMENTS; GM-CMS PROJECT
param,mextsee,1
SPC = 100
LOAD = 1000
DISP = ALL
K2GG=KAAX
M2GG=MAAX
BEGIN BULK
param,marcnd99,-1
force, 1000, 844, , 0.1, 0., 0., 1.
SPC1    100     12346     840     848
$2345678 2345678 2345678
mesuper      100 extse2a.pch
mesuper      200 extse2b.pch
include 'OUTDIR:extse2a.asm'
include 'OUTDIR:extse2b.asm'
include 'OUTDIR:extse2a.pch'
include 'OUTDIR:extse2b.pch'
ENDDATA
```

MFLUID Fluid Volume Properties

Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.

Format:

1	2	3	4	5	6	7	8	9	10
MFLUID	SID	CID	ZFS	RHO	ELIST1	ELIST2	PLANE1	PLANE2	
	RMAX	FMEXACT							

Example:

MFLUID	3	2	15.73	1006.	3	4	S	N	
		100.							

Field	Contents
SID	Set identification number. (Integer > 0)
CID	Identification number of rectangular coordinate system used to specify the orientation of the free surface (normal to X_3) and of planes of symmetry, if any. (Integer $\geq$ 0 or blank)
ZFS	Intercept of the free surface on the X_3 axis of the coordinate system referenced by CID. If X3 > ZFS then there is no fluid. See Remark **3.** (Real; Default means that the free surface is located at an infinitely large positive value of X_3.)
RHO	Density of the fluid. (Real)
ELIST1	Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on one side by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. See Remarks **3.** and **5.** (Integer $\geq$ 0)
ELIST2	Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on both sides by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. (Integer $\geq$ 0; ELIST1 + ELIST2 > 0)

Field	Contents
PLANE1, PLANE2	Planes of symmetry, antisymmetry, or no symmetry. “S” means that plane 1, which is the plane containing the X_1 and X_3 axes of CID, is a plane of symmetry. “A” means that plane 1 is a plane of antisymmetry. “N” means that it is neither. See Remark 5. Plane 2 uses “S”, “A”, or “N” for the X_2 and X_3 plane. (Character: “S”, “A”, or “N”)
RMAX	Characteristic length. Interactions between elements with separation that is greater than RMAX will be neglected. (Real > 0.0; Default = 1.0E 10)
FMEXACT	Exact integration is used if the distance between two elements is less than FMEXACT times the square root of the area of the larger element. Otherwise, center point integration is used by default. (Real; Default = 1.0E 15)

Remarks:

1. The MFLUID entry must be selected with the Case Control command MFLUID = SID.
2. Several MFLUID entries corresponding to different fluid volumes can be used simultaneously.
3. The wetted side of an element in ELIST1 is determined by the presence or absence of a minus sign preceding the element’s ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element’s positive normal, as determined by applying the right-hand rule to the sequence of its corner points. The same element can appear on two ELIST entries, indicating that it forms a barrier between the unconnected fluids.
4. The fluid volume may be finite (interior) or infinite (exterior). The volume may be bounded by a free surface and one or two planes of structural symmetry. If structural symmetry is used, the structure must have the symmetric or antisymmetric boundary corresponding to the selection in fields 8 and 9. Interior fluids must have ELIST1 data and a free surface or plane of antisymmetry.
5. The planes of symmetry and / or antisymmetry defined in fields 8 and 9 must be planes of symmetry for the entire analysis. The user may apply appropriate structural boundary conditions at all grid points lying in these planes.
6. The current list of elements that may be placed in ELIST1 and ELIST2 include CTRIA3 and CQUAD4.

7. The continuation entry is optional.
8. If there is ELIST1 data and no free surface nor plane of antisymmetry, the program assumes a special form of external fluid. These special external fluids must have a CID (field 3) such that the origin of the fluid coordinate system is near the center of the enclosed volume, since the singularity for volume change will be placed at the origin. Special external fluids are supported only in SOLs 103 and 107 through 112. If used in conventional solution sequences, System Fatal Message 3001 results for file 205.
9. See PARAM,VMOPT in "**Parameters**" on page 1409. VMOPT controls when the virtual mass is included in the mass matrix.
10. If any MFLUID entry is changed or added on restart then a complete re-analysis may be performed. Therefore, MFLUID entry changes or additions are not recommended on restart.
11. A tolerance is computed for each wetted element, with the value of $0.01 \cdot \text{sqrt}(2. \cdot A)$, where A is the area of the element. If any grid point connected to the element lies within TOL below the free surface it is moved to the free surface.
12. Any element that has all grids on or above the free surface, after the grid points are moved by the procedures given in Remark **11.** is removed from the ELIST. It is not included in the VM effects, and will produce no pressure output.

MKAERO1 Mach Number - Frequency Table

Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

Format:

1	2	3	4	5	6	7	8	9	10
MKAERO1	m1	m2	m3	m4	m5	m6	m7	m8	
	k1	k2	k3	k4	k5	k6	k7	k8	

Example:

MKAERO1	.1	.7							
	.3	.6	1.0						

Field	Contents
mi	List of from 1 to 8 Mach numbers. (Real $\geq$ 0.0)
kj	List of from 1 to 8 reduced frequencies. (Real > 0.0)

Remarks:

1. Blank fields end the list, and thus cannot be used for 0.0.
2. All combinations of (mi, kj) will be used.
3. The continuation entry is required.
4. Multiple MKAERO1 entries are permitted.
5. For the lifting surface theories (Doublet-Lattice and Mach Box), the maximum value of kj should be less than one quarter of the number of boxes on a representative chord (i.e., $\text{MAX(kj)} < \bar{C}/4\Delta x \text{MAX(kj)} < \bar{C}/4\Delta x$ where $\bar{C}$ is the reference chord and Δx is a typical box chord length).
6. In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.
7. The very low nonzero value of kj required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.

MKAERO2 Mach Number - Frequency Table

Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

Format:

1	2	3	4	5	6	7	8	9	10
MKAERO2	m1	k1	m2	k2	m3	k3	m4	k4	

Example:

MKAERO2	.10	.30	.10	.60	.70	.30	.70	1.0	

Field	Contents
mi	Mach numbers. (Real $\geq$ 0.0)
ki	Reduced frequencies. (Real > 0.0)

Remarks:

1. MKAERO2 will cause the aerodynamic matrices to be computed for the given sets of parameter pairs. Embedded blank pairs are skipped.
2. No continuation entries are allowed, but multiple MKAERO2 entries are permitted.
3. In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.
4. The very low nonzero value of ki required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.

MNF600 Defines Auxiliary Data for MSC.Adams MNF Files

Generated by the MSC.Marc portion of a SOL 600 execution. Used in MD Nastran Implicit Nonlinear (SOL 600). (See also the **MDMIOUT (SOL 600)** Bulk Data entry.)

Format:

1	2	3	4	5	6	7	8	9	10
MNF600	ID	ISTRESS	ISTRAIN	ISHELL	MASSU	LENGU	TIMEU	FORCU	

Example:

MNF600	100	1	1	3	2	7	3	2	

Field	Contents
ID	Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example, if the case control contains SUBCASE 20, ID would be 20. Must match the MDMIOUT entry. At present, ID is ignored and the first entry will apply to all subcases. (Integer, Default = 1)
ISTESS (2)	Flag to compute stress and place in the MNF file (0=stress, 1=stress) (Integer, Default = 0)
ISTRAIN (3)	Flag to compute strain and place in the MNF file (0 - no strain, 1=strain) (Integer, Default = 0)
ISHELL (4)	For shell elements, this entry describes which location the stresses or strains will be output to the MNF file. 1=top, 2=center, 3=bottom (Integer, Default = 1)
MASSU [2,1]	Mass units for MNF file. (Integer, no Default) The following possible values may be entered: 1:kilogram (Default) 2:pound mass 3:slug 4:gram 5:ounce mass 6:kpound mass 7:megagram 8:dozen slug

Field	Contents
LENGU	Length units for MNF file (Integer, no Default) The following possible values may be entered: 1:kilometer 2:meter (Default) 3:centimeter 4:millimeter 5:mile 6:foot 7:inch
TIMEU [2,3]	Time units for MNF file (Integer, no Default). The following possible values may be entered: 1:hour 2:minute 3:second (Default) 4:millisecond
FORCEU [2,4]	Force units for MNF file (Integer, no Default). The following possible values may be entered: 1:Newton (Default) 2:poind force 3:kilogram force 4:ounce force 5:dyne 6:kNewton 7:kpound force

Remarks:

1. The MDMIOUT entry is the primary entry which generates an MNF file. This entry, MNF600 is only necessary if one or more of the fields is required to define non-default values, for example to generate stresses or strains or to specify the units.
2. The ID must be the same as the MDMIOUT ID.
3. Only one MNFDAT entry is allowed in an input file. If more than one is entered, the first will be used.
4. All remarks concerning MNF files for the MDMIOUT entry are also applicable to this entry.
5. (i) Indicates the corresponding field of MSC.Marc's MNF Parameter.

6. [i,j] Indicates the corresponding datablock and field of MSC.Marc's MNF units entry.

MODTRAK Mode Tracking Parameters

Specifies parameters for mode tracking in design optimization (SOL 200).

Format:

1	2	3	4	5	6	7	8	9	10
MODTRAK	SID	LOWRNG	HIGHRNG	MTFILTER					

Example:

MODTRAK	100	1	26	0.80					

Field	Contents
SID	Sets identification number that is selected in the Case Control Section with the MODTRAK command. See Remark **1.** (Integer; No Default)
LOWRNG	Lowest mode number in range to search. See Remark **2.** (Integer $\geq$ 0, Default = 0. If nonzero, LOWRNG < HIGHRNG.)
HIGHRNG	Highest mode number in range to search. See Remark **2.** (Integer > 0, Default =number of eigenvalues extracted. If nonzero, LOWRNG< HIGHRNG.)
MTFILTER	Filtering parameter used in mode cross-orthogonality check. See Remark **3.** (Real, Default = 0.9)

Remarks:

1. Only the designed modes for the subcase will be tracked. A designed mode is one that is used in the design model (in connection with either objective or constraints) and, therefore, identified on a DRESP1 entry.
2. The range of modes LOWRNG through HIGHRNG, inclusive, will be used to track the designed modes. If LOWRNG and HIGHRNG are both blank, then all computed modes will be used to search for the designed modes. Since large numbers of computed modes will result in higher computational costs, limiting the search range with LOWRNG and HIGHRNG is recommended.
3. Modes are considered to correlate if their mass normalized cross orthogonalities are greater than MTFILTER.

MOMAX Conical Shell Static Moment

Defines a static concentrated moment load on a ring of a conical shell.

Format:

1	2	3	4	5	6	7	8	9	10
MOMAX	SID	RID	HID	S	MR	MP	MZ		

Example:

MOMAX	1	2	3	1.0	0.1	0.2	0.3		

Field	Contents
SID	Load set identification number. (Integer > 0)
RID	Ring identification number. See the RINGAX entry. (Integer > 0)
HID	Harmonic identification number or a sequence of harmonics. See Remark **5.** (Integer $\geq$ 0 or Character)
S	Scale factor. (Real)
MR, MP, MZ	Moment components in the r, ϕ, z directions. (Real)

Remarks:

1. MOMAX is allowed only if an AXIC entry is also present.
2. Load sets must be selected with the Case Control command LOAD=SID.
3. A separate entry is needed for the definition of the moment associated with each harmonic.
4. For a discussion of the conical shell problem, see “**Conical Shell Element (RINGAX)**” on page 155 of the *MSC.Nastran Reference Guide.*
5. If a sequence of harmonics is to be placed in HID, the form is as follows: “Sn1Tn2", where n1 is the start of the sequence and n2 is the end of the sequence; i.e., for harmonics 0 through 10, the field would contain “S0T10".

MOMENT Static Moment

Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		

Example:

MOMENT	2	5	6	2.9	0.0	1.0	0.0		

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
CID	Coordinate system identification number. (Integer $\geq$ 0 or blank)
M	Scale factor. (Real)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni $\neq$ 0.0 unless M is zero)

Remarks:

1. The static moment applied to grid point G is given by

$$\vec{m} = M\vec{N}$$

where $\vec{N}$ is the vector defined by (N1, N2, N3). The magnitude of $\vec{m}$ is equal to M times the magnitude of $\vec{N}$.

2. In the static solution sequences, the load set ID (SID) is selected by the Case Control command, LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
3. A CID of zero or blank references the basic coordinate system.
4. For scalar points see SLOAD.

MOMENT1 Static Moment, Alternate Form 1

Defines a static concentrated moment at a grid point by specifying a magnitude and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT1	SID	G	M	G1	G2				

Example:

MOMENT1	6	13	-2.93	16	13				

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
M	Magnitude of moment. (Real)
G1, G2	Grid point identification numbers used to define the unit vector $\vec{n}$. (Integer > 0; G1 and G2 cannot be coincident.)

Remarks:

1. The static concentrated moment applied to grid point G is given by

$$\vec{m} = M\vec{n}$$

where $\vec{n}$ is a unit vector parallel to a vector from G1 to G2.

2. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "**FOLLOWK**" on page 710). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if

geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

MOMENT2 Static Moment, Alternate Form 2

Defines a static concentrated moment at a grid point by specification of a magnitude and four grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT2	SID	G	M	G1	G2	G3	G4		

Example:

MOMENT2	6	13	-2.93	16	13	17	13		

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
M	Magnitude of moment. (Real)
Gi	Grid point identification numbers used to determine the unit vector $\hat{n}$. (Integer > 0; G1 and G2 cannot be coincident; G3 and G4 cannot be coincident.)

Remarks:

1. The static concentrated moment applied to grid point G is given by

$$\vec{m} = M\hat{n}$$

where $\hat{n}$ is the unit vector parallel to the cross product of the vectors from G1 to G2, and G3 to G4.

2. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "**FOLLOWK**" on page 710). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear

transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

MONDSP1 Displacement Monitor Point

Defines a virtual point displacement response at a user-defined reference location (ordinates and coordinates system) as a weighted average of the motions at a set of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
MONDSP1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD	INDDOF	

Example:

MONDSP1	WING195	Wing twist at station 150.							
	5	WING150	1001	120	150.0	17.0	1002		

Field	Contents
NAME	Unique character string of up to 8 characters identifying the monitor point (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point.
AXES	Component axes to monitor. (Any unique combination of the integers 1 through 6 with no embedded blanks.)
COMP	The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined.
CP	The identification number of a coordinate system in which the input (x,y,z) ordinates (and resulting displacement components) are defined.
X,Y,Z	The coordinates in the CP coordinate system at which the displacement is to be monitored.
CD	The identification number of a coordinate system in which the resulting displacement components are output.
INDDOF	Component numbers of al the independent grids from which the derived, dependent, monitor DOF's are to be computed. (Any unique combination of the integers 1 through 6 with no embedded blanks.) See Remark 3. (Default = 123)

Remarks:

1. The MONDSP1 is available only for SOLs 101, 144, and 146.
2. The LABEL is a 56 character string that should be unique among all MONDSP1 entries.
3. The INDDOF field defines the Ci field on the virtual RBE3; that is, it defines the components of the grids on the AECOMP that will be sampled to define the dependent (monitor point) displacement. Typically, the default is the correct choice. However, there is only a single grid point, all six DOF's can be used.

MONPNT1 Integrated Load Monitor Point

Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system. The integrated loads about this point over the associated nodes will be computed and printed for statics, dynamics and static aeroelastic trim analyses and form integrated loads on the nonlinear static aeroelastic database.

Format:

1	2	3	4	5	6	7	8	9	10
MONPNT1	NAME	LABEL							
	AXES	COMP	CP	X	Y	Z	CD		

Example:

MONPNT1	WING155	Wing Integrated Load to Butline 155							
	34	WING		0.0	155.0	15.0			

Field	Contents
NAME	Unique character string of up to 8 characters identifying the monitor point (Character, no Default)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point. (Character, optional)
AXES	Component axes to monitor. (Any unique combination of the integers 1 through 6 with no embedded blanks.)
COMP	The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined. (Character, no Default)
CP	The identification number of a coordinate system in which the input (x,y,z) ordinates (and resulting load components) are defined. (Integer $\geq$ 0)
X,Y,Z	The coordinates in the CP coordinate system about which the loads are to be monitored. (Real, Default = 0.0).
CD	The identification number of a coordinate system in which the resulting load components are output. (Integer $\geq$ 0)

Remarks:

1. The MONPNT1 is available only for SOLs 101, 144, and 146.

2. The LABEL is a 56 character string that should be unique among all MONPT1. It is used as additional label information in the printed output.

MONPNT2 Internal Load Monitor Point

Element Monitor Output Results Item.

Format:

1	2	3	4	5	6	7	8	9	10
MONPNT2	NAME	LABEL							
	TABLE	TYPE	NDDLitem	EID					

Example:

MONPNT2	SB100	Leading edge stringer at root							
	STRESS	CBAR	SX2A	100					

Field	Contents
NAME	A character string of up to 8 characters identifying the monitor point (Character, no Default)
LABEL	An optional string comprising of no more than 56 characters (fields 3 through 9) that identifies the monitor point. (Character, Default=blank)
TABLE	Type of output to be monitored. Options are STRESS, FORCE or STRAIN. (Character, no Default)
TYPE	Element type (Character, no Default)
NDDLitem	Component for this type to be monitored. This is the NDDL label for the particular Table and element type. (Character, no Default)
EID	Element ID. (Integer > 0)

Remarks:

1. The MONPNT2 is available only for SOL 101 and SOL 144 (Statics and Static Aeroelasticity).
2. Most element types have some items that can be monitored.
3. An assumption is made that the desired component is linear with respect to the displacement vector. If this assumption is not valid, the results will be approximate.
4. Fictitious Table / Type / NDDLitems / EID generate a warning message and are ignored.

5. NDDL descriptions for Table=FORCE can be found in the ***MD Nastran 2006 DMAP Programmer's Guide*** within the OEF datablock description. Table=STRESS and STRAIN are contained in the OES datablock description.

   Once within the datablock description you can search for the element name (or better yet, element number, see the following table) you are interested in. There can be several different descriptions for an element type. For example, real vs. complex, thermal, stress vs. strain (within the OES description), linear vs nonlinear. In addition, the shell class of elements (quad4, quad8, quadr, tria3, tria6, triar) will have a composite form (quad4lc, quud8lc, quadrlc, tria3lc, .., i.e., basename + "LC"), corner or bilin stresses (basename + "C").

   By looking at the comments contained in the text make sure you are reading from the appropriate section. The NDDLitem is labeled as the 'NAME' field within the ***MD Nastran 2006 DMAP Programmer's Guide***.

   You can also print the NDDL description for the entire database by running the following 4 statement bulk data file.

```
sol loadnddl
compile nddl=nddl,list
cend
enddata
```

MONPNT3 Integrated Load Monitor Point

Sums select Grid Point Forces to a user chosen monitor point.

Format:

1	2	3	4	5	6	7	8	9	10
MONPNT3	NAME	LABEL							
	AXES	GRIDSET	ELEMSET	CID	X	Y	Z	XFLAG	

Example:

MONPNT3	t0	Fuselage station 1105							
	1223456	1	2	0	30.0			ASM	

Field	Contents
NAME	A character string of up to 8 characters identifying the monitor point (Character, required)
LABEL	A optional string comprising of no more than 56 characters (fields 3 through 9) that identifies the monitor point.
AXES	Component axes about which to sum. Any unique combination of the integers 1 through 6 with no embedded blanks. (Integer, required)
GRIDSET	Refers to a SET1 entry that has a list of Grid Point Force grids to include at the monitored point. (Integer, required)
ELEMSET	Refers to a SET1 entry that has a list of elements to include at the monitored point. (Integer, optional)
CID	The identification number of a coordinate system in which the (x,y,z) ordinates are defined. (Default Global system). (Integer, optional)
X,Y,Z	The coordinates in the CID coordinate system about which the forces are to be summed. (Real, Default = 0.0).
XFLAG	Exclusion flag. Exclude the indicated Grid Point Force types from summation at the monitor point. Default = blank (no type excluded). S = SPCforces M = MPC forces A, L, or P = applied loads D = dmig's (and any other type not described above) at the monitored point.

Remarks:

1. The MONPNT3 is available only for SOLs 101, 144 and 146.
2. If ELEMSET is blank, no contributions are made from the set of elements attached to the grid.
3. Fictitious grids or elements do not produce error or warning messages.

MPC Multipoint Constraint

Defines a multipoint constraint equation of the form

$$\sum_j Aj\, u_j = 0$$

where u_j represents degree-of-freedom Cj at grid or scalar point Gj.

Format:

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		
		G3	C3	A3	-etc.-				

Example:

MPC	3	28	3	6.2	2		4.29		
		1	4	-2.91					

Field	Contents
SID	Set identification number. (Integer > 0)
Gj	Identification number of grid or scalar point. (Integer > 0)
Cj	Component number. (Any one of the Integers 1 through 6 for grid points; blank or zero for scalar points.)
Aj	Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)

Remarks:

1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
2. The first degree-of-freedom (G1, C1) in the sequence is defined to be the dependent degree-of-freedom. A dependent degree-of-freedom assigned by one MPC entry cannot be assigned dependent by another MPC entry or by a rigid element.
3. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

4. The m-set degrees-of-freedom specified on this entry may not be specified on other entries that define mutually exclusive sets. See the "**Degree-of-Freedom Sets**" on page 1557 for a list of these entries.
5. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in "**Parameters**" on page 1409). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,3.

MPCADD Multipoint Constraint Set Combination

Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.

Format:

1	2	3	4	5	6	7	8	9	10
MPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc.-						

Example:

MPCADD	101	2	3	1	6	4			

Field	Contents
SID	Set identification number. (Integer > 0)
Sj	Set identification numbers of multipoint constraint sets defined via MPC entries. (Integer > 0)

Remarks:

1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
2. The Sj must be unique and may not be the identification number of a multipoint constraint set defined by another MPCADD entry.
3. MPCADD entries take precedence over MPC entries. If both have the same SID, only the MPCADD entry will be used.
4. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in "**Parameters**" on page 1409). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,101.

MPCAX Conical Shell Multipoint Constraint

Defines a multipoint constraint equation of the form

$$\sum_j \text{Aj}\ u_j = 0$$

for conical shell coordinates, where u_j represents the degree-of-freedom Cj at ring RIDj and harmonic HIDj.

Format:

1	2	3	4	5	6	7	8	9	10
MPCAX	SID				RID1	HID1	C1	A1	
	RID2	HID2	C2	A2	-etc.-				

Example:

MPCAX	32				17	6	1	1.0	
	23	4	2	-6.8					

Field	Contents
SID	Set identification number. (Integer > 0)
RIDj	Ring identification number. (Integer > 0)
HIDj	Harmonic identification number. (Integer > 0)
Cj	Component number. ($1 \leq$ Integer ≤ 6)
Aj	Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)

Remarks:

1. MPCAX is allowed only if an AXIC entry is also present.
2. The first degree-of-freedom in the sequence is assumed to be the dependent degree-of-freedom. A dependent degree-of-freedom assigned by one MPCAX entry cannot be assigned dependent by another MPCAX entry.
3. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
4. Dependent degrees-of-freedom appearing on MPCAX entries may not appear on OMITAX, SPCAX, or SUPAX entries.

5. See "**Conical Shell Element (RINGAX)**" in Chapter 3 of the *MSC.Nastran Reference Guide* for further discussion of the problem.
6. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in "**Parameters**" on page 1409). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,32.

MPROCS (SOL 600)

Defines a processor entry to be used in the SOL 600 MSC.Marc Parameter Section

Defines a processor entry to be used in the SOL 600 MSC.Marc Parameter Section. It controls the use of vectorization and parallelization in the element assembly phase in MSC.Marc. Used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MPROCS	I2	I3	I4	I5	I6				
	IRF	IFGF	IAXIS	WGHT					
	Dx	Dy	Dz	X	Y	Z			

Example:

MPROCES	1	1	1	1	11				

Field	Contents
I2	Value of MSC.Marc's 2nd processor field. Number of CPUs to use. (Integer, Default = 0)
I3	Value of MSC.Marc's 3rd processor field. Optimal vector length (Defaults to 32 or 64 depending on the computer system). (Integer, Default = 0)
I4	Value of MSC.Marc's 4th processor field (enter 1 if beta matrices are to be formed in parallel). (Integer, Default = 0)
I5	Value of Marc's 6th processor field (enter 1 to use DDM single file input). (Integer, Default = 0)
I6	Value of Marc's 7th processor field -- Domain Decomposition Method. (Integer, Default = 0) Enter 11 to use Metis Best (best method of 12, 13 or 14) decomposition; Default. Enter 12 to use Metis Element-Based decomposition. Enter 13 to use Metis Node-Based decomposition. Enter 14 to use Vector decomposition. Enter 15 to use Radial decomposition. Enter 16 to use Angular decomposition.

IRF	Island removal flag for domain decomposition. (Default = 0) 0 - Do not remove islands 1 - Attempt to remove islands
IFGF	Fine graph flag for domain decomposition. (Default = 0) 0 - Coarse graph 1 - Fine graph
IAXIS	Control of point on axis of roation for radial/angular domain decomposition 0 - Use centroid of the boundary box of the model 1 - User supplied point (supply X, Y, Z below)
WGHT	Element coefficient weight. Controls balance between computational costs of domains, ragne is 0.0 to 1.0. (Default = 1.0, Use full element weight) 0.0 - Do not use element weight
Dx	First direction cosine of vector used for decomposition method 14, 15, or 16
Dy	Second direction cosine of vector used for decomposition method 14, 15, or 16
Dz	Third direction cosine of vector used for decomposition method 14, 15, or 16
X	X coordinate of point on axis
Y	Y coordinate of point on axis
Z	Z coordinate of point on axis

Remarks:

1. This entry should only be made for special cases when using DDM with MSC.Marc's single file parallel capability. Do not enter except for SOL 600 parallel executions.
2. Consult the MSC.Marc documentation volumes A, B, C for more detailed descriptions of this entry.
3. Enter only one of MPROCS entry in any given file. If more than one is entered, the first encountered will be used.
4. The continuation lines may be omitted if not required.

MSTACK (SOL 600) Defines the Direction in Which 3D Solid Composites are Stacked

Defines the direction in which 3D Solid Composites are stacked. Used in MD Nastran Implicit Noninear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
MSTACK	ID1	THRU	ID2	IDIR					

Example:

MSTACK	100	THRU	200	1					

Field	Contents
ID1	Starting solid element ID (Integer, required, no Default)
THRU	Enter the character string THRU if more than one element is desired
ID2	Ending solid element in a range of ID1 to ID2 (Integer, optional, no Default)
IDIR	Stacking direction for this range of solid elements (Integer, Required, Default = 1) 1 Thickness direction is 1-2-3-4 face to 5-6-7-8 face (for CHEXA) 2 Thickness direction is 1-5-8-4 face to 2-6-7-3 face (for CHEXA) 3 Thickness direction is 2-6-5-1 face to 3-7-8-4 face (for CHEXA)

Remarks:

1. Enter as many MSTACK lines as necessary to define composite. If IDIR is 1 for all elements in the range, this entry is not required.
2. This entry equates to MSC.Marc's EGEOM3 value for solid composite elements (for example elemnt 149).
3. This entry can only be used with MSC.Nastran 2005 or later.

MT16SPL (SOL 600)

Determines how to split a Marc t16 file into one or more smaller t16 files. Splitting of a large t16 file is sometimes necessary if the postprocessor aborts due to the large amount of data or if the results need to be transferred to another computer for postprocessing.

Format - This entry is available in small field format only:

1	2	3	4	5	6	7	8	9	10
MT16SPL	IOPT	NVECT	FNAME						
	ID1	START	LAST	INCR	(remark 4)				
	ID2	START	LAST	INCR	LastFew				

Examples:

MT16SPL	0								
	1	5	100	5	3				

MT16SPL	0		old1.mar	c.t16					
	1	10	80	10	3				
	2	90	100	5	4				

MT16SPL	1	1							
	1	100							

MT16SPL	3	2	old2.t16						
	1	10	100	10	3				

Create several new t16 files with 2 increments each using original increments 10, 20, ... 98, 99, 100. This is a restart job and uses old2.t16 (from a standalone Marc run). New files will be created as follows:

jid.0001.t16 with old increments 10, 20
jid.0002.t16 with old increments 30, 40
jid.0003.t16 with old increments 50, 60

jid.0004.t16 with old increments 70, 80
jid.0005.t16 with old increments 90, 98
jid.0006.t16 with old increments 99, 100

Field	Contents
IOPT	Option of how to split up the “old” t16 file. (Integer, Default = 0) 0 = Split into as many new t16 files as there are continuation lines of this entry, use the current jid.marc.t16 file, FNAME should not be entered. 1 = Split into as many new t16 files with NVECT original increments each, use the current jid.marc.t16 file, FNAME should not be entered. 2 = Split into as many new t16 files as there are continuation lines of this entry, use a previously generated t16 file whose name is specified by FNAME. 3 = Split into as many new t16 files with NVECT original increments each, use a previously generated t16 file whose name is specified by FNAME.
NVECT	Number of increments to be placed on each new t16 file (only used if IOPT is 1 or 3) (Integer, no Default)
FNAME	Original t16 filename - Only used if IOPT is 2 or 3. (Character, no Default) The filename may extend from fields 4-9. If it is more than 8 characters long, this entry must made in fixed format. the entire t16 filename including the t16 extension should be entered. No upper case letters may be used. (Integer > 0)
IDi	New t16 plot ID (must start with 1 and increase by 1 on each continuation line.) (Integer, no Default)
START	Starting Marc increment to be placed on new t16 file. (Integer, Default = 1)
LAST	Last Marc increment in range of start-last-incr to be n new t16 file. (Integer, default is last increment on original t16 file if FNAME is blank. If FNAME is not blank, LAST must be an accurate value which can be obtained from the .sts file of the original run that produced FNAME.)
INCR	Increments to be used for start-last-incr tgo b e on new t16 file. (Integer, Default = 1)
Last Few	In addition to START,LAST,INCR the last several increments may be placed on the t16 file. In the first example increments 5, 10, 15, ..., 100, 99, 98 are placed on the new t16 file. LastFew may only be entered on the last line. (Integer, Default = 2)

Remarks:

1. Sometimes large SOL 600 models do not converge on a user wishes to examine output at unknown time intervals. To determine what is happening, it is frequently necessary to plot the results at several output intervals. In fact, sometimes the last increment may have bogus results due to divergence. At present, MSC GUI programs sometimes are not capable of postprocessing the large amount of dta one would like to incude in a single f16 file. This option allows you to break up the t16 file into one or more smaller files.
2. GUI's might only be able to handle one increment per t16 file for extremely lalrge models. To specify this, only enter Id and START as in the 3rd example.
3. All t16 files will have the geometry as well as the specified output increments.
4. LastFew may only be entered on the last line.
5. Nastran may be restarted to perform this step. To do so, enter

   SOL 600,ID t16split=fname

   Where fname is the jid of the original job.
6. The new files will be named jid.ID.t16. Examples are as follows:

   Case 1 - Split up t16 as part of current run starting with jid1.dat as the Nastran input SOL 600 will create jid1.marc.t16 with a full set of output increments (unless reduced by you).

   jid1.0001.t16
   jid1.0002.t16
   etc.

   Case 2 - Split up a t16 file formed by a previous SOL 600 run named jid1.dat (the t16 file is named jid1.marc.t16). The current Nastran input file to split up the original t16 is named jid2.dat. The new t16 files will be designated:

   jid2.00001.t16
   jid2.00002.t16
   etc.
7. Nastran will spawn Marc's pldump2000 program to split up the original t16 file.
8. If FNAME is entered (IOPT=2 or 3), OUTR options (on SOL600,ID) will be ignored. If FNAME is blank and IOPT=0 or 1 and OUTR options are requested, the t16op2 translator process the full (unsplit) jiid.mar.t15 file. Future implementations may allow processing of the split t16 files.

9. Only one MT16SPL (plus many continuation lines as necessary) is allowed per job.

NLAUTO (SOL 600) Parameters for Automatic Load / Time Stepping

Defines parameters for automatic load / time stepping used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
NLAUTO	ID	TINIT	TFINAL	RSMALL	RBIG	TSMIN	TSMAX	NSMAX	
	NRECYC	IENHAN	IDAMP	NSTATE	NCUT	LIMTAR	IFINISH	FTEMP	
	SFACT	IFLAG	IDTAB	DAMP	IDMPFLG		PHSYS	I313	
	CRITERIA	SETID	Y1	X1	Y2	X2	Y3	X3	
		Y4	X4						

Field	Contents
ID	Identification number referenced by a NLPARM, TSTEPNL, or TSTEP Case Control command for the applicable subcase. See Remarks 1 and 3. Include a NLPARM or TSTEPNL entry for the subcase in addition to the NLAUTO entry. See Remarks 1 and 3. (Integer > 0; no Default)
TINIT (2,1)	Initial time step. (Real; Default is 1% of total time).
TFINAL (2,2)	Total time period. (Real; Default is 1.0)
RSMALL (2,3)	Smallest ratio between steps. (Real; Default = 0.1)
RBIG (2,4)	Largest ratio between steps. (Real; Default = 10.0)
TSMIN (2,5)	Minimum time step. (Real; Default is total time divided by number of time steps)
TSMAX (2,6)	Maximum time step. (Real; Default is total time)
NSMAX (2,7)	Maximum number of steps allowed. (Integer; Default = 999999)
NRECYC (2,8)	Desired number of recycles per increment. (Integer; Default = 10)

Field	Contents
IENHAN (2,9)	Enter 1 to activate the enhanced scheme, 0 otherwise. (Integer)
IDAMP (2,10)	Enter 1 to use artificial damping for static, 0 otherwise. (Integer, Default = 1) Enter 0 to not use artificial damping Enter 1 to use artificial damping if time step is less than miniumu time step Enter 2 to always use artificial damping. The value to use is given by DMAP which scales the damping matrix. Enter 4 to always use artificial damping which is determined by the strain energy in the first increment of the present load case. DAMP is used to scale the strain energy.
NSTATE (3,1)	Number of states for post file. (Integer; enter only if ienhan = 1)
NCUT (3,2)	Maximum number of times to cut down time step in an increment. (Integer) (Default = 10; enter only if ienhan = 1)
LIMTAR (3,3)	Enter 0 to create criteria as limits, 1 to treat criteria as targets. (Integer; (enter only if ienhan = 1)
IFINISH (3,4)	Enter 1 to finish time period when all nodal temperatures fall below FTEMP. Enter -1 to all nodal temperatures should exceed FTEMP. Enter 0 to omit temperature check. (Integer; enter only if ienhan = 1, Default = 0)
FTEMP (3,5)	Finish temperature, use with IFINISH. (Real; enter only if ienhan = 1, Default = 0.0)
SFACT (3,6)	Scale factor for time step changes other than changes due to user criteria. (Real) (Default = 1.2; enter only if ienhan = 1)
IFLAG (3,7)	Enter flag to override CREEP and DYNAMIC parameters as specified in the MSC.Marc input parameter section for this load case. (Integer) 0 Do not override parameters. 1 Turn off CREEP and DYNAMICS. 2 Turn off CREEP. 3 Turn off DYNAMICS.
IDTAB (3,8)	Table ID scaling damping factor (see next item) (Integer)

Field	Contents
DAMP (3,9)	Damping factor for artificial damping. The number entered here depends on the IDAMP option. If IDAMP is 1, the damping matrix is scaled by setting this factor to be the ratio of the initial damping energy to the initial strain energy (Defaults to 1e-5). If IDAMP is 2, the damping matrix is directly scaled by this factor. If IDAMP is 4, the estimated total damping energy in the subcase will be this factor times the estimated total strain energy. (Default value of 2.e-4 is used.)
IDMPFLG (3,10)	Enter 1 to put states reached by the above IDMPFLG flag on the post file. (Integer)
IPHYS (3,12)	Flag to determine if automatic physical criteria should be added and how analysis should proceed if they are not satisfied. (Integer) 2 Do not add automatic physical criteria. Stop when any user criteria are not satisfied. 1 Add automatic physical criteria. Stop when any user criteria are not satisfied. -1 Add automatic physical criteria. Continue when any user criteria are not satisfied. -2 Do not add automatic physical criteria. Stop when any user criteria are not satisified.
I313 (3,13)	An integer flag corresponding to MSC.Marc field 13.
CRITERIA (4,1)	Enter an integer corresponding to the criteria desired. See Remark 2. (Integer; no Default)
SETID (4,2)	Case Control Set ID of nodes or elements for which this criteria will apply. Restriction: Must be one of first 25 sets entered in the case control. Leave blank if “ALL” is desired. (Integer; Default is all)
Y1 (4,3)	New time step = Calculated time step / Y1 if time is less than X1. (Real)
X1 (4,4)	Time for which Y1 adjustment is applied. (Real)
Y2 (4,5)	New time step = Calculated time step / Y2 if time is between X1 and X2. (Real)
X2 (4,6)	Time for which Y2 adjustment is applied. (Real)

Field	Contents
Y3 (4,7)	New time step = Calculated time step / Y3 if time is between X2 and X3. (Real)
X3 (4,8)	Time for which Y3 adjustment is applied. (Real)
Y4 (4,9)	New time step = Calculated time step / Y4 if time is greater than X4. (Real)
X4 (4,10)	Time for which Y4 adjustment is applied. (Real)

Remarks:

1. The entry is currently only recognized by MD Nastran Implicit Nonlinear (SOL 600) can only be used by MSC.Marc 2001 and later. Currently, only one such entry may be entered.
2. Enter the following index in the CRITERIA field (a limit of 9 criteria may be specified and the usual option is to specify none).

1	Strain Increment	2	Plastic Strain Increment
3	Creep Strain Increment	4	Normalized Creep Strain Increment
5	Stress Increment	7	Strain Energy Increment
8	Temperature Increment	9	Displacement Increment
10	Rotation Increment		

3. Values entered on NLAUTO override values with the same meaning if entered elsewhere (for example on the NLPARM or TSTEPNL entry)
4. If the NLAUTO entry is used, there should also be a corresponding NLPARM or TSTEPNL Bulk Data entry for the applicable subcase (or above all subcases if applicable to all subcases. The matching NLPARM entry must have KMETHOD=AUTO (or blank). If TSTEPNL is the matching entry, then field 6 must be blank or have the value ADAPT.
5. Values such as (3,7) indicated corresponding item on MSC.Marc's AUTOSTEP data block 3 field 7.
6. Items (3,7) to (3,13) were implemented starting with MSC.Nastran 2004.0.4 and are not in previous versions.

NLDAMP (SOL 600) Damping Constants

Defines damping constants for nonlinear analysis when MSC.Marc is executed from MD Nastran used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
NLDAMP	EID1	EID2	ALPHA	BETA	GAMMA				

Examples:

NLDAMP	1	2000	.025	4.5	1.0				

Field	Contents
EID1	First element for which the damping values will be used. (Integer $\geq$ 0; required)
EID2	Last element for which the damping values will be used. (Integer $\geq$ 0 or blank)
ALPHA	Mass Matrix Multiplier. (Real; Default= 0.0)
BETA	Stiffness Matrix Multiplier. (Real; Default = 0.0)
GAMMA	Numerical Damping Multiplier. (Real; Default = 0.0)

Remarks:

1. This entry matches MSC.Marc's Damping definition.
2. NLDAMP is recognized only when MSC.Marc is executed from MD Nastran Implicit Nonlinear (SOLNINL and SOL 600).

NLPARM Parameters for Nonlinear Static Analysis Control

Defines a set of parameters for nonlinear static analysis iteration strategy.

Format:

1	2	3	4	5	6	7	8	9	10
NLPARM	ID	NINC	DT	KMETHOD	KSTEP	MAXITER	CONV	INTOUT	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS	LSTOL	
	MAXBIS				MAXR		RTOLB		

Example:

NLPARM	15	5		ITER					

Field	Contents
ID	Identification number. (Integer > 0)
NINC	Number of increments. See Remark **16.** (0 < Integer < 1000; Default = 10)
DT	Incremental time interval for creep analysis. See Remark **3.** (Real ≥ 0.0; Default = 0.0 for no creep.)
KMETHOD	Method for controlling stiffness updates. See Remark **4.** (Character = “AUTO”, “ITER”, or “SEMI”; Default = “AUTO”.)
KSTEP	Number of iterations before the stiffness update for ITER method. See Remark **5.** (Integer ≥ 1; Default = 5)
MAXITER	Limit on number of iterations for each load increment. See Remark **6.** (Integer > 0; Default = 25)
CONV	Flags to select convergence criteria. See Remark **7.** (Character = “U”, “P”, “W”, or any combination; Default = “PW”.)
INTOUT	Intermediate output flag. See Remark **8.** (Character = “YES”, “NO”, or “ALL”; Default = NO)
EPSU	Error tolerance for displacement (U) criterion. See Remark **16.** and **17.** (Real > 0.0; Default = 1.0E-2;)
EPSP	Error tolerance for load (P) criterion. See Remark **16.** and **17.** (Real > 0.0; Usual Default = 1.0E-2)
EPSW	Error tolerance for work (W) criterion. See Remark **16.** and **17.** (Real > 0.0; Usual Default = 1.0E-2)

Field	Contents
MAXDIV	Limit on probable divergence conditions per iteration before the solution is assumed to diverge. See Remark **9.** (Integer $\neq$ 0; Default = 3)
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark **10.** (Integer $\geq$ 0; Default = MAXITER)
MAXLS	Maximum number of line searches allowed for each iteration. See Remark **11.** (Integer $\geq$ 0; Default = 4)
FSTRESS	Fraction of effective stress $(\bar{\sigma})$ used to limit the subincrement size in the material routines. See Remark **12.** (0.0 < Real < 1.0; Default = 0.2)
LSTOL	Line search tolerance. See Remark **12.** (0.01 < Real < 0.9; Default = 0.5)
MAXBIS	Maximum number of bisections allowed for each load increment. See Remark **13.** (-10 $\leq$ MAXBIS $\leq$ 10; Default = 5)
MAXR	Maximum ratio for the adjusted arc-length increment relative to the initial value. See Remark **14.** (1.0 $\leq$ MAXR $\leq$ 40.0; Default = 20.0)
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark **15.** (Real > 2.0; Default = 20.0)

Remarks:

1. The NLPARM entry is selected by the Case Control command NLPARM = ID. Each solution subcase requires an NLPARM command.
2. In cases of static analysis (DT = 0.0) using Newton methods, NINC is the number of equal subdivisions of the load change defined for the subcase. Applied loads, gravity loads, temperature sets, enforced displacements, etc., define the new loading conditions. The differences from the previous case are divided by NINC to define the incremental values. In cases of static analysis (DT = 0.0) using arc-length methods, NINC is used to determine the initial arc-length for the subcase, and the number of load subdivisions will not be equal to NINC. In cases of creep analysis (DT > 0.0), NINC is the number of time step increments.
3. The unit of DT must be consistent with the unit used on the CREEP entry that defines the creep characteristics. Total creep time for the subcase is DT multiplied by the value in the field NINC; i.e., DT*NINC.
4. The stiffness update strategy is selected in the KMETHOD field.

- If the AUTO option is selected, the program automatically selects the most efficient strategy based on convergence rates. At each step the number of iterations required to converge is estimated. Stiffness is updated, if (i) estimated number of iterations to converge exceeds MAXITER, (ii) estimated time required for convergence with current stiffness exceeds the estimated time required for convergence with updated stiffness, and (iii) solution diverges. See Remarks **9.** and **13.** for diverging solutions.
- If the SEMI option is selected, the program for each load increment (i) performs a single iteration based upon the new load, (ii) updates the stiffness matrix, and (iii) resumes the normal AUTO option.
- If the ITER option is selected, the program updates the stiffness matrix at every KSTEP iterations and on convergence if KSTEP $\leq$ MAXITER. However, if KSTEP > MAXITER, stiffness matrix is never updated. Note that the Newton-Raphson iteration strategy is obtained by selecting the ITER option and KSTEP = 1, while the Modified Newton-Raphson iteration strategy is obtained by selecting the ITER option and KSTEP = MAXITER.

5. For AUTO and SEMI options, the stiffness matrix is updated on convergence if KSTEP is less than the number of iterations that were required for convergence with the current stiffness.
6. The number of iterations for a load increment is limited to MAXITER. If the solution does not converge in MAXITER iterations, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e., MAXBIS is attained or MAXBIS = 0) and MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next load increment. If MAXDIV is negative, the analysis is terminated.
7. The test flags (U = displacement error, P = load equilibrium error, and W = work error) and the tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All the requested criteria (combination of U, P, and / or W) are satisfied upon convergence. See the *MSC.Nastran Handbook for Nonlinear Analysis* for more details on convergence criteria.
8. INTOUT controls the output requests for displacements, element forces and stresses, etc. YES or ALL must be specified in order to be able to perform a subsequent restart from the middle of a subcase.

INTOUT	Output Processed
YES	For every computed load increment.
NO	For the last load of the subcase or step.
ALL	For every computed and user-specified load increment.

- For the Newton family of iteration methods (i.e., when no NLPCI command is specified), the option ALL is equivalent to option YES since the computed load increment is always equal to the user-specified load increment.
- For arc-length methods (i.e., when the NLPCI command is specified) the computed load increment in general is not going to be equal to the user-specified load increment, and is not known in advance. The option ALL allows the user to obtain solutions at the desired intermediate load increments.

9. The ratio of energy errors before and after the iteration is defined as divergence rate (E^i), i.e.,

$$E^i = \frac{\{\Delta u^i\}^T \{R^i\}}{\{\Delta u^i\}^T \{R^{i-1}\}}$$

Depending on the divergence rate, the number of diverging iteration (NDIV) is incremented as follows:

If $E^i \geq 1$ or $E^i < -10^{12}$, then NDIV = NDIV + 2

If $-10^{12} < E^i < -1$, then NDIV = NDIV + 1

The solution is assumed to diverge when NDIV $\geq$ |MAXDIV|. If the solution diverges and the load increment cannot be further bisected (i.e., MAXBIS is attained or MAXBIS is zero), the stiffness is updated based on the previous iteration and the analysis is continued. If the solution diverges again in the same load increment while MAXDIV is positive, the best solution is computed and the analysis is continued to the next load increment. If MAXDIV is negative, the analysis is terminated on the second divergence.

10. The BFGS update is performed if MAXQN > 0. As many as MAXQN quasi-Newton vectors can be accumulated. The BFGS update with these QN vectors provides a secant modulus in the search direction. If MAXQN is reached, no additional ON vectors will be accumulated. Accumulated QN vectors are purged when the stiffness is updated and the accumulation is resumed.

11. The line search is performed as required, if MAXLS > 0. In the line search, the displacement increment is scaled to minimize the energy error. The line search is not performed if the absolute value of the relative energy error is less than the value specified in LSTOL.

12. The number of subincrements in the material routines (elastoplastic and creep) is determined so that the subincrement size is approximately FSTRESS $\cdot \bar{\sigma}$ (equivalent stress). FSTRESS is also used to establish a tolerance for error correction in the elastoplastic material; i.e.,

    $$\text{error in yield function} < \text{FSTRESS} \cdot \bar{\sigma}$$

    If the limit is exceeded at the converging state, the program will exit with a fatal message. Otherwise, the stress state is adjusted to the current yield surface.

13. The number of bisections for a load increment / arc-length is limited to the absolute value of MAXBIS. Different actions are taken when the solution diverges depending on the sign of MAXBIS. If MAXBIS is positive, the stiffness is updated on the first divergence, and the load is bisected on the second divergence. If MAXBIS is negative, the load is bisected every time the solution diverges until the limit on bisection is reached. If the solution does not converge after |MAXBIS| bisections, the analysis is continued or terminated depending on the sign of MAXDIV. See Remark **9.**

14. MAXR is used in the adaptive load increment / arc-length method to define the overall upper and lower bounds on the load increment / arc-length in the subcase; i.e.,

    $$\frac{1}{\text{MAXR}} \leq \frac{\Delta l_n}{\Delta l_o} \leq \text{MAXR}$$

    where Δl_n is the arc-length at step n and Δl_o is the original arc-length. The arc-length method for load increments is selected by an NLPCI Bulk Data entry. This entry must have the same ID as the NLPARM Bulk Data entry.

15. The bisection is activated if the incremental rotation for any degree-of-freedom ($\Delta\theta_x$, $\Delta\theta_y$, or $\Delta\theta_z$) exceeds the value specified by RTOLB. This bisection strategy is based on the incremental rotation and controlled by MAXBIS.

16. Default tolerance sets are determined based on model type and desired accuracy. Accuracy is under user control and can be specified on the PARAM, NLTOL entry. NLTOL's value is used only if the CONV, EPSU, EPSP and EPSW fields are blank, and if NINC is set to a value of 10 or larger. Otherwise, the NLTOL selection will be overridden. The tables below list tolerances according to NLTOL selections:

Default Tolerances for Static Nonlinear SOL 106 Models Without Gaps, Contact or Heat Transfer

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
1	High	PW	_____	1.0E-2	1.0E-3
2	Engineering	PW	_____	1.0E-2	1.0E-2
3	Prelim Design	PW	_____	1.0E-1	1.0E-1
None	Engineering	PW	_____	1.0E-2	1.0E-2

Default Tolerances for Static Nonlinear SOL 106 Models With Gaps or Contact (Enter NLTOL Values of 0 or 2 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
2	Engineering	PW	_____	1.0E-3	1.0E-5
None	Engineering	PW	_____	1.0E-3	1.0E-5

Default Tolerances for Static Nonlinear SOL 106 or 153 Models With Heat Transfer (Enter NLTOL Value of 0 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
None	Very high	PW	_____	1.0E-3	1.0E-7

Default Tolerances for Static Nonlinear SOL 400 Models Without Gaps, Contact or Heat Transfer

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	-	1.0E-3	1.0E-3
1	High	PW	-	1.0E-2	1.0E-3
2	Engineering	PW	-	1.0E-2	1.0E-2
3	Prelim Design	PW	-	1.0E-1	1.0E-1
None	Engineering	PW	-	1.0E-2	1.0E-2

Default Tolerances for Static Nonlinear SOL 400 Models With Gaps or Contact (Enter NLTOL Values of 0 or 2 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	-	1.0E-3	1.0E-3
2	Engineering	PW	-	1.0E-3	1.0E-3
None	Engineering	PW	-	1.0E-3	1.0E-3

17. The method to compute the energy (work) error is different for SOL 106 and SOL 400. For SOL 106, the energy error is computed based on the residue forces. While, for SOL 400, the energy error computed is the total energy error, which is based on the nonlinear forces acting on the structure. At the start of the iteration, these two methods give approximately the same value. However, near convergence, the SOL 106 method will field a much smaller value than that provided by the SOL 400 method. The difference in these two methods is reflected in the default values shown in Remark **16.** The reason for a new method used in SOL 400 is that it gives the true error of the physical energy. On the other hand, the error computed in SOL 106 has no counter part in the physical world.
18. For SOL 600, the only fields used are ID, NINC, DT (creep only) INTOUT, however, PARAM,MARCOTIM is recommended instead of INTOUT. For other fields, advanced convergence controls are available using NLAUTO, NLSTRAT and PARAM,MARCDEF Bulk Data entries. For SOL 600, if INTOUT is specified all NLPARM's in the file must use the same values. The first INTOUT encountered is what is actually used. For SOL 600, the initial time step for each subcase is 1/NINC of the NLPARM applicable to that subcase. If TINIT or the NLAUTO entry is entered it overrides 1/NINC as the initial time step. For arc length methods NLPCI with the same ID as

NLPARM must be entered and if AIFRACT or the NLSTRAT entry is entered it will override 1/NINC as the initial increment size. Beware that NLSTRAT entries, if used, must be entered for each subcase as well as for “subcase zero”. The ID of NLSTRAT do not correspond to the NLPARM Id or to the subcase ID but are numbered sequentially starting with zero for Marc increment zero, one for the first subcase (regardless of its ID) etc.

NLPCI Parameters for Arc-Length Methods in Nonlinear Static Analysis

Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106). This entry will be used if a subcase contains an NLPARM command (NLPARM = ID).

Format:

1	2	3	4	5	6	7	8	9	10
NLPCI	ID	TYPE	MINALR	MAXALR	SCALE		DESITER	MXINC	

Example:

NLPCI	10	CRIS	1.0	1.0			12	10	

Field	Contents
ID	Identification number of an associated NLPARM entry. (Integer > 0)
TYPE	Constraint type. See Remark **2.** (Character: “CRIS”, “RIKS”, or “MRIKS”; Default = “CRIS”)
MINALR	Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks **3.** and **4.** ($0.0 < \text{Real} \leq 1.0$; Default = 0.25)
MAXALR	Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks **3.** and **4.** ($\text{Real} \geq 1.0$; Default = 4.0)
SCALE	Scale factor (w) for controlling the loading contribution in the arc-length constraint. ($\text{Real} \geq 0.0$; Default = 0.0)
DESITER	Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. See Remarks **3.** and **4.** (Integer > 0; Default = 12)
MXINC	Maximum number of controlled increment steps allowed within a subcase. See Remark **5.** (Integer > 0; Default = 20)

Remarks:

1. The NLPCI entry is selected by the Case Control command NLPARM = ID. There must also be an NLPARM entry with the same ID. However, for creep analysis (DT $\neq$ 0.0 in NLPARM entry), the arc-length methods cannot be activated, and the NLPCI entry is ignored if specified. The NLPCI entry is not recommended for heat transfer analysis in SOL 153.

2. The available constraint types are as follows:

   TYPE = “CRIS”:

$$\left\{u_n^i - u_n^0\right\}^T \left\{u_n^i - u_n^0\right\} + w^2(\mu^i - \mu^0)^2 = \Delta l_n^2$$

   TYPE = “RIKS”:

$$\left\{u_n^i - u_n^{i-1}\right\}^T \left\{u_n^1 - u_n^0\right\} + w^2 \Delta\mu^i = 0$$

   TYPE = “MRIKS”:

$$\left\{u_n^i - u_n^{i-1}\right\}^T \left\{u_n^{i-1} - u_n^0\right\} + w^2 \Delta\mu^i(\mu^{i-1} - \mu^0) = 0$$

   where:

   w = the user-specified scaling factor (SCALE)

   μ = the load factor

   Δl = the arc-length

   The constraint equation has a disparity in the dimension by mixing the displacements with the load factor. The scaling factor (w) is introduced as user input so that the user can make constraint equation unit-dependent by a proper scaling of the load factor μ. As the value of w is increased, the constraint equation is gradually dominated by the load term. In the limiting case of infinite w, the arc-length method is degenerated to the conventional Newton’s method.

3. The MINALR and MAXALR fields are used to limit the adjustment of the arc-length from one load increment to the next by:

$$\text{MINALR} \le \frac{\Delta l_{\text{new}}}{\Delta l_{\text{old}}} \le \text{MAXALR}$$

   The arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence) and the change in stiffness. For constant arc-length during analysis, use MINALR = MAXALR = 1.

4. The arc-length Δl for the variable arc-length strategy is adjusted based on the number of iterations that were required for convergence in the previous load increment (I_{max}) and the number of iterations desired for convergence in the current load increment (DESITER) as follows:

$$\Delta l_{new} = \Delta l_{old} \sqrt{\frac{\text{DESITER}}{I_{max}}}$$

5. The MXINC field is used to limit the number of controlled increment steps in case the solution never reaches the specified load. This field is useful in limiting the number of increments computed for a collapse analysis.

NLRGAP Nonlinear Transient Load Proportional to Gap

Defines a nonlinear transient radial (circular) gap.

Format:

1	2	3	4	5	6	7	8	9	10
NLRGAP	SID	GA	GB	PLANE	TABK	TABG	TABU	RADIUS	

Example:

NLRGAP	21	3	4	XY	3	10	6	1.6	

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GA	Inner (e.g., shaft) grid for radial gap. (Integer > 0)
GB	Outer (e.g., housing) grid for radial gap. (Integer > 0)
PLANE	Radial gap orientation plane: XY, YZ, or ZX. (Character, Default = XY.)
TABK	Table ID of gap stiffness vs. time. (Integer > 0) Table ID of force vs. penetration. (Integer < 0)
TABG	Table ID for radial gap clearance as function of time. (Integer > 0)
TABU	Table ID for radial coefficient of friction as function of time. (Integer > 0)
RADIUS	Shaft radius. (Real $\geq$ 0.0, Default = 0.0)

Remarks:

1. NLRGAP must be selected with the Case Control command NONLINEAR = SID.
2. Multiple NLRGAP entries with the same SID are allowed.
3. The NLRGAP is not an element, but a nonlinear load similar to the NOLINi Bulk Data entries. It computes the relative displacements of GA and GB in the selected plane and applies appropriate nonlinear loads to simulate the radial contact.
4. The degrees-of-freedom in the XY, YZ, or ZX planes (depending on the PLANE) of GA and GB must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation. If RADIUS is > 0.0, then the in-plane rotation degree-of-freedom must also be in the solution set.

5. As with the NOLINi entries, the NLRGAP is limited to use in direct transient response solution sequences.
6. The XY, YZ and ZX planes are relative to the displacement coordinates systems of GA and GB. GA and GB should be coincident grids with parallel displacement coordinate systems. MD Nastran does not check nor enforces this. Wrong answers can occur if this rule is not followed.
7. The shaft radius is used only for the computation of friction induced torque.
8. In the underlying equations, a positive coefficient of friction is consistent with counter-clockwise shaft rotation from axis 1 towards axis 2 (anti-clockwise). A negative coefficient of friction is consistent with clockwise shaft rotation from axis 2 towards axis 1 (clockwise). See **Figure 8-128**.
9. Nonlinear forces for the grids referenced on the NLRGAP can be output with the NLLOAD Case Control command. See **Figure 8-128** for the sign conventions.

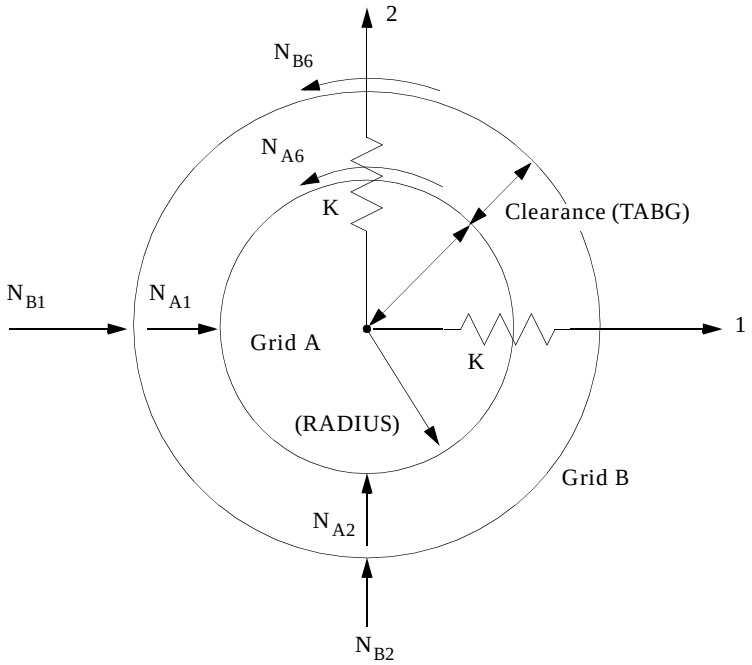


Figure 8-128 Radial Gap Orientation and Nonlinear Load Sign Conventions

10. The time step algorithm in transient solution sequences may loose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

NLRSFD Nonlinear Transient Load Proportional to a Squeeze Film Damper

Defines a nonlinear transient radial squeeze film damper.

Format:

1	2	3	4	5	6	7	8	9	10
NLRSFD	SID	GA	GB	PLANE	BDIA	BLEN	BCLR	SOLN	
	VISCO	PVAPCO	NPORT	PRES1	THETA1	PRES2	THETA2	NPNT	
	OFFSET1	OFFSET2							

Example:

NLRSFD	100	1001	1002	XY	1.0	2.0	0.05	LONG	
	2.1	300.0	1	100.0	30.0	120.0	90.0	51	
	0.01	0.0							

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0, Required)
GA	Inner (e.g., damper journal) grid for squeeze film damper. (Integer > 0, Required)
GB	Outer (e.g., housing) grid for squeeze film damper. (Integer > 0, Required)
PLANE	Radial gap orientation plane: XY, XZ, or ZX. See Remark 1. (Character, Default = XY)
BDIA	Inner journal diameter. (Real > 0.0, Required)
BLEN	Damper length. (Real > 0.0, Required)
BCLR	Damper radial clearance. (Real > 0.0, Required)
SOLN	Solution option: LONG or SHORT bearing. (Character, Default = LONG)
VISCO	Lubricant viscosity. (Real > 0.0, Required)
PVAPCO	Lubricant vapor pressure. (Real, Required)
NPORT	Number of lubrication ports: 1 or 2 (Integer, no Default)
PRES1	Boundary pressure for port 1. (Real $\geq$ 0.0, Required if NPORT = 1 or 2)

THETA1	Angular position for port 1. (0.0 ≤ Real > 360.0, Required if NPORT = 1 or 2). See Remark 2.
PRES2	Boundary pressure for port 2. (Real ≥ 0.0, Required if NPORT = 2).
THETA2	Angular position for port 2. See Remark 2. (0.0 ≤ Real < 360.0, Required if NPORT = 2)
NPNT	Number of finite difference points for damper arc. (Odd integer ≤ 201, Default = 101)
OFFSET1	Offset in the SFD direction 1. (Real, Default = 0.0)
OFFSET2	Offset in the SFD direction 2. (Real, Default = 0.0)

Remarks:

1. The XY, YZ, and ZX planes are relative to the displacement coordinates of GA and GB. The plane coordinates correspond to the NLRSFD directions 1 and 2. GA and GB should be coincident grids with parallel displacement coordinate systems. Wrong answers will be produced if this rule is not followed.
2. The angular measurement is counterclockwise from the displacement x-axis for the XY plane, the y-axis for the YZ plane, and the z-axis for the ZX plane.
3. OFFSET1 = Damper housing ID center offset displacement relative to OD center in the horizontal direction. Entered as a positive value for horizontally to the left (negative x-direction) displacement (inches).
4. OFFSET2 = Damper housing ID center offset displacement relative to OD center in the vertical direction. Entered as a positive value for downward (negative y-direction) displacement (inches). Positive entry typically used for -1 g compensation.
5. The time step algorithm in transient solution sequences may loose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

NLSTRAT (SOL 600) Strategy Parameters for Nonlinear Structural Analysis

Defines strategy parameters for nonlinear structural analysis used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
NLSTRAT	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	etc				

Example:

NLSTRAT	501	CONVTYP	4	RESPF	.015	ALPHA	.05		
	KNONPOS	1							

Field	Contents
ID	Identification number referenced by a Case Control command with regard to time steps or load steps (such as SUBCASE). If ID=0, the values entered will be used for MSC.Marc increment zero. For the first subcase ID=1; for the second subcase ID=2, etc. If there are no subcases in the model, enter ID=1. If NLSTRAT with ID > 0 is entered and if the ISOLVER option is entered, another NLSTRAT entry with ID=0 and the same ISOLVER option must be entered. If NLSTRAT is used, there must be an NLSTRAT entry for each subcase. (Integer $\geq$ 0)
PARAMi	Name of the NLSTRAT parameter. Allowable names are given in **Table 8-26**. (Character).
VALi	Value of the parameter. See **Table 8-26**. (Real or Integer)

Table 8-26 Parameters

Name	Description, Type and Value Convergence Criteria
CONVTYP	Convergence Criteria -- (Integer) - If not set, value will be determined by NLPARM or TSTEPNL entry - CONTROL(2,4) The possibilities are: 0 -- Convergence based on residuals 1 -- Convergence based on displacements 2 -- Convergence based on energy 4 -- Convergence based on residuals or displacements 5 -- Convergence based on residuals and displacements
IRELABS	Flag for relative or absolute convergence criteria (Integer) CONTROL(2,5) 0 -- Testing is done on relative error 1 -- Testing is done on absolute value 2 -- Testing is done on relative error testing unless reactions or incremental displacements are below minimum value, in which case absolute tolerance testing is used.
RCK1	Used for Relative Checking - Maximum residual force ratio (maximum value of residual force divided by maximum residual force) or displacement ratio (maximum allowable value of displacement increment divided by displacement increment) depending on convtyp (Real $\geq$ 0) CONTROL(3,1); Default = 0.1
RCK2	Used for Relative Checking - Maximum residual moment ratio or rotation ratio depending on convtyp (Real $\geq$ 0) CONTROL(3,2); Default = no checking
RCK3	Used for Relative Checking - Minimum reaction force ratio or minimum displacement ratio depending on convtyp (Real $\geq$ 0) CONTROL(3,3); no Default, if 0.0, checking is bypassed or absolute testing is performed
RCK4	Used for Relative Checking - Minimum moment ratio or rotation ratio depending on convtyp (Real $\geq$ 0) CONTROL(3,4)
ABCK1	Used for Absolute Checking - Maximum residual force ratio or displacement ratio depending on convtyp (Real $\geq$ 0) CONTROL(3,5); Default = no checking

Table 8-26 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
ABCK2	Used for Absolute Checking - Maximum residual moment ratio or rotation ratio depending on convtyp (Real ≥ 0) CONTROL(3,6) - Default = no checking
MAXDI	Maximum change in displacement increment divided by displacement increment (Real) Default is no checking (real) - CONTROL (3a-1) - Enter only if convtyp is 4 or 5.
MAXRI	Maximum change in rotational increment divided by rotational increment (Real) Default is no checking (Real) - CONTROL (3a-2) - Enter only if convtyp is 4 or 5.
MINDI	Minimum change in displacement increment divided by displacement increment (real) Default is no checking (Real) - CONTROL (3a-3) - Enter only if convtyp is 4 or 5.
MINRI	Minimum change in rotational increment divided by rotational increment (real) Default is no checking (Real) - CONTROL (3a-4) - Enter only if convtyp is 4 or 5.
MAXD	Maximum value of displacement increment (Real) Default is no checking (real) - CONTROL (3a-5) - Enter only if convtyp is 4 or 5.
MAXR	Minimum value of rotational increment (real) Default is no checking (Real) - CONTROL (3a-6) - Enter only if convtyp is 4 or 5.
IPRCONV	Flag controls printing of convergence. CONTROL(2,9) (Integer; Default = 0 = no print)
AUTOSW	Flag to turn on or off MSC.Marc's Auto Switch (Integer) Controls switching between convergence testing of residuals and displacements when residuals are small - CONTROL (2,11). 0 -- Off (Default unless NLAUTO entry is entered) 1 -- On (Default only if NLAUTO entry is entered)
Newton Iterations	
MAXSTEP	Maximum number of load steps. CONTROL(2,1) (Integer, Default = 9999)

Table 8-26 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
MAXREC	Maximum number of recycle steps per load step (Integer; Default = 3) If set to a negative value, if convergence is not obtained after maxrec recycles, a warning is issued and the analysis proceeds to the next step (not recommended) CONTROL(2,2)
MINREC	Minimum number of recycle steps per load step. CONTROL(2,3) (Integer; Default = 0)
IKMETH	Newton method (Integer; Default = 1) 1= Full Newton. 2 = Modified Newton, 3 = Newton-Raphson with strain correction, 8 = Secant stiffness CONTROL(2,6)
IKUPD	Reassembly interval of stiffness and mass. AUTO LOAD (2,2) DYNAMIC CHANGE (2,5) (Integer)
IKNONPOS	Solve a non positive definite stiffness (1) or not (0). CONTROL(2,7) (Integer)
IKINIT	Initial stiffness control (Integer) CONTROL (2,10) 0 - Normal full contribution 1 - For Mooney material, reduced contribution of hydrostatic pressure based on initial stress 2 - No initial stiffness 3 - Use stress at beginning of increment but not in the last iteration 4 - Use only positive stresses in initial stress stiffness (faster than option 0 and is always stable for thin shell structures).
General Parameter	
STRAINS	Scale factor for strain increments. PARAMETERS(2,1) (Real $\geq$ 0; Default = 1.0)
PENBOUN	Penalty value to enforce certain boundary conditions. PARAMETERS(2,2) (Real)
FSTRESS	Fraction of the hydrostatic pressure subtracted from the stress tensor in initial stress calculation. PARAMETERS(3,5) (Real)

Table 8-26 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
Load Step or Time Step Control	
MAXTSC	Maximum number of allowable time step cuts. AUTO LOAD (2,3) (Integer ≥ 0) 0 - No automatic restart from the previously converged step >1 - Maximum number of time step cutbacks allowed.
Transient Analysis Damping Parameters	
BETA	Beta parameter used by Newmark-beta procedure. PARAMETERS (2,5) (Real; Default = 0.25)
GAMMA	Gamma parameter used by Newmark-beta procedure. PARAMETERS(2,6) (Real; Default = 0.5)
GAMMA1	Gamma1 parameter used by Single Step Houbolt procedure. PARAMETERS (2,7) (Real; Default = 1.5)
GAMMA2	Gamma parameter used by Single Step Houbolt procedure. PARAMETERS (2,8) (Real; Default = -0.5)
Solver-Related Parameters	
ISOLVER	Type of solver (Integer ≥ 0) SOLVER(2,1) 0 - Profile Direct Solver 2 - Spares Iterative 4 - Sparse Direct 6 - Hardware provided direct sparse 8 - Multifrontal direct sparse (Default) 9 - The CASI iterative solver will be used
ISYMM	Nonsymmetric solver option (Integer ≥ 0; Default = 0) SOLVER (2,2) 0 - Symmetric solver 1 - Non symmetric solver
NONPOS	Nonpositive Definite solver option (Integer ≥ 0; Default = 0) SOLVER (2,3) 0 - Error if system is nonpositive-definite 1 - Solve nonpositive definite systems if possible

Table 8-26 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
MBYTE	Solver type 6 or 8 memory option (Integer ≥ 0) SOLVER (2,8) Enter the number of 4-byte words in millions to be used if solver type 6 (SGI only) or solver type 8 (all systems) is to be used. For example, if 96MB is needed, enter 96.
MAXITER	Maximum number of iterations (Iterative solver only). (Integer; Default = 1000) Enter a negative value if program is to continue even though iterations have not fully converged. SOLVER (3,1)
PREVITER	Enter 1 if the previous solution is to be used as the initial trial value (Iterative solver only) (Integer > 0; Default = 0) SOLVER (3,2)
PRECOND	Preconditioner Option (Iterative solver only) (Integer > 0) SOLVER (3,3) 0 - CASI solver with standard preconditioner 1 - CASI solver with primal preconditioner 3 - Use diagonal preconditioner 4 - Use scaled-diagonal preconditioner 5 - Use incomplete Cholesky preconditioner
CJTOL	Enter Conjugate Gradient Convergence Tolerance (Iterative solver only) (Real; Default = 0.001) SOLVER (4,1)
Arc Length and Other Parameters for MSC.Marc's AUTO INCREMENT Option	
AITYPE	Arc Length Method. AUTO INCREMENT (2,8) (Integer ≥ 0; Default = 3) 0 standard load control 1 - Crisfield quadratic constraint method 2 - Riks / Ramm linear constraint method 3 - Modified Riks / Ramm (linear constrain method) 4 - Crisfield, switch to modified Riks / Ramm if no real root found
AIMAXCUT	Maximum number or time step cutbacks. AUTO INCREMENT (2,9) (Integer ≥ 0) 0 - No automatic restart from previous converged step >0 - Maximum number of load step cutbacks
AIFRACT	Fraction of total load increment that is applied in the first cycle of the first increment. AUTO INCREMENT (2,1) (Real)

Table 8-26 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
AIMAXINC	Maximum number of increments. This value is normally determined from the NLPARM entry. AUTO INCREMENT (2,2) (Integer > 0)
AINRECYC	Desired number of recycles per increment used to increase or decrease load steps. AUTO INCREMENT (2,3) (Integer > 0; Default = 3)
AIMAXF	Maximum fraction of the total load that can be applied in any increment.It is recommended that for most nonlinear problems, this value be 0.1 or smaller. AUTO INCREMENT (2,4) (Real; Default=0.05 if the model does not have contact and 0.01 if the model has contact).
AIARCM	Maximum arc length multiplier (norm of displacement vector to initial arc length). AUTO INCREMENT (2,5) (Real; Default is fraction of load divided by initial fraction of load)
AITOTT	Total Time period to be covered, used in conjunction with contact analysis. AUTO INCREMENT (2,6) (Real; Default = 1.0)
AIARC0	Fraction of the initial arc length to define a minimal arc length. AUTO INCREMENT (2,7) (Real; Default = 0.01)
Fully Coupled Heat-Structural Analysis Controls	
TCHANGE	Maximum nodal temperature change allowed. CONTROL(4,1) (Real; Default = 20.0)
TEVAL	Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. CONTROL(4,2) (Real; Default = 100.0)
TERROR	Maximum error in temperature estimates used for property evaluation. CONTROL(4,3) (Real; Default = 0.0, which bypasses the test)
Contact-Related Parameters	
ANG2D	Angle at which a node separates from a convex corner or becomes stuck in a concave corner in 2D contact (Real, Default = 8.625 degrees) PARAMETERS (3,1)

Table 8-26 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
ANG3D	Angle at which a node separates from a convex corner or becomes stuck in a concave corner in 3D contact (Real, Default = 20.0 degrees) PARAMETERS (3,2)
Other Parameters	
DRILLF	The factor used to calculate the drilling mode for shell elements (types 22, 75, 138, 139, 140) (Real, Default = 0.0001) PARAMETERS (3,6)
REZONEF	Incremental displacement scale factor after a rezoning increment (Real, Default=1.0). Note that a value of 1.0 improves friction convergence but may result in an inside-out element. PARAMETERS (3,7)
UGAS	Universal Gas Constant (Real, Default = 8.314 J $mol^{1}K^{1}$ PARAMETERS (3,1)
TOFSET	Offset temperature between user units and absolute zero temperature (Real, Default=273.15 which is correct for Centigrade). If temperature units are Kelvin (K) or Rankine (R), enter a negative value and the temperature offset is set to zero PARAMETERS (3,2)
TWEIGHT	Thermal properties evaluation weight (Real, Default = 0.5) PARAMETERS (3,3)
SPFACT	Surface projection factor for single step Houbolt method (Real, Default = 0.0) PARAMETERS (3,4)
STEFAN	Stefan Boltzman Constant (Real, Default = 5.67051E-8 $W/m^{2}K^{4}$ PARAMETERS (3,5)
New Items for Version 2005 r3	
IASMBL	Assembly flag. If set to 1, the stiffness matrix is assembled each iteration. Note that this switches off the modified Newton-Raphson procedure if chosen. (Integer)
INNER	For some material models, such as damage, cracking, and Chaboche, there is an inner iteration loop to insure accuracy. The maximum number of iterations allowed can be set here. (Integer; Default = 50)

Table 8-26 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
RIGLNK	Rigid Link Rotation Tolerance: Maximum allowable value of the change in rotation increment at the retained nodes of RBE2, rigid link 80 or beam-shell offset nodes. Default is 0.0, in which case, no checking on rigid link rotations takes place. (Real)
RLROTT	Rigid Link Rotation Tolerance: Maximum allowable value of the change in rotation increment at the retained nodes of RBE2, rigid link 80 or beam-shell offset nodes. Default is 0.0, in which case, no checking on rigid link rotations takes place. (Real) **Note:** If CONVTYP is 4 or 5, the rigid link rotation tolerance entered here circumvents the corresponding value RIGLNK above. For all versions with PARAM,MARCVER less than 12, the rigid link rotation tolerance if left at 0, is reset to 0.001 radians to ensure backward compatibility for RBE2. In this case, the rigid link rotation tolerance should be set to a negative number to by-pass the check.
ENRGCH	Maximum allowable value of the change is energy increment. Enter only if CONVTYP=2. (Real; Default = 0.1)

Remarks:

1. This entry matches MSC.Marc's CONTROL, AUTO LOAD, DYNAMIC CHANGE, PARAMETERS, and SOLVER definitions.
2. NLSTRAT is recognized only when MSC.Marc is executed from MD Nastran Implicit Nonlinear (SOL 600).

3. Correlation between NLSTRAT names and MSC.Marc CONTROL entry fields

2-1 MAXSTEP	3-1 RCK1	4-1 MAXDI	5-1 MAXENRG	6-1 TCHANGE
2-2 MAXREC	3-2 RCK2	4-2 MAXRI		6-2 TEVAL
2-3 MINREC	3-3 RCK3	4-3 MINDI		6-3 TERROR
2-4 CONVTYP	3-4 RCK4	4-4 MINRI		6-4 VOLTMAX
2-5 IRELABS	3-5 ABCK1	4-5 MAXD		
2-6 IKMETH	3-6 ABCK2	4-6 MAXR		7-1 ESRELER
2-7 IKNONPOS	4-7 RIGLNK	4-7 RLROTT		7-2 ESABSER
2-8 Not Used				
2-9 IPRCONVs				ENRGCH
2-10 IKINT				
2-11 AUTOSW				
2-12 IASMBL				
2-13 INERR				

- (Items 2-12,2-13,3-7,4-7,5-1,6-4,7-1,7-2 are available starting with MSC.Marc and MSC.Nastran version 2005 r3. Items 6-4,7-1,7-2 are not presently available using SOL 600).
- 3-1 to 3-6 is entered only if CONVTYP=0, 4 or 5.
- 4-1 to 4-6 is entered only if CONVTYP=1, 4 or 5.
- 5-1 is entered only if CONVTYP=2.

4. Correlation between NLSTRAT names and MSC.Marc PARAMETERS entry fields

2-1 STRAINS	3-1 ANG2D	4-1 UGAS
2-2 PENBOUN	3-2 ANG3D	4-2 TOFSET
2-3 PFPLAS	3-3 RATE0	4-3 TWEIGHT
2-4 PFFLUID	3-4 RATEC	4-4 SPFACT
2-5 BETA	3-5 FSTRESS	4-5 STEFAN
2-6 GAMMA	3-6 DRILLF	
2-7 GAMMA1	3-7 REZONEF	
2-8 GAMMA2		

5. Correlation between NLSTRAT names and MSC.Marc SOLVER entry fields

   2-1 ISOLVER 3-1 MAXITER 4-1 CJTOL

   2-2 ISYMM 3-2 PREVITER

   2-3 NONPOS 3-3 PRECOND

   2-8 MBYTE

6. Correlation between NLSTRAT names and MSC.Marc AUTO INCREMENT entry fields

   2-1 AIFRACT 2-6 AITOTT

   2-2 AIMAXINC 2-7 AIARC0

   2-3 AINRECYC 2-8 AITYPE

   2-4 AIMAXF 2-9 AIMAXCUT

   2-5 AIARCM

7. Correlation between NLSTRAT names and MSC.Marc AUTO LOAD entry fields

   2-2 IKUPD

   2-3 MAXTSC

8. Correlation between NLSTRAT names and MSC.Marc DYNAMIC CHANGE entry fields

   2-5 IKUPD

9. The ISOLVER must be the same for all load cases (and Phase 0) or MSC.Marc will abort. It is recommended that all other SOLVER items also be the same for the entire run. If ISOLVER is a value other than 8, NLSTART with ID=0 must be entered with the desired ISOLVER type. Multiply NLSTRAT entries with ID=1, 2, etc. may be used to change other values if desired.

10. ISYMM must be the same for all load cases. All discussions from note 10 apply to ISYMM.

NOLIN1 Nonlinear Transient Load as a Tabular Function

Defines nonlinear transient forcing functions of the form

$$\text{Function of displacement: } P_i(t) = S \cdot T(u_j(t)) \qquad \text{Eq. 8-5}$$

$$\text{Function of velocity: } P_i(t) = S \cdot T(\dot{u}_j(t)) \qquad \text{Eq. 8-6}$$

where $u_j(t)$ and $\dot{u}_j(t)$ are the displacement and velocity at point GJ in the direction of CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN1	SID	GI	CI	S	GJ	CJ	TID		

Example:

NOLIN1	21	3	4	2.1	3	10	6		

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 < Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

TID	Identification number of a TABLEDi entry. (Integer > 0)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.
2. Nonlinear loads may not be referenced on DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN1 entries must be members of the solution set. This means the e-set (EPOINT entry) for modal formulation and the d-set for direct formulation.
4. Nonlinear loads as a function of velocity (**Eq. 8-6**) are denoted by components ten greater than the actual component number; i.e., a component of 11 is component 1 (velocity). The velocity is determined by

$$\dot{u}_{j,\,t} = \frac{u_{j,\,t} - u_{j,\,t-1}}{\Delta t}$$

   where Δt is the time step interval and $u_{j,\,t-1}$ is the displacement of GJ-CJ for the previous time step.
5. The time step algorithm in transient solution sequences may loose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

NOLIN2 Nonlinear Transient Load as the Product of Two Variables

Defines nonlinear transient forcing functions of the form

$$P_i(t) = S \cdot X_j(t) \cdot X_k(t)$$

where $X_j(t)$ and $X_k(t)$ can be either displacement or velocity at points GJ and GK in the directions of CJ and CK.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN2	SID	GI	CI	S	GJ	CJ	GK	CK	

Example:

NOLIN2	14	2	1	2.9	2	1	2		

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 ≤ Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ, GK	Grid, scalar, or extra point identification number. (Integer > 0)
CJ, CK	Component number for GJ, GK according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR=SID.
2. Nonlinear loads may not be referenced on a DLOAD entry.

3. All degrees-of-freedom referenced on NOLIN2 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. GI-CI, GJ-CJ, and G
   K-CK may be the same point.
5. Nonlinear loads may be a function of displacement ($X = u$) or velocity ($X = \dot{u}$). Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_t = \frac{u_t - u_{t-1}}{\Delta t}$$

   where Δt is the time step interval and u_{t-1} is the displacement of GJ-CJ or GK-CK for the previous time step.
6. The time step algorithm in transient solution sequences may loose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

NOLIN3 Nonlinear Transient Load as a Positive Variable Raised to a Power

Defines nonlinear transient forcing functions of the form

$$P_i(t) = \begin{cases} S \cdot [X_j(t)]^A, & X_j(t) > 0 \\ 0, & X_j(t) \le 0 \end{cases}$$

where $X_j(t)$ may be a displacement or a velocity at point GJ in the direction of CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN3	SID	GI	CI	S	GJ	CJ	A		

Example:

NOLIN3	4	102		-6.1	2	15	-3.5		

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which the nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 ≤ Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ	Grid, scalar, extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

A	Exponent of the forcing function. (ReaI)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN3 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. Nonlinear loads may be a function of displacement $(X_j = u_j)$ or velocity $(X_j = \dot{u}_j)$. Velocities are denoted by a component number ten greater than the actual component number; e.g., a component of 16 is component 6 (velocity). The velocity is determined by

$$\dot{u}_{j,\,t} = \frac{u_{j,\,t} - u_{j,\,t-1}}{\Delta t}$$

   where Δt is the time step interval and $u_{j,\,t-1}$ is the displacement of GJ-CJ for the previous time step.
5. Use a NOLIN4 entry for the negative range of $X_j(t)$.
6. The time step algorithm in transient solution sequences may loose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

NOLIN4 Nonlinear Transient Load as a Negative Variable Raised to a Power

Defines nonlinear transient forcing functions of the form

$$P_i(t) = \begin{cases} -S \cdot [-X_j(t)]^A, & X_j(t) < 0 \\ 0, & X_j(t) \geq 0 \end{cases}$$

where $X_j(t)$ may be a displacement or a velocity at point GJ in the direction of CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN4	SID	GI	CI	S	GJ	CJ	A		

Example:

NOLIN4	2	4	6	2.0	101		16.3		

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 < Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

A	Exponent of forcing function. (Real)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN4 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. Nonlinear loads may be a function of displacement $(X_j = u_j)$ or velocity $(X_j = \dot{u}_j)$. Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_{j,\,t} = \frac{u_{j,\,t} - u_{j,\,t-1}}{\Delta t}$$

   where Δt is the time step interval and $u_{j,\,t-1}$ is the displacement of GJ-CJ for the previous time step.
5. Use a NOLIN3 entry for the positive range of $X_j(t)$.
6. The time step algorithm in transient solution sequences may loose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

NSM Non Structural Mass entry by ID

Defines a set of non structural mass.

Format:

1	2	3	4	5	6	7	8	9	10
NSM	SID	TYPE	ID	VALUE	ID	VALUE	ID	VALUE	

Example:

NSM	3	PSHELL	15	.022					

Field	Contents
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0)
VALUE	NSM value (Real)

Remarks:

1. Non structural mass sets must be selected with Case Control command NSM = SID.
2. For CCONEAX the element ID is $1000 \cdot ID + i$, where i = 1 to number of harmonics.
3. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSM1 Alternate Form for NSM Entry

Defines non structural mass entries by VALUE,ID list.

Format:

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	ID	ID	ID	ID	ID	
	ID	ID	ID	etc.	-				

Example:

NSM1	3	ELEMENT	.044	1240	1500				

Alternate Form and Example(s):

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	ID	THRU	ID			

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	ALL					

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	ID	THRU	ID	BY	N	

Field	Contents
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
VALUE	NSM value (Real)
ID	Property or Element ID. (Integer > 0 or “ALL” or “THRU” or “BY” or N (the BY increment))

Remarks:

1. Non structural mass sets must be selected with Case Control command NSM = SID.
2. For CCONEAX the element ID is $1000 \cdot ID + i$, where i = 1 to number of harmonics.
3. PBEAML and PBCOMP are treated as PBEAM, PBARL is treated as PBAR, and PCOMP is treated as PSHELL; therefore a command such as:

   NSM1,12,PCOMP,0.045,ALL

   would for example get all PSHELLs in the file. The converted PCOMPs plus any existing PSHELLS would have .045 added to their nonstructural mass.
4. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSMADD Non Structural Mass Set Combination

Defines non structural mass as the sum of the sets listed.

Format:

1	2	3	4	5	6	7	8	9	10
NSMADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	S10	etc.	-				

Example(s):

NSMADD	3	17	18	19	20	22	23	24	
	25	26	27	28					
NSMADD	3	29	40	50	55				

Field	Contents
SID	Identification number of non structural mass set. (Integer > 0)
Si	Identification numbers of non structural mass sets defined via NSM, NSML, NSM1, and NSML1 entries. (Integer > 0; SID ≠ Si

Remarks:

1. The non structural mass sets must be selected with the Case Control command NSM = SID.
2. No Si may be the identification number of a non structural mass set defined by another NSMADD entry.
3. NSMADD entries take precedence over NSM, NSML, NSM1 or NSML1 entries. If both have the same set ID, only the NSMADD entry will be used.
4. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSML Lumped Non Structural Mass Entry by ID

Defines a set of lumped non structural mass.

Format:

1	2	3	4	5	6	7	8	9	10
NSML	SID	TYPE	ID	VALUE	ID	VALUE	ID	VALUE	

Example:

NSML	3	PSHELL	15	.66					

Field	Contents
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0)
VALUE	A lumped mass value to be distributed (Real).

Remarks:

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with Area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.
2. For Area elements the calculation is NSM = VALUE / $\Sigma_{elements} Area$ and for Line elements the calculation is NSM = VALUE / $\Sigma_{elements} Length$.
3. Non structural mass sets must be selected with Case Control command NSM = SID.
4. This entry is not allowed for the CCONEAX element.
5. This entry will cause an equivalent NSM entry to be generated using the computed value for NSM.
6. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

NSML1 Alternate Form for NSML Entry

Defines lumped non structural mass entries by VALUE,ID list.

Format:

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID	ID	ID	ID	ID	
	ID	ID	ID	etc.	-				

Example:

NSML1	3	ELEMENT	.044	1240	1500				

Alternate Form and Example(s):

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID	THRU	ID	ID	THRU	
	ID	ID	THRU	ID	ID	THRU	ID	ID	
	THRU	ID	...						

1	2	3	4	5	6	7	8	9	10
NSML1	15	PSHELL	.067	1240	THRU	1760			
	2567	THRU	2568	35689	THRU	40998			
	76	THRU	300						

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID	THRU	ID	BY	N	
	ID	THRU	ID	BY	N	...			

1	2	3	4	5	6	7	8	9	10
NSML1	3	PSHELL	.067	1240	THRU	1760	1763	1764	
	2567	THRU	2568	35689	TO	40999	BY	2	
	76666	76668	79834						

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ALL					

1	2	3	4	5	6	7	8	9	10
NSML1	59	PTUBE	.0123	ALL					

Field	Contents
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
VALUE	A lumped mass value to be distributed (Real)
ID	Property or Element ID. (Integer > 0 or “ALL” or “THRU” or “TO” or “BY” or N (the BY increment))

Remarks:

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with Area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.
2. For Area elements the calculation is NSM = VALUE / $\Sigma_{elements} Area$ and for Line elements the calculation is NSM = VALUE / $\Sigma_{elements} Length$.
3. For NSML1 entries with multiple “THRU” and “THRU,BY” and “ID” lists or any such combination of entries, the NSM = VALUE / $\Sigma_{elements} Area$ and for Line elements the calculation is NSM = VALUE / $\Sigma_{elements} Length$ is based on the individual parent card plus all continuation entries. If an element appears more then once in these multiple combinations, its area or length will be used multiple times in the sun.
4. Nonstructural mass sets must be selected with Case Control command NSM=SID.
5. This entry is not allowed for the CCONEAX element.
6. PBEAML and PBCOMP are treated as PBEAM, PBARL is treated as PBAR, and PCOMP is treated as PSHELL; therefore a command such as:

   NSML1,12,PCOMP,1.35,ALL

   would, for example, get all PSHELLs in the file. The converted PCOMPs plus any existing PSHELLS would have a mass of 1.35 added to their nonstructural mass.

7. The ELSUM Case Control command will give a summary of both structural and non structural mass by element or property type.
8. With the “THRU” and “THRU”, “BY” forms, blanks fields are allowed for readability. Any combination of a list of IDs and “THRU” and “THRU”, “BY” is allowed. “TO” and “THROUGH” are substitutes for “THRU”. The “THRU” and “BY” lists may have missing IDs. That is the list of IDs in a THRU range need not be continuous.

NTHICK (SOL 600) Nodal Thickness Values

Defines nodal thickness values for beams, plates and / or shells. This is the MSC.Marc's nodal thickness option used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
NTHICK	ID1	ID2	THICK						

Example:

NTHICK	151	180	0.255						

Field	Contents
ID1	First Nodal ID to which the thickness applies. (Integer > 0)
ID2	Last Nodal ID to which the thickness applies. (Integer, Default = EID1)
THICK	Thickness for all beam, plate or shell elements connecting the nodes specified. (Real > 0.0)

Remarks:

1. This entry only applies when MSC.Marc is executed from MD Nastran using MD Nastran Implicit Nonlinear (SOL 600) and is ignored for other solutions.
2. The option allows specification of beam, plate and / or shell thickness on a nodal basis. Thickness values specified on property entries overrides values specified by this entry.
3. For all elements including composite elements, nodal thickness is the total thickness.
4. Discontinuities must be modeled using property entries.

OMIT Omitted Degrees-of-Freedom

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
OMIT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

OMIT	16	2	23	3516			1	4	

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; zero or blank for scalar points.)

Remarks:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 1557 for a list of these entries.
2. Up to 24 degrees-of-freedom may be specified on a single entry.
3. In many cases it may be more convenient to use OMIT1, ASET, or ASET1 entries.
4. In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.

OMIT1 Omitted Degrees-of-Freedom, Alternate Form 1

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	-etc.-					

Example:

OMIT1	3	2	1	3	10	9	6	5	
	7	8							

Alternate Format and Example:

OMIT1	C	G1	“THRU”	G2					
OMIT1	0	17	THRU	109					

Field	Contents
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; zero or blank for scalar points.)
Gi	Grid or scalar point identification number. (Integer > 0; for “THRU” option, G1 < G2.)

Remarks:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 1557 for a list of these entries.
2. If the alternate format is used, not all points in the range G1 through G2 have to be defined. Undefined points will collectively produce a warning message but will otherwise be ignored.
3. In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.

OMITAX Omitted Conical Shell Degrees-of-Freedom

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
OMITAX	RID1	HID1	C1	RID2	HID2	C2			

Example:

OMITAX	2	6	3	4	7	1			

Field	Contents
RIDi	Ring identification number. (Integer > 0)
HIDi	Harmonic identification number. (Integer ≥ 0)
Ci	Component number(s). (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

Remarks:

1. OMITAX is allowed only if an AXIC entry is also present.
2. Up to 12 degrees-of-freedom may be specified on this entry.
3. Degrees-of-freedom appearing on OMITAX entries may not appear on MPCAX, SUPAX, or SPCAX entries.
4. For a discussion of the conical shell problem, see the “Conical Shell Element (RINGAX) on page 155 of the *MD Nastran Reference Guide*.

OUTPUT Output Control for Adaptive Analysis

Output control for p-adaptive analysis.

Format:

1	2	3	4	5	6	7	8	9	10
OUTPUT	SID								
	ELSET=n, cmd1=(option1, option2, etc.), cmd2=(option1, etc.), etc.								
	ELSET=m, -etc.-								

Example:

OUTPUT	127								
	ELSET=12, DISP=PRINT, STRESS=(PRINT,PUNCH), STRAIN=PUNCH								
	ELSET=42, STRESS=PRINT,BY=1								

Field	Contents	Type	Default
SID	DATAREC ID selected in Case Control. See Remark **1.**	Integer > 0	Required
ELSET	ID of SET entry containing sets of elements with results that will be processed. See Remark **1.**	Integer > 0	999999
cmdi	Output commands.	See below	
optioni	Specifies one or more of the following output options. The following options may be specified in any order without regard to field boundaries.	See below	Required
DISP	Request for calculating displacements. See Remark 1**a.**	Character	DISP = PRINT
VELO	Request for calculating velocities. See Remarks 1**a.**, **7.**, and **8.**	Character	VELO = NONE
ACCE	Request for calculating accelerations. See Remarks 1**a.**, **7.**, and **8.**	Character	ACCE = NONE
STRESS	Request for calculating stresses. See Remark 1**a.**	Character	STRESS = PRINT

Field	Contents	Type	Default
STRAIN	Request for calculating strains. See Remark 1a.	Character	STRAIN = NONE
FORCE	Request to output forces/length in shell elements or forces in beam elements.	Character	FORCE = NONE
ERROR	Request for error estimate table. See Remark 1a.	Character	ERROR = PRINT
PVAL	Request for new pval values. See Remark 1a.	Character	PVAL = PRINT
LAST	Request to print results of last analysis in an adaptive analysis. See Remark 1b.	Character	LAST = YES
BY	Request to print intermediate results in an adaptive analysis. See Remark 1c.	Integer≥0	BY = 0
FIRST	Request to print results of first analysis in an adaptive analysis. See Remark 1b.	Character	FIRST = YES

Remarks:

1. ELSET = n indicates the start of a new set of commands. Commands appearing after ELSET apply only to elements in SET n.
   a. For cmdi: DISP, VELO, ACCE, STRESS, STRAIN, FORCE, ERROR, and PVAL the allowable optioni are PRINT, PLOT, PUNCH, REAL, IMAG, PHASE, or NONE. If more than one option is desired, enclose in parentheses; e.g., DISP = (PRINT, PUNCH).
   b. For cmdi: STRAIN, the allowable optioni are STRCUR, FIBER, PRINT, PLOT, PUNCH, or NONE. If more than one option is desired, enclose in parentheses; e.g., STRAIN = (FIBER, PRINT, PUNCH). The options STRCUR and FIBER are for shell elements, they are ignored for other elements. For STRCUR membrane strain and curvature are output, for FIBER, strains in the fibers Z1 and Z2 are output. Z1 and Z2 are specified on the PSHELL Bulk Data entry. The Default is STRCUR. Either STRCUR or FIBER should be specified, but not both.
   c. For cmdi: FIRST and LAST, the allowable optioni are YES and NO. For example, “FIRST = YES”.
   d. For cmdi: BY, the allowable optioni is an integer greater than or equal to 0. optioni specifies that cmdi will be processed at every adaptive cycle that is a multiple of optioni.

2. Only the output (displacements, stresses, etc.) requested will be either printed or stored for postprocessing. optioni = PRINT (for print in F06 file), PUNCH (for print in punch file), and PLOT (for calculation to be used by postprocessing but not printing) can be used in any combination. For example, DISP = (PRINT), STRESS = (PRINT,PUNCH) will result in printing of displacement data in the F06 file and printing of the stress data in both the F06 file and the punch file.
3. If an element is specified in more than one ELSET = n, then the union of all commands will be performed on that element.
4. SET = 999999 is a reserved set that includes all elements.
5. A command and its options must be specified entirely on the same entry.
6. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
7. VELO and ACCE output commands are only available for transient and frequency response problems.
8. For modal transient and modal frequency analyses with the default matrix data recovery method, requests of velocity or acceleration output must be accompanied by the displacement request for the same set of elements (ELSET). THe complex output formats of displacements, velocities, and accelerations are specified by the REAL, IMAG, or PHASE option of the DISP command.
9. The REAL or IMAG option (the default) is used to request rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
10. The PHASE option is used to request polar format (magnitude and phase) of complex output. Phase output is in degrees.

OUTRCV Output Options for p-elements

Defines options for the output of displacements, stresses, and strains of p-elements.

Format:

1	2	3	4	5	6	7	8	9	10
OUTRCV	SID	SETID							
	OPTION1				OPTION2				

Examples:

OUTRCV	150	160							
	CID=2		VIEW=3*3*9						

OUTRCV	3	5							

Field	Contents	Type	Default
SID	Identification number. SID is selected by the OUTRCV Case Control command.	Integer > 0	Required
SETID	Set identification number of a SET Case Control command that appears after the SETS DEFINITION or OUTPUT(POST) command.	Integer > 0	999999
OPTIONi	Specifies one or more of the following options. The following options may be specified in any order without regard to field boundaries.	See CID and VIEW below	
CID	Specifies the output coordinate system for all stresses, strains, and displacements, except displacements at points defined by GRID entries. CID = 0 specifies the basic coordinate system; and CID = id specifies a CORDij entry. See Remark 4.	Integer $\geq$ 0	CID = 0

Field	Contents	Type	Default
VIEW	Specifies the intervals for displacement, stress, and strain $\xi \cdot \eta \cdot \zeta$ is the number of subdivisions in $\xi \cdot \eta \cdot \zeta$ of the element's output recovery parametric system. See Remark **5.**	Three Integers separated by "*"	VIEW = 3*3*3
PROJ	Specifies the orientation of a convective coordinate system for shells. PROJ = X specifies the coordinate axis in the CID system which is projected to define the x-axis of the convective coordinate system (tangent system) for shells and beams. Ignored for solids. A minus sign specifies the reverse direction. See Remarks **5.** and **10.** for more details.	Character; F, X, Y, Z, -X, -Y, -Z	PROJ = X
NORMAL	Specifies the positive direction of the outward normal for shell elements in the CID coordinate system. For NORMAL=R, the positive direction of the outward normal is the exiting arrow side of a radius vector from the origin of the CID system to the element center. For NORMAL = E, the positive direction of the outward normal is the z-axis of the element coordinate system. A minus sign specifies the reverse direction. See Remark **10.** for more details.	Character; R, E, -R, -E, X, Y, Z, -X, -Y, -Z	NORMAL = R
THETA	Angle in degrees which rotates the convective system defined with CID and PROJ. THETA is measured in the tangent plane of the shell from the projected axis (selected in PROJ) to the x-axis of the final output coordinate system. For shell elements only.	Real	THETA = 0.

Remarks:

1. OUTRCV is intended for p-elements only and specifies the coordinate system and density used for displacement, stress, strain, and force output. OUTRCV is used only for output and has no effect on the solution.
2. On the continuation entries, no commas can appear in columns 1 through 8 and the data in fields 2 through 9 must be specified in columns 9 through 72.
3. Sets referenced by SETID are defined on the SET command after the SETS DEFINITION or OUTPUT(POST) command. Any p-element not referenced by the SET = SETID Case Control command will use the defaults listed above for CID and VIEW.
4. If an element is referenced by more than one OUTRCV entry then a warning message will be issued and the last OUTRCV will be applied to the element.
5. ξ*η*j represents the ξ, η and ζ subdivisions in the solid element's output recovery parametric system. Both "*" delimiters are required. η is ignored for the CPENTA and CTETRA element and ξ is ignored for the CTETRA, CQUAD4, and CTRIA3 element.
6. The elements referenced by the SET = SETID command are labeled in the stress output as VUHEXA, VUPENTA, VUTETRA, VUQUAD, VUTRIA, and VUBEAM. They may be renamed via the PARAM,VUHEXA; PARAM,VUPENTA; PARAM,VUTETRA; PARAM,VUQUAD; PARAM,VUTRIA; and PARAM,VUBEAM entries.
7. Only one OUTRCV Case Control command is allowed. Multiple OUTRCV Bulk Data entries with the same SID are allowed to specify multiple element sets with different output coordinate systems.
8. The displacement output at locations defined by the GRID Bulk Data entry are determined by the CD value located on the GRID Bulk Data entry.
9. For p-version shell elements, the default output coordinates system is the convective coordinate system tangent to the shell mid surface. The x-axis of the convective system is the projected x-axis of the basic system. For p-version beam elements, the output system is the convective coordinate system tangent to the beam axis, oriented from grid A to grid B, as specified on the CBEAM entry.

10. The PROJ and NORMAL options for shells are described in the following list.

PROJ	Defines the orientation of the output coordinate system for stresses, strains and forces in shell elements. The reference system for PROJ is the CID coordinate system.
PROJ = F	Stresses, strains and forces of shells are output in the fixed CID. This option should be used if a postprocessor requires the results in terms of 3D vectors or tensors. For example, stress tensors with 6 components. The option does not produce output in the f06 file.
PROJ = X, Y, Z	The x- or y- or z-axis of the CID system is projected on to the shell tangent plane, the projected vector defines the x-axis of the convective coordinate system for output of stresses, strains and forces.
NORMAL	Specifies the positive normal direction of the output coordinate system for stresses, strains, and forces in shell elements. The reference system for NORMAL is the CID coordinate system.
NORMAL = R	The positive direction of the normal is the exiting arrow of the position vector from the origin of the CID system to the element center.
NORMAL = E	The positive direction of the normal is the z-axis of the element coordinate system.
NORMAL = X	The positive direction of the outward normal is the exiting arrow of the x-axis.
NORMAL = Y,Z	See above.

PAABSF Frequency-Dependent Absorbers Element Property

Format

1	2	3	4	5	6	7	8	9	10
PAABSF	PID	TZREID	TZIMID	S	A	B	K	RHOC	

Example:

PAABSF	44	38	47						

Field	Contents
PID	Property identification number that matches the identification number of the corresponding CAABSF entry. (Integer > 0)
TZREID	Identification number of a TABLEDi entry that defines the resistance as a function of frequency. The real part of the impedence. See Remark **1.** (Integer > 0)
TZIMID	Identification number of a TABLEDi entry that defines the reactance as a function of frequency. The imaginary part of the impedance. See Remark **1.** (Integer > 0)
S	Impedance scale factor. (Real; Default = 1.0)
A	Area factor when 1 or 2 grid points are specified on the CAABSF entry. (Real > 0.0; Default = 1.0)
B	Equivalent structural damping coefficient. (Real $\geq$ 0.0; Default = 0.0)
K	Equivalent structural stiffness coefficient. (Real $\geq$ 0.0; Default = 0.0)
RHOC	Constant used in data recovery for calculating an absorption coefficient. RHO is the media density, and C is the speed of sound in the media. (Real; Default = 1.0)

Remarks:

1. At least one of the four fields TZREID, TZIMID, B, or K must be specified.
2. If only one grid point is specified on the CAABSF entry, then the impedance $Z(f) = Z_R + iZ_I$ is the total impedance at the point. If two grids are specified, then the impedance is the impedance per unit length. If three or four points are specified, then the impedance is the impedance per unit area. $Z_R(f) = \text{TZREID}(f) + B$ and $Z_I(f) = \text{TZIMID}(f) - K/(2\pi f)$.
3. The resistance represents a damper quantity B. The reactance represents a quantity of the type $(\omega M - K/\omega)$. The impedance is defined as $= p/\dot{u}$ where p is the pressure and $\dot{u}$ is the velocity.
4. The impedance scale factor S is used in computing element stiffness and damping terms as:

$$k = \frac{A}{S} \cdot \frac{2\pi f Z_I(f)}{Z_R^2 + Z_I^2} \int \text{ (of shape functions)}$$

$$b = \frac{A}{S} \cdot \frac{Z_R(f)}{Z_R^2 + Z_I^2} \int \text{ (of shape functions)}$$

   The value of $(Z_R^2 + Z_I^2)$ must be greater than machine epsilon--a machine dependent constant in the neighborhood of 1.E-15. The scale factor S can be used to ensure this constraint while retaining the same units.

5. The output for the element is specified by the STRESS Case Control command and consists of the resistance, reactance, and absorption coefficient. The absorption coefficient is defined as:

$$a = \frac{4(Z_R/\rho c)}{(Z_R/\rho c + 1)^2 + (Z_I/\rho c)^2}$$

PACABS Acoustic Absorber Property

Defines the properties of the acoustic absorber element.

Format:

1	2	3	4	5	6	7	8	9	10
PACABS	PID	SYNTH	TID1	TID2	TID3	TESTAR	CUTFR	B	
	K	M							

Example:

PACABS	12		1	2	3	3.5	500.0		

Field	Contents
PID	Property identification number. (Integer > 0)
SYNTH	Request the calculation of B, K, and M from the tables TIDi below. (Character = “YES” or “NO”; Default = “YES”)
TID1	Identification of the TABLEDi entry that defines the resistance. See Remark 2. (Integer > 0 or blank)
TID2	Identification of the TABLEDi entry that defines the reactance. See Remark 2. (Integer > 0 or blank)
TID3	Identification of the TABLEDi entry that defines the weighting function. See Remark 2. (Integer > 0 or blank)
TESTAR	Area of the test specimen. (Real > 0.0; Default = 1 .0)
CUTFR	Cutoff frequency for tables referenced above. (Real > 0.0)
B, K, M	Equivalent damping, stiffness and mass values per unit area. (Real $\geq$ 0.0)

Remarks:

1. PACABS is referenced by a CHACAB entry only.
2. If SYNTH = “YES”, then TID1 and TID2 must be supplied (TID3 is optional) and the equivalent structural model will be derived from tables TIDi. If TID3 is blank, then the weighting function defaults to 1.0.
3. If SYNTH = “NO”, then the equivalent structural model will be derived from one of B, K, or M.

4. The continuation entry is optional.
5. All data defined in tables TIDi must be a function of frequency in cycles / unit time.

PACBAR Acoustic Barrier Property

Defines the properties of the acoustic barrier element.

Format:

1	2	3	4	5	6	7	8	9	10
PACBAR	PID	MBACK	MSEPTM	FRESON	KRESON				

Example:

PACBAR	12	1.0	0.01	400.0					

Field	Contents
PID	Property identification number. (Integer > 0)
MBACK	Mass per unit area of the backing material. (Real > 0.0)
MSEPTM	Mass per unit area of the septum material. (Real > 0.0)
FRESON	Resonant frequency of the sandwich construction in hertz. (Real > 0.0 or blank)
KRESON	Resonant stiffness of the sandwich construction. (Real > 0.0 or blank)

Remarks:

1. PACBAR is referenced by a CHACBR entry only.
2. Either FRESON or KRESON must be specified, but not both.

PACINF Acoustic Conjugate Infinite Element Property

Defines the properties of acoustic conjugate infinite elements.

Format:

1	2	3	4	5	6	7	8	9	10
PACINF	PID	MID	RIO	X1	X2	X3			

Field	Contents
PID	Property Identification Number of PACINF entry. (Integer > 0)
MID	Material Identification Number of a MAT10 entry. (Integer > 0)
XP, YP, ZP	Coordinates of the Pole of the Infinite Elements (in the Basic Coordinate System).

PAERO1 Aerodynamic Panel Property

Defines associated bodies for the panels in the Doublet-Lattice method.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO1	PID	B1	B2	B3	B4	B5	B6		

Example:

PAERO1	1	3							

Field	Contents
PID	Property identification number referenced by a CAERO1 entry. (Integer > 0)
Bi	Identification number of CAERO2 entries for associated bodies. (Integer $\geq$ 0 or blank)

Remarks:

1. The associated bodies must be in the same aerodynamic group, as specified in the IGID field on CAERO2 entry.
2. If there are no bodies, the entry is still required (with Bi fields blank).
3. The Bi numbers above must appear on a CAERO2 entry to define these bodies completely.

PAERO2 Aerodynamic Body Properties

Defines the cross-sectional properties of aerodynamic bodies.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO2	PID	ORIENT	WIDTH	AR	LRSB	LRIB	LTH1	LTH2	
	THI1	THN1	THI2	THN2	THI3	THN3			

Example:

PAERO2	2	Z	6.0	1.0	22	91	100		
	1	3							

Field	Contents
PID	Property identification number. (Integer > 0)
ORIENT	Orientation flag. Type of motion allowed for bodies. Refers to the aerodynamic coordinate system of ACSID. See AERO entry. (Character = “Z”, “Y”, or “ZY”)
WIDTH	Reference half-width of body and the width of the constant width interference tube. (Real > 0.0)
AR	Aspect ratio of the interference tube (height / width). (Real > 0.0)
LRSB	Identification number of an AEFACT entry containing a list of slender body half-widths at the end points of the slender body elements. If blank, the value of WIDTH will be used. (Integer > 0 or blank)
LRIB	Identification number of an AEFACT entry containing a list of slender body half-widths at the end points of the interference elements. If blank, the value of WIDTH will be used. (Integer > 0 or blank)
LTH1, LTH2	Identification number of AEFACT entries for defining θ arrays for interference calculations. (Integer ≥ 0)
THIi, THNi	The first and last interference element of a body to use the θ_1 array; the others use the θ_2 array. (Integer ≥ 0)

Remarks:

1. The half-widths (given on AEFACT entries referenced in fields 6 and 7) are specified at division points. The number of entries on an AEFACT entry used to specify half-widths must be one greater than the number of elements.
2. The half-width at the first point (i.e., the nose) on a slender body is usually 0.0; thus, it is recommended (but not required) that the LRSB data is supplied with a zero first value.
3. THIi and THNi are interference element numbers on a body. The first element is one for each body.
4. A body is represented by a slender body surrounded by an interference tube. The slender body creates the downwash due to the motion of the body, while the interference tube represents the effects upon panels and other bodies.

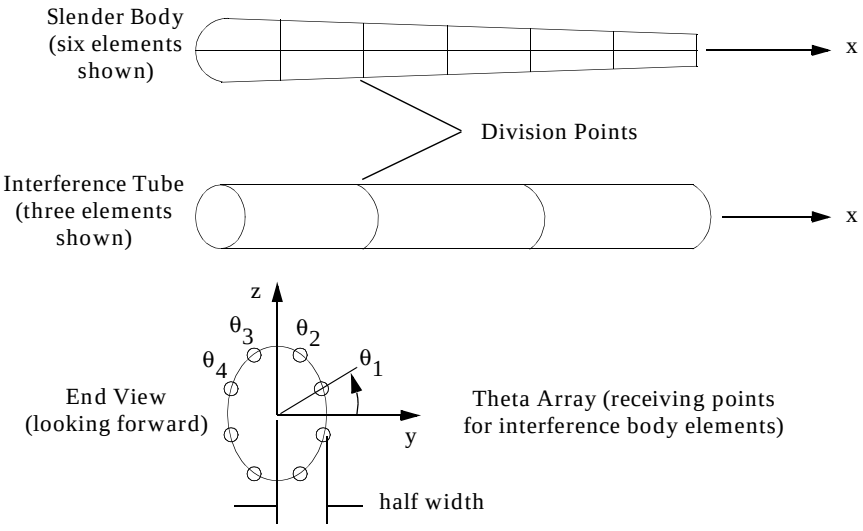


Figure 8-129 Idealization of Aerodynamic Body

5. The angles θ_1 and θ_2 are input in degrees using the aerodynamic element coordinate system as the reference coordinate system for defining the theta points.
6. Distribution of the theta points need not be uniform. A theta point must be placed a finite distance from any aerodynamic box edge; preferably the box edge would be equidistant from any two theta points. This aerodynamic coordinate system is defined on the AERO Bulk Data entry.
7. For half models, the theta arrays LTH1 and LTH2 should encompass a full 360 degree range.

PAERO3 Aerodynamic Panel Property

Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO3	PID	NBOX	NCTRL		X5	Y5	X6	Y6	
	X7	Y7	X8	Y8	X9	Y9	X10	Y10	
	X11	Y11	X12	Y12					

Example:

PAERO3	2001	15	1		0.	65.			
	78.	65.	108.	65.	82.	97.5	112.	97.5	
	86.	130.	116.	130.					

Field	Contents
PID	Property identification number. (Integer > 0)
NBOX	Number of Mach boxes in the flow direction. (0 < Integer < 50)
NCTRL	Number of control surfaces. (Integer 0, 1, or 2)
X5 through Y12	Locations of points 5 through 12, which are in the aerodynamic coordinate system, to define the cranks and control surface geometry. (Real)

Remarks:

1. For an illustration of the geometry, see the CAERO3 entry description.
2. If $Y5 \leq 0.0$, there is no leading edge crank. Also, if $Y6 \leq 0.0$, there is no trailing edge crank.
3. If NCTRL = 0, no continuations are required. If NCTRL = 1 or 2, then NCTRL continuations are required.
4. $Y7 \geq Y8$, $Y9 \geq Y10$, and $Y11 \geq Y12$.

5. The number of Mach boxes in the spanwise direction (NSB) may be found from the following formula:

$$NSB = \text{INT}\left[\frac{\beta \cdot y_{max}}{\left(\frac{x_{max}}{\text{NBOX} + 0.5}\right)} + 0.5\right]$$

where

$\beta = \sqrt{M^2 - 1}$

x_{max} = maximum chordwose direction

y_{max} = maximum spanwise direction

NBOX = initial number of boxes specified in field 3

The number of Mach boxes in the streamwise direction may then be computed from:

$$\text{NBOX} = \text{INT}\left[\frac{x_{max}}{\left(\frac{\beta \cdot y_{max}}{NSB - 0.5}\right)} + 0.999\right]$$

The number of chordwise boxes specified by the user (NBOX $\geq$ 50) will be replaced by a floating point number (usually slightly higher than NBOX). The method contracts the mesh equally in both dimensions until a box edge lies on the surface tip. This mesh size is then used to compute the number of chordwise boxes.

Note: A minimum of seven Mach boxes in the flow direction (NBOX) is recommended.

PAERO4 Aerodynamic Strip Properties

Defines properties of each strip element for Strip theory.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO4	PID	CLA	LCLA	CIRC	LCIRC	DOC1	CAOC1	GAPOC1	
	DOC2	CAOC2	GAPOC2	DOC3	CAOC3	GAPOC3	-etc.-		

Example:

PAERO4	6001	1	501	0	0	0.0	0.0	0.0	
	0.50	0.25	0.02	0.53	0.24	0.0			

Field	Contents
PID	Property identification number. (Integer > 0)
CLA	Select Prandtl-Glauert correction. (Integer = -1, 0, 1; Default = 0) -1 Compressibility correction made to lift curve slope data for a reference Mach number. 0 No correction and no list needed. (Default) +1 No correction and lift curve slope provided by a list as a function of strip location and Mach number.
LCLA	Identification number of the AEFACT entry that lists the lift curve slope on all strips for each Mach number on the MKAEROi entry. See Remark 7(**b.**) below. (Integer = 0 if CLA = 0, > 0 if CLA ≠ 0)
CIRC	Select Theodorsen's function $C(k)$ or the number of exponential coefficients used to approximate $C(k)$. (Integer = 0, 1, 2, 3; Default = 0. Must be zero if CLA ≠ 0.) 0 Theodorsen function. 1, 2, 3 Approximate function with $b_0, b_1, \beta_1, \ldots, b_n, \beta_n$ n = 1, 2, 3.
LCIRC	Identification number of the AEFACT entry that lists the b, β values for each Mach number. See Remark 7**c.**, 7**d.**, and 7**e.** below; variable b's and β's for each mi on the MKAEROi entry. (Integer = 0 if CIRC = 0, > 0 if CIRC ≠ 0)

Field	Contents
DOCi	d/c = distance of the control surface hinge aft of the quarter-chord divided by the strip chord (Real ≥ 0.0)
CAOCi	c_a/c = control surface chord divided by the strip chord. (Real ≥ 0.0)
GAPOCi	g/c = control surface gap divided by the strip chord. (Real ≥ 0.0)

Remarks:

1. PAERO4 is required for Strip theory with three fields (DOCi, CAOCi, GAPOCi) specified per strip.
2. If CLA = -1, lift curve slope data at one Mach number are needed on the AEFACT entry.
3. If CAOCi = 0.0, there is no control surface.
4. If GAPOCi = 0.0, there is no slot flow.
5. If GAPOCi < 0.01, then 0.01 is used.
6. Embedded blank fields are not permitted.
7. The following table lists the lift curve slope or lag function selection and the AEFACT entry formats used for Strip theory:

Table 8-27 Strip Theory Function Selections and AERACT Entry Formats

Theodorsen Function	Data Type Input	Parameter Combinations				Number of Words	Entry Format Index
		CLA	LCLA	CIRC	LCIRC		
Exact	Lift Curve Slope $c_{l_{\alpha i}} = 2\pi$	0	0	0	0	No AEFACT entry required	
	$c_{l_{\alpha i}}$ Input, Uses Prandtl-Glauert Correction	-1	ID	0	0	(NSTRIP+1)	a.
	$c_{l_{\alpha i}}$ Input, for All m's on MKAERO Entry	1	ID	0	0	(NSTRIP+1)*NMACH	b.
Approximate Coefficients	b_{0i}, b_{1i}, β_{1i}, etc.	0	0	1	ID	4*NMACH	c.
		0	0	2	ID	6*NMACH	d.
		0	0	3	ID	8*NMACH	e.

Entry Format

a. AEFACT, ID, $m_1, c_{l\alpha_1}, c_{l\alpha_2}, \ldots, c_{l\alpha_{\text{NSTRIP}}}$

b. AEFACT, ID, $m_1, c_{l\alpha_{11}}, c_{l\alpha_{21}}, \ldots, c_{l\alpha_{\text{NSTRIP1}}}$, $m_2, c_{l\alpha_{11}}, c_{l\alpha_{12}}, c_{l\alpha_{21}} c_{l\alpha_{22}}, \ldots, c_{l\alpha_{\text{NSTRIP1}}}, c_{l\alpha_{\text{NSTRIP2}}}$, for all m on MKAEROi data entry

c. AEFACT, ID, $m_1, b_{01}, b_{11}, \beta_{11}, m_2, b_{02}, b_{12}, P_{12}, m_3$, etc.

d. AEFACT, ID, $m_1, b_{01}, b_{11}, \beta_{11}, b_{21}, \beta_{21}, m_2$, etc.

e. AEFACT, ID, $m_1, b_{01}, b_{11}, \beta_{11}, b_{21}, \beta_{21}, b_{31}, \beta_{31} m_2$ etc.

8. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the ***MSC.Nastran Aeroelastic Analysis User's Guide***.

PAERO5 Aerodynamic Panel Property

Defines properties of each strip element for Piston theory.

Format:

1	2	3	4	5	6	7	8	9	10
PAERO5	PID	NALPHA	LALPHA	NXIS	LXIS	NTAUS	LTAUS		
	CAOC1	CAOC2	CAOC3	CAOC4	CAOC5				

Example:

PAERO5	7001	1	702	1	701	1	700		
	0.0	0.0	5.25	3.99375	0.0				

Field	Contents
PID	Property identification number. (Unique Integer > 0)
NALPHA	Number of angle of attack (α) values to be input for each Mach number (mi) on the MKAERO1 or MKAERO2 entry. (Integer > 0)

NALPHA	Meaning
1	α is the same value for all strips; enter one value, in units of degrees, on the AEFACT entry for each Mach number.
Number of Strips	α is different for each strip; enter α's, in units of degrees, in the following order: $m_1, \alpha_1, \alpha_2, \ldots,$ $_2, \alpha_1, \alpha_2, \ldots,$ etc.

LALPHA	ID number of the AEFACT entry that lists the α's for the strips at each Mach number in the MKAERO1 or MKAERO2 entry. (Integer > 0)
NXIS	Number of dimensionless chord coordinates (ξ) to be input. (Integer $\geq$ 0, Default = 0)

NXIS	Meaning
0	No ξ's are required. (Default)
1	ξ's are the same for all strips; enter values for one strip on the AEFACT entry (ξ_h if NTHICK > 0, or ξ_m and ξ_h if NTHICK = 0)
Number of Strips	ξ's have to be input for each strip (ξ_{h1}, ξ_{h2}, ..., ξ_{hNSPAN}, if NTHICK > 0, or ξ_{m1}, ξ_{h1}, ξ_{m2}, ξ_{h2}, ..., ξ_{mNSPAN}, ξ_{hNSPAN}, β_{hNSPAN} if NTHICK = 0)

LXIS Identification number of AEFACT entry that lists the ξ values for the strip in order indicated by values of NXIS and NTHICK. (Integer = 0 if c_a = 0 and NTHICK > 0 or LXIS>0 if c_a = 0 and / or NTHICK = 0)

NTAUS Parameter used to select the number of thickness ratio (τ) values to be input. (Integer $\geq$ 0, Default = 0)

NTAUS	Meaning
0	No τ's are required. (Default)
1	τ's are the same for all strips; enter (τ_1, τ_{h1}, τ_{t1}) values for one strip on AEFACT entry.
Number of Strips	τ's must to be input for each strip on an AEFACT entry in the following order: (τ_1, τ_{h1}, τ_{t1}, τ_2, τ_{h2}, τ_{t2}, ..., τ_{NSPAN}, τ_{hNSPAN}, τ_{tNSPAN})

LTAUS Identification number of AEFACT entry that lists the τ values for the strips. (Integer = 0 or blank if NTAUS = 0, LTAUS > 0 if NTAUS > 0)

CAOCi c_a / c = control surface chord divided by the strip chord. (Real $\geq$ 0.0)

Remarks:

1. The continuation entry is required for Piston theory with one entry (CAOCi) per strip.

2. Embedded blank fields are not allowed on the continuation entry.
3. If CAOCi = 0.0, there is no control surface.
4. **Table 8-28** lists the thickness data input and AEFACT entry format used for Piston theory.

Table 8-28 Thickness Data Input and AEFACT Entry Format for Piston Theory

Type of Input	Parameter Combinations						Number of Words	Entry Format Index
	CAOG	NGHICK	NXIS	LXIS	NTAUS	LTAUS		
No control surfaces, Integrals input are same for all strips	0.	ID(a)	0	0	0	0	6	a.
With control surfaces, Integrals input, same hinge on all strips	≠ 0	ID(b)	1	ID(c)	0	0	12 1	b. c.
With control surfaces, Integrals input, variable hinge	≠ 0	ID(b)	NSTRIP	ID(d)	0	0	12 NSTRIP	b. d.
No control surfaces, thickness inputs are same for all strips	0.0	0	1	ID(f)	1	ID(e)	3 2	e. f.
With control surfaces, thickness inputs are same for all strips	≠ 0.0	0	1	ID(f)	1	ID(e)	3 2	e. f.
With control surfaces, thickness inputs vary for strips	≠ 0.0	0	NSTRIP	ID(h)	NSTRIP	ID(g)	3*NSTRIP 2*NSTRIP	g. h.

Entry Format

a. AEFACT, ID, I_1, I_2, I_3, I_4, I_5, I_6

b. AEFACT, ID, I_1, …, I_6, J_1, …., $J_6 I_1$, I_2, I_3, I_4, I_5, I_6

c. AEFACT, ID, ξ_h

d. AEFACT, ID, $\xi_{h1}, \xi_{h2}, \xi_{h3}, \ldots, \xi_{h\,\mathrm{NSTRIP}}$

e. AEFACT, ID, τ_m, τ_h, τ_t

f. AEFACT, ID, ξ_m, ξ_h

g. AEFACT, ID, $\tau_{m1}, \tau_{h1}, \tau_{t1}, \tau_{m2}, \tau_{h2}, \tau_{t2}, \ldots, \tau_{m\,\mathrm{NSTRIP}}, \tau_{h\,\mathrm{NSTRIP}}, \tau_{t\,\mathrm{NSTRIP}}$
$\tau_{m1}, \tau_{h1}, \tau_{t1}, \tau_{m2}, \tau_{h2}, \tau_{t2}, \ldots, \tau_{m\,\mathrm{NSTRIP}}, \tau_{h\,\mathrm{NSTRIP}}, \tau_{t\,\mathrm{NSTRIP}}$

h. AEFACT, ID, $\xi_{m1}, \xi_{h1}, \xi_{m2}, \xi_{h2}, \ldots, \xi_{m\,\mathrm{NSTRIP}}, \xi_{h\,\mathrm{NSTRIP}}$

5. The following table lists the angle-of-attack distribution and AEFACT entry formats used for Piston theory.

Type of Distribution	Parameter Combinations		Number of Words	Entry Format Index
	NALPHA	LALPHA		
Equal angle of attack on all strips	1	ID	2*NMACH	a.
Unequal angle of attack	NSTRIP	ID	(1 + NSTRIP) * NMACH	b.

Entry Format

a. AEFACT, ID, $m_1, \alpha_1, m_2, \alpha_2, \ldots$,

b. AEFACT, ID, $m_1, \alpha_{11}, \alpha_{21}, \alpha_{31}, \ldots, \alpha_{NSTRIP1}, m_2, \alpha_{12}, \alpha_{22}, \ldots, \alpha_{NSTRIP2}, m_2$, etc., for all m on MKAEROi entry.

c. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the *MSC.Nastran Aeroelastic Analysis User's Guide.*

PANEL Panel Definition for Coupled Fluid-Structural Analysis

Defines one or more panels by referencing sets of grid points, elements or properties.

Format:

1	2	3	4	5	6	7	8	9	10
PANEL	NAME1	SETID1	NAME2	SETID2	NAME3	SETID3	NAME4	SETID4	

Example:

PANEL	BKDOOR	103							

Field	Contents
NAMEi	Panel label. (Character)
SETIDi	Identification number of a SET1 or SET3 entry that lists the grid points, elements or properties of the panel. (Integer > 0)

Remarks:

1. If a set of grid points is referenced, the set must list only structural grid points.
2. If an element is assigned to a panel, it is recommended that all of its connections points beloing to the same panel.
3. If a set of elements is referenced, the set must list only structural elements. The panel will consist of all grid points that are connection points of these elements.
4. If a set of property identifiers is referenced, the properties must be referenced by structural elements. The panel will consist of all grid points that are connection points of elements referencing one of the properties contained in the set.
5. NAMEi is used only for labeling the output of the panel participation factors (cf. the description of the PFMODE and PFPANEL Case Control commands).

PARAM Parameter

Specifies values for parameters used in solution sequences or user-written DMAP programs.

Format:

1	2	3	4	5	6	7	8	9	10
PARAM	N	V1	V2						

Example:

PARAM	IRES	1							

Field	Contents
N	Parameter name (one to eight alphanumeric characters, the first of which is alphabetic).
V1, V2	Parameter value based on parameter type, as follows:

Type	V1	V2
Integer	Integer	Blank
Real, single-precision	Real	Blank
Character	Character	Blank
Real, double-precision	Double-precision real	Blank
Complex, single-precision	Real or blank	Real or blank
Complex, double-precision	Double-precision real	Double-precision real

Remarks:

1. See “**Parameters**” on page 659 for a list of parameters used in solution sequences that may be set by the user on PARAM entries.
2. If the large field entry format is used, the second physical entry must be present, even though fields 6 through 9 are blank.

PARAMARC (SOL 600) Parallel Domain Decomposition in MSC.Marc

Specifies parallel regions for domain decomposition in nonlinear analysis when MSC.Marc is executed from MD Nastran. Used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
PARAMARC	ID	KIND	NPROC						
	IDP	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	IDP	ID8	ID9	etc.					

Alternate Format:

1	2	3	4	5	6	7	8	9	10
PARAMARC	ID	KIND	NPROC						
	IDP	ID1	THRU	ID2	BY	INC			

Example: To create 4 parallel processes using MSC.Marc's single file input

PARAMARC	51		4						

Example: To create 4 parallel processes automatically using MD Nastran ACMS

PARAMARC	101	1	4						

Example: To create 2 parallel processes by specifying element numbers

PARAMARC	201	2	2						
	1	1	THRU	5000	BY	1			
	2	5001	5002	5005	5010	5012	5013	5015	
	2	5020	5030	5030	5040	5050			

Field	Contents
ID	Identification number of the PARAMARC entry -- Not presently used (Integer)
KIND	Designates how parallel domains are created. (Integer $\geq$ 0, Default = 0) 0=Parallel processing is accomplished using MSC.Marc's single file input. (MSC.Marc Version 2005 and subsequent must be used. The command line to execute MSC.Marc is changed from -np N (or -nprocd N) to -nps N where N is the number of processors. The maximum number of processors for MSC.Marc is 256.) Continuation lines may not be entered for KIND=0.
NPROC	Number of parallel processes requested (Integer > 0, Default = 1)
IDP	Parallel process ID for following group of elements or grid IDs (Integer > 0, required -- see Remark 6.)
ID(i)	Element number (if KIND=2) or Grid ID (if KIND=3), Property ID (KIND=4) or Material ID (KIND=5) (Integer > 0, required)
THRU	Enter THRU in field 4 (may be omitted if I2 and INC are not required. The word TO instead of THRU may also be used.)
ID2	Ending element number (if KIND=2) or Grid ID (if KIND=3), Property ID (KIND=4) or Material ID (KIND=5) (Integer > 0)
BY	Enter BY in field 6 (may be omitted if INC=1)
INC	Increment IDs (Integer, Default = 1, may be negative but not zero)

Remarks:

1. The PARAMARC entry is recognized only when MSC.Marc is executed from within MD Nastran Implicit Nonlinear (SOL 600).
2. All PARAMARC entries found in the bulk data will be used to form parallel domains and associated MSC.Marc input decks if an option to create MSC.Marc input or execute MSC.Marc is found in the Executive Control or Case Control. There should only be one PARAMARC first line entry in the Bulk Data. If more than one is found, only the first is used.
3. Use of KIND=2-5 provides more control than does the automatic (KIND=1) option.
4. Several MSC.Marc input data files are produced. If the MD Nastran input file is named a.dat or a.bdf, and 3 parallel domains are specified (NPROC=3) the following files are produced:

a.marc.dat
1a.marc.dat
2a.marc.dat
3a.marc.dat

The file (a.marc.dat) is a stub model which contains only the control portion of the model. The other files (1a.marc.dat, 2a.marc.dat, 3a.marc.dat) contain the MSC.Marc input data for each domain and along with a.marc.dat will be executed in parallel. MD Nastran will automatically build the command string to execute MSC.Marc with the requested number of processors. A simple parallel run with the same number of processors should be successfully completed before attempting large models.

5. If parallel jobs are run on different computers across a network, as opposed to using multiple processors in the same box, a host file is normally needed. consult MSC technical support to determine how to setup a host file for your computer system. Use of the host file is triggered by Bulk Data PARAM,MARCHOST,Name.
6. The continuation entries should be entered as many times as necessary to completely define each parallel region for KIND=2-5.
7. If the KIND=1 (automatic parallel) is requested and if 3D contact is present all contact regions must be identified using BCTABLE entries. All BCTABLE entries found in the bulk data will be used to determine the domains. For the case where every element may potentially contact every other element in the model, it is presently not possible to define parallel regions, so the full model will be executed.
8. If MPC's, rigid elements or contact prevent the requested number of domains from being formed, NPROC will be adjusted and MSC.Marc executed using the adjusted value.
9. Continuation lines should not be entered if KIND=1.
10. The string TO may be substituted for THRU if so desired.

PBAR Simple Beam Property

Defines the properties of a simple beam element (CBAR entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

Example:

PBAR	39	6	2.9		5.97				
			2.0	4.0					

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks **2.** and **3.** (Integer > 0)
A	Area of bar cross section. (Real; Default = 0.0)
I1, I2, I12	Area moments of inertia. See **Figure 8-130**. (Real; I1 $\geq$ 0.0, I2 $\geq$ 0.0, I1*I2 > $I12^2$; Default = 0.0)
J	Torsional constant. See **Figure 8-130**. (Real; Default = $\frac{1}{2}(I_1 + I_2)$ for SOL 600 and 0.0 for all other solution sequences)
NSM	Nonstructural mass per unit length. (Real)
Ci, Di, Ei, Fi	Stress recovery coefficients. (Real; Default = 0.0)
K1, K2	Area factor for shear. See Remark **5.** (Real or blank)

Remarks:

1. Both continuation entries may be omitted.
2. For structural problems, MID must reference a MAT1 material entry.
3. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
4. See the CBAR entry description for a discussion of bar element geometry.

5. The transverse shear stiffnesses per unit length in planes 1 and 2 are K1*A*G and K2*A*G, respectively, where G is the shear modulus. The default values for K1 and K2 are infinite; in other words, the transverse shear flexibilities are set equal to zero. K1 and K2 are ignored if I12 $\neq$ 0. K1 and K2 must be blank if A = 0.0.
6. The stress recovery coefficients C1 and C2, etc., are the y and z coordinates in the bar element coordinate system of a point at which stresses are computed. Stresses are computed at both ends of the bar.
7. For response spectra analysis on stress recovery coefficients, the CBEAM element entry should be used because bar element results will be inaccurate.
8. **Figure 8-130** describes the PBAR element coordinate system.

   Note:

   $I1 = I_{zz_{elem}}$

   $I2 = I_{yy_{elem}}$

   $I12 = I_{zy_{elem}}$

   $J = I_{xx_{elem}}$

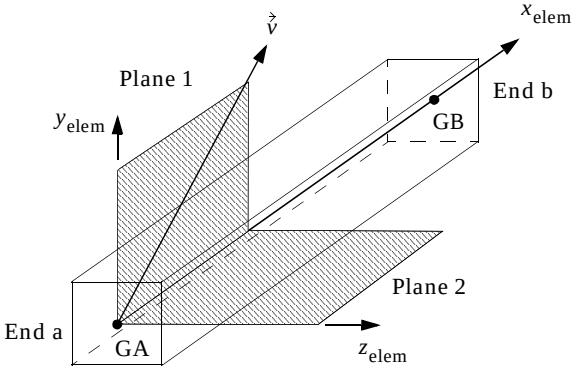


Figure 8-130 PBAR Element Coordinate System

9. By definition, the shear center and neutral axes coincide.
10. Mass moment of inertial formulation has changed in Version 2003. System (398) may be used to select the formulation in pre-Version 2004 systems.

PBARL Simple Beam Cross-Section Property

Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.

Format:

1	2	3	4	5	6	7	8	9	10
PBARL	PID	MID	GROUP	TYPE					
	DIM1	DIM2	DIM3	DIM4	DIM5	DIM6	DIM7	DIM8	
	DIM9	-etc.-	NSM						

Example:

PBARL	39	6		I					
	14.	6.	.5	.5	.5	.5	.2		

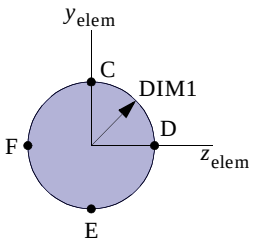
Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. See Remarks **6.** and **8.** (Character; Default = “MSCBML0")
TYPE	Cross-section type. See Remarks **6.** and **8.** and **Figure 8-131**. (Character: “ROD”, “TUBE”, “I”, “CHAN”, “T”, “BOX”, “BAR”, “CROSS”, “H”, “T1", “I1", “CHAN1", “Z”, “CHAN2", “T2", “BOX1", “HEXA”, “HAT”, “HAT1”, “DBOX” for GROUP=“MSCBML0")
DIMi	Cross-sectional dimensions. (Real > 0.0 for GROUP = “MSCBMLO”)
NSM	Nonstructural mass per unit length. NSM is specified after the last DIMi. (Default = 0.0)

Remarks:

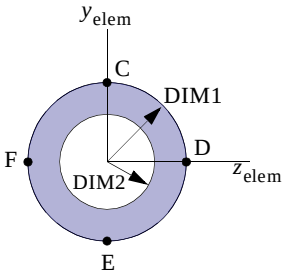
1. For structural problems, PBARL entries must reference a MAT1 material entry.
2. PID must be unique with respect to all other PBAR and PBARL property identification numbers.
3. See CBAR entry for a discussion of bar element geometry.

4. For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
5. For response spectra analysis on stress recovery coefficients, the CBEAM element should be used because results for the CBAR element will not be accurate.
6. The GROUP is associated with an FMS CONNECT statement that specifies the evaluator. A reserved GROUP name is “MSCBML0". Users may create their own cross-section types. Each of the types will require one or more subroutines to convert DIMi information to geometric property information contained on a PBAR entry and optimization information. See “**Building and Using the Sample Programs**” on page 255 of the *MD Nastran 2006 Installation and Operations Guide* for a discussion on how to include a user-defined beam library.
7. A function of this entry is to derive an equivalent PBAR entry. Any sorted echo request will also cause printout and / or punch of the derived PBAR.
8. For GROUP = “MSCBML0", the cross-sectional properties, shear flexibility factors, and stress recovery points (C, D, E, and F) are computed using the TYPE and DIMi as shown in **Figure 8-131**. The origin of element coordinate system is centered at the shear center of the cross-section oriented as shown. The PBARL does not account for offsets between the neutral axis and the shear center. Therefore, the CHAN, CHAN1 and CHAN2 cross-sections may produce incorrect results. The PBEAML is recommended.
9. For DBOX section, the default value for DIM5 to DIM10 are based on the following rules:
   a. DIM5, DIM6, DIM7 and DIM8 have a default value of DIM4 if not provided.
   b. DIM9 and DIM10 have a default value of DIM6 if not provided.

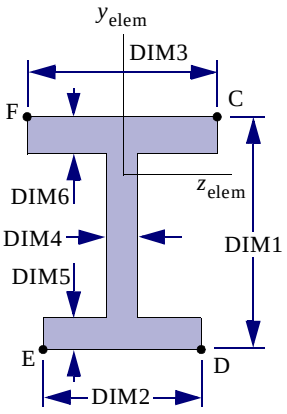
   NOTE: The above default value rules for DIM5 to DIM10 are not applicable to design optimization property value update.
10. Finite element formulation (FEF), utilized for arbitrary beam cross section is selected as default method for computing sectional properties for all supported cross section types of PBARL if GROUP=MSCBML0. The original beam equations which are based on thin-walled assumption can be accessed via Bulk Data entry ‘MDLPRM,TWBRBML,1’. For optimization, individual DIMx of PBARL can be selected as design property even with finite element formulation.



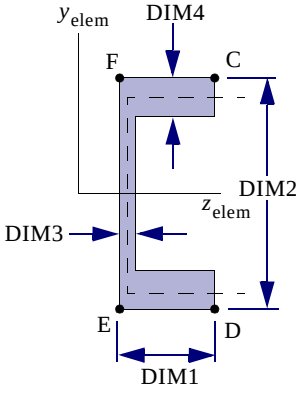
TYPE="ROD"



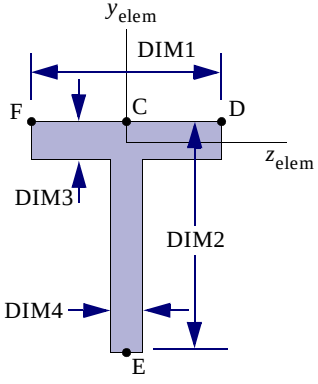
TYPE="TUBE"



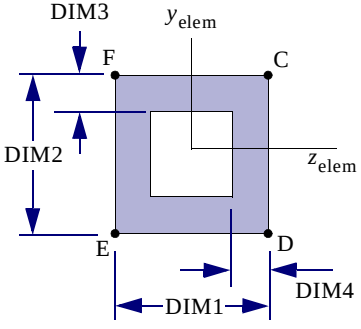
TYPE="I"



TYPE="CHAN"

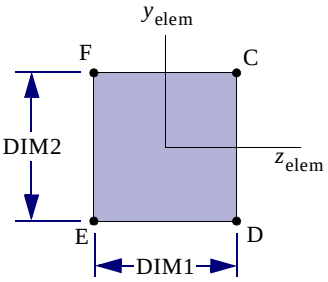


TYPE="T"

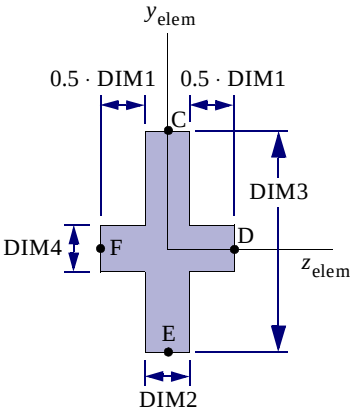


TYPE="BOX"

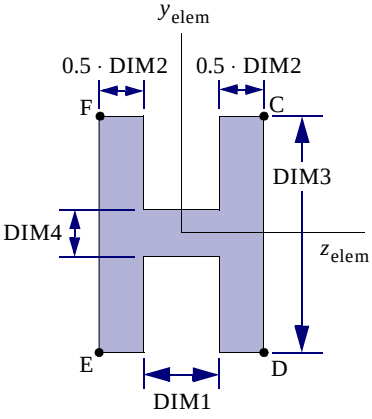
Figure 8-131 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"



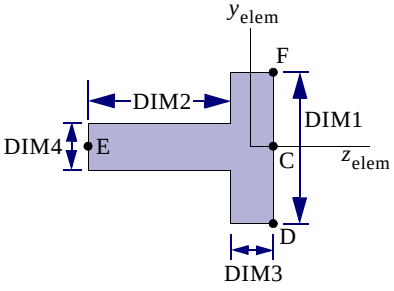
TYPE=”BAR”



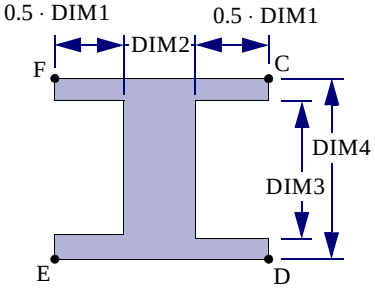
TYPE=”CROSS”



TYPE=”H”

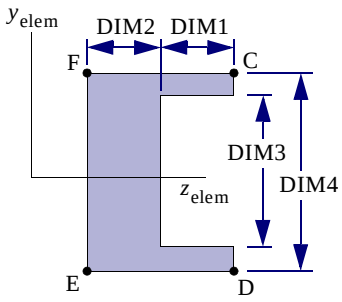


TYPE=”T1”

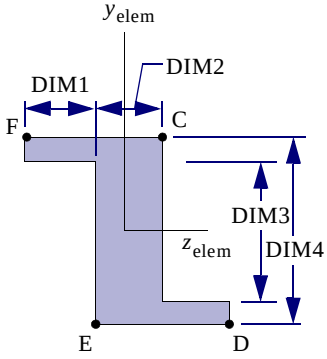


TYPE=”I1”

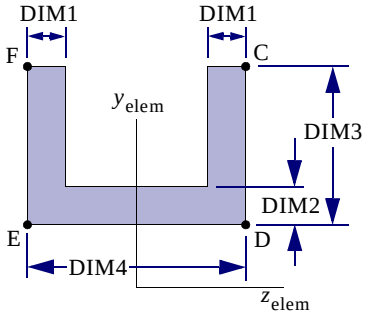
Figure 8-131 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = “MSCBML0" (continued)



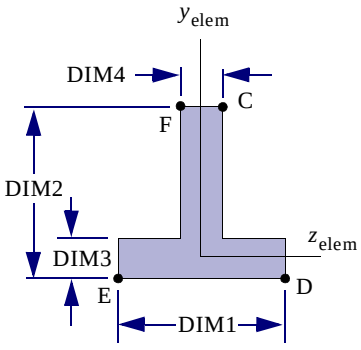
TYPE=”CHAN1”



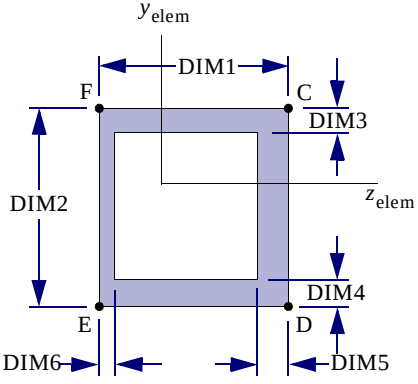
TYPE=”Z”



TYPE=”CHAN2”

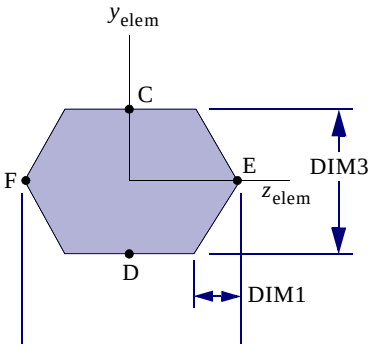


TYPE=”T2”

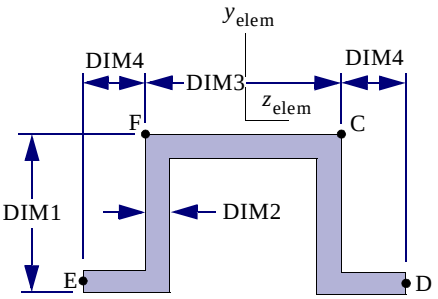


TYPE=”BOX1”

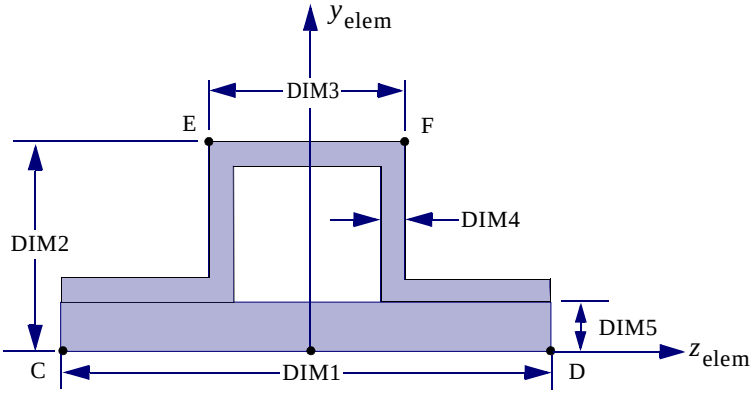
Figure 8-131 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = “MSCBML0" (continued)



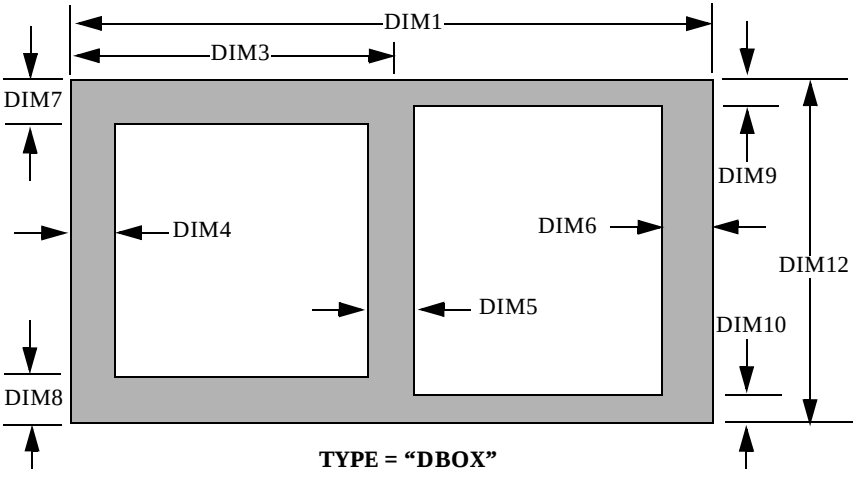
TYPE=”HEXA”



TYPE=”HAT”



TYPE=”HAT1”



TYPE = “DBOX”

Figure 8-131 **Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = “MSCBML0" (continued)**

PBCOMP Beam Property (Alternate Form of PBEAM)

Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry. This entry is also used to specify lumped areas of the beam cross section for nonlinear analysis and / or composite analysis.

Format:

1	2	3	4	5	6	7	8	9	10
PBCOMP	PID	MID	A	I1	I2	I12	J	NSM	
	K1	K2	M1	M2	N1	N2	SYMOPT		
	Y1	Z1	C1	MID1					
	Y2	Z2	C2	MID2					
	-etc.-								

Example:

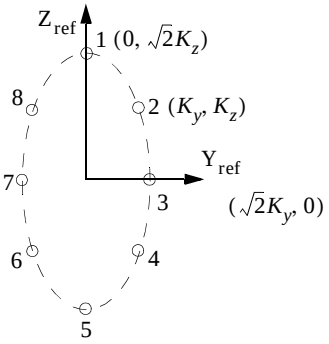
PBCOMP	39	6	2.9						
							1		
	-0.5	1.2	0.1	18					
	0.2	0.9	0.15						

Field	Contents
PID	Property identification number. See Remark **1.** (Integer > 0)
MID	Material identification number. See Remarks **2.** and **5.** (Integer > 0)
A	Area of beam cross section. (Real > 0.0)
I1	Area moment of inertia in plane 1 about the neutral axis. See Remark **6.** (Real > 0.0)
I2	Area moment of inertia in plane 2 about the neutral axis. See Remark **6.** (Real > 0.0)
I12	Area product of inertia. See Remark **6.** (Real; Default = 0.0, but $I1 \cdot I2 - (I12)^2 > 0.0$)
J	Torsional stiffness parameter. See Remark **6.** (Real > 0.0; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real > 0.0; Default = 0.0)
K1, K2	Shear stiffness factor K in K*A*G for plane 1 and plane 2. See Remark **4.** (Real > 0.0; Default = 1.0)

Field	Contents
M1, M2	The (y,z) coordinates of center of gravity of nonstructural mass. See the figure in the CBEAM entry description. (Real; Default = 0.0)
N1, N2	The (y,z) coordinates of neutral axis. See the figure in the CBEAM entry description. (Real; Default = 0.0)
SYMOPT	Symmetry option to input lumped areas for the beam cross section. See **Figure 8-133** and Remark **7.** ($0 \le$ Integer ≤ 5; Default = 0)
Yi, Zi	The (y,z) coordinates of the lumped areas in the element coordinate system. See Remark **1.** (Real)
Ci	Fraction of the total area for the i-th lumped area. (Real > 0.0; Default = 0.0)
MIDi	Material identification number for the i-th integration point. See Remark **5.** (Integer > 0)

Remarks:

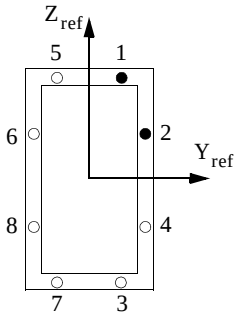
1. The PID number must be unique with respect to other PBCOMP entries as well as PBEAM entries. The second continuation entry may be repeated 18 more times. If SECTION = 5 a maximum of 21 continuation entries is allowed; i.e., a maximum of 20 lumped areas may be input. If SECTION = 1 through 4, the total number of areas input plus the total number generated by symmetry must not exceed 20. If these are not specified, the program defaults, as usual, to the elliptically distributed eight nonlinear rods. See **Figure 8-132**.



SYMOPT = 0 (default)
Symmetric about Y_{ref} and Z_{ref}

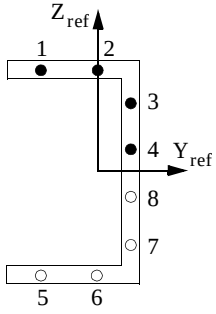
$K_y = \sqrt{\frac{I_{zz}}{A}}$, $K_z = \sqrt{\frac{I_{yy}}{A}}$, C1 = $\frac{1}{8}$

I_{zz} - moment of inertia about z-axis
I_{yy} - moment of inertia about y-axis



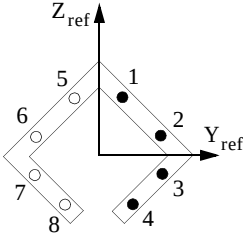
SYMOPT = 1
(w / continuation entry)
Symmetric about Y_{ref} and Z_{ref}

Y1 = Y3 = -Y5 = -Y7
Z1 = -Z3 = Z5 = -Z7, etc.



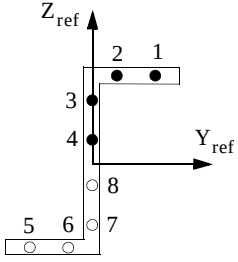
SYMOPT = 2
Symmetric about Y_{ref}

Y1 = Y5
Z1 = -Z5, etc.



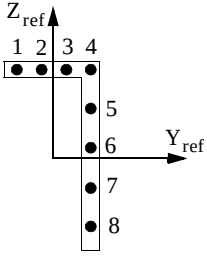
SYMOPT = 3
Symmetric about Z_{ref}

Y1 = -Y5, Z1 = Z5, etc.



SYMOPT = 4
Mirror Symmetry about Y_{ref} and Z_{ref}

Y1 = -Y5, Z1 = -Z5, etc.



SYMOPT = 5
No symmetry

Figure 8-132 PBCOMP Entry SYMOPT Type Examples with 8 Lumped Areas

Figure Example Notes:

Integration points (lumped area) are numbered 1 through 8.

User-specified points are denoted by ● and the program default point is denoted by ○ .

2. For structural problems, MID and MIDi must reference a MAT1 material entry. For material nonlinear analysis, the material should be perfectly plastic since the plastic hinge formulation is not valid for strain hardening. For heat transfer problems, MID and MIDi must reference a MAT4 or MAT5 material entry.
3. For the case where the user specifies I1, I2 and I12 on the parent entry, the stress-output location may also be specified on continuation entries. The (y,z) coordinates specified on these entries will serve as stress output locations with the corresponding Ci's set to 0. Stress output is provided at the first four lumped area locations only. If one of the symmetry options is used and fewer than four lumped areas are input explicitly, the sequence of output locations in the imaged quadrants is shown in **Figure 8-132**. For one specific example in the model shown in Remark **7.** (**Figure 8-133**), output can be obtained at points 1 and 2 and in the image points 3 and 4.
4. Blank fields for K1 and K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear stiffness becomes rigid and the transverse shear flexibilities are set to 0.0.
5. The values E_0 and G_0 are computed based on the value of MID on the parent entry. MID is will follow the same symmetry rules as Ci depending on the value of SECTION. If the MIDi field on a continuation entry is blank, the value will be that of MID on the parent entry. MIDi values may be input on continuations without the corresponding Yi, Zi, and Ci values to allow different stress-strain laws.
6. If the lumped cross-sectional areas are specified, fields I1, I2, and I12 will be ignored. These and other modified values will be calculated based on the input data (Yi, Zi, Ci, MIDi) as follows:

$$y_{NA} = \frac{\sum_{i=1}^{n} \text{Yi Ci Ei}}{\sum_{i=1}^{n} \text{Ci Ei}}$$

$$z_{NA} = \frac{\sum_{i=1}^{n} \text{Zi Ci Ei}}{\sum_{i=1} \text{Ci Ei}}$$

$$\bar{A} = A \sum_{i=1}^{n} \frac{\text{Ci Ei}}{\text{E}_0}$$

$$\bar{I}_1 = A \sum_{i=1}^{n} \frac{\text{Ci Ei (Yi} - y_{\text{NA}})^2}{\text{E}_o}$$

$$\bar{I}_2 = A \sum_{i=1}^{n} \frac{\text{Ci Ei (Yi} - z_{\text{NA}})^2}{\text{E}_o}$$

$$\bar{I}_{12} = A \sum_{i=1}^{n} \frac{\text{Ci Ei}(\text{Yi} - y_{NA})(\text{Zi} - z_{NA})}{\text{E}_o}$$

$$J = \text{J} \sum_{i=1}^{n} \frac{\text{Ci Gi}}{\text{G}_o}$$

where n is the number of lumped cross-sectional areas specified.

7. As can be seen from **Figure 8-132**, if the user chooses to leave the SECTION field blank, the program defaults to the elliptically distributed eight nonlinear rods, similar to the PBEAM entry. For this particular case it is illegal to supply Ci and MIDi values. For a doubly symmetric section (SECTION = 1), if the lumped areas are specified on either axis, the symmetry option will double the areas. For example, for the section shown in **Figure 8-133**, points 2 and 4 are coincident and so are points 6 and 8. In such cases, it is recommended that users input the value of area as half of the actual value at point 2 to obtain the desired effect.

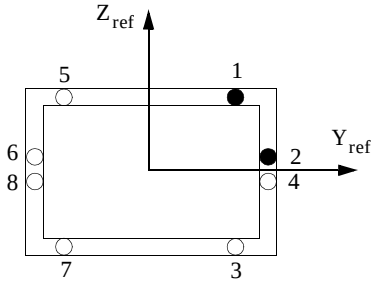


Figure 8-133 Doubly Symmetric PBCOMP Section

8. For SECTION = 5, at least one Yi and one Zi must be nonzero.

PBDISCR (SOL 700)

Defines properties for 6 DOF discrete beam elements.

Format:

1	2	3	4	5	6	7	8	9	10
PBDISCR	PID	MID	SCOOR						
	VOL	INER	CID	CA	OFFSET	RRCON	SRCON	TRCON	

Example:

PBDISCR	12	64							
	1.0	21	1.0	22					

Field	Contents
PID	Property ID. PID is referenced on the CBEAM entry and must be unique. (I > 0, Default = Required)
MID	Material ID. (I > 0, Default = Required) Material types allowed are: MATD066 ➔ LINEAR_ELASTIC_DISCRETE_BEAM MATD067 ➔ NONLINEAR_ELASTIC_DISCRETE_BEAM MATD068 ➔ NONLINEAR_PLASTIC_DISCRETE_BEAM MATD069 ➔ SID_DAMPER_DISCRETE_BEAM (for Side Impact Dummy) MATD070 ➔ HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM MATD071 ➔ CABLE_DISCRETE_BEAM MATD074 ➔ ELASTIC_SPRING_DISCRETE_BEAM MATD093 ➔ ELASTIC_6DOF_SPRING_DISCRETE_BEAM MATD094 ➔ INELASTIC_SPRING_DISCRETE_BEAM MATD095 ➔ INELASTIC_6DOF_SPRING_DISCRETE_BEAM MATD097 ➔ GENERAL_JOINT_DISCRETE_BEAM

Field	Contents
SCOOR	Location of triad for tracking the rotation of the discrete beam element, see the parameter CID below. The force and moment resultants in the output databases are referenced to this triad. The flags -3.0, -1.0, 0.0, 1.0, and 3.0 are inactive if the option to update the local system is active in the CID definition. Type: (Real, Default = Required) = -3.0: beam node 1, the angular velocity of node 1 rotates triad, = -2.0:beam node 1, the angular velocity of node 1 rotates triad but the r-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams., = -1.0: beam node 1, the angular velocity of node 1 rotates triad, = 0.0: centered between beam nodes 1 and 2, the average angular velocity of nodes 1 and 2 is used to rotate the triad, = +1.0:beam node 2, the angular velocity of node 2 rotates triad. = +2.0:beam node 2, the angular velocity of node 2 rotates triad. but the r-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams. =+3.0:beam node 2, the angular velocity of node 2 rotates triad. *If the magnitude of SC00R is less than or equal to unity then zero length discrete beams are assumed with infinitestimal separation between the nodes in the deformed state. For large separations or nonzero length beams set* $\|SCOOR\|$ *to 2 or 3.*
VOL	Volume of discrete beam. If the mass density of the material model for the discrete beam is set to unity, the magnitude of the lumped mass can be defined here instead. This lumped mass is partitioned to the two nodes of the beam element. The translational time step size for the type 6 beam is dependent on the volume, mass density, and the translational stiffness values, so it is important to define this parameter. Defining the volume is also essential for mass scaling if the type 6 beam controls the time step size. (R > 0.0, Default = Required)

Field	Contents
INER	Mass moment of inertia for the six degree of freedom discrete beam. This lumped inertia is partitioned to the two nodes of the beam element. The rotational time step size for the type 6 beam is dependent on the lumped inertia and the rotational stiffness values, so it is important to define this parameter if the rotational springs are active. Defining the rotational inertia is also essential for mass scaling if the type 6 beam rotational stiffness controls the time step size. (R > 0.0, Default = Required)
CID	Coordinate system ID for orientation (materials MATD066-69, 93, 95, 97). If CID=0, a default coordinate system is defined in the global system or on the third node of the beam, which is used for orientation. This option is not defined for material types than act between two nodal points, such as cable elements. The coordinate system rotates with the discrete beam, see SCOOR above. (I $\geq$ 0, Default = Required)
CA	Cable area, material MATD071, for modeling a cable. (R > 0.0, Default = Required)
OFFSET	Offset for cable. For a definition see material MATD071. (R $\geq$ 0.0, Default = 0.0)
RRCON	r-rotational constraint for local coordinate system. (R $\geq$ 0.0, Default = 0.0) =0.0: Coordinate ID rotates about r axis with nodes. =1.0: Rotation is constrained about the r-axis
SRCON	s-rotational constraint for local coordinate system. (R $\geq$ 0.0, Default = 0.0) =0.0: Coordinate ID rotates about s axis with nodes. =1.0: Rotation is constrained about the s-axis
TRCON	t-rotational constraint for local coordinate system. (R $\geq$ 0.0, Default = 0.0) =0.0: Coordinate ID rotates about t axis with nodes. =1.0: Rotation is constrained about the t-axis

PBEAM Beam Property

Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	A(A)	I1(A)	I2(A)	I12(A)	J(A)	NSM(A)	
	C1 (A)	C2 (A)	D1 (A)	D2 (A)	E1 (A)	E2 (A)	F1 (A)	F2 (A)	

The next two continuations are repeated for each intermediate station as described in Remark **6.** and SO and X/XB must be specified.

	SO	X/XB	A	I1	I2	I12	J	NSM	
	C1	C2	D1	D2	E1	E2	F1	F2	

The last two continuations are:

	K1	K2	S1	S2	NSI(A)	NSI(B)	CW(A)	CW(B)	
	M1(A)	M2(A)	M1(B)	M2(B)	N1(A)	N2(A)	N1(B)	N2(B)	

Example:

Tapered beam with A=2.9 at end A and A=5.3 at end B.

PBEAM	39	6	2.9	3.5	5.97				
			2.0	-4.0					
	YES	1.0	5.3	56.2	78.6				
			2.5	-5.0					
			1.1		2.1		0.21		
					0.5		0.0		

Field	Contents	Default Values
PID	Property identification number. (Integer > 0)	Required
MID	Material identification number. See Remarks **1.** and **3.** (Integer > 0)	Required

Field	Contents	Default Values
A(A)	Area of the beam cross section at end A. (Real > 0.0)	Required
I1(A)	Area moment of inertia at end A for bending in plane 1 about the neutral axis. See Remark **10.** (Real > 0.0)	Required
I2(A)	Area moment of inertia at end A for bending in plane 2 about the neutral axis. See Remark **10.** (Real > 0.0)	Required
I12(A)	0.0. Area product of inertia at end A. See Remark **10.** (Real, but $I1 \cdot I2 - (I12)^2 > 0.0$)	
J(A)	Torsional stiffness parameter at end A. See Remark **10.** (Real $\geq$ 0.0 but > 0.0 if warping is present)	Default=$\frac{1}{2}(I_1 + I_2)$ for SOL 600 and 0.0 for all other solution sequences
NSM(A)	Nonstructural mass per unit length at end A. (Real)	0.0
Ci(A), Di(A) Ei(A), Fi(A)	The y and z locations (i = 1 corresponds to y and i = 2 corresponds to z) in element coordinates relative to the shear center (see the diagram following the remarks) at end A for stress data recovery. (Real)	y = z = 0.0
SO	Stress output request option. See Remark **9.** (Character) "YES" Stresses recovered at points Ci, Di, Ei, and Fi on the next continuation. "YESA" Stresses recovered at points with the same y and z location as end A. "NO" No stresses or forces are recovered.	Required*
X/XB	Distance from end A in the element coordinate system divided by the length of the element See **Figure 8-134** in Remark **10.** (Real > 0.0)	Required* See Remark **6.**

Field	Contents	Default Values
A, I1, I2, I12, J, NSM	Area, moments of inertia, torsional stiffness parameter, and nonstructural mass for the cross section located at x. (Real; J > 0.0 if warping is present.)	See Remark 7.
Ci, Di, Ei, Fi	The y and z locations (i = 1 corresponds to y and i = 2 corresponds to z) in element coordinates relative to the shear center (see Figure 8-134 in Remark 10.) for the cross section located at X/XB. The values are fiber locations for stress data recovery. Ignored for beam p-elements. (Real)	
K1, K2	Shear stiffness factor K in K*A*G for plane 1 and plane 2. See Remark 12. (Real)	1.0, 1.0
S1, S2	Shear relief coefficient due to taper for plane 1 and plane 2. Ignored for beam p-elements. (Real)	0.0, 0.0
NSI(A), NSI(B)	Nonstructural mass moment of inertia per unit length about nonstructural mass center of gravity at end A and end B. See Figure 8-134. (Real)	0.0, same as end A
CW(A), CW(B)	Warping coefficient for end A and end B. Ignored for beam p-elements. See Remark 11. (Real)	0.0, same as end A
M1(A), M2(A), M1(B), M2(B)	(y,z) coordinates of center of gravity of nonstructural mass for end A and end B. See Figure 8-134. (Real)	0.0 (no offset from shear center), same values as end A
N1(A), N2(A), N1(B), N2(B)	(y,z) coordinates of neutral axis for end A and end B. See Figure 8-134. (Real)	0.0 (no offset from shear center), same values as end A

Remarks:

1. For structural analysis, MID must reference a MAT1 material entry (SOLs 600 and 700). The beam may be described by any valid stress-strain law. A plastic hinge is not used for SOLs 600 and 700; instead, a standard nonlinear analysis is performed

2. For material nonlinear analysis, MID may also reference a MATS1 entry, but the material properties must be defined as elastic-perfectly plastic; for example, H = 0.0 on the MATS1 entry. Also, only one-eighth of the length at each end of the element abides by material nonlinear law; i.e., the element is modeled as a plastic hinge. Any other type of material property specification may yield inaccurate results.
3. For heat transfer analysis, MID must reference a MAT4 or MAT5 material entry.
4. If no stress data at end A is to be recovered and a continuation with the SO field is specified, then the first continuation entry, which contains the fields C1(A) through F2(A), may be omitted.
5. If SO is "YESA" or "NO", the third continuation entry, which contains the fields C1 through F2, must be omitted. If SO is "YES", the continuation for Ci, Di, Ei, and Fi must be the next entry. The blank fields are defaulted to 0.0 on these continuations.
6. The rules for the continuations entries are:
    - The second and third continuation entries, which contain fields SO through F2, may be repeated nine more times for intermediate X/XB values for linear beam elements. The order of these continuation pairs is independent of the X/XB value; however, one value of X/XB must be 1.0, corresponding to end B. The intermediate stress output requests will be ignored in the nonlinear solution sequences (SOLs 106 and 129).
    - The fourth and fifth continuation entries, which contain fields K1 through N2(B), are optional and may be omitted if the default values are appropriate.
7. If any fields 4 through 9 are blank on the continuation with the value of X/XB = 1.0, then the values for A, I1, I2, I12, J and NSM are set to the values given for end A. For the continuations that have intermediate values of X/XB between 0.0 and 1.0 and use the default option (any of the fields 4 through 9 are blank), a linear interpolation between the values at ends A and B is performed to obtain the missing section properties.
8. Blank fields for K1, K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear flexibilities are set to 0.0 and field G on the MAT1 entry selected by MID must be nonzero.
9. If end B forces are desired, put "YESA" in the SO field even when no end A stress points are input.

10. **Figure 8-134** describes the PBEAM element coordinate system.

Note:

I1 = $I_{(zz)_{na}}$	N1(A) = y_{na}	N1(B) = y_{nb}
I2 = $I_{(yy)_{na}}$	N2(A) = z_{na}	N2(B) = z_{nb}
I12 = $I_{(zy)_{na}}$	M1(A) = y_{ma}	M1(B) = y_{mb}
J = $I_{(xx)_{na}}$	M2(A) = z_{ma}	M2(B) = z_{mb}

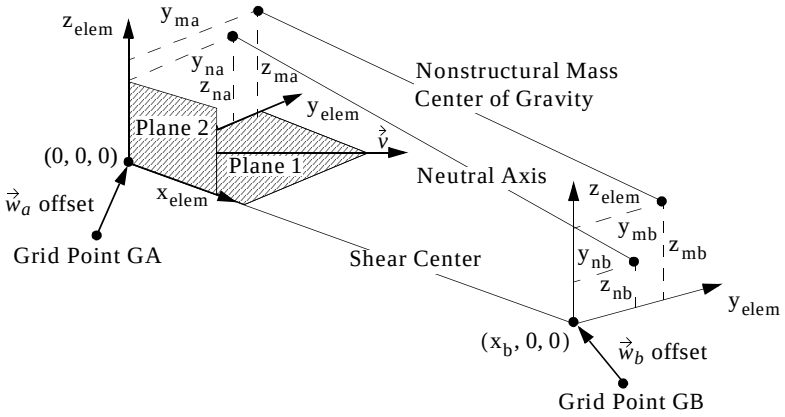


Figure 8-134 PBEAM Element Coordinate System

11. The warping coefficient CW is represented in the following differential equation for the torsion of a beam about the axis of the shear centers:

$$G\frac{d}{dx}\left(J\frac{d\theta}{dx}\right)-E\frac{d^2}{dx^2}\left(CW\frac{d^2\theta}{dx^2}\right) = m$$

where

G = shear modulus

J = torsional stiffness

E = Young's modulus

θ = angle of rotation at any cross-section

m = applied torsional moment per unit length

Note: CW has units of $(\text{length})^6$.

12. The shear stiffness factors K_1 and K_2 adjust the effective transverse shear cross-section area according to the Timoshenko beam theory. Their default values of 1.0 approximate the effects of shear deformation. To neglect shear deformation (i.e., to obtain the Bernoulli-Euler beam theory), the values of K_1 and K_2 should be set to 0.0.
13. In nonlinear analysis the location of the 8 plastic rods is the same on the PBEAM as on the PBCOMP SYMOPT = 0.
14. For SOLs 600 and 700, For structural analysis, MID must reference a MAT1 material entry.

PBEAM3 Three-node Beam Property

Defines the properties of a three-node beam element (CBEAM3 entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM3	PID	MID	A(A)	IZ(A)	IY(A)	IYZ(A)	J(A)	NSM(A)	
	CY(A)	CZ(A)	DY(A)	DZ(A)	EY(A)	EZ(A)	FY(A)	FZ(A)	
	SO(B)		A(B)	IZ(B)	IY(B)	IYZ(B)	J(B)	NSM(B)	
	CY(B)	CZ(B)	DY(B)	DZ(B)	EY(B)	EZ(B)	FY(B)	FZ(B)	
	SO(C)		A(C)	IZ(C)	IY(C)	IYZ(C)	J(C)	NSM(C)	
	CY(C)	CZ(C)	DY(C)	DZ(C)	EY(C)	EZ(C)	FY(C)	FZ(C)	
	KY	KZ	NY(A)	NZ(A)	NY(B)	NZ(B)	NY(C)	NZ(C)	
	MY(A)	MZ(A)	MY(B)	MZ(B)	MY(C)	MZ(C)	NSIY(A)	NSIZ(A)	
	NSIYZ(A)	NSIY(B)	NSIZ(B)	NSIYZ(B)	NSIY(C)	NSIZ(C)	NSIYZ(C)	CW(A)	
	CW(B)	CW(C)	STRESS						
	WC(A)	WYC(A)	WZC(A)	WD(A)	WYD(A)	WZD(A)	WE(A)	WYE(A)	
	WZE(A)	WF(A))	WYF(A)	WZF(A)	WC(B)	WYC(B)	WZC(B)	WD(B)	
	WYD(B)	WZD(B)	WE(B)	WYE(B)	WZE(B)	WF(B)	WYF(B)	WZF(B)	
	WC(C)	WYC(C)	WZC(C)	WD(C)	WYD(C)	WZD(C)	WE(C)	WYE(C)	
	WZE(C)	WF(C)	WYF(C)	WZF(C)					

Example:

PBEAM3	1010	2	2.9	3.5	5.97			1.0	
	0.2	3.0	-1.2	2.6	2.0	0.5			
	YES		1.0	23.6	34.7				
	1.1	3.2							
	YESA		3.2	2.1	3.2			1.0	
	0.8		0.5						
	0.9	1.0		1.5					
		1.0							

Field	Contents
PID	Property identification number. (Integer > 0, Required)
MID	Material identification number. See Remark 1. (Integer > 0; Required)
A(A)	Area of the beam cross-section at end A. (Real > 0.0; Required)
IZ(A)	Area moment of inertia at end A about local z-axis and the neutral axis. (Real > 0.0; Required)
IY(A)	Area moment of inertia at end A about local y-axis and the neutral axis. (Real > 0.0; Required)
IYZ(A)	Area product of inertia at end A about local y- and z-axes and the neutral axis. If y- and z- axes are principal axes, then IYZ(A)=0.0. (Real, but $I_y \cdot I_z - I_{yz}^2 > 0.0$; Default = 0.0)
J(A)	Torsional stiffness parameter at end A. (Real > 0.0; Default = IZ+IY)
NSM(A)	Nonstructural mass per unit length at end A. (Real; Default = 0.0)
C*i*(*f*), D*i*(*j*) E*i*(*j*), F*i*(*j*)	The local y and z coordinates (*i*=Y, Z) at point *j* (*j*=A, B, C), used for stress output. (Real; Default = 0.0)
A(j), IZ(j), IY(j) IYZ(j), J(j), NSM(j)	Area, moments of inertia, torsional stiffness parameter and nonstructural mass for the cross-section at *j* (*j*=B, C). (Real; See Remark 2)
SO(j)	Stress output request optioon at *j* (*j*=B, C). (Character; Default = “YESA”) “YES”: Stresses are recovered at C*i*, D*i*, E*i*, and F*i* on the next continuation. “YESA”: Stresses are recovered at points with the same (*y, z*) location at end A
KY, KZ	Shear effectiveness factors for local y- and z-directions. (Real > 0.0, Default = 1.0)
NY(j), NZ(j)	Local (*y, z*) coordinates of neutral axis for *j* (*j*=A, B, C). (Real, Default = 0.0 at end A and same values as end A for *j* = B, C)
MY(*j*), MZ(*j*)	Local (y, z) coordinates of nonstructural mass cneter of gravity at *j* (*j*=A, B, C). (Real, Default = 0.0 at end A and same values as end A for *j*=B,C)

Field	Contents
NSIY(j), NSIZ(j)	Nonstructural mass moments of inertia per unit length about local y and z-axes, respectively, with regard to the nonstructural mass center of gravity at *j* (*j*=A, B, C). (Real, Default = 0.0 at end A and same values as end A for *j*=B, C)
NSIYZ(j)	Nonstructural mass product of inertia per unit length about local y and z-axes, respectively, with regard to the nonstructural mass center of gravity at *j* (*j*=A, B, C). (Real, Default = 0.0 at end A and same values as end A for *j*=B, C)
CW(j)	Warping coefficient at *j* (*j*=A, B, C). (Real $\geq$ 0.0; Default = 0.0 at end A; same values as end A for *j* = B, C)
STRESS	Location selection for stress, strain and force output. (Character, Default = "GRID", See Remark 3)
Wi(j)	Values of warping function at stress recovery points *i* = C, D, E and F, at location *j*=A, B, and C. (Real; Default = 0.0 at end A and same values as end A for *j*=B, C)
WYi(j), WZi(j)	Gradients of warping function in the local (y, z) coordinate system at stress recovery points *i*=C, D, E, and F, at location *j*=A, B, and C. (Real; Default = 0.0 at end A and same values as end A for *j*=B, C.

Remarks:

1. For structural analysis, MID must reference a MAT1, MAT2 or MAT8 material entry.
2. If any fields 4 through 9, for values of A, IZ, IY, IYZ, J and NSM at end B or C, are blank, then those values for end B or C are set to the values given for end A.
3. If STRESS="GRID", then the stresses, strains and forces are recovered at A, B and C. If STRESS="GAUSS", then the stresses, strains and forces are recovered at Gauss integration points, $\xi = \{1/\sqrt{3}, 1/\sqrt{3}, 0\}$. The beam cross-section properties at these points are interpolated from those at A, B and C.
4. If all fields of W*i*(*j*), WY*i*(*j*) and WZ*i*(*j*) (*i*=C, D, E, F and *j*=A, B, C), are left blank, both stresses and strains due to the warping effect will not be recovered at the stress recovery points.

PBEAM71 (SOL 700) (Alternate Format 1) Beam Properties (Belystschko-Schwer Beams)

Defines complex beam properties that cannot be defined using the PBAR or PBEAM entries. These entries are to be used only for Belytschko-Schwer elements.

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM71	PID	MID	FORM				SHFACT	SECT	
	A	I1	I2	J	ZPZ	ZPY			
	CS1	CS2	CS3	CS4	CS5	CS6			

Example:

PBEAM71	1	7	BELY			0.9	RECT		

Field	Contents	Type	Default
PID	Unique property number.	I > 0	Required
MID	Material number.	I > 0	PID
FORM	Element formulation.	C	Required
	BELY Belytschko-Schwer.		
SHFACT	Shear factor for the section.	R	0.83333
SECT	Type of section. See Remark 4.	C	RECT
A	Area of the section.	R	Blank
I1	The moment of inertia about the element z-axis.	R	Blank
I2	The moment of inertia about the element y-axis.	R > 0.	Blank
J	The torsional stiffness of the section.	R ≥ 0.	Blank
ZPZ	Plastic modulus *Zp* about the element z-axis.	R > 0.	Blank
ZPY	Plastic modulus *Zp* about the element y-axis.	R > 0.	Blank
CSi	Geometrical definition of the cross section. The data in these fields depends on the type of the section.	R ≥ 0.	See Remark 4.

Remarks:

1. Only the entries that are relevant for Belytschko-Schwer beam definition are listed. PBEAM1 entries that apply to Hughes-Liu beams appear earlier in this PBEAM1 discussion.
2. Note the following:

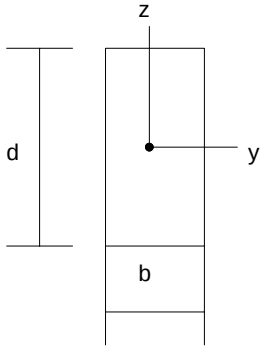
$$I1 = I_{zz}$$

$$I2 = I_{yy}$$

$$J = I_{xx}$$

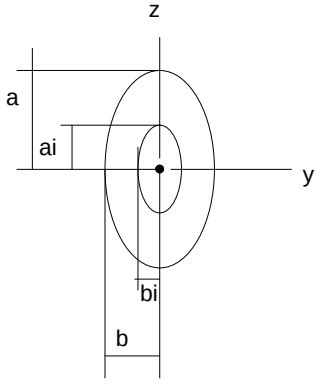
3. The cross-sectional properties are calculated as follows:
   a. If the geometry is defined in the fields CSi, the values of A, I1, I2, J, ZPZ and ZPY are calculated automatically.
   b. If a value is defined in the fields A, I1, I2, J, ZPZ, ZPY, these values override the values as calculated in step a.
   c. All values of CSi for a particular cross section (see Remark 4) must be entered for the geometry to be defined. If not all values of CSi are supplied, then values for A, I1, I2 and J are required, and ZPZ, ZPY will have a default value of 1.E20.
4. The geometrical definitions for the various cross sections are defined in the element coordinate system as follows:

SECT = RECT



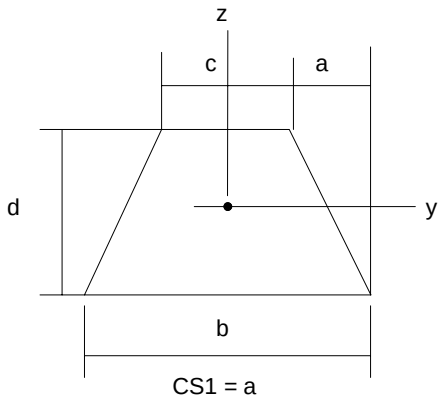
CS1 = b
CS2 = d

SECT = TUBE



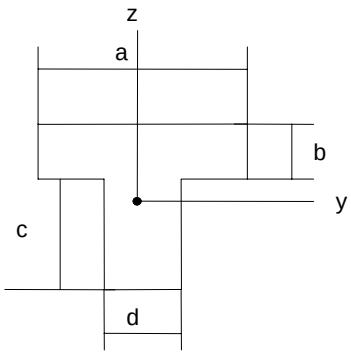
CS1 = b
CS2 = a
CS3 = bi
CS4 = ai

SECT = TRAPZ



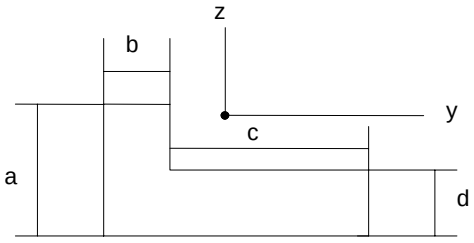
CS1 = a
CS2 = b
CS3 = c
CS4 = d

SECT = TSECT



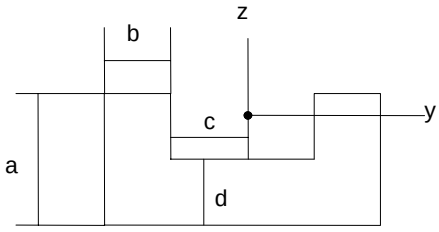
CS1 = a
CS2 = b
CS3 = c
CS4 = d

SECT = LSECT



CS1 = a
CS2 = b
CS3 = c
CS4 = d

SECT = USECT

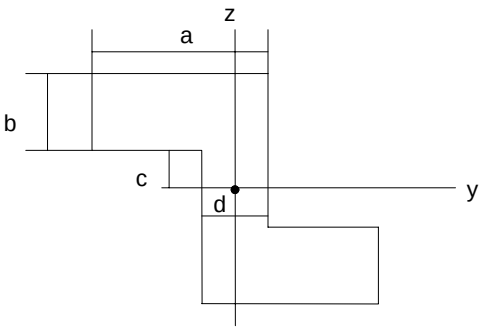


CS1 = a
CS2 = b
CS3 = c
CS4 = d

PBEAM71 (SOL 700)

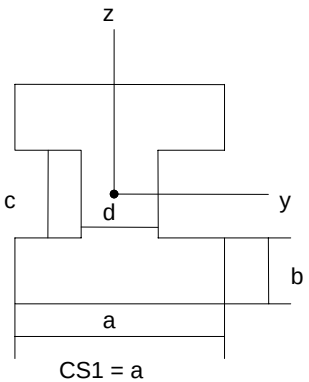
(Alternate Format 1) Beam Properties (Belystschko-Schwer Beams)

SECT = ZSECT



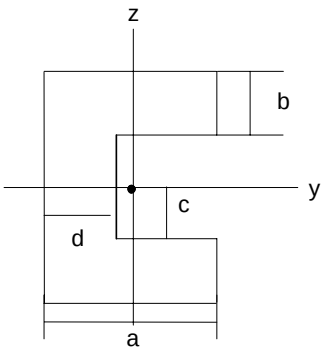
CS1 = a
CS2 = b
CS3 = c
CS4 = d

SECT = ISECT



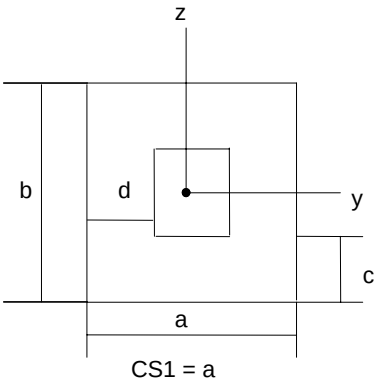
CS1 = a
CS2 = b
CS3 = c
CS4 = d

SECT = CSECT



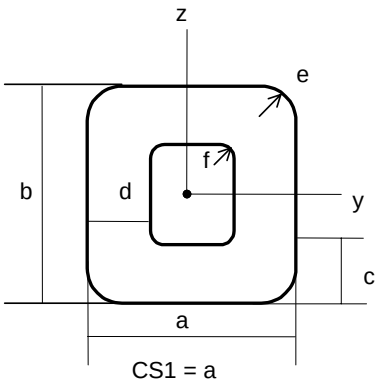
CS1 = a
CS2 = b
CS3 = c
CS4 = d

SECT = BOXSECT



CS1 = a
CS2 = b
CS3 = c
CS4 = d

SECT = RCBSECT



CS1 = a
CS2 = b
CS3 = c
CS4 = d
CS5 = e
CS6 = f

PBEAM71 (SOL 700) (Alternate Format 2) Beam Properties (Predefined Hughes-Liu Cross Sections)

Defines more complex beam properties that cannot be defined using the PBAR or PBEAM entries. The following entries are for predefined cross sections of Hughes-Liu beam elements only.

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM71	PID	MID	FORM	DATABASE		SHFACT	SECT		
	V1	V2	V3	V4	V5	V6	V7	V8	
	N1(A)	N2(1)	N1(B)	N2(B)					

Example:

PBEAM71	1	7	HELSECTS	DYTRAN		0.9	ISECT		
	30.1	30.1	10.0	10.0					
	2.0		2.0						

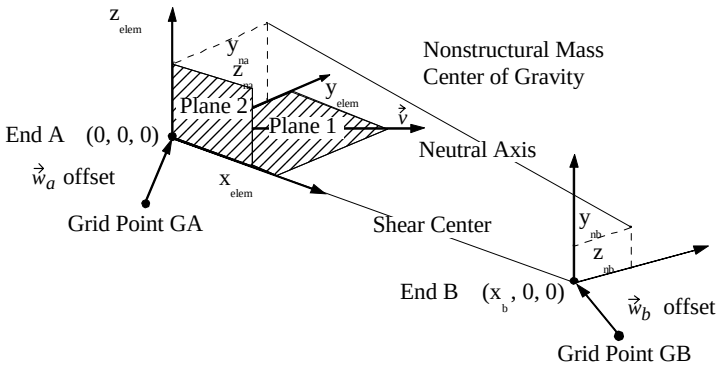
Field	Contents	Type	Default
PID	Unique property number.	I > 0	Required
MID	Material number.	I > 0	PID
FORM	Element formulation.	C	Required
	HLSECTS Predefined Hughes-Liu cross sections.		
DATABASE	Cross-section database.	C	Required
	DYTRAN See Figures in Remark 3 for available cross sections.		
	NASTRAN See Figures in Remark 4 for available cross sections.		
SHFACT	Shear factor for the section.	R	0.83333
SECT	Type of section.	C	Required
	TSECT "T" cross section.		
	LSECT "L" cross section.		
	USECT "U"/"CHAN2" cross section.		

Field	Contents	Type	Default
	ZSECT "Z" cross section.		
	ISECT "I" cross section.		
	CSECT "C" cross section. (NASTRAN Database only)		
	BOXSECT "BOX" cross section. (NASTRAN Database only)		
	HATSECT "HAT" cross section. (NASTRAN Database only)		
	RCBSECT Round Corners BOX" cross section. (NASTRAN Database only)		
V1–V4	Geometric properties of the beam. The data in these fields depends on the beam formulation and the type of cross section.		
	For Hughes formulations of the DYTRAN database cross sections.		
	V1-V4 Cross Section Dimensions at end A.	R	Required
	V5-V8 Cross Section Dimensions at end B.	R	Same as V1-V4
	For Hughes formulations of the NASTRAN Database cross sections:		
	V1-V6Cross Section Dimensions of beam.	R	Required
N1(A), N2(A), N1(B), N2(B)	(y,z) coordinates of neutral axis for end A and end B. See the figure in Remark2.	R	0.0

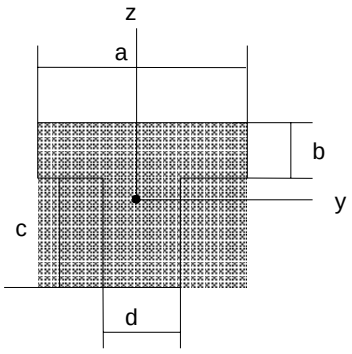
Remarks:

The cross sections TUBE and RECT can be defined in the regular Hughes-Liu PBEAM1 entry.

1.

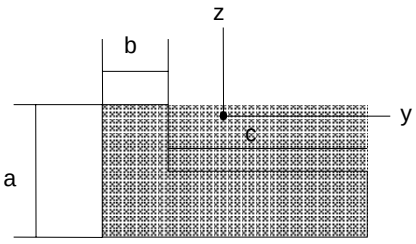

z elem
y na
z na
y elem
Plane 2
Plane 1
Nonstructural Mass
Center of Gravity
End A (0, 0, 0)
$\vec{v}$
Neutral Axis
$\vec{w}_a$ offset
x elem
Grid Point GA
Shear Center
y nb
z nb
End B (x b, 0, 0)
$\vec{w}_b$ offset
Grid Point GB

SECT = TSECT



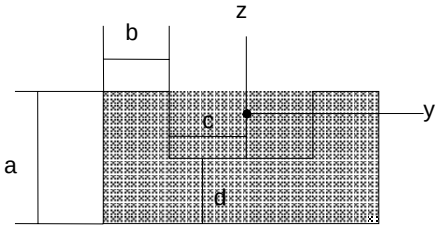
End A	End B
V1 = a	V5 = a
V2 = b	V6 = b
V3 = c	V7 = c
V4 = d	V8 = d

SECT = LSECT



End A	End B
V1 = a	V5 = a
V2 = b	V6 = b
V3 = c	V7 = c
V4 = d	V8 = d

SECT = USECT

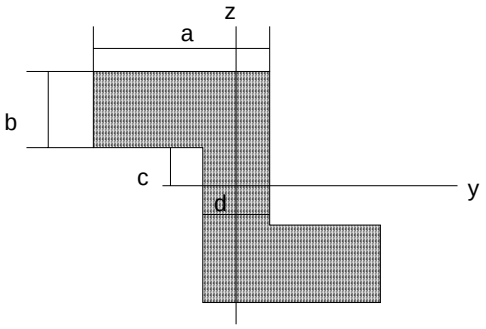


End A	End B
V1 = a	V5 = a
V2 = b	V6 = b
V3 = c	V7 = c
V4 = d	V8 = d

PBEAM71 (SOL 700)

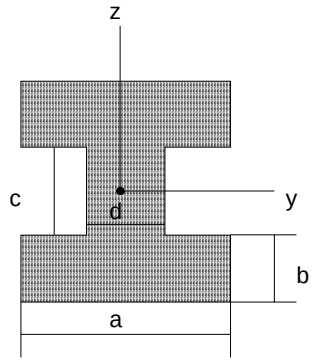
(Alternate Format 2) Beam Properties (Predefined Hughes-Liu Cross Sections)

SECT = ZSECT

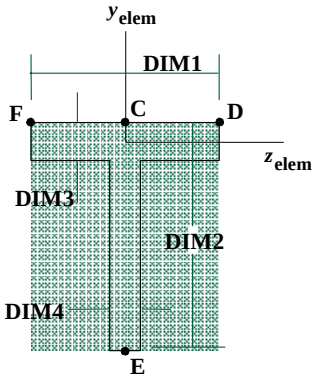


End A	End B
V1 = a	V5 = a
V2 = b	V6 = b
V3 = c	V7 = c
V4 = d	V8 = d

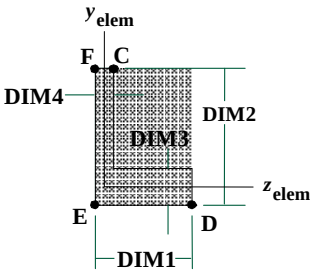
SECT = ISECT



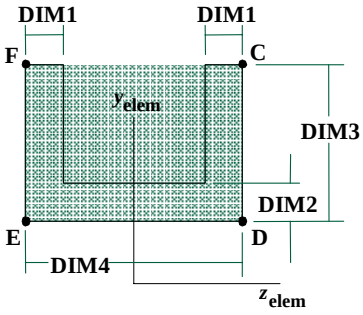
End A	End B
V1 = a	V5 = a
V2 = b	V6 = b
V3 = c	V7 = c
V4 = d	V8 = d



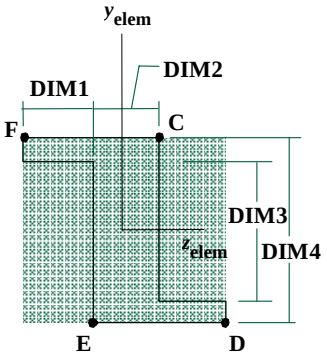
TYPE="TSECT"



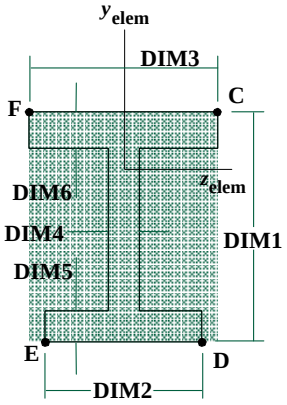
TYPE="LSECT"



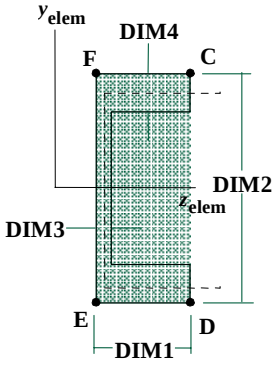
TYPE="USECT"



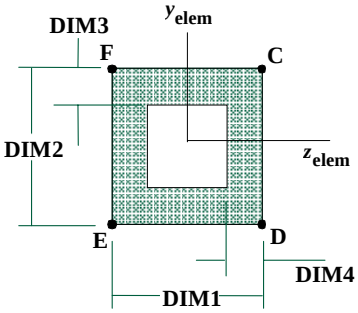
TYPE="ZSECT"



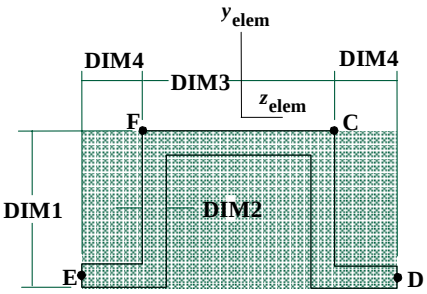
TYPE="ISECT"



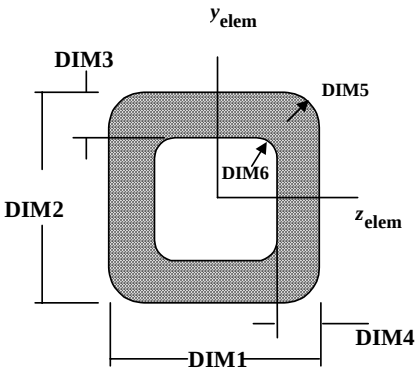
TYPE="CSECT"



TYPE="BOXSECT"



TYPE="HATSECT"



TYPE="RCBSECT"

PBEAM71 (SOL 700) (Alternate Format 3) Beam Properties (Hughes-Liu Beams)

Defines more complex beam properties that cannot be defined using the PBAR or PBEAM entries. The following entries are for Hughes-Liu beam elements only.

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM71	PID	MID	FORM	QUAD	NUMB	SHFACT	SECT		
	V1	V2	V3	V4					

Example:

PBEAM71	1	7	HUGHES	QUAD		0.9	SECT		
	30.1	30.1	10.0	10.0					

Field	Contents	Type	Default
PID	Unique property number.	I > 0	Required
MID	Material number.	I > 0	PID
FORM	Element formulation.	C	Required
	HUGHES Hughes-Liu.		
QUAD	Type of quadrature.	C	GAUSS
	GAUSS Gauss quadrature.		
	LOBATTO Lobatto quadrature.		
NUMB	The number of integration points for Hughes-Liu beams. For Gauss integration, the following can be specified:	I > 0	3
	1 1 point (rod element).		
	2 2x2 points (4-point circle, if tubular).		
	3 3x3 points (9-point circle, if tubular).		
	4 4x4 points (16-point circle, if tubular).		
	At present only 3x3 points are available with the Lobatto quadrature.		
SHFACT	Shear factor for the section.	R	0.83333

Field	Contents	Type	Default
SECT	Type of section. RECT Rectangular cross section. TUBE Tubular cross section.	C	RECT
V1–V4	Geometric properties of the beam. The data in these fields depends on the beam formulation and the type of cross section. For Hughes formulations with rectangular cross sections. V1 The thickness in the element y direction at grid point 1. V2 The thickness in the element y direction at grid point 2. V3 The thickness in the element z direction at grid point 1. V4 The thickness in the element z direction at grid point 2. For Hughes formulations with tubular cross sections: V1 The outer diameter at grid point 1. V2 The outer diameter at grid point 2. V3 The inner diameter at grid point 1. V4 The inner diameter at grid point 2.	R	Required

PBEAMD (SOL 700)

Defines cross-sectional properties for beam, truss, discrete beam, and cable elements.

Format 1 (ELFORM = 1, integrated beam type 1):

1	2	3	4	5	6	7	8	9	10
PBEAMD	PID	MID	ELFORM	SHRF	QR/IRID	CST	SCOOR	NSM	
	TS1	TS2	TT1	TT2	NSLOC	NTLOC			

Format 2 (ELFORM = 2 or 3, resultant formulation):

PBEAMD	PID	MID	ELFORM	SHRF	QR/IRID	CST	SCOOR	NSM	
	A	ISS	ITT	IRR	SA				

Format 3 (ELFORM = 2 or 3, resultant formulation, alternate entry):

PBEAMD	PID	MID	ELFORM	SHRF	QR/IRID	CST	SCOOR	NSM	

	STYPE	D1	D2	D3	D4	D5	D6		

Format 4 (ELFORM = 4or 5, integrated beam formulation):

PBEAMD	PID	MID	ELFORM	SHRF	QR/IRID	CST	SCOOR	NSM	
	TS1	TS2	TT1	TT2					

Example:

PBEAMD	11	2	1	0.5	2	1.0	0.0		
	0.7	.07	0.6	.06	0.0	0.0			

Field	Contents	Type	Default
PID	Property ID. Must be unique.	Integer > 0	Required
MID	Material ID.	Integer > 0	Required

Field	Contents	Type	Default
ELFORM	Element formulation options: EQ.1: Hughes-Liu with cross section integration (Default), EQ.2: Belytschko-Schwer resultant beam (resultant), EQ.3: truss (resultant), see Remark 2. EQ.4: Belytschko-Schwer full cross-section integration, EQ.5: Belytschko-Schwer tubular beam with cross-section integration, Note that the 2D and 3D element types must not be mixed, and different types of 2D elements must not be used together. For example, the plane strain element type must not be used with the axisymmetric element type. In 3D the different beam elements types, i.e., 1-6 and 9 can be freely mixed together.	Integer > 0	1
SHRF	Shear factor. This factor is not needed for truss, resultant beam, discrete beam. The recommended value for rectangular sections is 5/6, the default is 1.0.	Real > 0	1.0
QR/IRID	Quadrature rule of rule number for user-defined rule for integrated beams: EQ.1.0: one integration point, EQ.2.0: 2x2 Gauss quadrature (default beam) EQ.3.0: 3x3 Gauss quadrature, EQ.4.0: 3x3 Lobatto quadrature, EQ.5.0: 4x4 Gauss quadrature	Real > 0	2.0
CST	Cross-section type, not needed for truss, resultant beam, and discrete beam. EQ.0.0: rectangular, EQ.1.0: tubular (circular only) EQ.2.0: arbitrary (user-defined integration rule).	Real > 0.0	0.0

Field	Contents	Type	Default
SCOOR	Location of triad for tracking the rotation of the discrete beam element, see the parameter CID below. The force and moment resultants in the output databases are referenced to this triad. The flags -3.0, -1.0, 0.0, 1.0, and 3.0 are inactive if the option to update the local system is active in the CID definition. EQ.-3.0: beam node 1, the angular velocity of node 1 rotates triad, EQ.-2.0: beam node 1, the angular velocity of node 1 rotates triad but the r-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams., EQ.-1.0: beam node 1, the angular velocity of node 1 rotates triad, EQ. 0.0: centered between beam nodes 1 and 2, the average angular velocity of nodes 1 and 2 is used to rotate the triad, EQ.+1.0:beam node 2, the angular velocity of node 2 rotates triad. EQ.+2.0:beam node 2, the angular velocity of node 2 rotates triad. but the r-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams. EQ.+3.0:beam node 2, the angular velocity of node 2 rotates triad. *If the magnitude of SC00R is less than or equal to unity then zero length discrete beams are assumed with infinitestimal separation between the nodes in the deformed state. For large separations or nonzero length beams set |SCOOR| to 2 or 3.*		

Field	Contents	Type	Default
NSM	Nonstructural mass per unit length. This option applies to beam types 1-5 and does not apply to discrete, 2D, and spotweld beams, respectively.	Real > 0.0	0.0
TS1	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n1.	Real > 0.0	0.0
TS2	Beam thickness (CST=0.0, 2.0) or outer diameter (CST = 1.0) in s direction at node n2.	Real > 0.0	0.0
TT1	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n1.	Real > 0.0	0.0
TT2	Beam thickness (CST=0.0, 2.0) or inner diameter (CST = 1.0) in t direction at node n2.	Real > 0.0	0.0
NSLOC	Location of reference surface normal to s axis for Hughes-Liu beam elements only: EQ.1.0: side at s=1.0, EQ.0.0: center, EQ.-1.0: side at s = -1.0.	Real > 0.0	0.0
NTLOC	Location of reference surface normal to t axis for Hughes-Liu beam elements only: EQ.1.0: side at t =1.0, EQ.0.0: center, EQ.-1.0: side at t = -1.0.	Real > 0	0.0
A	Cross-sectional area.	Real > 0	0.0
ISS	I_{ss}.	Real > 0	0.0
ITT	I_{tt}.	Real > 0	0.0
IRR	I_{rr} (J) polar inertia. If IRR is zero, then IRR is reset to the sum of ISS+ITT as an approximation.	Real > 0.0	ISS+ITT
SA	Shear area.	Real > 0	0.0
STYPE	Section type (A format): EQ.SECTION_01: I-shape EQ.SECTION_12: Cross	I > 0	Required

Field	Contents	Type	Default
	EQ.SECTION_02: Channel EQ.SECTION_13: H-shape		
	EQ.SECTION_03: L-shap EQ.SECTION_14: T-shape1		
	EQ.SECTION_04: T-shape EQ.SECTION_15: I-shape2		
	EQ.SECTION_05: Tubular box EQ.SECTION_16: Channel1		
	EQ.SECTION_06: Z-shape EQ.SECTION_17: Channel2		
	EQ.SECTION_07:Trapezoidal EQ.SECTION_18: T-shape2		
	EQ.SECTION_08: Circular EQ.SECTION_19: Box-shape1		
	EQ.SECTION_09: Tubular EQ.SECTION_20: Hexagon		
	EQ.SECTION_10: I-shape1 EQ.SECTION_21: Hat-shape		
	EQ.SECTION_11: Solid box EQ.SECTION_22: Hat-shape1		
D1-D6	Input parameters for section option using STYPE above.	Real > 0	Required
VOL	Volume of discrete beam. If the mass density of the material model for the discrete beam is set to unity, the magnitude of the lumped mass can be defined here instead. This lumped mass is partitioned to the two nodes of the beam element. The translational time step size for the type 6 beam is dependent on the volume, mass density, and the translational stiffness values, so it is important to define this parameter. Defining the volume is also essential for mass scaling if the type 6 beam controls the time step size.	Real > 0	0.0

Field	Contents	Type	Default
INER	Mass moment of inertia for the six degree of freedom discrete beam. This lumped inertia is partitioned to the two nodes of the beam element. The rotational time step size for the type 6 beam is dependent on the lumped inertia and the rotational stiffness values, so it is important to define this parameter if the rotational springs are active. Defining the rotational inertia is also essential for mass scaling if the type 6 beam rotational stiffness controls the time step size.	Real > 0	0.0
CID	Coordinate system ID for orientation (material types 66-69, 93, 95, 97), see CORDx. If CID=0, a default coordinate system is defined in the global system or on the third node of the beam, which is used for orientation. This option is not defined for material types than act between two nodal points, such as cable elements. The coordinate system rotates with the discrete beam, see SCOOR above.	I > 0	0
RRCON	r-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about r axis with nodes. EQ.1.0: Rotation is constrained about the r-axis	Real > 0	0.0
SRCON	s-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about s axis with nodes. EQ.1.0: Rotation is constrained about the s-axis	Real > 0	0.0
TRCON	t-rotational constraint for local coordinate system EQ.0.0: Coordinate ID rotates about t axis with nodes. EQ.1.0: Rotation is constrained about the t-axis	Real > 0	0.0
DOFN1	Active degree-of-freedom at node 1, a number between 1 and 6 where 1 in x-translation and 4 is x-rotation.	I > 0	0

Field	Contents	Type	Default
DOFN2	Active degree-of-freedom at node 2, a number between 1 and 6.	I > 0	0
PRINT	Output spot force resultant from spotwelds.	Real > 0	0.0
	EQ.0.0: Data is output to SWFORC file.		
	EQ.1.0: Output is suppressed.		

Remarks:

1. For implicit calculations all of the beam element choices are implemented:
2. For the truss element, define the cross-sectional area, A, only.
3. The local coordinate system rotates as the nodal points that define the beam rotate. In some cases this may lead to unexpected results if the nodes undergo significant rotational motions. In the definition of the local coordinate system using CORDi2, if the option to update the system each cycle is active then this updated system is used. This latter technique seems to be more stable in some applications.

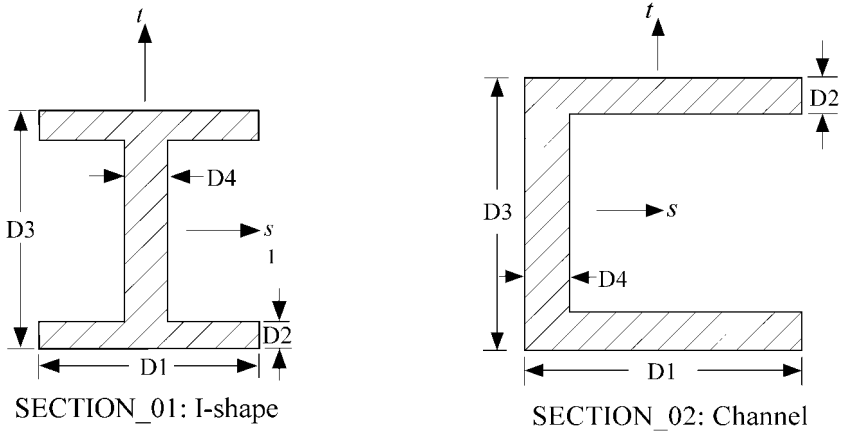


Figure 8-135 Properties of beam cross section for several common cross sections.

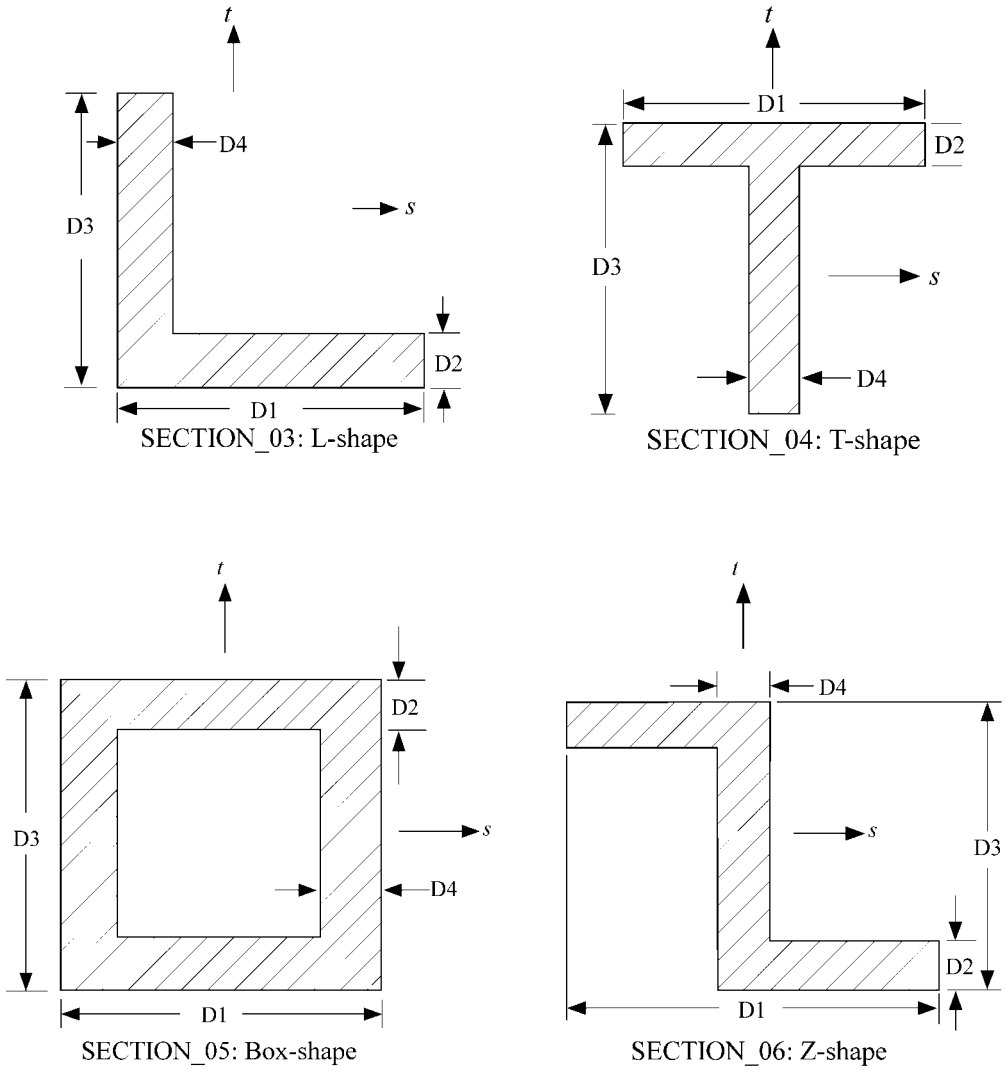


Figure 135. Properties of beam cross section for several common cross sections. (Cont.)

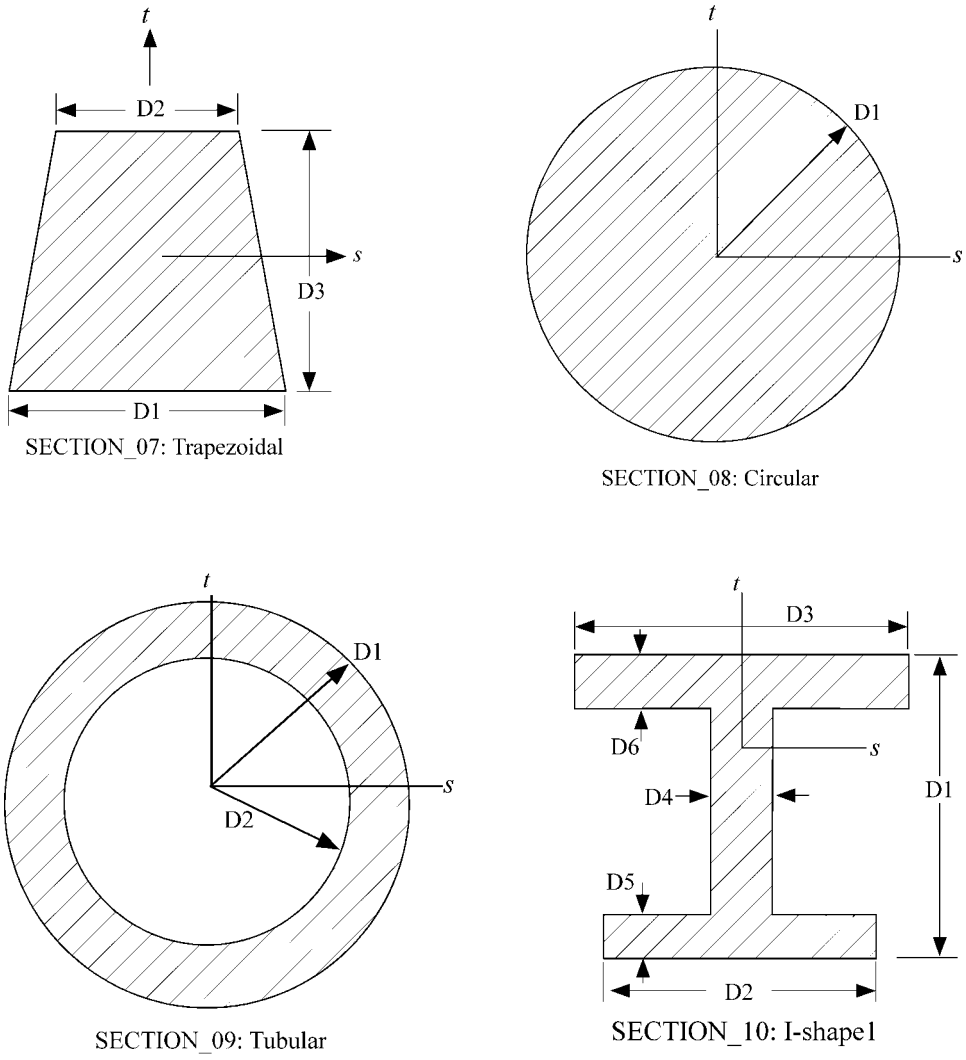


Figure 135. Properties of beam cross section for several common cross sections. (Cont.)

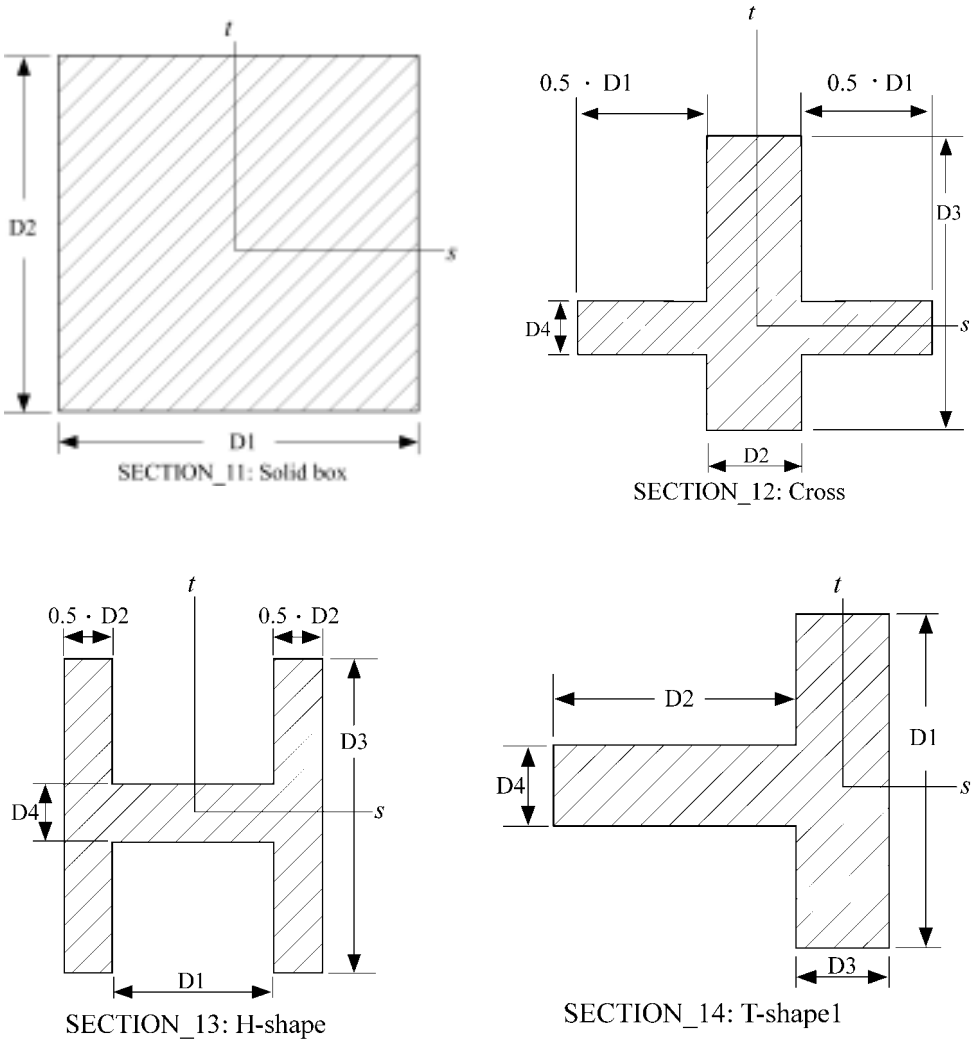


Figure 135. Properties of beam cross section for several common cross sections. (Cont.)

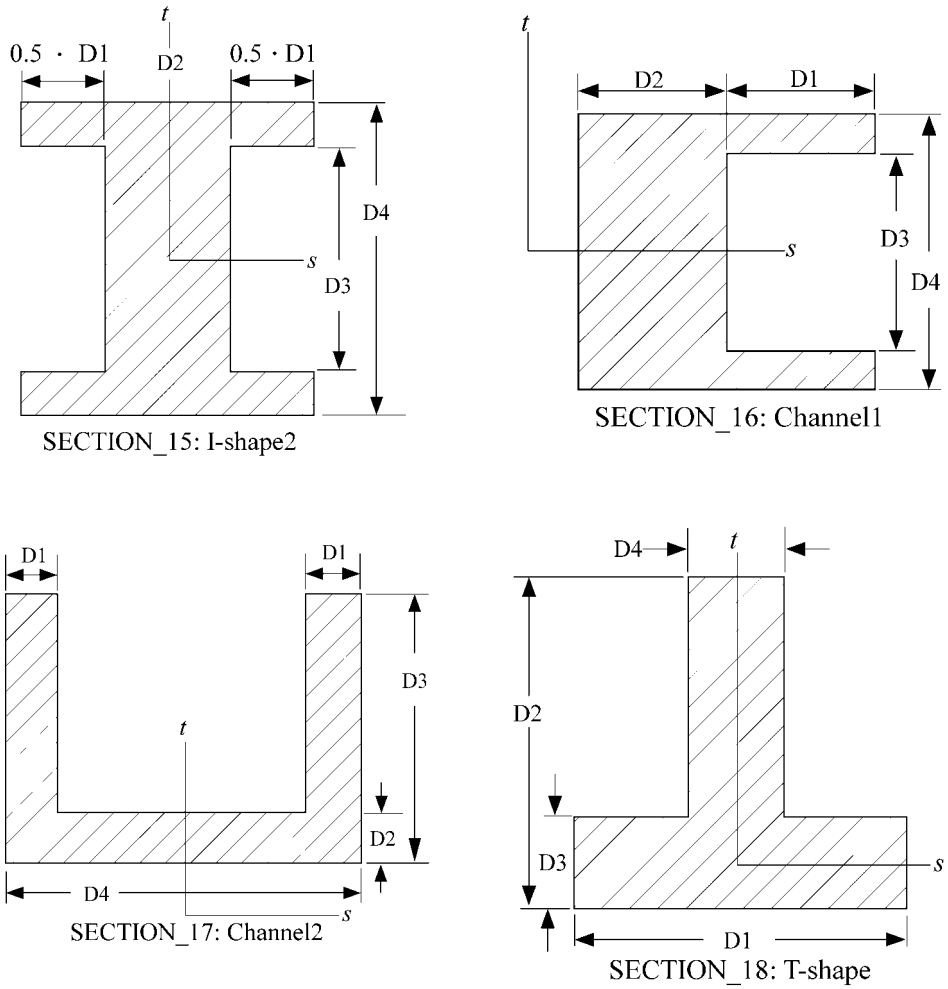


Figure 135. Properties of beam cross section for several common cross sections. (Cont.)

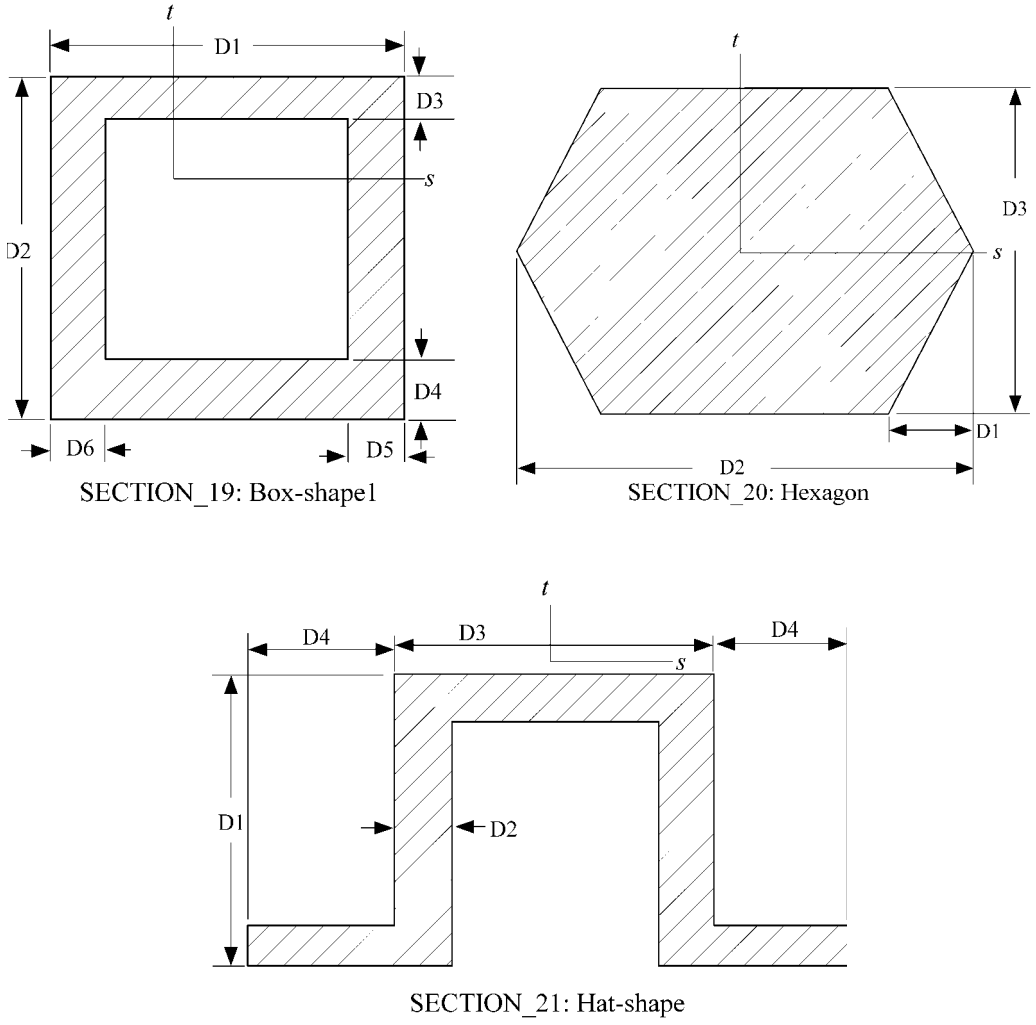


Figure 135. Properties of beam cross section for several common cross sections. (Cont.)

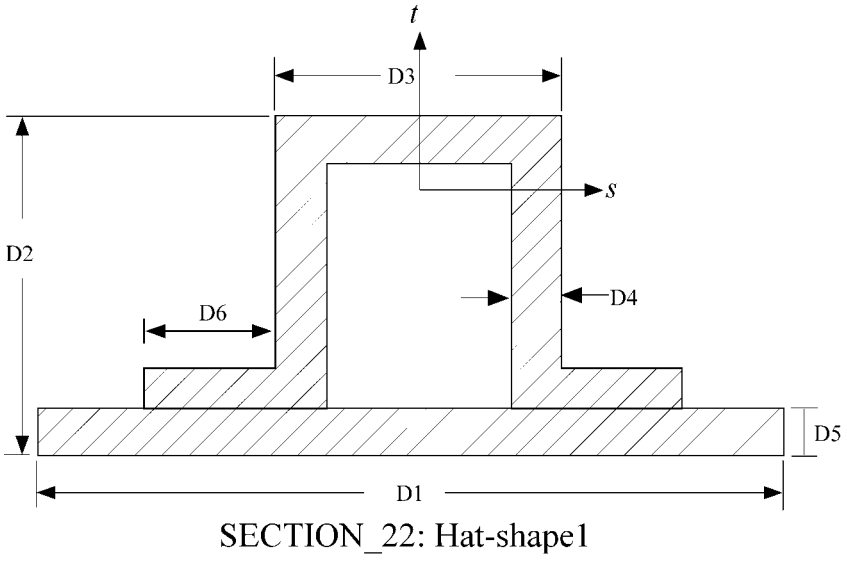


Figure 135. Properties of beam cross section for several common cross sections. (Cont.)

PBEAML Beam Cross-Section Property

Defines the properties of a beam element by cross-sectional dimensions.

Format:

(Note: n = number of dimensions and m = number of intermediate stations)

1	2	3	4	5	6	7	8	9	10
PBEAML	PID	MID	GROUP	TYPE					
	DIM1(A)	DIM2(A)	-etc.-	DIMn(A)	NSM(A)	SO(1)	X(1)/XB	DIM1(1)	
	DIM2(1)	-etc.-	DIMn(1)	NSM(1)	SO(2)	X(2)/XB	DIM1(2)	DIM2(2)	
	-etc.-	DIMn(2)	NSM(m)	-etc.-	SO(m)	X(m)/XB	DIM1(m)	-etc.-	
	DIMn(m)	NSM(m)	SO(B)	1.0	DIM1(B)	DIM2(B)	-etc.-	DIMn(B)	
		NSM(B)							

Example:

PBEAML	99	21		T					
	12.	14.8	2.5	2.6		NO	0.4	6.	
	7.	1.2	2.6		YES	0.6	6.	7.8	
	5.6	2.3		YES					

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. (Character; Default = “MSCBML0")
TYPE	Cross-section shape. See Remark 4. (Character: “ROD”, “TUBE”, “L”, “I”, “CHAN”, “T”, “BOX”, “BAR”, “CROSS”, “H”, “T1", “I1", “CHAN1", “Z”, “CHAN2", “T2", “BOX1", “HEXA”, “HAT”, “HAT1”, “DBOX” for GROUP = “MSCBML0")
DIMi(j)	Cross-section dimensions at end A, intermediate station j and end B. (Real > 0.0 for GROUP = “MSCBML0")
NSM(j)	Nonstructural mass per unit length. (Default = 0.0)
SO(j),SO(B)	Stress output request option for intermediate station j and end B. (Character; Default = “YES”)

Field	Contents
	YES: Stresses recovered at all points on next continuation and shown in **Figure 8-136** as C, D, E, and F.
	NO: No stresses or forces are recovered.
X(j) / XB	Distance from end A to intermediate station j in the element coordinate system divided by the length of the element. (Real>0.0; Default = 1.0)

Remarks:

1. For structural problems, PBEAML entries must reference a MAT1 material entry.
2. PID must be unique with respect to all other PBEAM and PBEAML property identification numbers.
3. For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
4. See the PBEAM entry description for a discussion of beam-element geometry.
5. If any of the fields NSM(B), DIMi(B) are blank on the continuation entry for End B, the values are set to the values given for end A. For the continuation entries that have values of X(j) / XB between 0.0 and 1.0 and use the default option (blank field), a linear interpolation between the values at ends A and B is performed to obtain the missing field.
6. The GROUP is associated with a FMS CONNECT statement, which specifies the evaluator. A reserved GROUP name is “MSCBML0". Users may create their own cross-section types. Each of the types will require a one or more subroutines to convert DIMi information to geometric property information contained on a PBEAM entry. See “**Building and Using the Sample Programs**” on page 255 of the *MD Nastran 2006 Installation and Operations Guide* for a discussion of how to include a user-defined beam library.
7. For GROUP = “MSCBML0", the cross-sectional properties, shear flexibility factors and stress recovery points are computed using the TYPE and DIMi as shown in **Figure 8-136**. The element coordinate system is located at the shear center.
8. A function of this entry is to derive an equivalent PBEAM entry. Any sorted echo request will also cause printout and / or punch of the derived PBEAM.

9. Beams can have no more than 14 dimensions per station. The total number of dimensions at all stations must be less than 200. The transfer of data with the beam server is limited to 4000 words.

   None of these limits are exceeded with the MSC beam library, but a user defined beam library could.

10. Finite element formulation (FEF), utilized for arbitrary beam cross section, is selected as default method for computing sectional properties for all supported cross section types of PBEAML if GROUP=MSCBML0. The original beam equations which are based on thin-walled assumption can be accessed via Bulk Data entry ‘MDLPRM,TWBRBML,1’. For optimization, individual DIMx of PBEAML can be selected as design property even with finite element formulation.

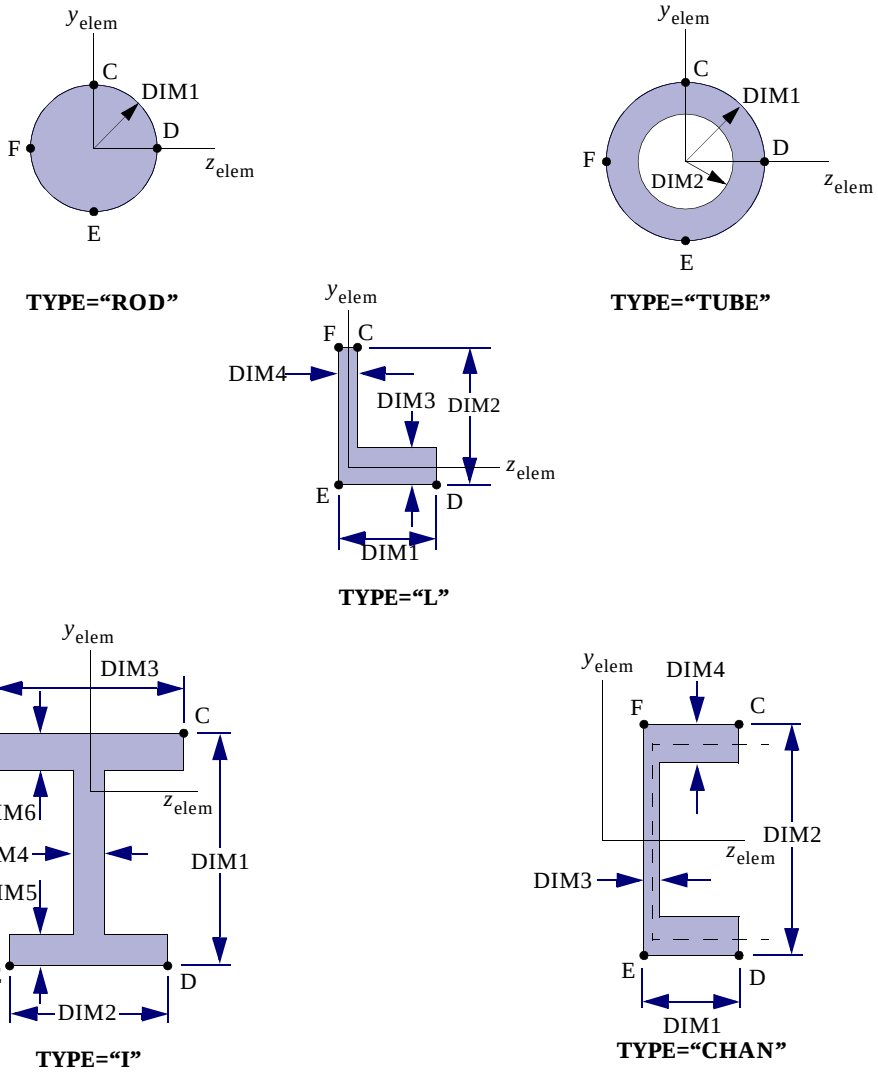
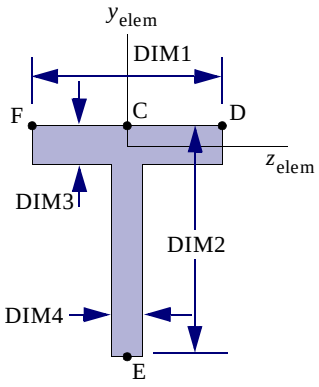
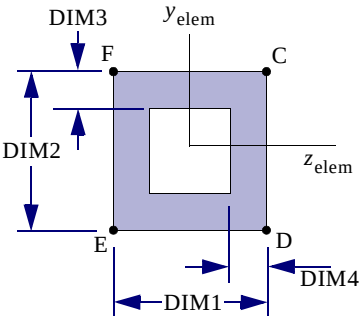


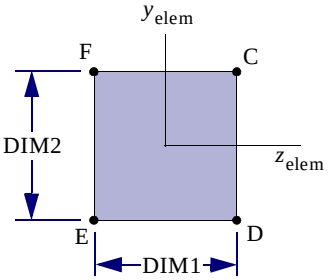
Figure 8-136 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"



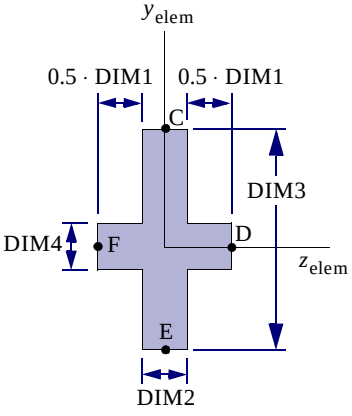
TYPE="T"



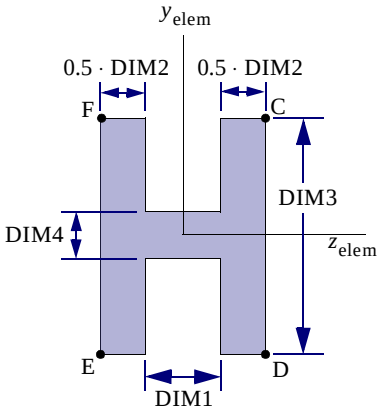
TYPE="BOX"



TYPE="BAR"

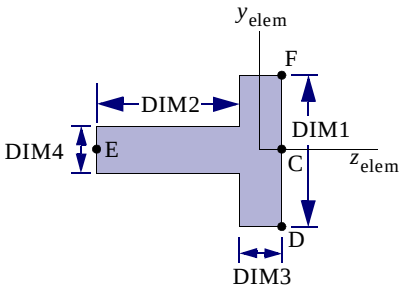


TYPE="CROSS"

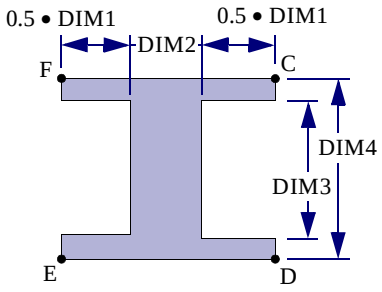


TYPE="H"

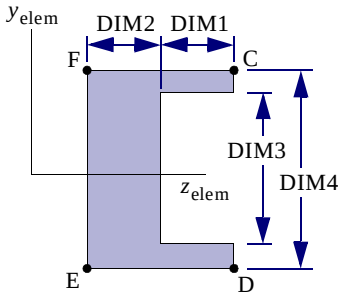
Figure 8-136 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)



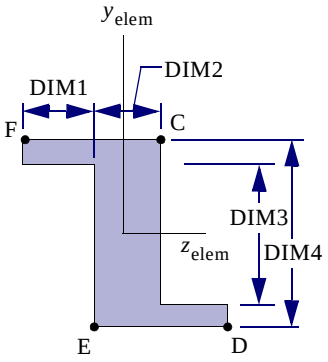
TYPE=”T1”



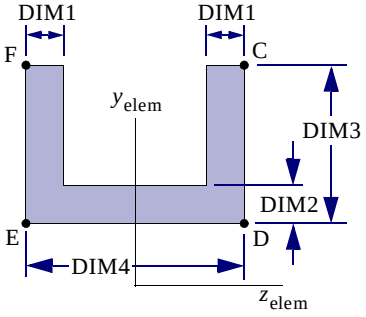
TYPE=”I1”



TYPE=”CHAN1”

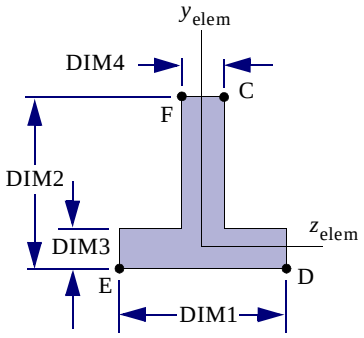


TYPE=”Z”

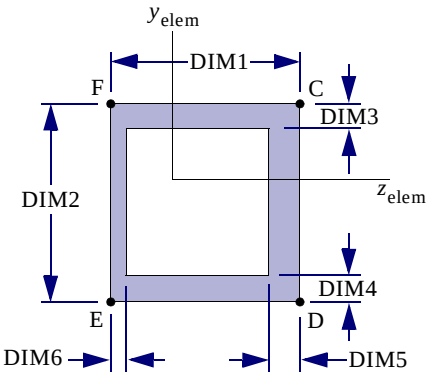


TYPE=”CHAN2”

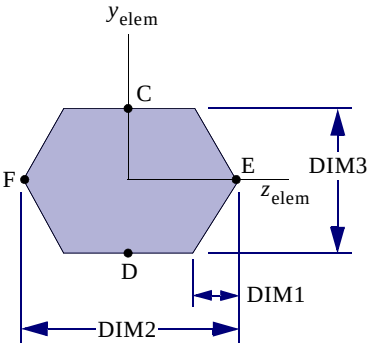
Figure 8-136 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = “MSCBML0" (continued)



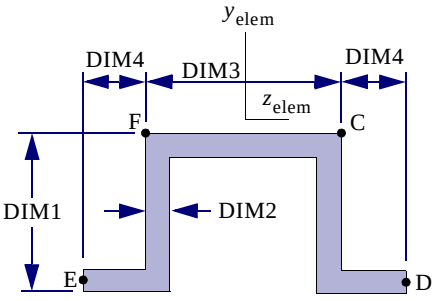
TYPE=”T2”



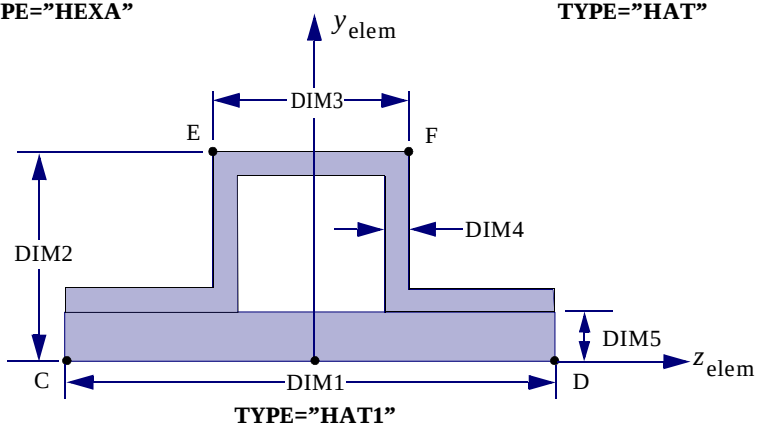
TYPE=”BOX1”



TYPE=”HEXA”



TYPE=”HAT”



TYPE=”HAT1”

Figure 8-136 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = “MSCBML0" (continued)

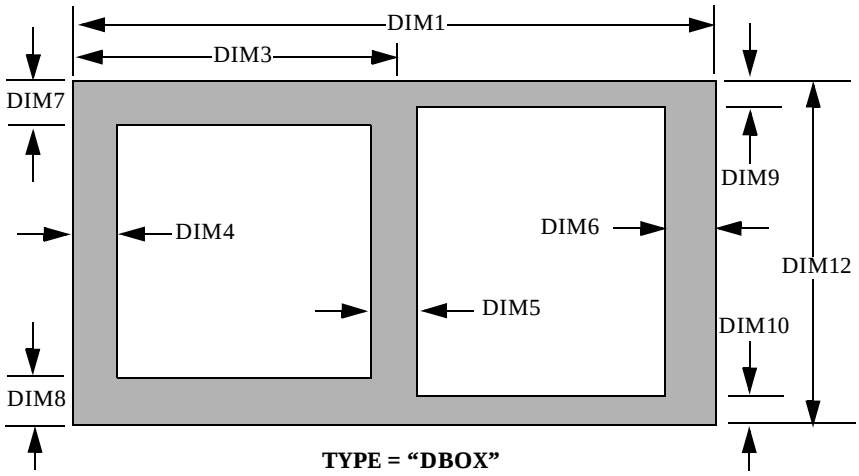


Figure 8-136 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = “MSCBML0" (continued)

PBELTD (SOL 700) Seat Belt Property

Defines section properties for the seat belt elements. These definitions must correspond with the material type selection for the elements.

Format:

1	2	3	4	5	6	7	8	9	10
PBELTD	PID	MID							

Example:

PBELTD	12	21							

Field	Contents
PID	Property ID. PID is referenced on the CROD / CBELT entry and must be unique. (Integer, Required)
MID	Material ID. (Integer, Required)

Remark:

Corresponds to Ls-Dyna input *SECTION_SEATBELT.

PBEND Curved Beam or Pipe Element Property

Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBEND	PID	MID	A	I1	I2	J	RB	THETAB	
	C1	C2	DI	D2	E1	E2	F1	F2	
	K1	K2	NSM	RC	ZC	DELTAN			

Example:

PBEND	39	1	0.8	0.07	0.04	0.04	10.		
	0.5	0.4	-0.5	0.4					
	0.6	0.6				0.1			

Alternate Format and Example for Elbows and Curved Pipes:

PBEND	PID	MID	FSI	RM	T	P	RB	THETAB	
			NSM	RC	ZC				

PBEND	39	1	1	0.5	0.02	10.	10.		
					0.1				

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks **1.** and **2.** (Integer > 0)
A	Area of the beam cross section. (Real > 0.0)
I1, I2	Area moments of inertia in planes 1 and 2. (Real > 0.0)
J	Torsional stiffness. (Real > 0.0)
FSI	Flag selecting the flexibility and stress intensification factors. See Remark **3.** (Integer = 1, 2, or 3)
RM	Mean cross-sectional radius of the curved pipe. (Real > 0.0)
T	Wall thickness of the curved pipe. (Real $\geq$ 0.0; RM + T/2 < RB)

Field	Contents
P	Internal pressure. (Real)
RB	Bend radius of the line of centroids. (Real. Optional, see CBEND entry.)
THETAB	Arc angle of element. (Real, in degrees. Optional, see CBEND entry.)
Ci, Di, Ei, Fi	The r,z locations from the geometric centroid for stress data recovery. See Remark **8.** (Real)
K1, K2	Shear stiffness factor K in K*A*G for plane 1 and plane 2. (Real)
NSM	Nonstructural mass per unit length. (Real)
RC	Radial offset of the geometric centroid from points GA and GB. See **Figure 8-138**. (Real)
ZC	Offset of the geometric centroid in a direction perpendicular to the plane of points GA and GB and vector v. See **Figure 8-138**. See Remark **9.** (Real)
DELTAN	Radial offset of the neutral axis from the geometric centroid, positive is toward the center of curvature. See **Figure 8-138**. See Remark **9.** (Real; Default is described in Remark **5.**)

Remarks:

1. For structural problems, MID must reference a MAT1 material entry.
2. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
3. Flexibility and stress intensification factors are selected by specification of FSI.

FSI = 1 The flexibility factor is set to unity. The stress intensification factor in plane 2 using the general or alternate format is set to unity and for plane 1 it is set to the following:

$$S1 = \frac{I1}{A \cdot RB}\left[\frac{1}{r_{elem}} + \frac{RB - \Delta N}{\Delta N(RB + r_{elem})}\right]$$

where r_{elem} is C1, D1, E1 or F1 of the stress recovery points. If any of these values is zero then $S1 = 1$.

FSI = 2 ASME code Section III, NB-3687.2, NB-3685.2., 1977.

FSI = 3 Empirical factors from the Welding Research Council Bulletin 179, by Dodge and Moore.

4. The transverse shear stiffness in planes 1 and 2 are K1*A*G and K2*A*G, respectively. The default values for K1 and K2 on the first format are zero, which means the transverse shear flexibilities (1/Ki*A*G) are set equal to zero. Transverse shear stiffness for the alternate format are automatically calculated for the curved pipe.
5. The neutral axis radial offset from the geometric centroid is default to the

$$\Delta N = \frac{I1}{A \cdot RB}$$

It is recommended that the default be used whenever

$$\frac{(RB)^2 A}{I1} < 15$$

in which case the default value of ΔN is within 5% of the exact expression for circular or rectangular cross sections. For the alternate format, the neutral axis offset is calculated from an analytical expression for a hollow or solid circular cross section.

The user may compute an exact value for N as follows:

$$\Delta N = \frac{RB}{1 + \frac{(RB)^2 A}{Z}}$$

where

$$Z = \int \frac{r^2 dA}{1 + \frac{r}{RB}}$$

The integration is carried out over the cross section of the element.

6. If T is zero, a solid circular cross section of radius RM is assumed and FSI must be 1.
7. If the first format is used, third-order moments are neglected for the consistent mass matrix. These moments are zero whenever the cross section of the beam is symmetric about both the r and z axes.

8. If the circular cross-sectional property entry format is used, the stress points are automatically located at the points indicated in **Figure 8-137**.
9. Offset vectors are treated like rigid elements and are therefore subject to the same limitations.
   - Offset vectors are not affected by thermal loads.
   - The specification of offset vectors is not recommended in solution sequences that compute differential stiffness because the offset vector remains parallel to its original orientation. (Differential stiffness is computed in buckling analysis provided in SOLs 105 and 200; SOLs 103 and 107 through 112 with STATSUB; and also nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1.)

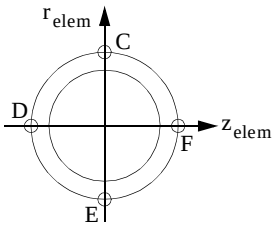


Figure 8-137 PBEND Circular Cross Section

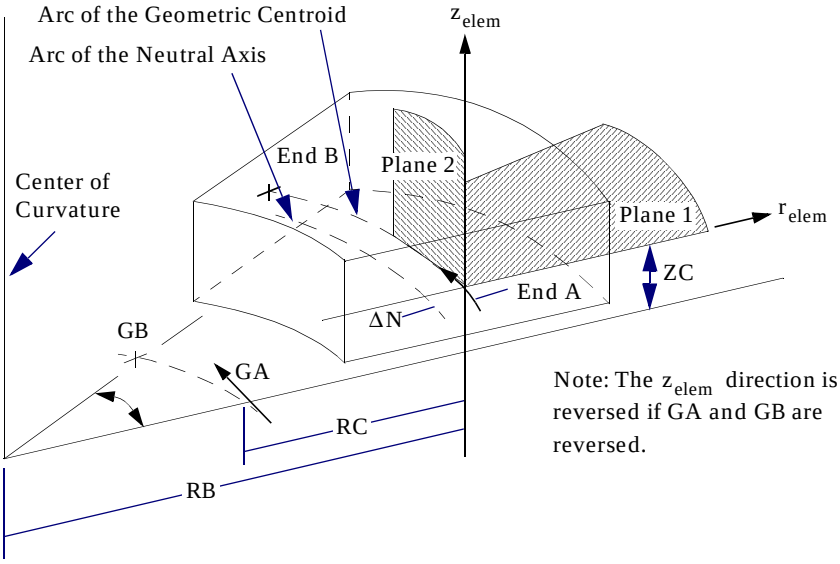


Figure 8-138 PBEND Element Coordinate System

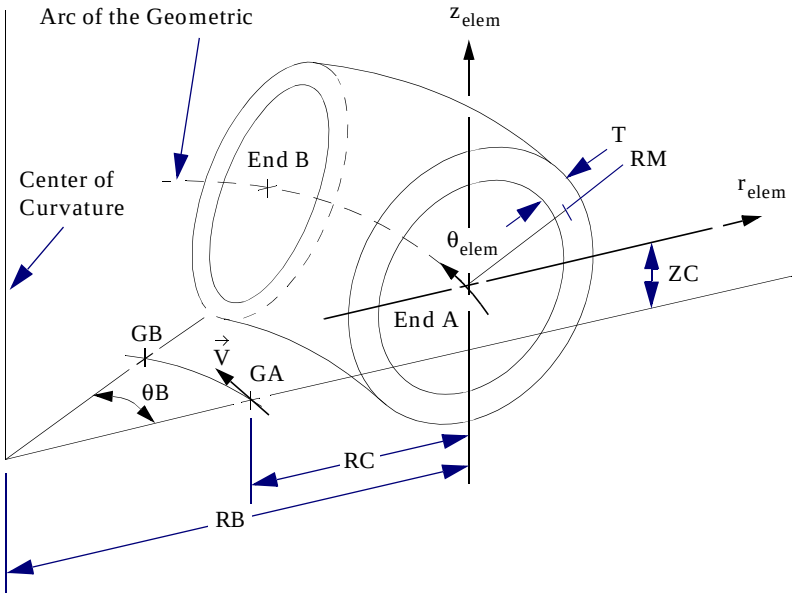


Figure 8-139 PBEND Circular Cross Section Element Coordinate System

PBMSECT Arbitrary Cross-Section for CBEAM

Defines the shape of arbitrary cross-section for CBEAM element.

Format:

1	2	3	4	5	6	7	8	9	10
PBMSECT	PID	MID	FORM						
	Data description for arbitrary section								

Example:

PBMSECT	1	10	GS						
	OUTP=10,INP=20								
PBMSECT	1	10	CP						
	OUTP=10,BRP=20,T=1.0,T(11)=[1.2,PT=(123,204)], NSM=0.01								

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
FORM	Cross-section form. (Character) See Remark 1.

Remarks:

1. Options for FORM are

GS	General Section
OP	Open Profile
CP	Closed Profile

2. Keywords for describing the arbitrary cross-section:

   For GS, OP and CP:

   OUTP = value(Integer > 0); points to ID of a SET1 or SET3 that defines the OUTer Perimeter for FORM=GS or the center line for FORM=CP (or OP) by traversing through all the POINTs in the SET.

For GS only:

INP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that defines a INner Perimeter by traversing through all the POINTs in the SET.

For OP and CP:

BRP(id) = value(Integer > 0) ; points to the ID of a SET1 or SET3 that specifies a BRanch. Note that a branch must be starting from and / or ending to OUTP.

T(id) = [value(Real > 0.0),PT=(pid1,pid2)]; specifies the thickness of a segment in profile. PT=(pid1,pid2) defines the end points of a straight line segment. For CP and OP, it is a requirement to have a T(id) without PT=(pid1,pid2) to serve as a default thickness for all segments which do not have specific thickness associated with them. This requirement is valid even when the thickness for every segment is specified.

(id) = integer (>0) identifies INP, BRP or T which is not required if a single entity appears in the PBMSECT entry. For T, the T(id) can be used to identify the particular thickness to be designed in SOL 200.

3. Stress data recovery points are selected automatically from all points of a PBMSECT with GS form. The points with maximum and / or minimum coordinates in X and Y axes are more likely to be picked. For PBMSECT with CP or OP form, the stress data recovery points are selected from points with computed coordinates that actually encircle the profile. Similar to GS form, the points with extreme coordinates are more likely to be selected.
4. Only the POINT entry ID should be listed under SET1 or SET3 entries which, in turn, are referenced by OUTP, INP and BRP. In addition, the POINT entry for defining an arbitrary beam cross section must have the CP and X3 fields left blank.

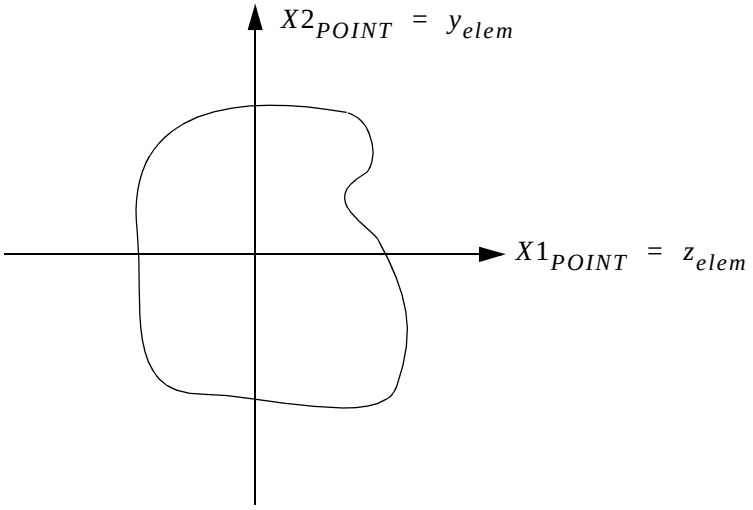


Figure 8-140 Arbitrary Cross-Section Definition

5. Current implementation of PBMSECT supports constant section beam only.

PBRSECT Arbitrary Cross-Section for CBAR

Defines the shape of arbitrary cross-section for CBAR element.

Format:

1	2	3	4	5	6	7	8	9	10
PBRSECT	PID	MID	FORM	NSM					
	Data description for arbitrary section								

Example:

PBRSECT	1	10	GS						
	OUTP=10,INP=20								
PBRSECT	1	10	CP						
	OUTP=10,BRP=20,T=1.0,T(11)=[1.2,PT=(123,204)]								

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
FORM	Cross-section form. (Character) See Remark 1.
NSM	Non-structural mass per unit length. (Real ≥ 0.0; Default = 0.0)

Remarks:

1. Options for FORM are

GS	General Section
OP	Open Profile
CP	Closed Profile

2. Keywords for describing the arbitrary cross-section:

   For GS, OP and CP:

   OUTP = value(Integer > 0); points to ID of a SET1 or SET3 that defines the OUTer Perimeter for FORM=GS or the center line for FORM=CP (or OP) by traversing through all the POINTs in the SET.

For GS only:

INP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that defines a INner Perimeter by traversing through all the POINTs in the SET.

For OP and CP:

BRP(id) = value(Integer > 0) ; points to the ID of a SET1 or SET3 that specifies a BRanch. Note that a branch must be starting from and / or ending to OUTP.

T(id) = [value(real > 0.0),PT=(pid1,pid2)]; specifies the thickness of a segment in profile. PT=(pid1,pid2) defines the end points of a straight line segment. For CP and OP, it is a requirement to have a T(id) without PT=(pid1,pid2) to serve as a default thickness for all segments which do not have specific thickness associated with them. This requirement is valid even when the thickness for every segment is specified.

(id) = integer (>0) identifies INP, BRP or T which is not required if a single entity appears in the PBRSECT entry. For T, the T(id) can be used to identify the particular thickness to be designed in SOL 200.

3. Stress data recovery points are selected automatically from all points of a PBRSECT with GS form. The points with maximum and / or minimum coordinates in X1 and / or X2 axes are more likely to be picked. For PBRSECT with CP or OP form, the stress data recovery points are selected from points with computed coordinates that actually encircle the profile. Similar to GS form, the points with extreme coordinates are more likely to be selected.
4. Only the POINT entry ID should be listed under SET1 or SET3 entries which, in turn, are referenced by OUTP, INP and BRP. In addition, the POINT entry for defining an arbitrary beam cross section must have the CP and X3 fields left blank.

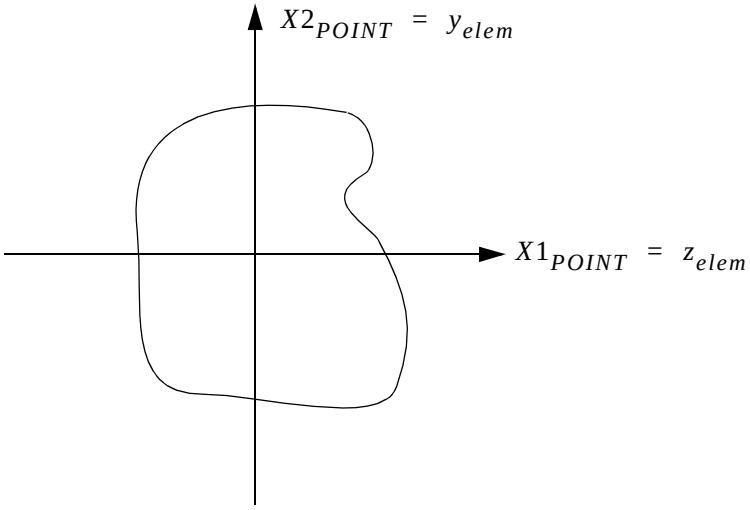


Figure 8-141 Arbitrary Cross-Section Definition

PBUSH Generalized Spring-and-Damper Property

Defines the nominal property values for a generalized spring-and-damper structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH	PID	“K”	K1	K2	K3	K4	K5	K6	
		“B”	B1	B2	B3	B4	B5	B6	
		“GE”	GE1	GE2	GE3	GE4	GE5	GE6	
		“RCV”	SA	ST	EA	ET			

Example 1:

Stiffness and structural damping are specified.

PBUSH	35	K	4.35	2.4				3.1	
		GE	.06						
		RCV	7.3	3.3					

Example 2:

Damping force per unit velocity are specified.

PBUSH	35	B	2.3						

Field	Contents
PID	Property identification number. (Integer > 0)
“K”	Flag indicating that the next 1 to 6 fields are stiffness values in the element coordinate system. (Character)
Ki	Nominal stiffness values in directions 1 through 6. See Remarks **2.** and **3.** (Real; Default = 0.0)
“B”	Flag indicating that the next 1 to 6 fields are force-per-velocity damping. (Character)
Bi	Nominal damping coefficients in direction 1 through 6 in units of force per unit velocity. See Remarks **2.**, **3.**, and **9.** (Real; Default = 0.0)

Field	Contents
“GE”	Flag indicating that the next fields, 1 through 6 are structural damping constants. See Remark 7. (Character)
GEi	Nominal structural damping constant in directions 1 through 6. See Remarks 2. and 3. (Real; Default = 0.0)
“RCV”	Flag indicating that the next 1 to 4 fields are stress or strain coefficients. (Character)
SA	Stress recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ST	Stress recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)
EA	Strain recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ET	Strain recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)

Remarks:

1. Ki, Bi, or GEi may be made frequency dependent for both direct and modal frequency response by use of the PBUSHT entry.
2. The nominal values are used for all analysis types except frequency response. For modal frequency response, the normal modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.
3. If PARAM,W4 is not specified, GEi is ignored in transient analysis.
4. The element stresses are computed by multiplying the stress coefficients with the recovered element forces. $\sigma_i = F_i \cdot SA$ or $\sigma_i = M_i \cdot ST$
5. The element strains are computed by multiplying the strain coefficients with the recovered element displacements. $\varepsilon_i = U_i \cdot EA$ or $\varepsilon_i = \theta_i \cdot ET$
6. The “K”, “B”, “GE”, or “RCV” entries may be specified in any order.
7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
8. Applicable fields refer to directions in the element’s coordinate system.

9. For upward computability, if ONLY GE1 is specified on ALL PBUSH entries and GEi, $i = 2 \rightarrow 6$ are blank on ALL PBUSH entries, then a single structural damping for each PBUSH applied to all defined Ki for each PBUSH is assumed. If ANY PBUSH entry has a GEi, $i = 2 \rightarrow 6$ specified, then the GEi fields are considered variable on ALL PBUSH entries.

PBUSH1D Rod Type Spring-and-Damper Property

Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH1D	PID	K	C	M		SA	SE		
	“SHOCKA”	TYPE	CVT	CVC	EXPVT	EXPVC	IDTS		
			IDETS	IDECS	IDETSD	IDECSD			
	“SPRING”	TYPE	IDT	IDC	IDTDU	IDCDU			
	“DAMPER"	TYPE	IDT	IDC	IDTDV	IDCDV			
	“GENER”		IDT	IDC	IDTDU	IDCDU	IDTDV	IDCDV	

Example:

PBUSH1D	35	3000.	200.	300.					
	SHOCKA	TABLE	2.2	1.2	1.		200		

The continuation entries are optional. The four options, SHOCKA, SPRING, DAMPER, and GENER can appear in any order.

Field	Contents	Default
PID	Property identification number. (Integer > 0).	Required
K	Stiffness. (Real ≥ 0).	See Remark **1.**
C	Viscous damping. (Real ≥ 0).	See Remarks **1.** and **2.**
M	Total mass of the element. (Real ≥ 0).	Blank
SA	Stress recovery coefficient [1 / area]. (Real ≥ 0).	Blank
SE	Strain recovery coefficient [1 / length]. (Real ≥ 0).	Blank

Field	Contents	Default
SHOCKA	Character string specifying that the next 10 fields are coefficients of the following force versus velocity / displacement relationship. (Character). $F(u, v) = C_v \cdot S(u) \cdot \text{sign}(v) \cdot \lvert v \rvert^{\text{EXPV}}$ The force F, the displacement u, and the velocity v, are in the axial direction of the damper. The axis of the damper is defined by the two connecting grid points GA and GB on the CBUSH1D Bulk Data entry. The displacement u and the velocity v, are the relative displacement and the relative velocity with respect to the grid point GA. The scale factor S(u) must be defined with a table or with an equation.	
TYPE	Character string indicating the type of definition. (Character). For TYPE = EQUAT, the fields IDETS, IDECS, IDETSD, and IDECSD are identification numbers of DEQATN entries. For TYPE = TABLE the field IDTS is an identification number of a TABLEDi entry. If no character string is provided (blanks), TYPE = TABLE is set.	TABLE
CVT	Viscous damping coefficient C_V for tension v > 0, force per unit velocity. (Real).	Required for SHOCKA
CVC	Viscous damping coefficient C_V for compression v > 0, force per unit velocity. (Real).	CVT
EXPVT	Exponent of velocity EXPV for tension v > 0. (Real).	1.
EXPVC	Exponent of velocity EXPV for compression v < 0. (Real).	EXPVT
IDTS	Identification number of a TABLEDi entry for tension and compression if TYPE = TABLE. The TABLEDi entry defines the scale factor S, versus displacement u.	Required for SHOCKA and TYPE=TABLE

Field	Contents	Default
IDETS	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for tension u > 0.	Required for SHOCKA and TYPE=EQUAT
IDECS	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for compression u < 0.	IDETS
IDETSD	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for tension u>0.	Required for SHOCKA and TYPE=EQUAT
IDECSD	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for compression u < 0.	IDETSD
SPRING	Character string specifying that the next 5 fields define a nonlinear elastic spring element in terms of a force versus displacement relationship. (Character). $F(u) = F_T(u)$ Tension is u > 0 and compression is u < 0.	
DAMPER	Character string specifying that the next 5 fields define a nonlinear viscous element in terms of a force versus velocity relationship. (Character). $F(v) = F_T(\mathrm{v})$ Tension is v > 0 and compression is v < 0.	

Field	Contents	Default
GENER	Character string specifying that the next 7 fields define a general nonlinear elastic spring and viscous damper element in terms of a force versus displacement and velocity relationship. (Character). For this element, the relationship can only be defined with TYPE=EQUAT. $F(u, v) = F_T(u, v)$ Tension is u > 0 and compression is u < 0. For SPRING, DAMPER, and GENER, the remaining fields are	
TYPE	Character string indicating the type of definition. (Character). For TYPE = EQUAT the following fields are identification numbers of DEQATN entries. For TYPE = TABLE the following field is an identification number of a TABLEDi entry. TYPE is ignored for GENER.	Required for SPRING or DAMPER
IDT	Identification number of a DEQATN entry for tension if TYPE = EQUAT. Identification number of a TABLEDi entry for tension and compression if TYPE = TABLE.	Required for SPRING, DAMPER, and GENER
IDC	Identification number of a DEQATN entry for compression if TYPE = EQUAT. Is ignored for TYPE = TABLE.	IDT
IDTDU	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for tension u > 0. For SPRING and GENER only.	Required if TYPE=EQUAT
IDCDU	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for compression u < 0. For SPRING and GENER only.	IDTDU

Field	Contents	Default
IDTDV	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for tension v > 0. For DAMPER and GENER only.	Required if TYPE=EQUAT
IDCDV	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for compression v < 0. For DAMPER and GENER only.	IDCDT

Remarks:

1. Either the stiffness K or the damping C must be specified.
2. The damping C and mass M are ignored in static solution sequences.
3. The parameters defined on the continuation entries are used in nonlinear solution sequences only.
4. The linear parameters K and C are used in all solution sequences unless parameters on continuation entries are defined and a nonlinear solution sequence is used. Then, the parameters K and C are used for initial values in the first iteration of the first load step and the parameters from continuation entries overwrite the linear parameters thereafter. When SHOCKA, SPRING or GENER are specified, K is overwritten. When SHOCKA, DAMPER or GENER is specified, C is overwritten.
5. PBUSH1D may only be referenced by CBUSH1D elements in the residual structure which do not attach to omitted degrees-of-freedom.
6. The continuation entries SHOCKA, SPRING, DAMPER and GENER may be specified in any order. If more than one continuation entry is defined, then the forces of SHOCKA, SPRING, etc. are added. Multiple continuation entries of the same kind are not allowed, for example, multiple SPRING continuation entries.
7. For TYPE = TABLE, values on the TABLEDi entry are for tension and compression. If table values $f(u)$ are provided only for positive values u > 0, then it is assumed that $f(-u) = -f(u)$.
8. For TYPE = EQUAT, the equations for tension and compression can be different. If the identification numbers for compression are left blank, it is assumed that the equation for tension is also valid for compression.

PBUSH2D 2-D Linear / Nonlinear Connection Properties

Defines linear and nonlinear properties of a two-dimensional element (CBUSH2D entry).

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH2D	PID	K11	K22	B11	B22	M11	M22	PBUSH2D	PID
	‘SQUEEZE’	BDIA	BLEN	BCLR	SOLN	VISCO	PVAPCO		‘SQUEEZE’
	NPORT	PRES1	THETA1	PRES2	THETA2	OFFSET1	OFFSET2		NPORT

Example:

PBUSH2D	1000	50.0	150.0	0.02	0.02				
	“SQUEEZE”	1.0	2.0	0.05	LONG	2.1	300.0		
	2	100.0	30.0	120.0	90.0	0.01	0.0		

Field	Contents
PID	Property identification number (Integer > 0, Required).
K11	Nominal stiffness in T1 rectangular direction (Real, Required).
K22	Nominal stiffness in T2 rectangular direction (Real, Required).
B11	Nominal damping in T1 rectangular direction (Real, Default = 0.0).
B22	Nominal damping in T2 rectangular direction (Real, Default = 0.0).
M11	Nominal acceleration-dependent force in T1 direction (Real, Default = 0.0).
M22	Nominal acceleration-dependent force in T2 direction (Real, Default = 0.0).
‘SQUEEZE’	Indicates that squeeze-film damper will be specified (Character, Required).
BDIA	Inner journal diameter. (Real > 0.0, Required)
BLEN	Damper length. (Real > 0.0, Required).
BCLR	Damper radial clearance (Real > 0.0, Required).
SOLN	Solution option: LONG or SHORT bearing (Character, Default = LONG).

Field	Contents
VISCO	Lubricant viscosity (Real > 0.0, Required).
PVAPCO	Lubricant vapor pressure (Real, Required).
NPORT	Number of lubrication ports: 1 or 2 (Integer, no Default).
PRES1	Boundary pressure for port 1 (Real $\geq$ 0.0, Required if NPORT= 1 or 2).
THETA1	Angular position for port 1 (0.0$\leq$ Real < 360.0, required if NPORT= 1 or 2). See Remark **2.**
PRES2	Boundary pressure for port 2 (Real $\geq$ 0.0, required if NPORT= 2).
THETA2	Angular position for port 2 (0.0$\leq$ Real < 360.0, required if NPORT= 2). See Remark **2.**.
OFFSET1	Offset in the SFD direction 1, see Remark **3.** (Real, Default = 0.0).
OFFSET2	Offset in the SFD direction 2, see Remark **3.** (Real, Default = 0.0)

Remarks:

1. Currently only the ‘SQUEEZE’ option is available.
2. The angular measurement is counterclockwise from the displacement x-axis for the XY plane, the y-axis for the YZ plane, and the z-axis for the XZ plane.
3. Offsets are measured from GB relative to GA. For example, if direction 2 is in the vertical direction and a gravity load is placed on GA, OFFSET2 will be a positive value (GB ‘moves’ toward GA in the positive direction 2).

PBUSHT Frequency Dependent or Nonlinear Force Deflection Spring and Damper Property

Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PBUSHT	PID	“K”	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
		“B”	TBID1	TBID2	TBID3	TBID4	TBID5	TBID6	
		“GE”	TGEID1	TGEID2	TGEID3	TGEID4	TGEID5	TGEID6	
		“KN”	TKNID1	TKIND2	TKNID3	TKIND4	TKIND5	TKIND6	

Example:

PBUSHT	35	K	72						
		B	18						

Field	Contents
PID	Property identification number that matches the identification number on a PBUSH entry. (Integer > 0)
“K”	Flag indicating that the next 1 to 6 fields are stiffness frequency table identification numbers. (Character)
TKIDi	Identification number of a TABLEDi entry that defines the stiffness vs. frequency relationship in directions 1 through 6. (Integer $\geq$ 0; Default = 0)
“B”	Flag indicating that the next 1 to 6 fields are force per velocity frequency table identification numbers. (Character)
TBIDi	Identification number of a TABLEDi entry that defines the force per unit velocity damping vs. frequency relationship in directions 1 through 6 . (Integer $\geq$ 0; Default = 0)
“GE”	Flag indicating that the next field is a structural damping frequency table identification number. (Character)
TGEIDi	Identification number of a TABLEDi entry that defines the non-dimensional structural damping vs. frequency relationship. (Integer $\geq$ 0; Default = 0)

Field	Contents
"KN"	Flag indicating that the next 1 to 6 fields are nonlinear force-deflection table identification numbers. (Character)
TKNIDi	Identification number of a TABLEDi entry that defines the force vs. deflection relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)

Remarks:

1. The "K", "B", and "GE" fields are associated with same entries on the PBUSH entry.
2. PBUSHT may only be referenced by CBUSH elements in the residual structure which do not attach to any omitted degrees-of-freedom.
3. The nominal values are used for all analysis types except frequency response and nonlinear analyses. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency. For nonlinear analysis the nominal values for Ki should agree with the initial slope of the nonlinear force-displacement relationship defined by the PBUSHT, or the results will be unpredictable.
4. The "K", "B", "GE" or "KN" fields may be specified in any order.
5. The PBUSHT entry is ignored in all solution sequences except frequency response or nonlinear analyses.
6. For upward computability, if ONLY TGEID1 is specified on ALL PBUSHT entries and TGEIDi, $i = 2 \rightarrow 6$ are blank on ALL PBUSHT entries, then a single structural damping table for each PBUSHT applied to all defined Ki for each PBUSH is assumed. If ANY PBUSHT entry has a TGEIDi, $i = 2 \rightarrow 6$ specified, then the GEi fields on the PBUSH and the TGEIDi fields on the PBUSHT are considered variable on ALL PBUSH and PBUSHT entries.
7. For nonlinear analysis, only the "KN" field is used.
8. For frequency responses, only the "K", "B" and / or "GE" fields are used.

PCOMP Layered Composite Element Property

Defines the properties of an n-ply composite material laminate.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	MID3	T3	THETA3	SOUT3	-etc.-				

Example of multiple plies per line format:

PCOMP	181	-0.224	7.45	10000.0	HOFF				
	171	0.056	0.	YES			45.		
			-45.				90.		

Example of single ply per line format:

PCOMP	181	-0.224	7.45	10000.	HOFF				
	171	.056	0.	YES					
	171	.056	45.	YES					
	171	.056	-45.	YES					
	171	.056	90.	YES					

Field	Contents
PID	Property identification number. (0 < Integer < 10000000)
Z0	Distance from the reference plane to the bottom surface. See Remark **10.** (Real; Default = -0.5 times the element thickness.)
NSM	Nonstructural mass per unit area. (Real)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0)
FT	Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed) See Remark **7.**

Field	Contents
	"HILL" for the Hill theory. "HOFF" for the Hoffman theory. "TSAI" for the Tsai-Wu theory. "STRN" for the Maximum Strain theory.
TREF	Reference temperature. See Remark 3. (Real; Default = 0.0)
GE	Damping coefficient. See Remarks 4. and 12. (Real; Default = 0.0)
LAM	Laminate Options. (Character or blank, Default = blank). See Remarks 13. and 14.
	"Blank" All plies must be specified and all stiffness terms are developed.
	"SYM" Only plies on one side of the element centerline are specified. The plies are numbered starting with 1 for the bottom layer. If an odd number of plies are desired, the center ply thickness (T1) should be half the actual thickness.
	"MEM" All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed.
	"BEND" All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed.
	"SMEAR" All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set as blanks).
	"SMCORE" All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.
MIDi	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT2, or MAT8 Bulk Data entries. See Remarks 1. and 15. (Integer > 0 or blank, except MID1 must be specified.)
Ti	Thicknesses of the various plies. See Remarks 1. (Real or blank, except T1 must be specified.)

Field	Contents
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)
SOUTi	Stress or strain output request. See Remarks **5.** and **6.** (Character: “YES” or “NO”; Default = “NO”)

Remarks:

1. The default for MID2, ..., MIDn is the last defined MIDi. In the example above, MID1 is the default for MID2, MID3, and MID4. The same logic applies to Ti.
2. At least one of the four values (MIDi, Ti, THETAi, SOUTi) must be present for a ply to exist. The minimum number of plies is one.
3. The TREF specified on the material entries referenced by plies are not used. Instead TREF on the PCOMP entry is used for all plies of the element. If not specified, it defaults to “0.0.”

   If the PCOMP references temperature dependent material properties, then the TREF given on the PCOMP will be used as the temperature to determine material properties.

   TEMPERATURE Case Control commands are ignored for deriving the equivalent PSHELL and MAT2 entries used to describe the composite element.

   If for a nonlinear static analysis the parameter COMPMATT is set to YES, the temperature at the current load step will be used to determine temperature-dependent material properties for the plies and the equivalent PSHELL and MAT2 entries for the composite element. The TREF on the PCOMP entry will be used for the initial thermal strain on the composite element and the stresses on the individual plies. If the parameter EPSILONT is also set to INTEGRAL,TREF is not applicable.
4. GE given on the PCOMP entry will be used for the element and the values supplied on material entries for individual plies are ignored. The user is responsible for supplying the equivalent damping value on the PCOMP entry. If PARAM,W4 is not specified GE is ignored in transient analysis. See “**Parameters**” on page 659.

5. Stress and strain output for individual plies are available in all superelement static and normal modes analysis and requested by the STRESS and STRAIN Case Control commands.
6. If PARAM,NOCOMPS is set to -1, stress and strain output for individual plies will be suppressed and the homogeneous stress and strain output will be printed. See also Remark **10.**
7. In order to get failure index output the following must be present:
   a. ELSTRESS or ELSTRAIN Case Control commands,
   b. SB, FT, and SOUTi on the PCOMP Bulk Data entry,
   c. Xt, Xc, Yt, Yc, and S on all referenced MAT8 Bulk Data entries if stress allowables are used, or Xt, Xc, Yt, S, and STRN=1.0 if strain allowables are used.
8. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. Any sorted echo request will also cause printout and/or punch of the derived PSHELL and MATi entries in User Information Message 4379 and/or the punch file. (See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*, for proper interpretation of the output from User Information Message 4379.) However, if these equivalent PSHELL and MAT2 entries are input, then stress or strain output for individual plies is not available and PARAM,NOCOMPS,-1 must be supplied.

   Use the NASTRAN system cell (361) PRTPCOMP=1 to print equivalent PSHELL/MAT2 Bulk Data entries to the .f06 file. Use the ECHO=PUNCH Case Control command to write them to the .pch file.
9. The failure index for the boundary material is calculated as Failure Index = $(\tau_{1z}, \tau_{2z}) / \mathrm{SB}$.
10. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses are incorrect, while element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry.
11. An unsymmetric layup or the use of Z0 to specify an unsymmetric layup is not recommended in buckling analysis or the calculation of differential stiffness except with CQUAD4 and CTRIA3 elements in SOL 106.
12. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.

13. The SYM option for the LAM option computes the complete stiffness properties while specifying half the plies. The MEM, BEND, SMEAR and SMCORE options provide special purpose stiffness calculations. SMEAR ignores stacking sequence and is intended for cases where this sequence is not yet known, stiffness properties are smeared. SMCORE allows simplified modeling of a sandwich panel with equal face sheets and a central core.
14. Element output for the SMEAR and SMCORE options is produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress / strain results and prints results for the equivalent homogeneous element.
15. Temperature-dependent ply properties only available in SOL 106. See PARAM,COMPMATT for details.

PCOMPA (SOL 700)

Defines additional properties of a multi-ply laminate composite material.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPA	PID	FORM	SHFACT	REF					

Example:

PCOMPA	10	BLT							

Field	Contents
PID	Unique property number referring to a PCOMP property number. (Integer > 0, Required)
FORM	Element formulation (Character, Default = BLT)
	= 1or HUGHES: Hughes-Liu,
	= 2 or BLT: Belytschko-Tsay,
	= 3: BCIZ triangular shell,
	= 4: C0 triangular shell,
	= 5: Belytschko-Tsay membrane,
	= 6: S/R Hughes-Liu,
	= 7: S/R co-rotational Hughes-Liu,
	= 8: Belytschko-Leviathan shell,
	= 9: Fully integrated Belytschko-Tsay membrane,
	= 10: Belytschko-Wong-Chiang,
	= 11: Fast (co-rotational) Hughes-Liu,
	= 16: Fully integrated shell element (very fast),
	= 17: Fully integrated DKT, triangular shell element ,
	= 18: Fully integrated linear DK quadrilateral/triangular shell
	= 20: Fully integrated linear assumed strain C0 shell (See Remarks).
	= 21: Fully integrated linear assumed strain C0 shell (5 DOF).
	= 22: Linear shear panel element (3 DOF per node, see Remarks)

Field	Contents
SHFACT	Shear correction factor. (Real, Default = 0.83333)
REF	Reference surface (Character, Default = MID)
	TOP Reference surface is the top of the surface
	MID Reference surface is the central surface
	BOT Reference surface is the bottom surface

Remarks:

1. For CQUAD4 elements, the default formulation is Belytschko-Tsay. For CTRIA3 elements, the default formulation is C0-TRIA. See also Section 2.15 on application sensitive defaults.
2. If the failure mode is such that fiber and shear strength or matrix and shear strength are lost in all layers, the element is not included in the time-step calculation. If the element fails completely, the element is omitted from the time-step calculations, irrespective of the value entered in this field.

PCOMPG Layered Composite Element Property (Alternate to PCOMP Entry)

Defines global (external) ply IDs and properties for a composite material laminate.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPG	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	GPLYID1	MID1	T1	THETA1	SOUT1				
	GPLYID2	MID2	T2	THETA2	SOUT2				

Example of single ply per line format:

PCOMPG	181	-0.224	7.45	10000.	HOFF				
	1001	171	.056	0.	YES				
	101	171	.07	45.	YES				
	2002	171	.056	-45.	YES				
	102	171	0.55	90.	YES				

Field	Contents
PID	Property identification number. (0 < Integer < 10000000)
Z0	Distance from the reference plane to the bottom surface. See Remark **10.** (Real; Default = -0.5 times the element thickness.)
NSM	Nonstructural mass per unit area. (Real)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0)
FT	Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed) See Remark **7.** “HILL” for the Hill theory. “HOFF” for the Hoffman theory. “TSAI” for the Tsai-Wu theory. “STRN” for the Maximum Strain theory.
TREF	Reference temperature. See Remark **4.** (Real; Default = 0.0)
GE	Damping coefficient. See Remarks **5.** and **12.** (Real; Default = 0.0)
LAM	Laminate Options. (Character or blank, Default = blank). See Remarks **13.** and **14.**

Field	Contents	
	"Blank"	All plies must be specified and all stiffness terms are developed.
	"SYM"	Only plies on one side of the element centerline are specified. The plies are numbered starting with 1 for the bottom layer. If an odd number of plies are desired, the center ply thickness (T1 or TN) should be half the actual thickness.
	"MEM"	All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed.
	"BEND"	All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed.
	"SMEAR"	All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set to zero).
	"SMCORE"	All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.
GPLYIDi	User-defined Global (External) Ply ID. See Remark 1. (Integer > 0)	
MIDi	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT2, or MAT8 Bulk Data entries. See Remarks 2. and 17. (Integer > 0 or blank, except MID1 must be specified.)	
Ti	Thicknesses of the various plies. See Remark 2. (Real or blank, except T1 must be specified.)	
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)	
SOUTi	Stress or strain output request. See Remarks 6. and 7. (Character: "YES" or "NO"; Default = "NO")	

Remarks:

1. The global ply identification number should be unique with respect to other plies in the entry.
2. The default for MID2, ..., MIDn is the last defined MIDi. In the example above, MID1 is the default for MID2, MID3, and MId4. The same logic applies to Ti.
3. The global ply ID (GPLYIDi) and at least one of the four values (MIDi, Ti, THETAi, SOUTi) must be present for a ply to exist. The minimum number of plies is one.
4. The TREF specified on the material entries referenced by plies are not used. Instead TREF on the PCOMPG entry is used for all plies of the element. If not specified, it defaults to “0.0.”

   If the PCOMPG references temperature dependent material properties, then the TREF given on the PCOMPG will be used as the temperature to determine material properties.

   TEMPERATURE Case Control commands are ignored for deriving the equivalent PSHELL and MAT2 entries used to describe the composite element.

   If for a nonlinear static analysis the parameter COMPMATT is set to YES, the temperature at the current load step will be used to determine temperature-dependent material properties for the plies and the equivalent PSHELL and MAT2 entries for the composite element. The TREF on the PCOMPG entry will be used for the initial thermal strain on the composite element and the stresses on the individual plies. If the parameter EPSILONT is also set to INTEGRAL,TREF is not applicable.
5. GE given on the PCOMPG entry will be used for the element and the values supplied on material entries for individual plies are ignored. The user is responsible for supplying the equivalent damping value on the PCOMPG entry. If PARAM,W4 is not specified GE is ignored in transient analysis.
6. Stress and strain output for individual plies are available in all superelement static and normal modes analysis and requested by the STRESS and STRAIN Case Control commands.
7. If PARAM,NOCOMPS is set to -1, stress and strain output for individual plies will be suppressed and the homogeneous stress and strain output will be printed. See also Remark **11.**
8. In order to get failure index output the following must be present:

   a. ELSTRESS or ELSTRAIN Case Control commands,

b. SB, FT, and SOUTi on the PCOMPG Bulk Data entry,

c. Xt, Xc, Yt, Yc, and S on all referenced MAT8 Bulk Data entries if stress allowables are used, or Xt, Xc, Yt, S, and STRN=1.0 if strain allowables are used.

d. -1 - failure in the fiber direction
-2 - failure in the matrix direction
-12 - failure in the inplane shear.

9. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. Any sorted echo request will also cause printout and/or punch of the derived PSHELL and MATi entries in User Information Message 4379 and/or the punch file. (See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide*, for proper interpretation of the output from User Information Message 4379.) However, if these equivalent PSHELL and MAT2 entries are input, then stress or strain output for individual plies is not available and PARAM,NOCOMPS,-1 must be supplied.

   Use the NASTRAN system cell (361) PRTPCOMP=1 to print equivalent PSHELL/MAT2 Bulk Data entries to the .f06 file. Use the ECHO=PUNCH Case Control command to write them to the .pch file.

10. The failure index for the boundary material is calculated as Failure Index = $(\tau_{1z}, \tau_{2z})/\mathrm{SB}$.

11. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses are incorrect, while lamina stresses and element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry.

12. An unsymmetric layup or the use of Z0 to specify an unsymmetric layup, is not recommended in buckling analysis or the calculation of differential stiffness. Also, Z0 should not be used to specify an unsymmetric layup.

13. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.

14. The SYM option for the LAM option computes the complete stiffness properties while specifying half the plies. The MEM, BEND, SMEAR and SMCORE options provide special purpose stiffness calculations. SMEAR ignores stacking sequence and is intended for cases where this sequence is not yet known, stiffness properties are smeared. SMCORE allows simplified modeling of a sandwich panel with equal face sheets and a central core.

15. Element output for the SMEAR and SMCORE options is produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress / strain results and prints results for the equivalent homogeneous element.
16. This property type is not supported in SOL 200 Design Sensitivity and Optimization. May not be rerferenced on DVPRELi and DRESPi entries.
17. Temperature-dependent ply properties only available in SOL 106. See PARAM,COMPMATT for details.

PCONEAX Conical Shell Element Property

Defines the properties of a conical shell element described on a CCONEAX entry.

Format:

1	2	3	4	5	6	7	8	9	10
PCONEAX	ID	MID1	T1	MID2	I	MID3	T2	NSM	
	Z1	Z2	PHIl	PHI2	PHI3	PHI4	PHI5	PHI6	
	PHI7	PHI8	PHI9	PHI10	PHI11	PHI12	PHI13	PHI14	

Example:

PCONEAX	2	4	1.0	6	16.3	8	2.1	0.5	
	0.001	-0.002	23.6	42.9					

Field	Contents
ID	Property identification number. (Unique Integer > 0)
MIDi	Material identification number for membrane, bending, and transverse shear. (Integer $\geq$ 0)
T1	Membrane thickness. (Real > 0.0 if MID1 = 0)
T2	Transverse shear thickness. (Real > 0.0 if MID3 = 0)
I	Moment of inertia per unit width. (Real)
NSM	Nonstructural mass per unit area. (Real)
Z1, Z2	Fiber distances from the middle surface for stress recovery. (Real)
PHIi	Azimuthal coordinates (in degrees) for stress recovery. (Real)

Remarks:

1. PCONEAX is allowed only if an AXIC entry is also present.
2. PCONEAX entries may reference MAT1 or MAT2 material entries. However, only orthotropic material properties are consistent with axisymmetry. Therefore, G13 and G23 values on the MAT2 entry referenced by MID1 or MID2 and the G12 value on the MAT2 entry referenced by MID3 should be set to 0.0. In addition, the MID3 entry, if it references a MAT2 material matrix, should be of size 2 x 2.

3. If either MID1 = 0 or blank or T1 = 0.0 or blank, then both must be zero or blank.
4. If either MID2 = 0 or blank or I = 0.0 or blank, then both must be zero or blank.
5. If either MID3 = 0 or blank or T2 = 0.0 or blank, then both must be zero or blank.
6. A maximum of 14 azimuthal coordinates (PHIi) for stress recovery may be specified.
7. For a discussion of the conical shell problem, see Section 5.3.3 of the *MSC.Nastran Reference Manual.*
8. The following elastic relationships are assumed:

- In-plane forces per unit width

$$\{F\} = T1[G_1]\{\varepsilon\}$$

where $\{\varepsilon\}$ is the vector of strains in the middle surface.

- Bending moments per unit width

$$\{M\} = I[G_2]\{\chi\}$$

where $\{\chi\}$ is the vector of curvatures.

- Transverse shear forces per unit width

$$\{V\} = T2[G_3]\{\gamma\}$$

where $\{\gamma\}$ is the vector of transverse shear strains.

$[G_1]$, $[G_2]$ and $[G_3]$ are the stress-strain matrices defined by MID1, MID2, and MID3, respectively.

PCONV Convection Property Definition

Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
PCONV	PCONID	MID	FORM	EXPF	FTYPE	TID			
	CHLEN	GIDIN	CE	E1	E2	E3			

Examples:

PCONV	53	2	0	.25					

PCONV	4				1	101			

PCONV	38	21			2	54			
	2.0	235	0	1.0	0.0	0.0			

Alternate Format and Examples:

1	2	3	4	5	6	7	8	9	10
PCONV	PCONID	MID	FORM	EXPF	“3”	H1	H2	H3	
	H4	H5	H6	H7	H8				

PCONV	20				3	10.0			

PCONV	7				3	10.32	10.05	10.09	
	10.37								

Field	Contents
PCONID	Convection property identification number. (Integer > 0)
MID	Material property identification number. (Integer > 0)
FORM	Type of formula used for free convection. (Integer 0, 1, 10, 11, 20, or 21; Default = 0)

EXPF	Free convection exponent as implemented within the context of the particular form that is chosen. See Remark **3.** (Real $\geq$ 0.0; Default = 0.0)
FTYPE	Formula type for various configurations of free convection. See Remarks **2.** and **5.** (Integer $\geq$ 0; Default = 0)
TID	Identification number of a TABLEHT entry that specifies the two-variable tabular function of the free convection heat transfer coefficient. See Remark **5.** (Integer $\geq$ 0 or blank)
CHLEN	Characteristic length. See Remarks **6.** and **8.** (Real > 0.0 or blank)
GIDIN	Grid ID of the referenced inlet point. See Remarks **7.** and **8.** (Integer > 0 or blank)
CE	Coordinate system for defining the direction of boundary-layer flow. See Remarks **7.** and **8.** (Integer $\geq$ 0; Default = 0)
Ei	Component of the vector for defining the direction of boundary-layer flow in coordinate system CE. See Remarks **7.** and **8.** (Real or blank)
Hi	Free convection heat transfer coefficient. See Remark **5.** (Real for H1 and Real or blank for H2 through H8; Default for H2 through H8 is H1)

Remarks:

1. Every surface to which free convection is to be applied must reference a PCONV entry. PCONV is referenced on the CONV Bulk Data entry.
2. MID is used to supply the convection heat transfer coefficient (H) for FTYPE=0, or the thermal conductivity (K) for FTYPE=2. MID is ignored for FTYPE=1.
3. EXPF is the free convection temperature exponent.
   - If FORM = 0, 10, or 20, EXPF is an exponent of (T - TAMB), where the convective heat transfer is represented as

$$q = H \cdot u_{\text{CNTRLND}} \cdot (T - \text{TAMB})^{\text{EXPF}} \cdot (T - \text{TAMB}) .$$

   - If FORM = 1, 11, or 21,

$$q = H \cdot u_{\text{CNTRLND}} \cdot (T^{\text{EXPF}} - \text{TAMB}^{\text{EXPF}})$$

   where T represents the elemental grid point temperatures and TAMB is the associated ambient temperature.
4. FORM specifies the formula type and the reference temperature location used in calculating the convection film coefficient if FLMND = 0.

- If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperatures (average).
- If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).
- If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperatures).

5. FTYPE defines the formula type used in computing the convection heat transfer coefficient h.
   - If FTYPE = 0, h is specified in the MAT4 Bulk Data entry referenced by MID.
   - If FTYPE = 1, h is computed from $h = f(T_w, T_a)$, where f is a two-variable tabular function specified in the TABLEHT Bulk Data entry referenced by TID, T_w is the wall temperature, and T_a is the ambient temperature.
   - If FTYPE = 2, h is computed from $Nu = f(T_w, T_a)$, where $Nu_L = hL/K$ or $Nu_x = hX/K$ is the Nusselt number, f is a two-variable tabular function specified in the TABLEHT Bulk Data entry referred by TID, T_w is the wall temperature, and T_a is the ambient temperature.
   - If FTYPE=3, hi is the free convection heat transfer coefficient applied to grid point Gi of the referenced HBDY surface element.
6. CHLEN specifies the characteristic length used to compute the average heat transfer coefficient $\bar{h}$. The following table lists typical values of CHLEN for various convection configurations.

Convection Configuration	Characteristic Length CHLEN
Free convection on a vertical plate or cylinder	Height of the plate or cylinder
Free convection from horizontal tubes	Diameter of the pipes
Free convection from horizontal square plates	Length of a side
Free convection from horizontal rectangular plates	Average length of four sides

Convection Configuration	Characteristic Length CHLEN
Free convection from horizontal circular disks	0.9d, where d is the diameter of the disk.
Free convection from horizontal unsymmetric plates	A / P, where A is the surface area and P is the perimeter of the surface.

7. GIDIN, CE and Ei are used to define the distance from the leading edge of heat transfer. GIDIN specifies the referenced grid ID where heat transfer starts. CE and Ei define the direction of boundary-layer flow. If CE field is blank, the default is CE=0 for basic coordinate system. If E1, E2, and E3 fields are blank, the defaults are Ei = < 1.0, 0.0, 0.0 >, i.e. the flow is in the x direction.
8. CHLEN, GIDIN, CE, and Ei are required only for free convection from flat plates with FTYPE = 2. In this case, if the heat transfer coefficient is spatial dependent, GIDIN must be specified. Otherwise, CHLEN has to be defined for the computation of average heat transfer coefficient $\bar{h}$. For free convection from tubes (CHBDYP elements with TYPE="ELCY", “TUBE” or “FTUBE”), CHLEN, GIDIN, CE, and Ei need not be specified, because MD Nastran will use the average diameter of tubes as the characteristic length while computing Nu. CHLEN, GIDIN, CE, and Ei are ignored for FTYPE $\neq 2$.

PCONVM Forced Convection Property Definition

Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
PCONVM	PCONID	MID	FORM	FLAG	COEF	EXPR	EXPPI	EXPPO	

Example:

PCONVM	3	2	1	1	.023	0.80	0.40	0.30	

Field	Contents
PCONID	Convection property identification number. (Integer > 0)
MID	Material property identification number. (Integer > 0)
FORM	Type of formula used for convection. (Integer = 0, 1, 10, 11, 20, or 21; Default = 0)
FLAG	Flag for mass flow convection. (Integer = 0 or 1; Default = 0)
COEF	Constant coefficient used for forced convection. (Real > 0.0)
EXPR	Reynolds number convection exponent. (Real $\geq$ 0.0; Default = 0.0)
EXPPI	Prandtl number convection exponent for heat transfer into the working fluid. (Real $\geq$ 0.0; Default = 0.0)
EXPPO	Prandtl number convection exponent for heat transfer out of the working fluid. (Real $\geq$ 0.0; Default = 0.0)

Remarks:

1. Every surface to which forced convection is applied must reference a PCONVM entry. PCONVM is referenced on the CONVM entry.
2. MID specifies material properties of the working fluid at the temperature of the point FLMND. FLMND is specified on the CONVM entry.
3. The material properties are used in conjunction with the average diameter and mass flow rate (mdot). MID references the material properties and supplies the fluid conductivity (k), heat capacity (cp), and viscosity (μ) needed to compute the Reynolds (Re) and Prandtl (Pr) numbers as follows:

$$Re = 4 \cdot |mdot| / (\pi \cdot diameter \cdot \mu)$$
$$Pr = cp \cdot \mu / k$$

4. FORM controls the type of formula used in determination of the forced convection film coefficient h. There are two cases:
   - If FORM = 0, 10, or 20 than $h = coef \cdot Re^{EXPR} \cdot Pr^{EXPP}$.
   - If FORM = 1, 11, or 21 then the above h is multiplied by k and divided by the average hydraulic diameter.
   - FORM also specifies the reference temperature used in calculating material properties for the fluid if FLMND = 0.
   - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperature (average).
   - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).
   - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperature).
5. In the above expression, EXPP is EXPPI or EXPPO, respectively, for heat flowing into or out of the working fluid. This determination is performed internally.
6. FLAG controls the convective heat transfer into the downstream point (the second point as identified on the CHBDYi statement is downstream if mdot is positive).
   - FLAG = 0, no convective flow (stationary fluid).
   - FLAG = 1, convective energy flow that is consistent with the Streamwise Upwind Petrov Galerkin (SUPG) element formulation.
7. No phase change or internal heat generation capabilities exist for this element.

PDAMP Scalar Damper Property

Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.

Format:

1	2	3	4	5	6	7	8	9	10
PDAMP	PID1	B1	PID2	B2	PID3	B3	PID4	B4	

Example:

PDAMP	14	2.3	2	6.1					

Field	Contents
PIDi	Property identification number. (Integer > 0)
Bi	Force per unit velocity. (Real)

Remarks:

1. Damping values are defined directly on the CDAMP2 and CDAMP4 entries, and therefore do not require a PDAMP entry.
2. A structural viscous damper, CVISC, may also be used for geometric grid points.
3. Up to four damping properties may be defined on a single entry.
4. For a discussion of scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide.*

PDAMP5 Scalar Damper Property for CDAMP5

Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only.

Format:

1	2	3	4	5	6	7	8	9	10
PDAMP5	PID	MID	B						

Example:

PDAMP5	2	3	4.0						

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number of a MAT4 or MAT5 entry. (Integer > 0)
B	Damping multiplier. (Real > 0.0)

Remark:

1. B is the mass that multiplies the heat capacity CP on the MAT4 or MAT5 entry.

PDAMPT Frequency-Dependent Damper Property

Defines the frequency-dependent properties for a PDAMP Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
PDAMPT	PID1	TBID1							

Example:

PDAMPT	12	34							

Field	Contents
PID	Property identification number that matches the identification number on a PDAMP entry. (Integer > 0)
TBID1	Identification number of a TABLEDi entry that defines the damping force per-unit velocity versus frequency relationship. (Integer $\geq$ 0; Default = 0)

Remarks:

1. PDAMPT may only be referenced by CDAMP1 or CDAMP3 elements in the residual structure, which do not attach to any omitted degrees-of-freedom.
2. The PDAMPT entry is ignored in all solution sequences except frequency response analysis.

PDUMi Dummy Element Property

Defines the properties of a dummy element ($1 \leq i \leq 9$). Referenced by the CDUMi entry.

Format:

1	2	3	4	5	6	7	8	9	10
PDUMi	PID	MID	A1	A2	A3	A4	A5	A6	
	A7	-etc.-							

Example:

PDUM3	108	2	2.4	9.6	1.E4	15.		3.5	
	5		2						

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
Aj	Additional fields. (Real or Integer)

Remark:

1. The additional fields are defined in the user-written element subroutines.

PELAS1 (SOL 700)

Defines a spring property designated by a force-deflection curve for SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
PELAS1	PID	TID							

Example:

PELAS1	22	33							

Field	Contents
PID	Property identification number. (Integer, No Default, > 0)
TID	Identification number of a TABLED1 entry which defines the force deflection curve. (Integer, no Default, > 0)

Remarks:

1. Unlike PELAST, when PELAS1 is used, no PELAS entry is made.
2. All PELAS and PELAS1 ID's must be unique.
3. This entry may only be referenced by a CELAS1D entry.

PELAS Scalar Elastic Property

Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).

Format:

1	2	3	4	5	6	7	8	9	10
PELAS	PID1	K1	GE1	S1	PID2	K2	GE2	S2	

Example:

PELAS	7	4.29	0.06	7.92	27	2.17	0.0032		

Field	Contents
PIDi	Property identification number. (Integer > 0)
Ki	Elastic property value. (Real)
GEi	Damping coefficient, g_e. See Remarks **5.** and **6.** (Real)
Si	Stress coefficient. (Real)

Remarks:

1. Be careful using negative spring values.
2. Spring values are defined directly on the CELAS2 and CELAS4 entries, and therefore do not require a PELAS entry.
3. One or two elastic spring properties may be defined on a single entry.
4. For a discussion of scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide.*
5. If PARAM,W4 is not specified, GEi is ignored in transient analysis. See "**Parameters**" on page 659.
6. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0 by 2.0.
7. If PELAS is used in conjunction with PELAST, Ki > 0, and the initial slope of the nonlinear force-displacement relationship defined by the PELAST should agree with Ki.

PELAST Frequency Dependent Elastic Property

Defines the frequency dependent properties for a PELAS Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
PELAST	PID	TKID	TGEID	TKNID					

Example:

PELAST	44	38							

Field	Contents
PID	Property identification number that matches the identification number on a PELAS entry. (Integer > 0)
TKID	Identification number of a TABLEDi entry that defines the force per unit displacement vs. frequency relationship. (Integer > 0; Default = 0)
TGEID	Identification number of a TABLEDi entry that defines the nondimensional structural damping coefficient vs. frequency relationship. (Integer > 0; Default = 0)
TKNID	Identification number of a TABELDi entry that defines the nonlinear force vs. displacement relationship. (Integer > 0; Default = 0)

Remarks:

1. The PELAST entry may only be referenced by CELAS1 or CELAS3 elements in the residual structure which do not attach to any omitted degrees-of-freedom.
2. For frequency dependent modal frequency response the modes are computed using the nominal Ki values as specified on the PELAS entry.
3. The nominal values are used for all analysis types except frequency response and nonlinear analyses. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency. For nonlinear analysis the nominal values for Ki should agree with the initial slope of the nonlinear force-displacement relationship defined by the PBUSHT, or the results will be unpredictable.

4. The following table summarizes the usage PELAST entry in various solution sequences.

Field	Frequency Response	Nonlinear	Linear (Non-Frequency Response)
TKID	Used	Ignored	Ignored
TGEID	Used	Ignored	Ignored
TKNID	Ignored	Used	Ignored

5. The PELAST entry is ignored in all solution sequences except frequency response or nonlinear analyses.

PFAST CFAST Fastener Property

Defines the CFAST fastener property values.

Format:

1	2	3	4	5	6	7	8	9	10
PFAST	PID	D	MCID	MFLAG	KT1	KT2	KT3	KR1	
	KR2	KR3	MASS	GE					

Example:

PFAST	7	1.1	70		100000.	46000.	12300.		

Field	Contents
PID	Property identification number. (Integer > 0)
D	Diameter of the fastener. See Remark **2.** (Real > 0)
MCID	Specifies the element stiffness coordinate system. See Remark **1.** (Integer $\geq$ -1 or blank, Default = -1)
MFLAG	Defines if the coordinate system defined by MCID is absolute or relative. See Remark **1.** (Integer 0 or 1, Default = 0) If MFLAG = 0, MCID defines a relative coordinate system. See Remark 1**a.** If MFLAG = 1, MCID defines an absolute coordinate system. See Remark 1**c.**
KTi	Stiffness values in directions 1 through 3. (Real)
KRi	Rotational stiffness values in directions 1 through 3. (Real, Default = 0.0)
MASS	Lumped mass of fastener. (Real, Default = 0.0)
GE	Structural damping. (Real, Default = 0.0)

Remarks:

1.

a. If MCID > 0 and MFLAG = 0 (default), then the KT1 stiffness will be applied along the x_{elem} axis direction of the fastener defined as

$$\vec{e}_1 = \frac{\vec{x}_B - \vec{x}_A}{\left\| \vec{x}_B - \vec{x}_A \right\|}$$

The T2 direction defined by MCID will be used to define the orientation vector $\vec{v}$ of the fastener. Then the element z_{elem} axis will be defined as

$$\vec{e}_3 = \frac{\vec{e}_1 \times \vec{v}}{\left\| \vec{e}_1 \times \vec{v} \right\|}$$

The KT3 stiffness will lie along the z_{elem} axis. The element y_{elem} axis is defined as

$$\vec{e}_2 = \vec{e}_3 \times \vec{e}_1$$

The KT2 stiffness will lie along the y_{elem} axis

This option allows the user to define orthotropic material properties normal to the axis of the fastener that will “slide” with the curve of the patches.

b. If MICD = -1, MFLAG is ignored, and the following element system is defined: the x_{elem} axis direction of the fastener defined as

$$\vec{e}_1 = \frac{\vec{x}_B - \vec{x}_A}{\left\| \vec{x}_B - \vec{x}_A \right\|}$$

Relative to the basic system, find the smallest component j of the element x_{elem} axis unit vector. If two such components are equal, take the first one. Form a unit vector in the basic system. For example, assuming the $j = 3$ component of $\vec{e}_1$ was the smallest.

$$b_j = b_3 = \left\{ \begin{array}{c} 0 \\ 0 \\ 1 \end{array} \right\}$$

Form the following orthogonal vector:

$$\hat{e}_2 = \vec{b}_j - \frac{\vec{e}_1 \cdot \vec{b}_j}{\vec{e}_1 \cdot \vec{e}_1} \vec{e}_1$$

$$\vec{e}_2 = \frac{\hat{e}_2}{\left\| \hat{e}_2 \right\|}$$

Form $\vec{e}_3$ as

$$\vec{e}_3 = \vec{e}_1 \times \vec{e}_2$$

c. If MCID $\geq$ 0 and MFLAG = 1, then the material system directions will be used to compute stiffness. KT1 will be applied along the material T1 axis, KT2 along the material T2 axis, and KT3 along the material T3 axis. The element forces will be computed in the coordinate system defined in Remark 1b.

d. If the length of GA - GB is zero, then the element x_{elem} axis is defined to lie along the projected normal to patch A.

2. The diameter D is used along with the piercing points of GA and GB to determine the location of fictitious grid points to form a fictitious hexa volume that determines the elements and physical grids used for the fastener element. Four points are positioned at $\pm a$ positions parallel to the element axis where $a = f(D)$. The stiffness contribution of the fastener depends on both the stiffness values specified and the diameter D. It is a function of D, because the $\pm a$ positions are used along with the surface shape functions of the fictitious hexa to weight the contribution of the physical grids used to the grids GA and GB of the fastener element.

3. The CFAST element (see **Figure 8-142**), for stiffness and structural damping calculations, is designed to satisfy rigid body equilibrium requirements. When $\vec{x}_B - \vec{x}_A$ has finite length, internal rigid links connect grids GA and GB. This may result in coupling between translational and rotational degrees-of freedom even when no rotational stiffness (KR1-KR3) are specified.

   For mass calculations, half the specified mass value is placed directly onto the projected grid A and grid B translational degrees-of-freedom.

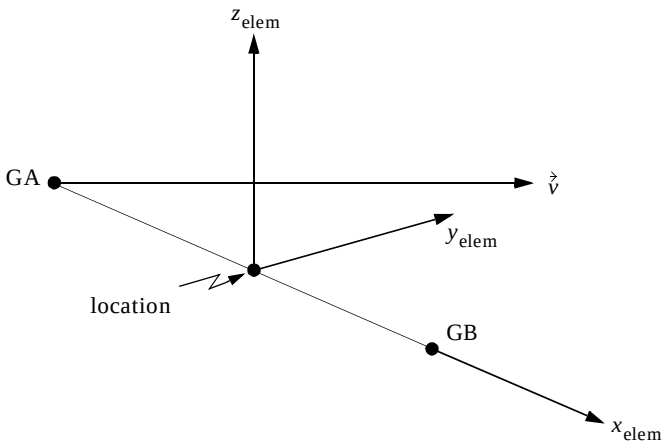


Figure 8-142 CFAST Element

4. The CFAST element lies midway between GA and GB.

5. Values for KTi and KRi are specified at the user's discretion. Assuming a short stubby beam where shear is dominate, possible values might be:

$$KT1 = \frac{EA}{L}$$

$$KT2 = \frac{G_2 A_s}{L}$$

$$KT3 = \frac{G_3 A_s}{L}$$

$$KR1 = \frac{GJ}{L}$$

$$KR2 = \frac{EI}{L} + \frac{G_2 A_s L}{3}$$

$$KR3 = \frac{EI}{L} + \frac{G_3 A_s L}{3}$$

where

$$A = \pi D^2 / 4$$

$$I = \tau D^4 / 64$$

$$J = \pi D^4 / 32$$

$$L = \left| \vec{x}_B - \vec{x}_A \right|$$

$$A_s = A_s = A / \alpha_s$$

$$\alpha_s = 4/3$$

E, G_2, G_3, and G are the material properties of the fastener.

The fastener stiffness is not, however, independent of the surrounding structure. The values of stiffness specified should not overwhelm the stiffness of the local structure or max ratio's will occur. One possible way to estimate the local stiffness S is by the relationship.

$$S = \frac{t_p E_p E}{E_p + E}$$

where t_p is a shell thickness and E_p is the modulus of the shell.

6. The element force and strain are computed as follows:

$$\{f_e\} = [K_e]\{u_e\} \text{ for statics}$$

$$\{f_e\} = ([K_e] + i(g + g_e)[K_e])(\{u_e\}_{\text{real}} + i\{u_e\}_{\text{imag}}) \text{ for frequency}$$

$$\{f_e\} = [K_e]\{u_e\} + \left(\frac{g}{w3} + \frac{g_e}{w4}\right)[K_e]\{\upsilon_e\} \text{ for transient}$$

where $[K_e]$ is the 6 x 6 element stiffness matrix, $\{u_e\} = \{u_b\} - \{u_a\}$ relative displacement in the element coordinate system, and $\{\upsilon_e\} = \{\upsilon_b\} - \{\upsilon_a\}$ relative velocity in the element coordinate system. The subscripts *a* and *b* stand for end A and end B of the fastener. g is defined by param,g; $w3$ is defined by param,w3, $w4$ is defined by param,w4; and g_e is the GE entry of the PFAST. $\{u_e\}$ is the strain output. Stress output is the same as force output.

PGAP Gap Element Property

Defines the properties of the gap element (CGAP entry).

Format:

1	2	3	4	5	6	7	8	9	10
PGAP	PID	U0	F0	KA	KB	KT	MU1	MU2	
	TMAX	MAR	TRMIN						

Example:

PGAP	2	.025	2.5	1.E6		1.E6	0.25	0.25	

Field	Contents
PID	Property identification number. (Integer > 0)
U0	Initial gap opening. See **Figure 8-144**. (Real; Default = 0.0)
F0	Preload. See **Figure 8-144**. (Real $\geq$ 0.0; Default = 0.0)
KA	Axial stiffness for the closed gap; i.e., $U_a - U_b > U0$. See **Figure 8-144**. (Real > 0.0)
KB	Axial stiffness for the open gap; i.e., $U_a - U_b < U0$. See **Figure 8-144**. See Remark **2.** (Real $\geq$ 0.0; Default = $10^{-14} \cdot KA$)
KT	Transverse stiffness when the gap is closed. See **Figure 8-145**. It is recommended that KT $\geq (0.1 \cdot KA)$. (Real $\geq$ 0.0; Default = MU1 $\cdot$ KA)
MU1	Coefficient of static friction (μ_s) for the adaptive gap element or coefficient of friction in the y transverse direction (μ_y) for the nonadaptive gap element. See Remark **3.** and **Figure 8-145**. (Real $\geq$ 0.0; Default = 0.0)
MU2	Coefficient of kinetic friction (μ_k) for the adaptive gap element or coefficient of friction in the z transverse direction (μ_z) for the nonadaptive gap element. See Remark **3.** and **Figure 8-145**. (Real $\geq$ 0.0 for the adaptive gap element, MU2 $\leq$ MU1; Default = MU1)

Field	Contents
TMAX	Maximum allowable penetration used in the adjustment of penalty values. The positive value activates the penalty value adjustment. See Remark 4. (Real; Default = 0.0)
MAR	Maximum allowable adjustment ratio for adaptive penalty values KA and KT. See Remark 5. ($1.0 < \text{Real} < 10^6$; Default = 100.0)
TRMIN	Fraction of TMAX defining the lower bound for the allowable penetration. See Remark 6. ($0.0 \le \text{Real} \le 1.0$; Default = 0.001)

Remarks:

1. Figure 8-143, Figure 8-144, and Figure 8-145 show the gap element and the force-displacement curves used in the stiffness and force computations for the element.
2. For most contact problems, KA (penalty value) should be chosen to be three orders of magnitude higher than the stiffness of the neighboring grid points. A much larger KA value may slow convergence or cause divergence, while a much smaller KA value may result in inaccurate results. The value is adjusted as necessary if TMAX > 0.0.
3. When the gap is open, there is no transverse stiffness. When the gap is closed and there is friction, the gap has the elastic stiffness (KT) in the transverse direction until the friction force is exceeded and slippage starts to occur.
4. There are two kinds of gap elements: adaptive gap and nonadaptive gap. If TMAX $\ge$ 0.0, the adaptive gap element is selected by the program. When TMAX = 0.0, penalty values will not be adjusted, but other adaptive features will be active (i.e., the gap-induced stiffness update, gap-induced bisection, and subincremental process). The value of TMAX = -1.0 selects the nonadaptive (old) gap element. The recommended allowable penetration TMAX is about 10% of the element thickness for plates or the equivalent thickness for other elements that are connected to the gap.
5. The maximum adjustment ratio MAR is used only for the adaptive gap element. Upper and lower bounds of the adjusted penalty are defined by

$$\frac{K^{init}}{\text{MAR}} \le K \le K^{init} \cdot \text{MAR}$$

where K^{init} is either KA or KT.

6. TRMIN is used only for the penalty value adjustment in the adaptive gap element. The lower bound for the allowable penetration is computed by TRMIN · TRMAX. The penalty values are decreased if the penetration is below the lower bound.

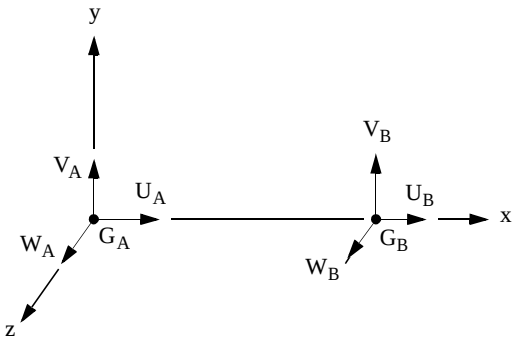


Figure 8-143 The CGAP Element Coordinate System

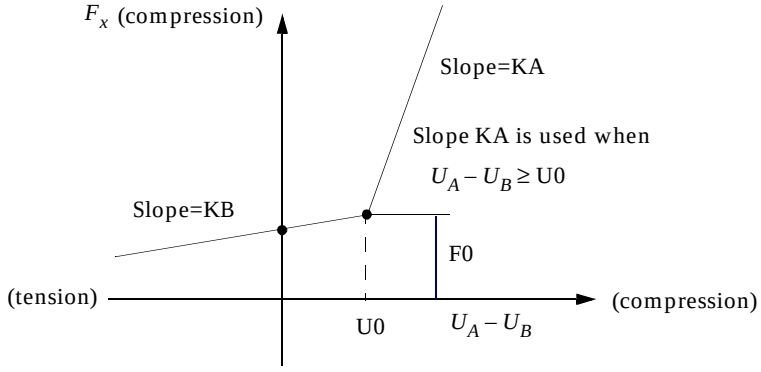


Figure 8-144 CGAP Element Force-Deflection Curve for Nonlinear Analysis

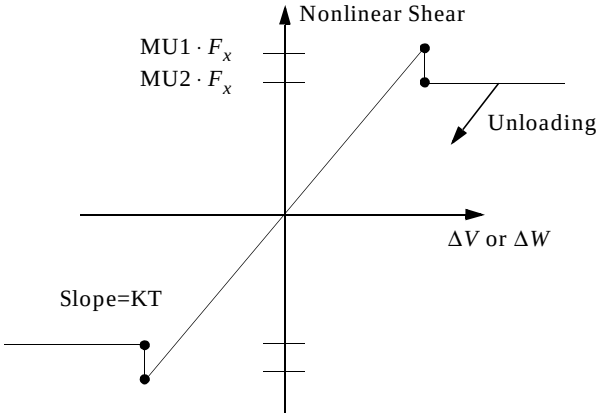


Figure 8-145 Shear Force for CGAP Element

7. If U0 is specified negative and GA and GB are not coincident, then the direction for closing must be controlled by the use of the CID field on the CGAP entry.

PHBDY CHBDYP Geometric Element Definition

A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements.

Format:

1	2	3	4	5	6	7	8	9	10
PHBDY	PID	AF	D1	D2					

Example:

PHBDY	2	.02	1.0	1.0					

Field	Contents
PID	Property identification number. (Unique Integer among all PHBDY entries). (Integer > 0)
AF	Area factor of the surface used only for CHBDYP element TYPE = “POINT”, TYPE = “LINE”, TYPE = “TUBE”, or TYPE = “ELCYL”. For TYPE = “TUBE”, AF is the constant thickness of the hollow tube. (Real > 0.0 or blank)
D1, D2	Diameters associated with the surface. Used with CHBDYP element TYPE = “ELCYL”, “TUBE”, and “FTUBE”. (Real > 0.0 or blank; Default for D2 = D1)

Remarks:

1. The PHBDY entry is used with CHBDYP entries.
2. AF
   - For TYPE = “POINT” surfaces, AF is the area.
   - For TYPE = “LINE” or TYPE = “ELCYL” surfaces, AF is the effective width: area = $AF \cdot (\text{length})$.
   - For TYPE = “FTUBE” and outer TYPE = “TUBE” surfaces

$$\text{area} = \pi \cdot \left(\frac{D1 + D2}{2}\right) \cdot \sqrt{(LGTH)^2 + \left(\frac{D1 - D2}{2}\right)^2}$$

3. D1 and D2 are used only with TYPE = “ELCYL”, TYPE = “TUBE”, and TYPE = “FTUBE” surfaces.

- For TYPE = “ELCYL” surfaces, D1 and D2 are the two diameters associated with the ellipse.
- For TYPE = “FTUBE” and outer TYPE = “TUBE” surfaces, D1 and D2 are the diameters associated with the first and second grid points, respectively.
- For inner TYPE = “TUBE” surfaces, the diameters are reduced by twice the thickness $(2 \cdot AF)$.

PINTC Properties of Geometric Interface -- Curve

Defines the properties for interface elements along curve interfaces between boundaries of multiple subdomains of p-elements.

Format:

1	2	3	4	5	6	7	8	9	10
PINTC	PID	TOL	DSCALE						

Example:

PINTC	1	0.01	1000.0						

Field	Contents
PID	Property identification number. (Integer > 0)
TOL	Tolerance for distance between interface elements and subdomain boundaries. See Remark **2.** (Real > 0.0; Default = 0.01)
DSCALE	Scaling parameter for Lagrange multiplier functions. See Remark **3.** (Real > 0.0; Default = 1000.0)

Remarks:

1. All PIDs must be unique.
2. TOL may be specified for the distance between the interface element and the boundaries. If the distance is greater than TOL, a warning message will occur. If the distance is less than TOL, but greater than the tolerance used by the geometric evaluator for the GMCURV method, a warning will be issued from the geometric evaluator.
3. DSCALE does not need to be specified unless the interface elements are poorly conditioned. Poor DSCALE conditioning can be determined from the epsilon value of the linear equation solution. A good value for DSCALE is two or three orders of magnitude less than the elastic moduli of the subdomain boundaries.

PINTS Properties of Geometric Interface -- Surface

Defines the properties for interface elements along surface interfaces between boundaries of multiple subdomains of p-elements.

Format:

1	2	3	4	5	6	7	8	9	10
PINTS	PID	TOL	DSCALE						

Example:

PINTS	1	0.01	1000.						

Field	Contents	Type	Default
PID	Property identification number	Integer > 0	Required
TOL	Tolerance for distance between interface element and subdomain boundaries.	Real > 0	0.01
DSCALE	Scaling parameter for Lagrange multiplier functions.	Real > 0	1000.

Remarks:

1. All PIDs must be unique.
2. The TOL tolerance may be specified for the distance between the interface element and the subdomain boundaries. If the distance is greater than the TOL, a warning will be issued. If the distance is less than the TOL, but greater than the tolerance used by the geometric evaluator for the GMSURF, a warning from the geometric evaluator will be issued.
3. The DSCALE scaling parameter for the Lagrange multipliers does not need to be changed unless the interface elements are poorly conditioned. This could be determined from the epsilon value of the linear equation solution. A good value for DSCALE, which has the units of elastic modulus, is two or three orders of magnitude less than the elastic modulus of the subdomain boundaries.

PLOAD Static Pressure Load

Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD	SID	P	G1	G2	G3	G4			

Example:

PLOAD	1	-4.0	16	32	11				

Field	Contents
SID	Load set identification number. (Integer > 0)
P	Pressure. (Real)
Gi	Grid point identification numbers. (Integer > 0; G4 may be zero or blank.)

Remarks:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. The grid points define either a triangular or a quadrilateral surface to which a pressure is applied. If G4 is blank, the surface is triangular.
3. In the case of a triangular surface, the assumed direction of the pressure is computed according to the right-hand rule using the sequence of grid points G1, G2, G3 illustrated in **Figure 8-146**.

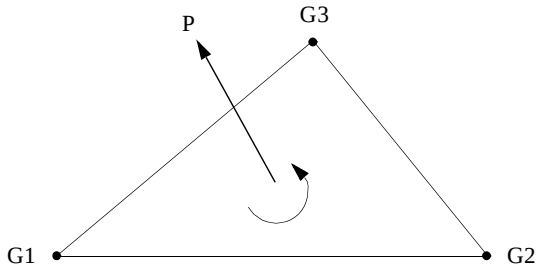


Figure 8-146 Pressure Convention for Triangular Surface of Surface Elements and/or the Faces of Solid Elements

The total load on the surface (see **Figure 8-147**), AP, is divided into three equal parts and applied to the grid points as concentrated loads. A minus sign in field 3 reverses the direction of the load.

4. In the case of a quadrilateral surface, the grid points G1, G2, G3, and G4 should form a consecutive sequence around the perimeter. The right-hand rule is applied to find the assumed direction of the pressure. Four concentrated loads are applied to the grid points in approximately the same manner as for a triangular surface. The following specific procedures are adopted to accommodate irregular and/or warped surfaces:

   - The surface is divided into two sets of overlapping triangular surfaces. Each triangular surface is bounded by two of the sides and one of the diagonals of the quadrilateral.
   - One-half of the pressure is applied to each triangle, which is then treated in the manner described in Remark **2.**

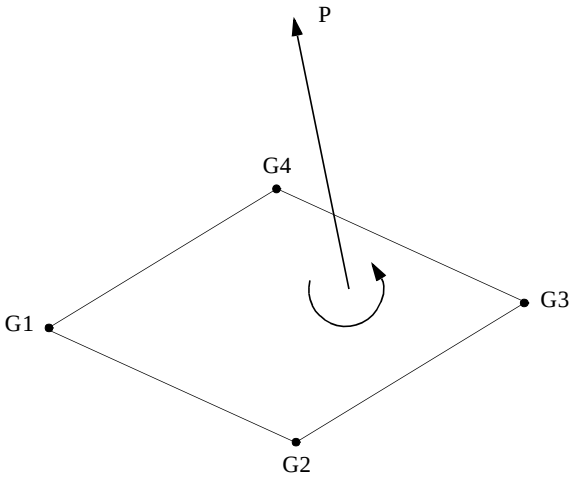


Figure 8-147 Pressure Convention for Quadrilateral Surface of Surface Elements and/or the Faces of Solid Elements

5. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "**FOLLOWK**" on page 710). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

PLOAD1 Applied Load on CBAR, CBEAM or CBEND Elements

Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	

Example:

PLOAD1	25	1065	MY	FRPR	0.2	2.5E3	0.8	3.5E3	

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	CBAR, CBEAM, or CBEND element identification number. (Integer > 0)
TYPE	Load type. (Character: “FX”, “FY”, “FZ”, “FXE”, “FYE”, “FZE”, “MX”, “MY”, “MZ”, “MXE”, “MYE”, “MZE”)
SCALE	Determines scale factor for X1, X2. (Character: “LE”, “FR”, “LEPR”, “FRPR”)
X1, X2	Distances along the CBAR, CBEAM, or CBEND element axis from end A. (Real; X2 may be blank; $0 \leq X1 \leq X2$)
P1, P2	Load factors at positions X1, X2. (Real or blank)

Remarks:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. If $X2 \neq X1$, a linearly varying distributed load will be applied to the element between positions X1 and X2, having an intensity per unit length of bar equal to P1 at X1 and equal to P2 at X2, except as noted in Remarks **8.** and **9.**
3. If X2 is blank or equal to X1, a concentrated load of value P1 will be applied at position X1.

4. If P1 = P2 and X2 ≠ X1, a uniform distributed load of intensity per unit length equal to P1 will be applied between positions X1 and X2 except as noted in Remarks **8.** and **9.**
5. Load TYPE is used as follows to define loads:

   “FX”, “FY” or “FZ”: Force in the x, y, or z direction of the basic coordinate system.

   “MX”, “MY” or “MZ”: Moment in the x, y, or z direction of the basic coordinate system.

   “FXE”, “FYE” or “FZE”: Force in the x, y, or z direction of the element’s coordinate system.

   “MXE”, “MYE” or “MZE”: Moment in the x, y, or z direction of the element’s coordinate system.
6. If SCALE = “LE” (length), the xi values are actual distances along the element axis, and, if $X1 \neq X2$, then Pi are load intensities per unit length of the element.
7. If SCALE = “FR” (fractional), the xi values are ratios of the distance along the axis to the total length, and (if $X2 \neq X1$) Pi are load intensities per unit length of the element.
8. If SCALE = “LEPR” (length projected), the xi values are actual distances along the element axis, and (if $X2 \neq X1$) the distributed load is input in terms of the projected length of the element.

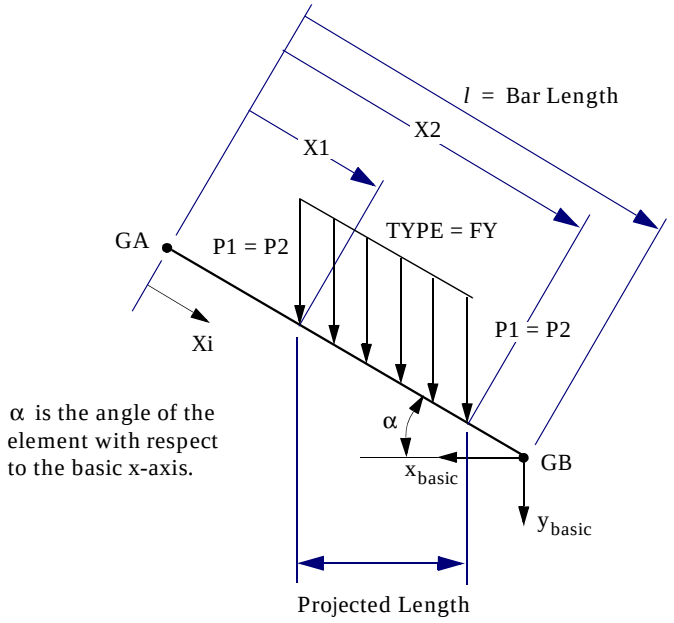


Figure 8-148 PLOAD1 Convention on Beam or Bar Elements

If SCALE = “LE”, the total load applied to the bar is P1 $(X2 - X1)$ in the y-basic direction.

If SCALE = “LEPR”, the total load applied to the bar is P1 $(X2 - X1)\cos\alpha$ in the y-basic direction.

9. If SCALE = “FRPR” (fractional projected), the Xi values are ratios of the actual distance to the length of the bar (CBAR entry), and if $X1 \neq X2$, then the distributed load is specified in terms of the projected length of the bar.
10. Element identification numbers for CBAR, CBEAM, and CBEND entries must be unique.
11. For the CBEND element, the following coordinate equivalences must be made for the element coordinates

$$R_{elem} \equiv X_{elem}$$
$$\theta_{elem} \equiv Y_{elem}$$

12. Only distributed loads applied over the entire length of the CBEND element may be applied.
13. Projected loads are not applicable to the CBEND element.

14. Loads on CBEAM elements defined with PLOAD1 entries are applied along the line of the shear centers.
15. If a CBARAO or PLOAD1 entry is specified and stress and / or force output is requested, then the stresses and / or forces will be calculated at each location Xi and output as a separate line. The force and stress locations Xi=0 and Xi = *l* will always be output. This output format will be used for all beam and bar elements.
16. If on the TYPE field of the PLOAD1 entry, the element coordinate system direction (e.g. TYPE = FYE) option is selected, then the projection (i.e. SCALE = FRPR or LEPR) option is ignored and the result is the same as the SCALE = FR (or LE) option.

PLOAD2 Uniform Normal Pressure Load on a Surface Element

Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	

Example:

PLOAD2	21	-3.6		4	16		2		

Alternate Format and Example:

PLOAD2	SID	P	EID1	“THRU”	EID2				
PLOAD2	1	30.4	16	THRU	48				

Field	Contents
SID	Load set identification number. (Integer > 0)
P	Pressure value. (Real)
EIDi	Element identification number. (Integer $\geq$ 0 or blank; for the “THRU” option, EID1 < EID2.)

Remarks:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. At least one positive EID must be present on each PLOAD2 entry.
3. If the alternate form is used, all elements EID1 through EID2 must be two-dimensional.
4. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element entry. Refer to the PLOAD entry.

5. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.
6. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "**FOLLOWK**" on page 710). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
7. The PLOAD2 entry may not be applied to p-elements. The PLOAD4 must be used.

PLOAD4 Pressure Load on Surface and Faces of Solid Elements

Defines a pressure load on a face of a CHEXA, CPENTA, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3 or G4	
	CID	N1	N2	N3	SORL	LDIR			

Example:

PLOAD4	2	1106	10.0	8.0	5.0		48		
	6	0.0	1.0	0.0					

Alternate Format and Example (See Remark 8.):

PLOAD4	SID	EID1	P1	P2	P3	P4	“THRU”	EID2	
	CID	N1	N2	N3	SORL	LDIR			

PLOAD4	2	1106	10.0	8.0	5.0		THRU	1143	
	6	0.0	1.0	0.0					

Field	Contents
SID	Load set identification number. (Integer > 0)
EID EID1 EID2	Element identification number. (Integer > 0; for the “THRU” option, EID1 < EID2)
P1, P2, P3, P4	Load per unit surface area (pressure) at the corners of the face of the element. (Real or blank; Default for P2, P3, and P4 is P1.)
G1	Identification number of a grid point connected to a corner of the face. Required data for solid elements only. (Integer > 0 or blank)

Field	Contents
G3	Identification number of a grid point connected to a corner diagonally opposite to G1 on the same face of a CHEXA or CPENTA element. Required data for quadrilateral faces of CHEXA and CPENTA elements only. G3 must be omitted for a triangular surface on a CPENTA element.
G4	Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded. This is required data and is used for CTETRA elements only. (Integer > 0)
CID	Coordinate system identification number. See Remark **2.** (Integer $\geq$ 0; Default = 0)
N1, N2, N3	Components of vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity. See Remark **2.** (Real)
SORL	The character string SURF or LINE. SURF means the surface load acting on the surface of the element and LINE means the consistent edge loads acting on the edges of the element. The default is SURF. Seee Remark **13.**
LDIR	Denote the direction of the line load (SORL=LINE), character string X, Y, Z, TANG, or NORM. The default is NORM. See Remark **14.**

Remarks:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. The continuation entry is optional. If fields 2, 3, 4, and 5 of the continuation entry are blank, the load is assumed to be a pressure acting normal to the face. If these fields are not blank, the load acts in the direction defined in these fields. Note that if CID is a curvilinear coordinate system, the direction of loading may vary over the surface of the element. The load intensity is the load per unit of surface area, not the load per unit of area normal to the direction of loading.

3. For the faces of solid elements, the direction of positive pressure (defaulted continuation) is inward. For triangular and quadrilateral faces, the load intensity P1 acts at grid point G1 and load intensities P2, P3, (and P4) act at the other corners in a sequence determined by applying the right-hand rule to the outward normal.
4. For plate elements, the direction of positive pressure (defaulted continuation) is in the direction of positive normal, determined by applying the right-hand rule to the sequence of connected grid points. The load intensities P1, P2, P3, (and P4) act respectively at corner points G1, G2, G3, (and G4) for triangular and quadrilateral elements. (See plate connection entries.)
5. If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1. P4 has no meaning for a triangular face and may be left blank in this case.
6. Equivalent grid point loads are computed by linear or bilinear interpolation of load intensity followed by numerical integration using isoparametric shape functions. Note that a uniform load intensity will not necessarily result in equal equivalent grid point loads.
7. G1 and G3 are ignored for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements.
8. The alternate format is available only for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements. The continuation entry may be used in the alternate format.
9. For triangular faces of CPENTA elements, G1 is an identification number of a corner grid point that is on the face being loaded and the G3 or G4 field is left blank. For faces of CTETRA elements, G1 is an identification number of a corner grid point that is on the face being loaded and G4 is an identification number of the corner grid point that is not on the face being loaded. Since a CTETRA has only four corner points, this point G4 will be unique and different for each of the four faces of a CTETRA element.
10. For the CQUADR and CTRIAR element, only pressure that acts normal to the element is computed properly. Surface tractions are not resolved into moments normal to the element.
11. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.

12. If fields 3 through 5 of the continuation entry are not blank, the load is assumed to have a fixed direction. If fields 2 through 5 of the continuation entry are left blank, the load is assumed to be a pressure load. In this case, follower force effects are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "**FOLLOWK**" on page 710). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
13. The SORL field is ignored by all elements except QUADR and TRIAR. For QUADR or TRIAR only, if SORL=LINE, the consistent edge loads are defined by the PLOAD4 entry. P1, P2, P3 and P4 are load per unit length at the corner of the element. If all four Ps are given, then the line loads along all four edges of the element are defined. If any P is blank, then the line loads for only two edges are defined. For example, if P1 is blank, the line loads of the two edges connecting to G1 are zero. If two Ps are given, then the line load of the edge connecting to the two grid points is defined. If only one P is given, the second P value default to the first P value. For example, P1 denotes that the line load along edge G1 and G2 has the constant value of P1.
14. The direction of the line load (SORL=LINE) is defined by either (CID, N1, N2, N3) or LDIR. Fatal error will be issued if both methods are given. TANG denotes that the line load is in tangential direction of the edge, pointing from G1 to G2 if the edge is connecting G1 and G2. NORM denotes that the line load is in the mean plan, normal to the edge, and pointing outward from the element. X, Y, or Z denotes the line load is in the X, Y, or Z direction of the element coordinate system. If both (CID, N1, n2, N3) and LDIR are blank, then the default is LDIR=NORM.

PLOADB3 Applied distributed load on CBEAM3 elements

Defines a distributed load to a CBEAM3 element over entire length of the beam axis.

Format:

1	2	3	4	5	6	7	8	9	10
PLOADB3	SID	EID	CID	N1	N2	N3	TYPE	SCALE	
	P(A)	P(B)	P(C)						

Example:

PLOADB3	10	1002	LOCAL	1.0			MOMENT		
	100.	90.	70.						

Field	Contents
SID	Load set identification number. (Integer>0; Required)
EID	CBEAM3 element identification number. (Integer>0, Required)
CID	Coordinate system for load definition. (Character or Integer; Default=”BASIC”)
	“LOCAL”: Local coordinate system;
	“ELEMENT”: Element coordinate system;
	“BASIC” or 0: Basic coordinate system;
n (n>0):	Any user-specified coordinate system identification number.
N1, N2, N3	Load vector components measured in coordinate system specified by CID. (Real; at least one $N_i \neq 0.0$)
TYPE	Type of applied load. (Character=”FORCE”, “MOMENT” or “BIMOMENT”; Required)
SCALE	Load vector scale factor. (Real; Default=1.0)
P(*j*)	Magnitudes of load at *j* (*j*=A, B and C). (Real; Default=0.0)

Remarks:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. The load vector is defined by $\vec{P}_j = SCALE \cdot P_j \cdot \vec{N}, (j = A, B, C)$. The orientation of load $\vec{P}$ is determined by vector $\vec{N}$ and the magnitude is equal to *SCALE•P* times magnitude of vector $\vec{N}$.
3. The distributed load is applied over the entire length of the beam axis, along the line of the shear center.

PLOADX1 Pressure Load on Axisymmetric Element

Defines surface traction to be used with the CQUADX, CTRIAX, and CTRIAX6 axisymmetric element.

Example:

PLOADX1	200	35	3.5	10.5	10	30	20.		

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	Element identification number. (Integer > 0)
PA	Surface traction at grid point GA. (Real)
PB	Surface traction at grid point GB. (Real; Default = PA)
GA, GB	Corner grid points. GA and GB are any two adjacent corner grid points of the element. (Integer > 0)
THETA	Angle between surface traction and inward normal to the line segment. (Real; Default = 0.0)

Remarks:

1. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. PLOADX1 is intended only for the CQUADX, CTRIAX, and CTRIAX6 elements.
3. The surface traction is assumed to vary linearly along the element side between GA and GB.
4. The surface traction is input as force per unit area.
5. THETA is measured counter-clockwise from the inward normal of the straight line between GA and GB, to the vector of the applied load, as shown in **Figure 8-149** and **Figure 8-150**. Positive pressure is in the direction of inward normal to the line segment.

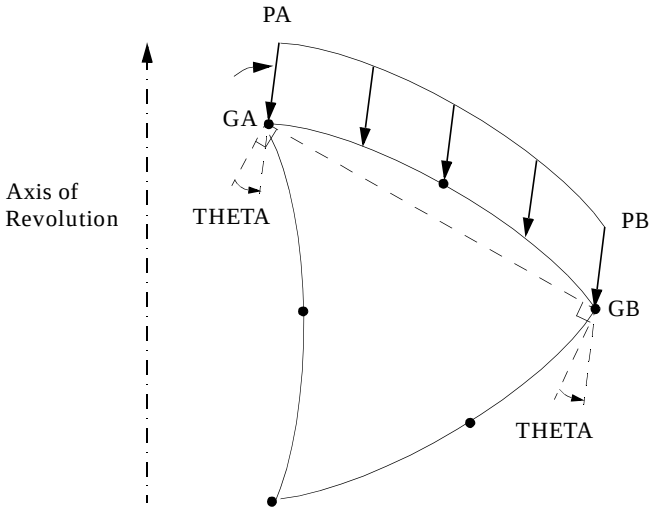


Figure 8-149 Pressure Load on CTRIAX6 or CTRIAX Element

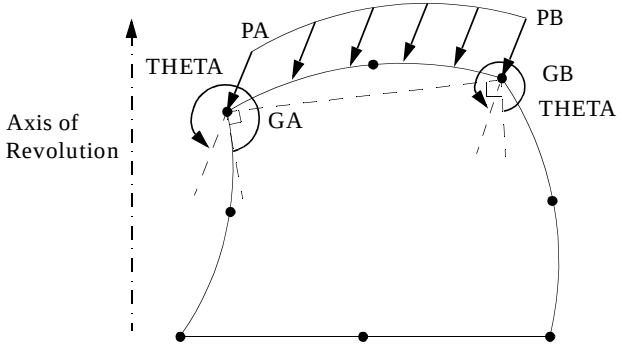


Figure 8-150 Pressure Load on CQUADX Element

PLOTEL Dummy Plot Element Definition

Defines a one-dimensional dummy element for use in plotting.

Format:

1	2	3	4	5	6	7	8	9	10
PLOTEL	EID	G1	G2						

Example:

PLOTEL	29	35	16						

Field	Contents
EID	Element identification number. (Integer > 0)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. This element is not used in the model during any of the solution phases of a problem. It is used to simplify plotting of structures with large numbers of colinear grid points, where the plotting of each grid point along with the elements connecting them would result in a confusing plot.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one PLOTEL element may be defined on a single entry.
4. In superelement analysis, PLOTELs, as well as other elements such as CBAR, CQUAD4, etc., will affect the formation of the superelement tree. The PLOTEL EIDs will also appear in the superelement map output; see the description of PARAM,SEMAPPRT in “**Parameters**” on page 659.
5. Only grid points connected by structural elements appear on structure plots. This does not include points connected only by rigid or general elements or MPCs. A plot element in parallel with elements that do not plot will cause these points to be present.

PLPLANE Fully Nonlinear Plane Element Properties

Defines the properties of a fully nonlinear (i.e., large strain and large rotation) hyperelastic plane strain or axisymmetric element.

Format:

1	2	3	4	5	6	7	8	9	10
PLPLANE	PID	MID	CID	STR					

Example:

PLPLANE	203	204	201						

Field	Contents
PID	Element property identification number. (Integer > 0)
MID	Identification number of a MATHP entry. (Integer > 0)
CID	Identification number of a coordinate system defining the plane of deformation. See Remarks **2.** and **3.** (Integer $\geq$ 0; Default = 0)
STR	Location of stress and strain output. (Character: “GAUS” or “GRID”, Default = “GRID”)

Remarks:

1. PLPLANE can be referenced by a CQUAD, CQUAD4, CQUAD8, CQUADX, CTRIA3, CTRIA6, or CTRIAX entry.
2. Plane strain hyperelastic elements must lie on the x-y plane of the CID coordinate system. Stresses and strains are output in the CID coordinate system.
3. Axisymmetric hyperelastic elements must lie on the x-y plane of the basic coordinate system. CID may not be specified and stresses and strains are output in the basic coordinate system.

PLSOLID Fully Nonlinear Solid Element Properties

Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.

Format:

1	2	3	4	5	6	7	8	9	10
PLSOLID	PID	MID	STR						

Example:

PLSOLID	20	21							

Field	Contents
PID	Element property identification number. (Integer > 0)
MID	Identification number of a MATHP entry. (Integer > 0)
STR	Location of stress and strain output. (Character: “GAUS” or “GRID”, Default = “GRID”)

Remarks:

1. PLSOLID can be referenced by a CHEXA, CPENTA, or CTETRA entry.
2. Stress and strain are output in the basic coordinate system.

PMASS Scalar Mass Property

Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).

Format:

1	2	3	4	5	6	7	8	9	10
PMASS	PID1	M1	PID2	M2	PID3	M3	PID4	M4	

Example:

PMASS	7	4.29	6	13.2					

Field	Contents
PIDi	Property identification number. (Integer > 0)
Mi	Value of scalar mass. (Real)

Remarks:

1. Mass values are defined directly on the CMASS2 and CMASS4 entries, and therefore do not require a PMASS entry.
2. Up to four mass values may be defined by this entry.
3. For a discussion of scalar elements, see "**Scalar Elements (CELASi, CMASSi, CDAMPi)**" on page 193 of the *MSC.Nastran Reference Guide*.

POINT Edge Point for FEEDGE Entry

Define edge point for FEEDGE or SELOC entries.

Format:

1	2	3	4	5	6	7	8	9	10
POINT	ID	CP	X1	X2	X3				

Example:

POINT	12	1	1.	2.	5.				

Field	Contents	Type	Default
ID	Point identification number.	Integer ≥ 0	Required
CP	Identification number of coordinate system in which the location of point is defined.	Integer ≥ 0	0
X1, X2, X3	Location of the point in coordinate system CP.	Real	0.0

Remarks:

1. POINT is used to specify additional geometric points for edges and can be used by p-version elements. There are no degrees-of-freedom assigned to a point.
2. FEEDGE entries can refer to POINT entries.
3. SELOC entries can refer to POINT entries in the residual or part superelements.
4. ID of POINTs must be unique with respect to ID of GRID entries.
5. POINT entries can be referenced on SET1 / SET3 for defining arbitrary beam cross section, ABCS, via PBRSECT / PBMSECT. Note that CP and X3 must be left blank for POINT entries used for ABCS.

POINTAX Conical Shell Point

Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested. These points are not subject to constraints via MPCAX, SPCAX, or OMITAX entries.

Format:

1	2	3	4	5	6	7	8	9	10
POINTAX	ID	RID	PHI						

Example:

POINTAX	2	3	30.0						

Field	Contents
ID	Point identification number. (Unique Integer > 0)
RID	Identification number of a RINGAX entry. (Integer > 0)
PHI	Azimuthal angle in degrees. (Real)

Remarks:

1. This entry is allowed only if an AXIC entry is also present.
2. POINTAX identification numbers must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.
3. For a discussion of the conical shell problem, see "**Conical Shell Element (RINGAX)**" on page 155 of the *MSC.Nastran Reference Guide*.

PRAC2D CRAC2D Element Property

Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PRAC2D	PID	MID	T	IPLANE	NSM	GAMMA	PHI		

Example:

PRAC2D	108	2	0.10	0	.17	.50	180.		

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
T	Element thickness. (Real > 0.0)
IPLANE	Plane strain or plane stress option. Use 0 for plane strain; 1 for plane stress. (Integer = 0 or 1)
NSM	Non-structural mass per unit area. (Real $\geq$ 0.0; Default = 0)
GAMMA	Exponent used in the displacement field. See Remark **4.** (Real; Default = 0.5)
PHI	Angle (in degrees) relative to the element x-axis along which stress intensity factors are to be calculated. See Remark **4.** (Real; Default = 180.0)

Remarks:

1. All PRAC2D property entries should have unique identification numbers with respect to all other property entries.
2. PRAC2D entry may refer to MAT1, MAT2, or MAT8 material property entries.
3. For plane strain analysis, only MAT1 type data should be used.
4. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

PRAC3D CRAC3D Element Property

Defines the properties of the CRAC3D structural element.

Format:

1	2	3	4	5	6	7	8	9	10
PRAC3D	PID	MID	GAMMA	PHI					

Example:

PRAC3D	108	2	.50	180.					

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GAMMA	Exponent used in the displacement field. See Remark **3.** (Real; Default = 0.5)
PHI	Angle (in degrees) relative to the element x axis along which stress intensity factors are to be calculated. See Remark **3.** (Real; Default = 180.0)

Remarks:

1. All PRAC3D property entries should have unique identification numbers with respect to all other property entries.
2. Either isotropic (MAT1) or anisotropic (MAT9) material entries may be referenced.
3. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

PRESAX Conical Shell Pressure Load

Defines the static pressure loading on a conical shell element.

Format:

1	2	3	4	5	6	7	8	9	10
PRESAX	SID	P	RID1	RID2	PHI1	PHI2			

Example:

PRESAX	3	7.92	4	3	20.6	31.4			

Field	Contents
SID	Load set identification number. (Integer > 0)
P	Pressure value. (Real)
RID1, RID2	Ring identification numbers. See RINGAX entry. (Integer > 0)
PHI1, PHI2	Azimuthal angles in degrees. (Real; PHI2 > PHI1)

Remarks:

1. PRESAX is allowed only if an AXIC entry is also present.
2. Load sets must be selected with the Case Control command LOAD = SID.
3. For a discussion of the conical shell problem, see "**Conical Shell Element (RINGAX)**" on page 155 of the *MSC.Nastran Reference Guide*.
4. For axisymmetric loading over 360 degrees, use PHI1 = 0.0 and PHI2 = 360.0.

PRESPT Fluid Pressure Point

Defines the location of pressure points in the fluid for recovery of pressure data.

Format:

1	2	3	4	5	6	7	8	9	10
PRESPT	IDF		IDP1	PHI1	IDP2	PHI2	IDP3	PHI3	

Example:

PRESPT	14		141	0.0			142	90.0	

Field	Contents
IDF	Fluid point (RINGFL entry) identification number. (Integer > 0)
IDPi	Pressure point identification number. (Integer > 0)
PHIi	Azimuthal position on fluid point referenced by IDF in fluid coordinate system. (Real)

Remarks:

1. PRESPT is allowed only if an AXIF entry is also present.
2. All pressure point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. The pressure points are used primarily for the identification of output data. They may also be used as points at which to measure pressure for input to control devices (see “**Additional Topics**” on page 555 of the *MSC.Nastran Reference Guide*).
4. One, two, or three pressure points may be defined per entry.
5. Output requests for velocity and acceleration of these degrees-of-freedom will result in derivatives of pressure with respect to time.

PROD Rod Property

Defines the properties of a rod element (CROD entry).

Format:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	NSM			

Example:

PROD	17	23	42.6	17.92	4.2356	0.5			

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks **2.** and **3.** (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)
C	Coefficient to determine torsional stress. (Real; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real)

Remarks:

1. PROD entries must all have unique property identification numbers.
2. For structural problems, MID must reference a MAT1 material entry.
3. For heat transfer problems, MID must reference a reference MAT4 or MAT5 entry.
4. The formula used to calculate torsional stress is

$$\tau = \frac{CM_\theta}{J}$$

where M_θ is the torsional moment.

PSET p-Version Element Polynomial Distribution

Describes polynomial order distribution and is selected by the ADAPT Case Control command.

Format:

1	2	3	4	5	6	7	8	9	10
PSET	SID	POLY1	POLY2	POLY3	CID	SETTYP	ID		

Example:

PSET	127	1	2	1			12		

Field	Contents	Type	Default
SID	ID selected in the ADAPT Case Control command.	Integer > 0	Required
CID	Coordinate system used to specify polynomial values in different directions. See Remark 1.	Integer ≥ 0	Remark 2.
POLYi	Polynomial order in 1, 2, 3 directions of the CID system.	Integer > 0	Remark 3.
SETTYP	Type of set provided (“SET” or “ELID”)	Character	“SET”
ID	SET ID or element ID with this p-value specification.	Integer > 0	999999

Remarks:

1. CID facilitates the specification of the p-order in curvilinear systems. For example, when modeling a thin cylinder, the user can restrict the p-order through the thickness of all elements to be 2 or 3 without specifically checking the connectivity of each element.
2. If the CID system is blank, the element’s topology is used to establish the 1, 2, 3 directions. The 1 direction is from the first to the second grid of the element, the 2 direction is from the first to the fourth, and, the 3 direction is from the first to the fifth. If CID is not blank then the following algorithm will

be used to determine the p-order of each edge: a vector will be defined in the CID system from the first to the second grid of every edge. (Curvilinear systems are evaluated at the midpoint of this vector.) The p-level of each edge is now determined by the nearest integer to

$$p = \sqrt{(n_1 \cdot \text{POLY1})^2 + (n_2 \cdot \text{POLY2})^2 + (n_3 \cdot \text{POLY3})^2}$$

where (n_1, n_2, n_3) are the components of this unit vector in the CID system.

3. The default value for POLY2 and POLY3 is POLY1.
4. Any overlap of the PSET specification will result in a warning message and the use of the PSET with the highest pi entry.
5. Whenever SETTYP = “SET”, a SET command must be defined in the SETS DEFINITION section of the Case Control Section.
6. SET = 999999 is a reserved set that includes all elements.
7. Whenever there are more than one PSET entries for a given element, then:
    - If CID on the PSET entries are the same, the entry with the maximum POLYi will be used.
    - If CID on the PSET entries are different, a fatal message is issued.

PSHEAR Shear Panel Property

Defines the properties of a shear panel (CSHEAR entry).

Format:

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM	F1	F2			

Example:

PSHEAR	17	23	42.6	17.92	4.236	0.5			

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number of a MAT1 entry. (Integer > 0)
T	Thickness of shear panel. (Real $\neq$ 0.0)
NSM	Nonstructural mass per unit area. (Real)
F1	Effectiveness factor for extensional stiffness along edges 1-2 and 3-4. See Remark **2.** (Real $\geq$ 0.0; Default = 0.0)
F2	Effectiveness factor for extensional stiffness along edges 2-3 and 1-4. See Remark **2.** (Real $\geq$ 0.0; Default = 0.0)

Remarks:

1. All PSHEAR entries should have unique identification numbers with respect to all other property entries.
2. The effective extensional area is defined by means of equivalent rods on the perimeter of the element. If F1 $\leq$ 1.01, the areas of the rods on edges 1-2 and 3-4 are set equal to $(F1 \cdot T \cdot PA)/(L12 + L34)$ where PA is the panel surface area-half the vector cross product area of the diagonals-and L12, L34 are the lengths of sides 1-2 and 3-4. Thus, if F1 = 1.0, the panel is fully effective for extension in the 1-2 direction. If F1 > 1.01, the areas of the rods on edges 1-2 and 3-4 are each set equal to $0.5 \cdot F1 \cdot T^2$.

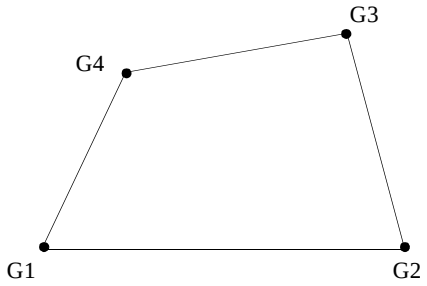


Figure 8-151 Extensional Area for Shear Panel

Thus, if F1 = 30, the effective width of skin contributed by the panel to the flanges on edges 1-2 and 3-4 is equal to 15T. The significance of F2 for edges 2-3 and 1-4 is similar.

3. Poisson's ratio coupling for extensional effects is ignored.

PSHELL Shell Element Property

Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12I/T**3	MID3	TS/T	NSM	
	Z1	Z2	MID4						

Example:

PSHELL	203	204	1.90	205	1.2	206	0.8	6.32	
	+.95	-.95							

Field	Contents
PID	Property identification number. (Integer > 0)
MID1	Material identification number for the membrane. (Integer $\geq$ 0 or blank)
T	Default membrane thickness for Ti on the connection entry. If T is blank then the thickness must be specified for Ti on the CQUAD4, CTRIA3, CQUAD8, and CTRIA6 entries. (Real or blank)
MID2	Material identification number for bending. (Integer $\geq$ -1 or blank)
12I/T**3	Bending moment of inertia ratio, $12I/T^3$. Ratio of the actual bending moment inertia of the shell, *I*, to the bending moment of inertia of a homogeneous shell, $T^3/12$. The default value is for a homogeneous shell. (Real > 0.0; Default = 1.0)
MID3	Material identification number for transverse shear. (Integer > 0 or blank; unless MID2 > 0, must be blank.)
TS/T	Transverse shear thickness ratio, T_s/T. Ratio of the shear thickness, (T_s), to the membrane thickness of the shell, *T*. The default value is for a homogeneous shell. (Real > 0.0; Default = .833333)
NSM	Nonstructural mass per unit area. (Real)

Z1, Z2 — Fiber distances for stress calculations. The positive direction is determined by the right-hand rule, and the order in which the grid points are listed on the connection entry. See Remark **11.** for defaults. (Real or blank)

MID4 — Material identification number for membrane-bending coupling. See Remarks **6.** and **13.** (Integer > 0 or blank, must be blank unless MID1 > 0 and MID2 > 0, may not equal MID1 or MID2.)

Remarks:

1. All PSHELL property entries should have unique identification numbers with respect to all other property entries.
2. The structural mass is calculated from the density using the membrane thickness and membrane material properties. If MID1 is blank, then the density is obtained from the MID2 material.
3. The results of leaving an MID field blank (or MID2 = -1) are:

   MID1 — No membrane or coupling stiffness

   MID2 — No bending, coupling, or transverse shear stiffness

   MID3 — No transverse shear flexibility

   MID4 — No bending-membrane coupling unless ZOFFS is specified on the connection entry. See Remark **6.**

   MID2=-1 — See Remark **12.**

   Note: MID1 and MID2 must be specified if the ZOFFS field is also specified on the connection entry.
4. The continuation entry is not required.
5. The structural damping (GE on the MATi entry) is obtained from the MID1 material. If MID1 is blank, then it is obtained from the MID2 material. If PARAM,SHLDAMP,DIFF or DIFF is any other character except SAME, then the structural damping κ^4 matrix is computed using the GE entries on the MATi entries according to rules in the following table. If a single PSHELL corresponds to row 8 of the table, then all PSHELLs in the model will follow the rule of row 8. Rows 1-7 is an attempt to maintain upward compatibility, if a user inadvertently places a SHLDAMP,DIFF in the model

Caution: Large values of damping associated with an MID4 entry, when using PARAM,SHLDAMP,DIFF, can cause structural instability in transient dynamics.

Table 8-29 SHELL Structural Damping Rules

SHELL Structural Damping Rules					
Row	**MID1**	**MID2**	**MID3**	**MID4**	**K^4 based on**
1*	v	v			MID1 GE value
2	v				MID1 GE value
3	v	-1			MID1 GE value
4	v	v			MID1 GE value
5		v			MID2 GE value
6		v	v		MID2 GE value
7	v1	v2	v3	v4	$n \rightarrow$ total number of non blank vi $m \rightarrow$ total number of non zero ge_i If: n = m and $ge_i = ge_2 = \ldots = ge_m$ Or: $m = 1$ and $ge_1 \neq 0$ Or: $m = 0$ MID1 GE value
8	v1	v2	v3	v4	Otherwise: $ge_1 \cdot$ membrane-stiff $+ ge_2 \cdot$ bending-stiff $+ ge_3 \cdot$ transverse shear-stiff $+ ge_4 \cdot$ bending-membrane-stiff is used
* v $\rightarrow$ MIDi values the same, vi $\rightarrow$ MIDi values different or blank $ge_i \rightarrow$ GE value from a MATj entry associated with MIDi If for row 8, a $ge_i = 0$ it is replaced by $ge_i = 1. - 8$					

6. The following should be considered when using MID4.
   - The MID4 field should be left blank if the material properties are symmetric with respect to the middle surface of the shell. If the element centerline is offset from the plane of the grid points but the material properties are symmetric, the preferred method for modeling the offset is by use of the ZOFFS field on the connection entry. Although the MID4 field may be used for this purpose, it may produce ill-conditioned stiffness matrices (negative terms on factor diagonal) if done incorrectly.
   - Only one of the options MID4 or ZOFFS should be used; if both methods are specified the effects are cumulative. Since this is probably not what the user intented, unexpected answers will result. Note that the mass properties are not modified to reflect the existence of the offset when the ZOFFS and MID4 methods are used. If the weight or mass properties of an offset plate are to be used in an analysis, the RBAR method must be used to represent the offset. See "**Shell Elements (CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR)**" on page 131 of the *MSC.Nastran Reference Guide*.
   - The effects of MID4 are not considered in the calculation of differential stiffness. Therefore, it is recommended that MID4 be left blank in buckling analysis.
7. This entry is referenced by the CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR entries via PID.
8. For structural problems, MIDi must reference a MAT1, MAT2, or MAT8 material property entry
9. If the transverse shear material MID3 or the membrane-bending coupling term MID4 references a MAT2 entry, then G33 must be zero. If MID3 references a MAT8 entry, then G1Z and G2Z must not be zero.
10. For heat transfer problems, MIDi must reference a MAT4 or MAT5 material property entry.
11. The default for Z1 is -T/2, and for Z2 is +T/2. T is the local plate thickness defined either by T on this entry or by membrane thicknesses at connected grid points, if they are input on connection entries.

12. For plane strain analysis, set MID2=-1 and set MID1 to reference a MAT1 entry. In-plane loads applied to plain strain elements are interpreted as line-loads with a value equal to the load divided by the thickness. Thus, if a thickness of “1.0” is used, the value of the line-load equals the load value. Pressure can be approximated with multiple line loads where the pressure value equals the line-load divided by the length between the loads.
13. For the CQUADR and CTRIAR elements, the MID4 field should be left blank because their formulation does not include membrane-bending coupling.
14. If MIDi is greater than or equal to 10^8, then parameter NOCOMPS is set to +1 indicating that composite stress data recovery is desired. (MIDi greater than 10^8 are generated by PCOMP entries.)
15. For a material nonlinear property, MID1 must reference a MATS1 entry and be the same as MID2, unless a plane strain (MID2 = -1) formulation is desired. Also, MID3 cannot reference a MATS1 entry.
16. If transverse shear flexibility is specified for a model with curved shells where the loading is dominated by twist, results will not converge and may be inaccurate. PARAM,SNORM should be set for this unique model condition.

PSHELL1 (SOL 700)

Defines the properties of SOL 700 shell elements that are much more complicated than the shell elements defined using the PSHELL entry.

Format:

1	2	3	4	5	6	7	8	9	10
PSHELL1	PID	MID	FORM	QUAD	NUMB	SHFACT			
	T1	T2	T3	T4					

Example:

PSHELL1	7	2	BLT	GAUSS	5	0.9			
	10.0	10.0	10.0	10.0					

Field	Contents
PID	Unique property number. (Integer, Required)
MID	Material number. See Remark 2. (Integer, Required)
FORM	Shell formulation. See Remark 2. (Character, Required)
	HUGHES Hughes-Liu.
	BLT Belytschko-Lin-Tsay.
	KEYHOFF Key-Hoff.
	C0-TRIA C0 triangle.
	MEMB Membrane element (no bending).
QUAD	Type of quadrature. (Character, Default = GAUSS)
	GAUSS Gauss quadrature.
	LOBATTO Lobatto quadrature.
NUMB	The number of integration points through the thickness. For Gauss and Lobatto quadrature: (Integer, Default = 3)
	1 1 point (membrane element)
	2 2 point
	3 3 point
	4 4 point
	5 5 point

Field	Contents
	6 6poinT
	7 7point
	8 8point
	9 9point
	10 10 point
SHFACT	Shear factor. (Real, Default = 0.83333)
T1 toT4	Element thickness at the grid points. See Remark 8. (Real, Default = 0.0)

Remarks:

1. Shells of constant thickness with three-point Gauss integration are more easily defined using the PSHELL entry.
2. For CQUAD4 elements, the default formulation is KEYHOFF. For CTRIA3 elements, the default formulation is CO-TRIA.
3. Make the property number unique with respect to all other properties.
4. If the thickness T is set to 9999., all elements with this property number are not treated as CQUAD4 and CTRIA3 elements but are converted to CSEG entries. This conversion allows CSEGs to be defined easily using standard preprocessors.
5. Membrane elements can only be triangular and must reference a DMAT or DMATEL material entry. In case the HUGHES shell formulation is used, only an elastic material can be referred to.
6. If the thickness is set to blank or 0.0, the thickness is defined on the CTRIA3 and CQUAD4 entry.

PSHELLD (SOL 700)

Defines properties for shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
PSHELLD	PID	MID1	ELFORM	SHRF	NIP		QR/IRID		
	T1	T2	T3	T4	NLOC	MAREA			

Example:

PSHELLD	10	53	1	.99	3				
	.2	.2	.2	.2					

Field	Contents	Type	Default
PID	Property ID. PID is referenced on the CQUAD4.	I > 0	Required
MID	Material ID.	I>0	Required
ELFORM	Element formulation options, see Remarks 1 and 2 below: EQ.1: Hughes-Liu, EQ.2: Belytschko-Tsay, EQ.3: BCIZ triangular shell, EQ.4: C0 triangular shell, EQ.5: Belytschko-Tsay membrane, EQ.6: S/R Hughes-Liu, EQ.7: S/R co-rotational Hughes-Liu, EQ.8: Belytschko-Leviathan shell, EQ.9: Fully integrated Belytschko-Tsay membrane, EQ.10: Belytschko-Wong-Chiang, EQ.11: Fast (co-rotational) Hughes-Liu,	I>0	1

Field	Contents	Type	Default
	EQ.16: Fully integrated shell element (very fast), EQ.17: Fully integrated DKT, triangular shell element , EQ.18: Fully integrated linear DK quadrilateral / triangular shell EQ.20: Fully integrated linear assumed strain C0 shell (See remarks). EQ.21: Fully integrated linear assumed strain C0 shell (5 DOF). EQ.22: Linear shear panel element (3 DOF per node, see remarks) The type 18 element is only for linear static and normal modes. It can also be used for linear springback in sheet metal stamping.		
SHRF	Shear correction factor which scales the transverse shear stress. The shell formulations in LS-DYNA, with the exception of the BCIZ and DK elements,are based on a first order shear deformation theory that yields constant transverse shear strains which violates the condition of zero traction on the top and bottom surfaces of the shell. The shear correction factor is attempt to compensate for this error. A suggested value is 5/6 for isotropic materials. This value is incorrect for sandwich or laminated shells; consequently, laminated / sandwich shell theory is now used in some of the constitutive model.	R>0	1.0
NIP	Number of through thickness integration points.	I>0	2

Field	Contents	Type	Default
	Either Gauss (default) or Lobatto integration can be used. The flag for Lobatto integration can be set on the control command, PARAM,LSDYNA,SHELL. The location of the Gauss and Lobatto integration points are tabulated below. EQ.0: set to 2 integration points for shell elements. EQ.1: 1 point (no bending) EQ.2: 2 point EQ.3: 3 point EQ.4: 4 point EQ.5: 5 point EQ.6: 6 point EQ.7: 7 point EQ.8: 8 point EQ.9: 9 point EQ.1: 10 point GT.1: trapezoidal or user defined rule Through thickness integration for the two-dimensional elements (options 12-15 above) is not meaningful; consequently, the default is equal to 1 integration point. Fully integrated two-dimensional elements are available for options 13 and 15 by setting NIP equal to a value of 4 corresponding to a 2 by 2 Gaussian quadrature. If NIP is 0 or 1 and the MATD098 model is used, then a resultant plasticity formulation is activated. NIP is always set to 1 if a constitutive model based on resultants is used.		
QR / IRID	Quadrature rule or Integration rule ID:	I	0

Field	Contents	Type	Default
	LT.0: absolute value is specified rule number, EQ.0: Gauss / Lobatto (up to 10 points are permitted), EQ.1: trapezoidal, *not recommend for accuracy reasons*.		
T1	Shell thickness at node n_1, unless the thickness is defined on the CQUAD4 / CTRIA3 entries.	R>0	0.0
T2	Shell thickness at node n_2, see comment for T1 above.	R>0	0.0
T3	Shell thickness at node n_3, see comment for T1 above.	R>0	0.0
T4	Shell thickness at node n_4, see comment for T1 above.	R>0	0.0
NLOC	Location of reference surface for three dimensional shell elements. If nonzero, the mid-surface of the shell is offset by a value equal to $offset = -0.50 \times NLOC \times (average\ shell\ thickness)$ Alternatively, the offset can be specified by using the OFFSET option in the CQUAD4 / CTRIA3 input section. EQ. 1: top surface, EQ. 0: mid-surface (default), EQ.-1: bottom surface.	I	0
MAREA	Non-structural mass per unit area. This is additional mass which comes from materials such as carpeting. This mass is not directly included in the time step calculation.	R>0	0.0

Gauss Integration Rule					
Number of Gauss Point	**1 Point**	**2 Point**	**3 Point**	**4 Point**	**5 Point**
#1	.0	-.5773503	.0	-.8611363	.0
#2		+.5773503	-.7745967	-.3399810	-.9061798
#3			+.7745967	+.3399810	-.5384693
#4				+.8622363	+.5384693
#5					+.9061798
Number of Gauss Point	**6 Point**	**7 Point**	**8 Point**	**9 Point**	**10 Point**
#1	-.9324695	-.9491080	-.9702896	-.9681602	-.9739066
#2	-.6612094	-.7415312	-.7966665	-.8360311	-.8650634
#3	-.2386192	-.4058452	-.5255324	-.6133714	-.6794096
#4	+.2386192	.0	-.1834346	-.3242534	-.4333954
#5	+.6612094	+.4058452	+.1834346	0.0	-.1488743
#6	+.9324695	+.7415312	+.5255324	+.3242534	+.1488743
#7		+.9491080	+.7966665	+.6133714	+.4333954
#8			+.9702896	+.8360311	+.6794096
#9				+.9681602	+.8650634
#10					+.9739066

Location of through thickness Gauss integration points. The coordinate is referenced to the shell midsurface at location 0. The inner surface of the shell is at -1 and the outer surface is at +1.

Lobatto Integration Rule					
Number of Integ. Point	**-**	**-**	**3 Point**	**4 Point**	**5 Point**
#1			.0	-1.0	.0
#2			-1.0	-.4472136	-1.0
#3			+1.0	+.4472136	-.6546537
#4				+1.0	+.6546537

Lobatto Integration Rule					
Number of Integ. Point	**-**	**-**	**3 Point**	**4 Point**	**5 Point**
#5					+1.0
Number of Integ. point	**6 POINT**	**7 POINT**	**8 POINT**	**9 POINT**	**10 POINT**
#1	-1.0	-1.0	-1.0	-1.0	-1.0
#2	-.7650553	-.8302239	-.8717401	-.8997580	-.9195339
#3	-.2852315	-.4688488	-.5917002	-.6771863	-.7387739
#4	+.2852315	.0	-.2092992	-.3631175	-.4779249
#5	+.7650553	+.4688488	+.2092992	.0	-.1652790
#6	+1.0	+.8302239	+.5917002	+.3631175	+.1652790
#7		+1.0	+.8717401	+.6771863	+.4779249
#8			+1.0	+.8997580	+.7387739
#9				+1.0	+.9195339
#10					+1.0

Location of through thickness Lobatto integration points. The coordinate is referenced to the shell midsurface at location 0. The inner surface of the shell is at -1 and the outer surface is at +1.

Remarks:

1. The linear elements consist of an assembly of membrane and plate elements. The elements have six d.o.f. per node and can therefore be connected to beams, or used in complex shell surface intersections. All elements possess the required zero energy rigid body modes and have exact constant strain and curvature representation, i.e. they pass all the first order patch tests. In addition, the elements have behavior approaching linear bending (cubic displacement) in the plate-bending configuration.
   a. The membrane component of all elements is based on an 8-node / 6-node isoparametric mother element which incorporates nodal in-plane rotations through cubic displacement constraints of the sides [Taylor, 1987; Wilson, 2000].

b. The plate component of element 18 is based on the Discrete Kirchhoff Quadrilateral (DKQ) [Batoz, 1982]. Because the Kirchhoff assumption is enforced, the DKQ is transverse shear rigid and can only be used for thin shells. No transverse shear stress information is available. The triangle is based on a degeneration of the DKQ. This element sometimes gives slightly lower eigenvalues when compared with element type 20.

c. The plate component of element 20 is based on the 8-node serendipity element. At the mid-side, the parallel rotations and transverse displacements are constrained and the normal rotations are condensed to yield a 4-node element. The element is based on thick plate theory and is recommended for thick and thin plates.

d. The quadrilateral elements contain a warpage correction using rigid links.

e. The membrane component of element 18 has a zero energy mode associated with the in-plane rotations. This is automatically suppressed in a non-flat shell by the plate stiffness of the adjacent elements. Element 20 has no spurious zero energy modes.

2. The linear shear panel element resist tangential in plane shearing along the four edges and can only be used with the elastic material constants of MATD001. Membrane forces and out-of-plane loads are not resisted.

PSOLID Properties of Solid Elements

Defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries).

Format:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CORDM	IN	STRESS	ISOP	FCTN		

Example:

PSOLID	2	100	6	TWO	GRID	REDUCED			

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Identification number of a MAT1, MAT4, MAT5, MAT9, or MAT10 entry. (Integer > 0)
CORDM	Identification number of the material coordinate system. See Remarks **3.** and **4.** (Integer; Default = 0, which is the basic coordinate system; see Remark **3.**)
IN	Integration network. See Remarks **5.**, **6.**, **7.**, and **9.** (Integer, Character, or blank)
STRESS	Location selection for stress output. See Remarks **8.** and **9.** (Integer, Character, or blank)
ISOP	Integration scheme. See Remarks **5.**, **6.**, **7.**, and **9.** (Integer, Character, or blank)
FCTN	Fluid element flag. (Character: “PFLUID” indicates a fluid element, “SMECH” indicates a structural element; Default = “SMECH.”)

Remarks:

1. PSOLID entries should have unique identification numbers with respect to all other property entries.
2. Isotropic (MAT1 or MAT4), anisotropic (MAT5 or MAT9), or fluid (MAT10) material properties may be referenced. If FCTN=“PFLUID”, then MID must reference a MAT10 entry. PFLUID is not available for SOL 600 or SOL 700.

3. See the CHEXA, CPENTA, or CTETRA entry for the definition of the element coordinate system. The material coordinate system (CORDM) may be the basic system (0 or blank), any defined system (Integer > 0), or the element coordinate system (-1). The default value for CORDM is zero unless it is overridden by the NASTRAN statement with the CORDM keyword. See "**nastran Command and NASTRAN Statement**" on page 1.
4. If MID references a MAT9 entry, then CORDM defines the material property coordinate system for Gij on the MAT9 entry. CORDM is ignored in the stress output labeled "NONLINEAR STRESSES" where only the element coordinate system is used.
5. For CHEXA and CPENTA elements with no midside nodes, reduced shear integration with bubble functions (ISOP = blank or "REDUCED" and IN = blank or "BUBBLE") is the default. This is recommended because it minimizes shear locking and Poisson's ratio locking and does not cause modes of deformation that lead to no strain energy. The effects of using nondefault values are as follows:
   a. IN = "THREE" or 3 produces an overly stiff element.
   b. If IN = "TWO" or 2 and the element has midside nodes, modes of deformation may occur that lead to no strain energy.
   c. Standard isoparametric integration (ISOP = "FULL" or 1 and IN = "TWO" or 2; or "THREE" or 3) produces an element overly stiff in shear. This type of integration is more suited to nonstructural problems.
6. IN = "BUBBLE" is not allowed for CTETRA elements or for CHEXA and CPENTA elements with midside nodes.
7. If you use IN="BUBBLE" for CTETRA elements, NASTRAN internally switch to IN=2 if you have 4-noded CTETRA element and IN=3 greater than 4 nodes.
8. Stress output may be requested at the Gauss points (STRESS = "GAUSS" or 1) of CHEXA and CPENTA elements with no midside nodes. Gauss point output is available for the CTETRA element with or without midside nodes.
9. The following tables indicate the allowed options and combination of options. If a combination not found in the table is used, then a warning message will be issued and default values will be assigned for all options.
10. The gauss point locations for the solid elements are documented in "" on page 568 of the *MSC.Nastran Reference Manual.*
11. Remarks 5 through 10, DO NOT APPLY TO SOL 600 or SOL 700.

Table 8-30 CHEXA Entry Options

CHEXA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability		
					SOLs 106, 129, 400	SOL 600	SOL 700
8 Node	2x2x2 Reduced Shear with Bubble Function (default)	BUBBLE or Blank or 0 (Default)	Blank or GRID or GAUSS or 1	Blank or REDUCED (Default*)	Yes	Yes**	Yes
	2x2x2 Reduced Shear Only	TWO or 2					
	2x2x2 Standard Isoparametric			FULL or 1		Yes	No
	3x3x3 Reduced Shear Only	THREE or 3	Blank or GRID	Blank or REDUCED	No	Yes	Yes
	3x3x3 Standard Isoparametric			FULL or 1		Yes**	No
9--20 Node	2x2x2 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or REDUCED	No	Yes	No
	2x2x2 Standard Isoparametric			FULL or 1		Yes	No
	3x3x3 Reduced Shear Only (default)	Blank or THREE or 3 (Default)		Blank or REDUCED (Default*)		No	No
	3 x3x3 Standard Isoparametric			FULL or 1		Yes	No
p-elements	Reduced (p-order) Bubble	0 or 1	Not applicable	0	No	No	No
	Bubble, P+ISOP Integration	1		$-10 \le ISOP \le 10$			
	No Bubble, P+ISOP Integration	2 or 3					

*REDUCED is the default only for structural elements (FCTN="SMECH").

** Requires PARAM,MRALIAS

Table 8-31 CPENTA Entry Options

CPENTA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability		
					SOLs 106, 129, 400	SOL 600	SOL 700
6 Node	2x3 Reduced Shear with Bubble Function (Default)	Blank or 0 or BUBBLE (Default)	GAUSS or 1 or Blank or GRID	Blank or REDUCED (Default*)	Yes	Yes**	Yes
	2x3 Reduced Shear Only	TWO or 2					
	2x3 Standard Isoparametric			FULL or 1		Yes	No
	3x7 Reduced Shear Only	THREE or 3	Blank or GRID	Blank or REDUCED	No	No	No
	3x7 Standard Isoparametric			FULL or 1		No	No
7-15 Node	2x3 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or REDUCED	No	Yes**	No
	2x3 Standard Isoparametric			FULL or 1		Yes	No
	3x7 Reduced Shear Only (default)	Blank or THREE or 3 (Default)		Blank or REDUCED (Default*)		No	No
	3x7 Standard Isoparametric			FULL or 1		No	No

** Requires PARAM,MRALIAS

Table 8-31 CPENTA Entry Options (continued)

CPENTA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability		
					SOLs 106, 129, 400	SOL 600	SOL 700
p-elements Bubble Function	2x3 for p=1, 1, 1 3x7 for p=2, 2, 2 (p+1)x(p) for all other	0 or 1	Not applicable	0	No	No	No
	2x3 for p=1, 1, 1 3x7 for p=2, 2,2	1		0			
	3x7 for p=1, 1, 1			1			
	(p+ISOP+1)x (p +ISOP) for all other			–10 ≤ ISOP ≤ –1 or 2 ≤ ISOP ≤ 10			
p-elements Standard Isopara-metric: (no bubble function)	2x3 for p=1, 1, 1 3x7 for p=2, 2, 2	2 or 3		0			
	3x7 for p=1, 1, 1			1			
	(p+ISOP+1)x (p +ISOP) for all other			–10 ≤ ISOP ≤ –1 or 2 ≤ ISOP ≤ 10			

*REDUCED is the default only for structural elements (FCTN=“SMECH”).

Table 8-32 CTETRA Entry Options

CTETRA	Integration	IN	STRESS (Default: GRID)	ISOP	Nonlinear Capability		
					SOLs 106, 129, 400	SOL 600	SOL 700
4 Node	1-Point Standard Isoparametric (Default)	Blank or TWO or 2 (Default)	GAUSS or 1 or Blank or GRID	Blank or FULL	Yes	Yes**	Yes
	5-Point Standard Isoparametric	THREE or 3	Blank or GRID		No	Yes	No
5-10 Node	5-Point Standard Isoparametric	Blank or THREE or 3 (Default)	GAUSS or 1 or Blank or GRID	Blank or FULL	No	Yes	No

Table 8-32 CTETRA Entry Options

CTETRA	Integration	IN	STRESS (Default: GRID)	ISOP	Nonlinear Capability		
					SOLs 106, 129, 400	SOL 600	SOL 700
p-elements	1-Point; P=1,1,1 5-Point; P=2,2,2 P+1 Cubic Point; P>2	0 or 1	Not applicable	0	No	No	No
	5-Point; P=1,1,1 P+1 Cubic for all other			1			
	P+ISOP Cubic			$-10 \le ISOP \le -1$ or $2 = ISOP \le 10$			

**Requires PARAM,MRALIAS

PSOLIDD (SOL 700) Additional Property Specification Data for Solid Elements

Additional (MSC.Dytran-lsdyna specific) property specification information may be provided using this entry when materials MATD010 or MATD015 are used.

Format:

1	2	3	4	5	6	7	8	9	10
PSOLIDD	PID	MID	ELFORM	EOSID					

Example:

PSOLIDD	123	12	1	123					

Field	Contents
PID	Property ID. PID is referenced on the CHEXA, CPENTA, or CTET entry and must be unique. (Integer, no Default)
MID	Material ID. (Integer, Default = 1)
ELFORM	Element formulation options: 0 One point corotational for MATD126. See Remark 2. 1 Constant stress solid element (Default) 2 Fully integrated S/R solid. See Remark 3. 3 Fully integrated quadratic 8 node element with nodal rotations. 4 S/R quadratic tetrahedron element with nodal rotations. 9 1 point corotational for MATD126. See Remark 2. 10 1 point tetrahedron. 13 1 point nodal pressure tetrahedron for bulk forming. 15 2 point pentahedron element. 16 5 point 10 noded tetrahedron. 18 8 point enhanced strain solid element for linear statics only.
EOSID	Equation of State ID. (Integer, Default = PID)

Remarks:

1. The keyword PARAM,LSDYNA,SOLID activates automatic sorting of tetrahedron and pentahedron elements into type 10 and 15 element formulation, respectively. These latter elements are far more stable than the degenerate solid element. The sorting is performed internally and is transparent to the user.
2. Element formulation 0 and 9, applicable only to DMAT126, behave essentially as nonlinear springs so as to permit severe distortions sometimes seen in honeycomb materials. In formulation 0, the local coordinate system follows the element rotation whereas in formulation 9, the local coordinate system is based on axes passing through the centroids of the element faces. Formulation 0 is preferred for severe shear deformation where the barrier is fixed in space. If the barrier is attached to a moving body, which can rotate, then formulation 9 is usually preferred.
3. The selective reduce deintegrated solid element, element type 2, assumes that pressure is constant throughout the element to avoid pressure locking during nearly incompressible flow. However, if the element aspect ratios are poor, shear locking will lead to an excessively stiff response. A better choice, given poor aspect ratios, is the one point solid element which work well for implicit and explicit calculations. For linear statics, the type 18 enhanced strain element works well with poor aspect ratios. Please note that highly distorted elements should always be avoided since excessive stiffness will still be observed even in the enhanced strain formulations.
4. The equation of state can be used for the following materials: MATD008, MATD009, MATD010, MATD011, MATD015, MATD016, MATD017, MATD065, MATD072, and MATD088.

Note: This form should be used only by MATD010 and MATD015.
ELFORM is not currently supported.
By default, EOSID will be the same as PID.

PSPRMAT (SOL 700)

Defines spring and damper elements for translation and rotation. These definitions must correspond with the material type selection for the elements.

Format:

1	2	3	4	5	6	7	8	9	10
PSPRMAT	PID	MID	DRO	KD	V0	CL	FD		
	CDL	TDL							

Example:

PSPRMAT	1	1		2.0					
		.1							

Field	Contents	Type	Default
PID	Property ID. PID is referenced on the CSPR entry.	I > 0	Required
MID	Material ID.	I>0	Required
	Material types allowed are:		
	MATDS01 ➔ SPRING_ELASTIC		
	MATDS02 ➔ DAMPER_VISCOUS		
DRO	Displacement/Rotation Option:	I>0	0
	EQ.0: the material describes a translational spring/damper,		
	EQ.1: the material describes a torsional spring/damper.		
KD	Dynamic magnification factor. See Remarks **1.** and **2.** below.	R	0.0
V0	Test velocity. See Remark **3.** below.	R	0.0
CL	Clearance. See Remark **3.** below.	R	0.0
FD	Failure deflection (twist for DRO=1). Negative for compression, positive for tension.	R	0.0

Field	Contents	Type	Default
CDL	Deflection (twist for DRO=1) limit in compression. See Remark 4. below.	R	0.0
TDL	Deflection (twist for DRO=1) limit in tension. See Remark 4. below.	R	0.0

Remarks:

1. The constants from KD to TDL are optional and do not need to be defined.
2. If k_d is nonzero, the forces computed from the spring elements are assumed to be the static values and are scaled by an amplification factor to obtain the dynamic value:

$$F_{dynamic} = 1. + k_d \frac{V}{V_0} F_{static}$$

where

V = absolute value of the relative velocity between the nodes.

V0 = dynamic test velocity.

For example, if it is known that a component shows a dynamic crush force at 15m/s equal to 2.5 times the static crush force, use $k_d = 1.5$ and $V_0 = 15$.

3. Here, “clearance” defines a compressive displacement which the spring sustains before beginning the force-displacement relation given by the load curve defined in the material selection. If a non-zero clearance is defined, the spring is compressive only.
4. The deflection limit in compression and tension is restricted in its application to no more than one spring per node subject to this limit, and to deformable bodies only. For example in the former case, if three springs are in series, either the center spring or the two end springs may be subject to a limit, but not all three. When the limiting deflection is reached, momentum conservation calculations are performed and a common acceleration is computed in the appropriate direction. An error termination will occur if a rigid body node is used in a spring definition where deflection is limited.

   Constrained boundary conditions like SPC entries must not be used for nodes of springs with deflection limits.

PTUBE Tube Property

Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).

Format:

1	2	3	4	5	6	7	8	9	10
PTUBE	PID	MID	OD	T	NSM	OD2			

Example:

PTUBE	2	6	6.29	0.25					

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks **3.** and **4.** (Integer > 0)
OD	Outside diameter of tube. (Real > 0.0)
T	Thickness of tube. (Real; $T \leq OD / 2.0$)
NSM	Nonstructural mass per unit length. (Real)
OD2	Diameter of tube at second grid point (G2) on CTUBE entry. (Real; Default = OD)

Remarks:

1. If T is zero, a solid circular rod is assumed.
2. PTUBE entries must all have unique property identification numbers.
3. For structural problems, MID must reference a MAT1 material entry.
4. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
5. Tapered OD tubes with constant wall thickness are available for heat transfer only. The effective diameter is given by:

$$D_{effective} = T + \frac{D_2 - D_1}{log_e\left(\frac{D_2 - T}{D_1 - T}\right)}$$

where:

D_1 = OD

$$D_2 = \begin{cases} \text{OD2 if OD2} \neq 0 \\ \text{OD if OD2} = 0 \text{ or blank} \end{cases}$$

PVAL p-Version Element Polynomial Order Distribution

Describes polynomial order distribution and is selected by the ADAPT Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
PVAL	ID	POLY1	POLY2	POLY3	CID	SETTYP	ID		

Example:

PVAL	127	1	2	1					

Field	Contents	Type	Default
ID	ID selected in ADAPT Bulk Data entry.	Integer > 0	Required
CID	Coordinate system used to specify polynomial values in different directions. See Remark **1.**	Integer ≥ 0	Remark **2.**
POLYi	Polynomial order in 1, 2, 3 directions of the CID system.	Integer > 0	Remark **3.**
SETTYP	Type of set provided (SET or ELID). See Remark **6.**	Character	“SET”
ID	SET ID or Element ID with these p value specifications. See Remark **6.**	Integer > 0	999999

Remarks:

1. CID facilitates the specification of the p-order in curvilinear systems. For example, when modeling a thin cylinder, the user can restrict the p-order through the thickness of all elements to be 2 or 3 without specifically checking the connectivity of each element.
2. If the CID system is blank, the element’s topology is used to establish the 1, 2, 3 directions. The 1 direction is from the first to the second grid of the element, the 2 direction is from the first to the fourth, and, the 3 direction is from the first to the fifth. If CID is not blank then the following algorithm will

be used to determine the p-order of each edge: a vector will be defined in the CID system from the first to the second grid of every edge. (Curvilinear systems are evaluated at the mid point of this vector). The p-level of each edge is now determined by the nearest integer to:

$$p = \sqrt{(n_1 \cdot \text{POLY1})^2 + (n_2 \cdot \text{POLY2})^2 + (n_3 \cdot \text{POLY3})^2}$$

where (n_1, n_2, n_3) are the components of this unit vector in the CID system.

3. For accuracy and efficiency the recommended minimum p-order is 3. The default value for POLY2 and POLY3 is POLY1.
4. Each finite element has to have a unique PVAL for PSTRTID, PMINID, PMAXID. Any overlap of the PVAL specification will result in a warning message and the use of the PVAL with the highest p_i entry.
5. The intermediate PVAL entries generated will have an identification number starting with ADGEN.
6. Whenever SETTYP = “SET”, a SET command must be defined under the SETS DEFINITION command in the Case Control Section.
7. SET = 999999 is a reserved set that includes all elements.
8. If there are more than one PVAL entries for a given element, then
    - If CID on the PVALs are the same, the entry with the maximum POLYi will be used.
    - If CID on the PVALs are different, a fatal message is issued.

PVISC Viscous Damping Element Property

Defines properties of a one-dimensional viscous damping element (CVISC entry).

Format:

1	2	3	4	5	6	7	8	9	10
PVISC	PID1	CE1	CR1		PID2	CE2	CR2		

Example:

PVISC	3	6.2	3.94						

Field	Contents
PIDi	Property identification number. (Integer > 0)
CE1, CE2	Viscous damping values for extension in units of force per unit velocity. (Real)
CR1, CR2	Viscous damping values for rotation in units of moment per unit velocity. (Real)

Remarks:

1. Viscous properties are material independent; in particular, they are temperature independent.
2. One or two viscous element properties may be defined on a single entry.

PWELD Connector Element Property

Defines the property of connector (CWELD) elements.

Format:

1	2	3	4	5	6	7	8	9	10
PWELD	PID	MID	D			MSET		TYPE	
	LDMIN	LDMAX							

Example:

PWELD	100	3	1.0						

Field	Contents	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 1.	Integer > 0	Required
D	Diameter of the connector. See Remark 1.	Real > 0	Required
MSET	Flag to eliminate m-set degrees-of-freedom (DOFs) =OFF m-set DOFs are eliminated, constraints are incorporated in the stiffness, see Remark 2. =ON m-set DOFs are not eliminated, constraints are generated.	Character	OFF
TYPE	Character string indicating the type of connection, see Remark 3. =blank general connector =”SPOT” spot weld connector	Character	Blank
LDMIN	Smallest ratio of length to diameter for stiffness calculation, see Remark 4.	Real or blank	0.2
LDMAX	Largest ratio of length to diameter for stiffness calculation.	Real or blank	5.0

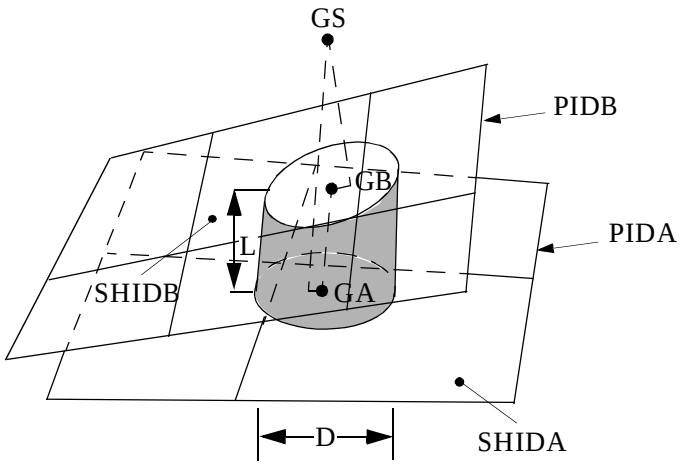


Figure 8-152 Length and Diameter of the CWELD Connector

Remarks:

1. The material MID, the diameter D, and the length are used to calculate the stiffness of the connector in 6 directions. MID can only refer to the MAT1 Bulk Data entry. The length is the distance of GA to GB (see **Figure 8-152**).
2. The parameter MSET is active only for the formats ELEMID and GRIDID (see "**CWELD**" on page 1350 for the format descriptions). MSET="OFF" incorporates constraints at the element stiffness matrix level avoiding explicit m-set constraint equations. For the formats PARTPAT and ELPAT, constraints are always eliminated on the element level. MSET="ON" generates explicit m-set constraints. For example, if a patch-to-patch connection is specified with the formats "GRIDID" or "ELEMID" on the CWELD entry, and MSET=ON is specified, 2x6 explicit constraints are generated that connect the 6 degrees-of-freedom of GA to the translational degrees-of-freedom of grid points GAi and the 6 degrees-of-freedom of GB to GBi. The 2x6 degrees-of-freedom of GA and GB are put into the m-set. The constraints are labeled "RWELD". The identification numbers of the generated RWELD constraint elements start with an offset of 100,001,001 by default. The offset number can be changed with PARAM, OSWELM. For MSET="OFF" or blank, the 2x6 constraint equations are built into the stiffness matrix of the CWELD element, thereby condensating the 2x6 degrees-of-freedom of GA and GB.

3. If TYPE=”SPOT” and if the formats PARTPAT, ELPAT, or ELEMID on the CWELD entry are used, then the effective length for the stiffness of the CWELD element is set to $L_e = 1/2 \cdot (t_A + t_B)$ regardless of the distance GA to GB. t_A and t_B are the shell thicknesses of shell A and B, respectively. The effective length is used to avoid excessively stiff or soft connections due to mesh irregularities.
4. If TYPE=blank, the effective length L_e of the CWELD is equal to the true length L, the distance of GA to GB, as long as the ratio of the length L to diameter D is in the range LDMIN $\leq$ L / D $\leq$ LDMAX. If L is below the range, the effective length is set to $L_e = \text{LDMIN} \cdot \text{D}$ and if L is above the range, the effective length is set to $L_e = \text{LDMAX} \cdot \text{D}$.

PWSEAM Seam Connector Element Property

Defines the property of seam connector (CWSEAM) elements.

Format:

1	2	3	4	5	6	7	8	9	10
PWSEAM	PID	MID	W	T					

Example:

PWSEAM	7	1	16.						

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
W	Width of the seam. See Remark **1.** (Real > 0.)
T	Thickness of the seam. See Remark **2.** (Real > 0. or blank)

Remarks:

1. The length of the seam is the distance between GS and GE. The width of the seam is measured perpendicular to the length and lies in the plane of the patches A and B (see **Figure 8-153**). The width is also used to find the projection of the seam on the two patches A and B.

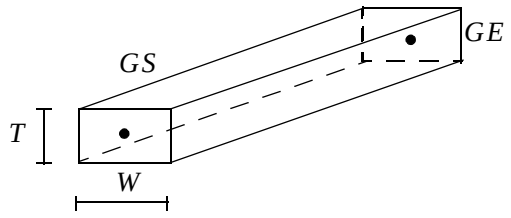


Figure 8-153 Dimensions of a CWSEAM Element

2. If left blank, the thickness will be computed as $T = T_A + T_B/2$ where T_A is the thickness of patch A and T_B is the thickness of patch B.

QBDY1 Boundary Heat Flux Load for CHBDYj Elements, Form 1

Defines a uniform heat flux into CHBDYj elements.

Format:

1	2	3	4	5	6	7	8	9	10
QBDY1	SID	Q0	EID1	EID2	EID3	EID4	EID5	EID6	

Example:

QBDY1	109	1.-5	721						

Alternate Format and Example:

QBDY1	SID	Q0	EID1	“THRU”	EID2				
QBDY1	109	1.-5	725	THRU	735				

Field	Contents
SID	Load set identification number. (Integer > 0)
Q0	Heat flux into element. (Real)
EIDi	CHBDYj element identification numbers. (Integer ≠ 0 or “THRU”. For “THRU” option EID2 > EID1.)

Remarks:

1. QBDY1 entries must be selected with the Case Control command LOAD = SID in order to be used in static analysis. The total power into an element is given by the equation:

$$_{in} = (\text{Effective area}) \cdot Q0$$

2. QBDY1 entries must be referenced on a TLOADi Bulk Data entry through the EXCITEID specification for use in transient analysis. The total power into an element is given by the equation:

$$P_{in}(t) = (\text{Effective area}) \cdot Q0 \cdot F(t - \tau)$$

where the function of time $F(t - \tau)$ is specified on a TLOADi entry.

3. The sign convention for Q0 is positive for heat input.

QBDY2 Boundary Heat Flux Load for CHBDYj Elements, Form 2

Defines grid point heat flux into CHBDYj elements.

Format:

1	2	3	4	5	6	7	8	9	10
QBDY2	SID	EID	Q01	Q02	Q03	Q04	Q05	Q06	
	Q07	Q08							

Example:

QBDY2	109	721	1.-5	1.-5	2.-5	2.-5			

Field	Contents
SID	Load set identification number. (Integer > 0)
EID	Identification number of an CHBDYj element. (Integer > 0)
Q0i	Heat flux at the i-th grid point on the referenced CHBDYj element. (Real or blank)

Remarks:

1. QBDY2 entries must be selected with the Case Control command LOAD=SID in order to be used in static analysis. The total power into each point i on an element is given by

$$P_i = AREA_i \cdot Q0i$$

2. QBDY2 entries must be referenced on a TLOADi Bulk Data entry through the EXCITEID specification for use in transient analysis. All connected grid points will have the same time function but may have individual delays. The total power into each point i on an element is given by

$$P_i(t) = AREA_i \cdot Q0i \cdot F(t - \tau_i)$$

where $F(t-\tau_i)$ is a function of time specified on a TLOADi entry.

3. The sign convention for Q0i is positive for heat flux input to the element.

QBDY3 Boundary Heat Flux Load for a Surface

Defines a uniform heat flux load for a boundary surface.

Format:

1	2	3	4	5	6	7	8	9	10
QBDY3	SID	Q0	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

Example:

QBDY3	2	20.0	10	1	THRU	50	BY	2	

Field	Contents
SID	Load set identification number. (Integer > 0)
Q0	Thermal heat flux load, or load multiplier. Q0 is positive for heat flow into a surface. (Real)
CNTRLND	Control point for thermal flux load. (Integer $\geq$ 0; Default = 0)
EIDi	CHBDYj element identification numbers. (Integer $\neq$ 0 or “THRU” or “BY”)

Remarks:

1. QBDY3 entries must be selected in Case Control (LOAD = SID) to be used in steady state. The total power into a surface is given by the equation:

   - if CNTRLND ≤ 0 then P_{in} = (Effective area) $\cdot$ $Q0$
   - if CNTRLND > 0 then P_{in} = (Effective area) $\cdot$ $Q0 \cdot u_{\text{CNTRLND}}$

     where u_{CNTRLND} is the temperature of the control point and is used as a load multiplier.

2. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time $F(t - \tau)$ defined on the TLOADi multiplies the general load, with τ specifying time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.

3. The CNTRLND multiplier cannot be used with any higher-order elements.
4. When using “THRU” or “BY”, all intermediate CHBDYE, CHBDYG, or CHBDYP elements must exist.

QHBDY Boundary Heat Flux Load

Defines a uniform heat flux into a set of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
QHBDY	SID	FLAG	Q0	AF	G1	G2	G3	G4	
	G5	G6	G7	G8					

Example:

QHBDY	2	AREA4	20.0		101	102	104	103	

Field	Contents
SID	Load set identification number. (Integer > 0)
FLAG	Type of face involved (must be one of the following: “POINT”, “LINE”, “REV”, “AREA3", “AREA4", “AREA6", “AREA8")
Q0	Magnitude of thermal flux into face. Q0 is positive for heat into the surface. (Real)
AF	Area factor depends on type. (Real > 0.0 or blank)
Gi	Grid point identification of connected grid points. (Integer > 0 or blank)

Remarks:

1. The continuation entry is optional.
2. For use in steady state analysis, the load set is selected in the Case Control Section (LOAD = SID).
3. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time $F(t - \tau)$ defined on the TLOADi entry multiplies the general load. τ specifies time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
4. The heat flux applied to the area is transformed to loads on the points. These points need not correspond to an HBDY surface element.

5. The flux is applied to each point i by the equation

$$P_i = AREA_i \cdot Q0$$

6. The number of connected points for the types are 1 (POINT), 2 (LINE, REV), 3 (AREA3), 4 (AREA4), 4-6 (AREA6), 5-8 (AREA8).
7. The area factor AF is used to determine the effective area for the POINT and LINE types. It equals the area and effective width, respectively. It is not used for the other types, which have their area defined implicitly and must be left blank.
8. The type of face (FLAG) defines a surface in the same manner as the CHBDYi data entry. For physical descriptions of the geometry involved, see the CHBDYG discussion.

QSET Generalized Degree-of-Freedom

Defines generalized degrees-of-freedom (q-set) to be used for dynamic reduction or component mode synthesis.

Format:

1	2	3	4	5	6	7	8	9	10
QSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

QSET	15	123456	1	3	9	2	105	6	

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
2. Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
3. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).

QSET1 Generalized Degree-of-Freedom (Alternate Form of QSET Entry)

Defines generalized degrees-of-freedom (q-set) to be used for generalized dynamic reduction or component mode synthesis.

Format:

1	2	3	4	5	6	7	8	9	10
QSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Example:

QSET1	123456	1	7	9	22	105	6	22	
	52	53							

Alternate Format and Example:

QSET1	C	ID1	“THRU”	ID2					
QSET1	0	101	THRU	110					

Field	Contents
C	Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2.)

Remarks:

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.
2. Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
3. When ASET, ASET1, QSET, and / or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).

QVECT Thermal Vector Flux Load

Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.

Format:

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q0	TSOUR	CE	E1 or TID1	E2 or TID2	E3 or TID3	CNTRLND	
	EID1	EID2	-etc.-						

Example:

QVECT	10	20.0	1000.0		1.0	1.0	1.0	101	
	20	21	22	23					

Field	Contents
SID	Load set identification number. (Integer > 0)
Q0	Magnitude of thermal flux vector into face. (Real or blank)
TSOUR	Temperature of the radiant source. (Real or blank)
CE	Coordinate system identification number for thermal vector flux. (Integer $\geq$ 0 or blank)
Ei	Vector components (direction cosines in coordinate system CE) of the thermal vector flux. (Real; Default = 0.0)
TIDi	TABLEDi entry identification numbers defining the components as a function of time. (Integer > 0)
CNTRLND	Control point. (Integer $\geq$ 0; Default = 0)
EIDi	Element identification number of a CHBDYE, CHBDYG, or CHBDYP entry. (Integer $\neq$ 0 or “THRU”)

Remarks:

1. The continuation entry is required.

2. If the coordinate system CE is not rectangular, then the thermal vector flux is in different directions for different CHBDYi elements. The direction of the thermal vector flux over an element is aligned to be in the direction of the flux vector at the geometric center of the element. The geometric center is measured using the grid points and includes any DISLIN specification on the VIEW entry for TYPE=LINE CHBDYi elements. The flux is presumed to be uniform over the face of each element; i.e., the source is relatively distant.
3. For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD = SID). The total power into an element is given by:
   - If CNTRLND = 0 then, $P_{in} = -\alpha A(\vec{e} \cdot \vec{n}) \cdot Q0$.
   - If CNTRLND > 0 then, $P_{in} = -\alpha A(\vec{e} \cdot \vec{n}) \cdot Q0 \cdot u_{\text{CNTRLND}}$.

   where

   α = face absorptivity (supplied from a RADM statement).

   A = face area as determined from a CHBDYi surface element.

   $\vec{e}$ = vector of direction cosines E1, E2, E3.

   $\vec{n}$ = face normal vector. See CHBDYi entries.

   $\vec{e} \cdot \vec{n}$ = 0 if the vector product is positive, (i.e., the flux is coming from behind the face).

   u_{cntrlnd} = temperature value of the control point used as a load multiplier.

4. If the absorptivity is constant, its value is supplied by the ABSORP field on the RADM entry. If the absorptivity is not a constant, the thermal flux is assumed to have a wavelength distribution of a black body at the temperature TSOUR.
   - For a temperature-dependent absorptivity, the element temperature is used to determine α.
   - For a wavelength-dependent absorptivity, the integration of the flux times α is computed for each wavelength band. The sum of the integrated thermal fluxes over all the wavelength bands is Q0. The wave bands are specified with the RADBND entry.
   - The user has the responsibility of enforcing Kirchhoff's laws.

5. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time $F(t-\tau)$ defined on the TLOADi entry multiplies the general load. τ provides any required time delay. $F(t-\tau)$ is a function of time specified on the TLOADi entry. The value of is calculated for each loaded grid point. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.

   The total power into an element is given by:

   - If CNTRLND = 0 then, $P_{in} = -\alpha A(\vec{e}(t) \cdot \vec{n}) \cdot Q0 \cdot F(t-\tau)$.
   - If CNTRLND > 0 then,

     $_{in} = -\alpha A(\vec{e}(t) \cdot \vec{n}) \cdot F(t-\tau) \cdot Q0 \cdot u_{\text{CNTRLND}}$.

6. If the referenced face is of TYPE = ELCYL, the power input is an exact integration over the area exposed to the thermal flux vector.
7. If the referenced face is of TYPE = REV, the thermal flux vector must be parallel to the axis of symmetry if an axisymmetric boundary condition is to be maintained.
8. When applied to a surface element associated with a radiation enclosure cavity, any incident energy that is not absorbed ($\alpha < 1.0$) is lost from the system and is not accounted for in a reflective sense ($\alpha + \rho = 1.0$).

QVOL Volume Heat Addition

Defines a rate of volumetric heat addition in a conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
QVOL	SID	QVOL	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

Example:

QVOL	5	10.0	101	10	12	11	9		

Field	Contents
SID	Load set identification. (Integer > 0)
QVOL	Power input per unit volume produced by a heat conduction element. (Real)
CNTRLND	Control point used for controlling heat generation. (Integer ≥ 0; Default = 0)
EIDi	A list of heat conduction elements. (Integer > 0 or “THRU” or “BY”)

Remarks:

1. EIDi has material properties (MAT4) that include HGEN, the element material property for heat generation, which may be temperature dependent. This association is made through the element EID. If HGEN is temperature dependent, it is based on the average element temperature.
2. QVOL provides either the constant volumetric heat generation rate or the load multiplier. QVOL is positive for heat generation. For steady-state analysis, the total power into an element is
   - If CNTRLND = 0, then P_{in} = volume · HGEN · QVOL.
   - If CNTRLND > 0, then P_{in} = volume · HGEN · QVOL · u_{CNTRLND}.

   where u_{CNTRLND} is the temperature multiplier.
3. For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD = SID).

4. In transient analysis SID is referenced by a TLOADi Bulk Data entry. A function of time $F[t - \tau]$ defined on the TLOADi entry multiplies the general load where τ specifies time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
5. For “THRU” or “BY”, all intermediate referenced heat conduction elements must exist.
6. The CNTRLND multiplier cannot be used with any higher-order elements.

RADBC Space Radiation Specification

Specifies an CHBDYi element face for application of radiation boundary conditions.

Format:

1	2	3	4	5	6	7	8	9	10
RADBC	NODAMB	FAMB	CNTRLND	EID1	EID2	EID3	-etc.-		

Example:

RADBC	5	1.0	101	10					

Field	Contents
NODAMB	Ambient point for radiation exchange. (Integer > 0)
FAMB	Radiation view factor between the face and the ambient point. (Real $\geq$ 0.0)
CNTRLND	Control point for radiation boundary condition. (Integer $\geq$ 0; Default = 0)
EIDi	CHBDYi element identification number. (Integer $\neq$ 0 or "THRU" or "BY")

Remarks:

1. The basic exchange relationship is:
   - if CNTRLND = 0, then $q = \sigma \cdot \text{FAMB} \cdot \varepsilon_e \cdot (T_e^4 - T_{amb}^4)$
   - if CNTRLND > 0, then

     $q = \sigma \cdot \text{FAMB} \cdot u_{\text{CNTRLND}} \cdot \varepsilon_e \cdot (T_e^4 - T_{amb}^4)$
2. NODAMB is treated as a black body with its own ambient temperature for radiation exchange between the surface element and space. No surface element that is a member of a radiation enclosure cavity may also have a radiation boundary condition applied to it.
3. Two PARAM entries are required when stipulating radiation heat transfer:
   - ABS defines the absolute temperature scale; this value is added internally to any specified temperature given in the problem. Upon solution completion, this value is subtracted internally from the solution vector.
   - SIGMA (σ) is the Stefan-Boltzmann constant.

4. RADBC allows for surface radiation to space in the absence of any cavity behavior. The emissivity is supplied from a RADM entry.
5. When using “THRU” or “BY”, all intermediate referenced CHBDYi surface elements must exist.

RADBND Radiation Wavelength Band Definition

Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.

Format:

1	2	3	4	5	6	7	8	9	10
RADBND	NUMBER	PLANCK2	LAMBDA1	LAMBDA2	LAMBDA3	LAMBDA4	LAMBDA5	LAMBDA6	
	LAMBDA7	etc.							

Example:

RADBND	6	14388.0	1.0	2.0	4.0	8.0	12.0		

Field	Contents
NUMBER	Number of radiation wave bands. See Remarks. (Integer > 1)
PLANCK2	Planck's second radiation constant. See Remarks. (Real > 0.0)
LAMBDAi	Highest wavelength of the i-th wave band. See Remarks. (Real $\geq$ 0.0)

Remarks:

1. Only one RADBND entry may be specified in the Bulk Data Section and must always be used in conjunction with the RADM entry.
2. PLANCK2 has the units of wavelength times temperature. The same units of length must be used for LAMBDAi as for PLANCK2. The units of temperature must be the same as those used for the radiating surfaces. For example: 25898.μm °R or 14388.μm °K.
3. The first wavelength band extends from 0 to LAMBDA1 and the last band extends from LAMBDAn to infinity, where n = NUMBER – 1.
4. Discontinuous segments in the emissivity versus wavelength piecewise linear curve must be treated as a wavelength band of zero width.
5. LAMBDAi must be specified in ascending order, and all LAMBDAi fields where i is greater than or equal to NUMBER must be blank.

RADCAV Radiation Cavity Identification

Identifies the characteristics of each radiant enclosure.

Format:

1	2	3	4	5	6	7	8	9	10
RADCAV	ICAVITY	ELEAMB	SHADOW	SCALE	PRTPCH	NFECI	RMAX	NCOMP	
	SET11	SET12	SET21	SET22	SET31	SET32	etc.		

Example:

RADCAV	1	1					.99		
	3	5	4	5	7	5			

Field	Contents
ICAVITY	Unique cavity identification number associated with enclosure radiation. (Integer > 0)
ELEAMB	CHBDYi surface element identification number for radiation if the view factors add up to less than 1.0. (Unique Integer > 0 among all CHBDYi elements or blank.)
SHADOW	Flag to control third body shading calculation during view factor calculation for each identified cavity. (Character = “YES” or “NO”; Default = “YES”)
SCALE	View factor that the enclosure sum will be set to if a view factor is greater than 1.0. ($0.0 \leq$ Real ≤ 1.0; Default = 0.0)
PRTPCH	Facilitates the blocking of view factor printing and punching onto RADLST and RADMTX entries. (Integer = 0, 1, 2, or 3; Default = 0):

PRTPCH	Print/Punch
0 (default)	print and punch
1	no print
2	no punch
3	no print or punch

Field	Contents
NFECI	Controls whether finite difference or contour integration methods are to be used in the calculation of view factors in the absence of a VIEW3D Bulk Data entry. (Character = “FD” or “CONT”; See Remark **4.** for default.)
RMAX	Subelement area factor. See Remark **5.** (Real ≥ 0.0; Default = 0.1)
NCOMP	Total number of computational element for one-half ring. See Remark **8.** (Default = 32)
SETij	Set identification pairs for the calculation of global view factors. Up to 30 pairs may be specified (i = 1 to 2 and j = 1 to 30). (Integer > 0)

Remarks:

1. For the surfaces of an incomplete enclosure (view factors add up to less than 1.0), a complete enclosure may be achieved (SUM = 1.0) by specifying an ambient element, ELEAMB. When multiple cavities are defined, each cavity must have a unique ambient element if ambient elements are desired. No elements can be shared between cavities.
2. Third-body shadowing is ignored in the cavity if SHADOW = “NO”. In particular, if it is known a priori that there is no third-body shadowing, SHADOW = NO overrides KSHD and KBSHD fields on the VIEW Bulk Data entry as well as reduces the calculation time immensely.
3. The view factors for a complete enclosure may add up to slightly more than 1.0 due to calculation inaccuracies. SCALE can be used to adjust all the view factors proportionately to acquire a summation equal to the value specified for SCALE. If SCALE is left blank or set to 0.0, no scaling is performed.
4. If the VIEW3D Bulk Data entry is not specified, the view factors are calculated using finite difference and contour integration methods. If NFECI = “FD”, then all view factors are calculated using the finite difference technique. NFECI = “CONT” invokes contour integration for all view factor calculations. If NFECI is blank, the program selects a method to use between any two particular elements based on RMAX.
5. The comparison value for RMAX is equal to A_s/d_{rs}^2 where A_s is the area of a subelement and d_{rs} is the distance between two subelements r and s for which view factors are being computed. When NFECI is blank, the program selects the contour integral method only if $A_s/d_{rs}^2 > \text{RMAX}$.

6. When a number of elements are grouped together and considered as a conglomerate surface, view factors can be calculated between these groups. These are referred to as global view factors. The SET1 Bulk Data entry is used to define the conglomerate. When using this feature, negative EIDs are not allowed.
7. If a RADLST and RADMTX entry exists for this cavity ID, new view factors are not computed and the existing RADLST and RADMTX are used in the thermal analysis.
8. The VIEW3D Bulk Data entry must be specified for the calculation of axisymmetric view factors. The process relies on the internal construction of a semi-circle of computational elements. NCOMP specifies the number of such elements desired.

RADLST Listing of Enclosure Radiation Faces

Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.

Format:

1	2	3	4	5	6	7	8	9	10
RADLST	ICAVITY	MTXTYP	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	-etc.-							

Example:

RADLST	2	1	2	3	4	5	6	7	

Field	Contents
ICAVITY	Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation faces. (Integer > 0)
MTXTYP	Type of radiation exchange matrix used for this cavity. (Integer ≤ 4 and $\neq 0$; Default = 1 for an enclosure without an ambient element. Default = 4 for an enclosure with an ambient element as specified on the RADCAV entry.) 1: Symmetric view factor matrix [F] and nonconservative radiation matrix [R]. 2: Symmetric exchange factor matrix [$\Im$] and conservative radiation matrix [R]. 3: Unsymmetric exchange factor matrix [$\Im$] and conservative radiation matrix [R]. 4: Symmetric view factor matrix [F] and conservative radiation matrix [R]. -n: The first n CHBDYi elements may lose energy to space but the remainder may not. Symmetric exchange factor matrix [F] and nonconservative radiation matrix [R].
EIDi	Identification numbers for the CHBDYi elements in this radiation cavity. (Integer $\neq$ 0 or “THRU”)

Remarks:

1. A radiation EIDi list isolates those CHBDYi surface element faces that are to communicate in a radiation enclosure. View-factor calculation and RADMTX formation for an enclosure is performed only for (or among) those faces identified within the same RADCAV.
2. A radiation exchange matrix (RADMTX) can only reference one radiative face list (RADLST). The companion RADCAV, RADLST, and RADMTX must share a unique ICAVITY.
3. For each EIDi, the appropriate CHBDYi element is located, and the proper RADM entry ID field found.
4. If the radiation exchange matrix or any radiation boundary conditions are available from an external source, the RADMTX must be user generated.
5. Multiple RADLST entries may be specified.
6. If any RADLST entry is changed or added on restart then a complete re-analysis may be performed. Therefore, RADLST entry changes or additions are not recommended on restart.

RADM Radiation Boundary Material Property

Defines the radiation properties of a boundary element for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RADM	RADMID	ABSORP	EMIS1	EMIS2	EMIS3	EMIS4	EMIS5	EMIS6	
	EMIS7	-etc.-							

Example:

RADM	11		.45	.33	.29	.20	.17	.13	

Field	Contents
RADMID	Material identification number. (Integer > 0)
ABSORP	Surface absorptivity or the temperature function curve multiplier if ABSORP is variable. See Remark 2. ($0.0 \leq$ Real ≤ 1.0)
EMISi	Surface emissivity at wavelength LAMBDAi or the temperature function curve multiplier if EMISi is variable (See the RADBND entry.) ($0.0 \leq$ Real ≤ 1.0)

Remarks:

1. The RADM entry is directly referenced only by one of the CHBDYE, CHBDYG, or CHBDYP type surface element entries.
2. For radiation enclosure problems, ABSORP is set equal to emissivity. For QVECT loads, absorptivity is specified by ABSORP.
3. If there is more than one EMISi, then:
   - There must be a RADBND entry.
   - The number of EMISi may not exceed the number of LAMBDAi on the RADBND entry.
   - The emissivity values are given for a wavelength specified by the corresponding LAMBDAi on the RADBND entry. Within each discrete wavelength band, the emissivity is assumed to be constant.
   - At any specific wavelength and surface temperature, the absorptivity is exactly equal to the emissivity.

4. To perform any radiation heat transfer exchange, the user must furnish PARAM entries for:
    - TABS to define the absolute temperature scale.
    - SIGMA (σ) to define the Stefan-Boltzmann constant in appropriate units.

RADMT Radiation Boundary Material Property Temperature Dependence

Specifies table references for temperature dependent RADM entry radiation boundary properties.

Format:

1	2	3	4	5	6	7	8	9	10
RADMT	RADMID	T(A)	$T(\varepsilon_1)$	$T(\varepsilon_2)$	$T(\varepsilon_3)$	$T(\varepsilon_4)$	$T(\varepsilon_5)$	$T(\varepsilon_6)$	
	$T(\varepsilon_7)$	-etc.-							

Example:

RADMT	11		1	2	3	4	5	6	

Field	Contents
RADMID	Material identification number. (Integer > 0)
T(A)	TABLEMj identifier for surface absorptivity. (Integer $\geq$ 0 or blank)
$T(\varepsilon_i)$	TABLEMj identifiers for surface emissivity. (Integer $\geq$ 0 or blank)

Remarks:

1. The basic quantities on the RADM entry of the same RADMID are always multiplied by the corresponding tabular function.
2. Tables T(A) and $T(\varepsilon_i)$ have an upper bound that is less than or equal to one and a lower bound that is greater than or equal to zero.
3. The TABLEMj enforces the element temperature as the independent variable. Blank or zero fields means there is no temperature dependence of the referenced property on the RADM entry.

RADMTX Radiation Exchange Matrix

Provides the $F_{ji} = A_j f_{ji}$ exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry.

Format:

1	2	3	4	5	6	7	8	9	10
RADMTX	ICAVITY	INDEX	Fi,j	Fi+1,j	Fi+2,j	Fi+3,j	Fi+4,j	Fi+5,j	
	Fi+6,j	-etc.-							

Example:

RADMTX	2	1	0.0	0.1	0.2	0.2	0.3	0.2	

Field	Contents
ICAVITY	Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation surface elements. (Integer > 0)
INDEX	Column number in the matrix. (Integer > 0)
$F_{k,j}$	If symmetric, the matrix values start on the diagonal (i = j) and continue down the column (k = i + 1, i + 2, etc.). If unsymmetric, the values start in row (i = 1). i refers to EIDi on the RADLST entry. (Real $\geq$ 0)

Remarks:

1. If the matrix is symmetric, only the lower triangle is input, and i = j = INDEX. If the matrix is unsymmetric, i = 1, and j = INDEX.
2. Only one ICAVITY may be referenced for those faces that are to be included in a unique radiation matrix.
3. Coefficients are listed by column with the number of columns equal to the number of entries in the RADLST.
4. All faces involved in any radiation enclosure must be defined with an CHBDYi element.
5. If any RADMTX entry is changed or added on restart then a complete re-analysis may be performed. Therefore, RADMTX entry changes or additions are not recommended on restart.
6. Set NASTRAN SYSTEM (87) = 3 is a new option in MSC.Nastran 2005 that prevents radiation energy from being lost to space.

RADSET Identifies a Set of Radiation Cavities

Specifies which radiation cavities are to be included for radiation enclosure analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RADSET	ICAVITY1	ICAVITY2	ICAVITY3	ICAVITY4	ICAVITY5	ICAVITY6	ICAVITY7	ICAVITY8	
	ICAVITY9	-etc.-							

Example:

RADSET	1	2	3	4					

Field	Contents
ICAVITYi	Unique identification number for a cavity to be considered for enclosure radiation analysis. (Integer > 0)

Remark:

1. For multiple radiation cavities, RADSET specifies which cavities are to be included in the analysis.

RANDPS Power Spectral Density Specification

Defines load set power spectral density factors for use in random analysis having the frequency dependent form

$$S_{jk}(F) = (X + iY)G(F)$$

Format:

1	2	3	4	5	6	7	8	9	10
RANDPS	SID	J	K	X	Y	TID			

Example:

RANDPS	5	3	7	2.0	2.5	4			

Field	Contents
SID	Random analysis set identification number. (Integer > 0)
J	Subcase identification number of the excited load set. (Integer > 0)
K	Subcase identification number of the applied load set. (Integer $\geq$ 0; K $\geq$ J)
X, Y	Components of the complex number. (Real)
TID	Identification number of a TABRNDi entry that defines $G(F)$. (Integer $\geq$ 0)

Remarks:

1. Set identification numbers must be selected with the Case Control command (RANDOM = SID).
2. For auto spectral density, J = K, X must be greater than zero and Y must be equal to zero.
3. For TID = 0, $G(F) = 1.0$.
4. RANDPS may only reference subcases included within a single loop (a change in direct matrix input is not allowed).
5. See the *MSC.Nastran Dynamics Users Guide* for a discussion of random analysis.
6. In the case of cyclic symmetry Solution Sequence 118, J and K must refer to the coded subcase IDs. See "**Additional Topics**" on page 555 of the *MSC.Nastran Reference Guide* for the coding procedure.

7. In superelement analysis, J and K must refer to the superelement subcases. For example, if superelement 10 has SUBCASEs 1 and 2 and superelement 20 has SUBCASEs 3 and 4, then a separate RANDPS entry is required for each superelement, even though X, Y, and TID may be identical.
8. For uncoupled PSDF (no J < K entries) only one J = K entry is allowed for unique value of J. For coupled PSDF (some J < K entries) any number of entries are allowed.

RANDT1 Autocorrelation Function Time Lag

Defines time lag constants for use in random analysis autocorrelation function calculation.

Format:

1	2	3	4	5	6	7	8	9	10
RANDT1	SID	N	T0	TMAX					

Example:

RANDT1	5	10	3.2	9.6					

Field	Contents
SID	Random analysis set identification number. (Integer > 0)
N	Number of time lag intervals. (Integer > 0)
T0	Starting time lag. (Real $\geq$ 0.0)
TMAX	Maximum time lag. (Real > T0)

Remarks:

1. Time lag sets must be selected with the Case Control command RANDOM = SID.
2. At least one RANDPS entry must be present with the same set identification number.
3. The time lags defined on this entry are given by

$$T_i = T0 + \frac{\text{TMAX} - T0}{N}(i - 1),\ i = 1,\ \text{N} + 2$$

4. See the *MSC.Nastran Dynamics Users Guide* for a discussion of random analysis.

RBAR Rigid Bar

Defines a rigid bar with six degrees-of-freedom at each end.

Format:

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CNA	CNB	CMA	CMB	ALPHA	

Example:

RBAR	5	1	2	123456				6.5-6	

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification number of connection points. (Integer > 0)
CNA, CNB	Component numbers of independent degrees-of-freedom in the global coordinate system for the element at grid points GA and GB. See Remark **3.** (Integers 1 through 6 with no embedded blanks, or zero or blank.)
CMA, CMB	Component numbers of dependent degrees-of-freedom in the global coordinate system assigned by the element at grid points GA and GB. See Remarks **4.** and **5.** (Integers 1 through 6 with no embedded blanks, or zero or blank.)
ALPHA	Thermal expansion coefficient. See Remark **11.** (Real $\geq$ 0.0 or blank)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, MD Nastran will create the Lagrange multiplier degrees-of-freedom internally in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.

3. For the linear method, the total number of components in CNA and CNB must equal six; for example, CNA = 1236, CNB = 34. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. For the Lagrange method, the total number of components must also be six. However, only CNA = 123456 or CNB = 123456 is allowed. If both CNA and CNB are blank, then CNA = 123456. For this method, RBAR1 gives the simpler input format.
4. If both CMA and CMB are zero or blank, all of the degrees-of-freedom not in CNA and CNB will be made dependent. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be member of the m-set, depending on the method selected in the RIGID Case Control command. However, the rules regarding the m-set described below apply to both methods.
5. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
6. Element identification numbers should be unique with respect to all other element identification numbers.
7. Rigid elements, unlike MPCs, are not selected through the Case Control Command, MPC.
8. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
9. Rigid elements are ignored in heat transfer problems.
10. See "**Rigid Elements and Multipoint Constraints (R-type, MPC)**" on page 167 of the *MSC.Nastran Reference Guide* for a discussion of rigid elements.
11. For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.

RBAR1 Rigid Bar

Alternative format for RBAR.

Format:

1	2	3	4	5	6	7	8	9	10
RBAR1	EID	GA	GB	CB	ALPHA				

Example:

RBAR1	5	1	2	123	6.5-6				

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB, which are constrained to move as the rigid bar. See Remark 4. (Integers 1 through 6 with no embedded blanks or blank.)
ALPHA	Thermal expansion coefficient. See Remark 8. (Real ≥ 0.0 or blank)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, MD Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CB.
3. RBAR1 is a preferred input format to define the Lagrange method for a rigid bar.
4. When CB = “123456” or blank, the grid point GB is constrained to move with GA as a rigid bar. For default CB = “123456”. Any number of degrees-of-freedom at grid point GB can be released not to move with the rigid body.
5. For the Lagrange method, the theory is formulated such that a consistent rigid body motion for grid points GA and GB will be computed even if these two points have different global coordinate systems.

6. For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.
7. Element identification numbers should be unique with respect to all other element identification numbers.
8. Rigid elements are ignored in heat transfer problems.

RBE1 Rigid Body Element, Form 1

Defines a rigid body connected to an arbitrary number of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RBE1	EID	GN1	CN1	GN2	CN2	GN3	CN3		
		GN4	CN4	GN5	CN5	GN6	CN6		
	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	-etc.-	ALPHA				

Example:

RBE1	59	59	123456						
	UM	61	246	6.5-6					

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GNi	Grid points at which independent degrees-of-freedom for the element are assigned. (Integer > 0)
CNi	Independent degrees-of-freedom in the global coordinate system for the rigid element at grid point(s) GNi. See Remark **1.** (Integers 1 through 6 with no embedded blanks.)
“UM”	Indicates the start of the dependent degrees-of-freedom. (Character)
GMj	Grid points at which dependent degrees-of-freedom are assigned. (Integer > 0)
CMj	Dependent degrees-of-freedom in the global coordinate system at grid point(s) GMj. (Integers 1 through 6 with no embedded blanks.)
ALPHA	Thermal expansion coefficient. See Remark **13.** (Real $\geq$ 0.0 or blank)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.

2. For the Lagrange method, MD Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CMj.
3. For the linear method, the total number of components in CN1 to CN6 must equal six; for example, CN1 = 123, CN2 = 3, CN3 = 2, CN4 = 3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element.The first continuation entry is not required if there are fewer than four GN points. For the Lagrange method, the total number of components must also be six. In addition, CNA must be 123456, and CN2 through CN6 must be blank.
4. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be member of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding to m-set described below apply to both types of methods.
5. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
6. A degree-of-freedom cannot be both independent and dependent for the same element. However, both independent and dependent components can exist at the same grid point.
7. Element identification numbers should be unique with respect to all other element identification numbers.
8. Rigid elements, unlike MPCs, are not selected through the Case Control command, MPC.
9. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
10. Rigid elements are ignored in heat transfer problems.
11. See "**Rigid Elements and Multipoint Constraints (R-type, MPC)**" on page 167 of the *MSC.Nastran Reference Guide* for a discussion of rigid elements.
12. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.

13. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the grid point GN1 and any dependent grid point are taken as the average temperature of the two connected grid points.

RBE2 Rigid Body Element, Form 2

Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	GM8	-etc.-	ALPHA				

Example:

RBE2	9	8	12	10	12	14	15	16	
	20	6.5-6							

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GN	Identification number of grid point to which all six independent degrees-of-freedom for the element are assigned. (Integer > 0)
CM	Component numbers of the dependent degrees-of-freedom in the global coordinate system at grid points GMi. (Integers 1 through 6 with no embedded blanks.)
GMi	Grid point identification numbers at which dependent degrees-of-freedom are assigned. (Integer > 0)
ALPHA	Thermal expansion coefficient. See Remark **11.** (Real $\geq$ 0.0 or blank)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, MD Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom which is obtained by CM multiplied with the number of dependent grid points.

3. For the linear method, the dependent degrees-of-freedom indicated by CM will be made members of the m-set at all grid points. For the Lagrange method, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
4. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
5. Element identification numbers should be unique with respect to all other element identification numbers.
6. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
8. Rigid elements are ignored in heat transfer problems.
9. See "**Rigid Elements and Multipoint Constraints (R-type, MPC)**" on page 167 of the *MSC.Nastran Reference Guide* for a discussion of rigid elements.
10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
11. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the grid point GN and any dependent grid point are taken as the average temperature of the two connected grid points.

RBE2A (SOL 700) Defines Extra Nodes for Rigid Body

Format:

1	2	3	4	5	6	7	8	9	10
RBE2A	EID	RIG							
	G1	G3	THRU	G4	BY	G5	G6	Gi	

Example:

RBE2A	25	R23							
	1	1001	THRU	2000	BY	2	5930		

Field	Contents
EID	Number of the nodal rigid-body. (Required, Integer > 0)
RIG	Rigid body ID. Use MR < ID > for MATRIG. (Required, Character)
Gi	Grid-point numbers. Thru indicates a range of grid points. by is the increment to be used within this range. (G3 < G4) (Required, Integer > 0)

RBE2D (SOL 700) Defines A Nodal Rigid Body

This is a rigid body that consists of defined grid points. The inertia tensor can be either user defined or computed from the nodal masses. Arbitrary motion of this rigid body is allowed. When the inertia tensor is user defined, constant translational and rotational velocities can be defined in a global or local coordinate system.

The first node in the nodal rigid body definition is treated as the master for the case where CMT and CMR are nonzero. The first node always has six degrees-of-freedom. The release conditions applied in the global system are sometimes convenient in small displacement linear analysis, but, otherwise, are not recommended. It is strongly recommended, that release conditions are only used for a two noded nodal rigid body.

Format:

1	2	3	4	5	6	7	8	9	10
RBE2D	EID	CID		PNODE	IPRT	CMT	CMR		
	CMO	CON1	CON2						
	XC	YC	ZC	TM		NODEID			
	IXX	IXY	IXZ	IYY	IYZ	IZZ			
	VTX	VTY	VTZ	VRX	VRY	VRZ			
	XL	YL	ZL	XLIP	YLIP	XLIP	CID2		
	G1	G2	THRU	G3	BY	G4	-etc.-		

Example:

RBE2D	25								
	450.5	350.4	-200.0						
	1.0E5			1.0E5		1.0E5			
	23	1001	THRU	2000	BY	2			

Field	Contents
EID	Number of the nodal rigid-body. This number must be unique with respect to other RBEx ID and with respect to any property ID used. (Required, Interer > 0)
CID	Optional coordinate system ID for the rigid body local system. Output of the rigid body data and the degree-of-freedom releases are done in this local system. This local system rotates with the rigid body. (Integer > 0, 0)
PNODE	An optional, possibly mass-less, grid point located at the mass center of the nodal rigid body. The initial nodal coordinates will be reset if necessary to ensure that they lie at the mass center. In the output files, the coordinates, accelerations, velocities, and displacements of this node will correspond to the mass center of the nodal rigid body. If CID is defined, the velocities and accelerations of PNODE will be output in the local system in the D3PLOT and D3THDT files unless PNODE is specified as a negative number in which case the global system is used. (Integer > 0, 0)
IPRT	Print flag. For nodal rigid bodies the following values apply: EQ.1: write data into RBDOUT EQ.2: do not write data into RBDOUT Printing is suppressed for two noded rigid bodies unless IPRT is set to unity. This is to avoid excessively large RBDOUT files when many, two-noded welds are used. (Integer > 0, 0)
CMT	Translation release flag for all grid-points except the first grid-point in the definition. Any combination of 1,2,3 and the + - sign is allowed, with following meaning (Integer > 0, 0): EQ.-123: release x, y, and z displacement in global system EQ.-13: release z and x displacement in global system EQ.-23: release y and z displacement in global system EQ.-12: release x and y displacement in global system EQ.-3: release z displacement in global system EQ.-2: release y displacement in global system EQ.-1: release x displacement in global system EQ. 0: off for rigid body behavior

Field	Contents
	EQ. 1: release x displacement in rigid body local system
	EQ. 2: release y displacement in rigid body local system
	EQ. 3: release z displacement in rigid body local system
	EQ. 12: release x and y displacement in rigid body local system
	EQ. 23: release y and z displacement in rigid body local system
	EQ. 13: release z and x displacement in rigid body local system
	EQ. 123: release x, y, and z displacement in rigid body local system
CMR	Rotation release flag for all grid-points except the first grid-point in the definition. Any combination of 4,5,6 and the + - sign is allowed, with following meaning (Integer > 0, 0):
	EQ.-456: release x, y, and z rotations in global system
	EQ.-46: release z and x rotations in global system
	EQ.-56: release y and z rotations in global system
	EQ.-45: release x and y rotations in global system
	EQ.-6: release z rotation in global system
	EQ.-5: release y rotation in global system
	EQ.-4: release x rotation in global system
	EQ. 0: off for rigid body behavior
	EQ. 4: release x rotation in rigid body local system
	EQ. 5: release y rotation in rigid body local system
	EQ. 6: release z rotation in rigid body local system
	EQ. 45: release x and y rotations in rigid body local system
	EQ. 56: release y and z rotations in rigid body local system
	EQ. 46: release z and x rotations in rigid body local system
	EQ. 456: release x, y, and z rotations in rigid body local system
CMO	Center of mass constraint option, CMO (Required > 0, 0.0):
	EQ.+1.0: constraints applied in global directions,
	EQ. 0.0: no constraints,
CON1	First constraint parameter (Required > 0, 0.0):
	If CMO=+1.0, then specify global translational constraint:

Field	Contents
	EQ.0: no constraints,
	EQ.1: constrained x displacement,
	EQ.2: constrained y displacement,
	EQ.3: constrained z displacement,
	Any combination of constraints are allowed, e.g., 12, 123.
	If CM0=-1.0, then specify local coordinate system ID. This coordinate system is fixed in time.
CON2	Second constraint parameter (Required > 0, 0.0):
	If CMO=+1.0, then specify global rotational constraint:
	EQ.0: no constraints,
	EQ.4: constrained x rotation,
	EQ.5: constrained y rotation,
	EQ.6: constrained z rotation,
	Any combination of global constraints are allowed, e.g., 45, 456.
	If CM0=-1.0, then specify local (SPC) constraint:
	EQ.0: no constraint,
	EQ.1: constrained x translation,
	EQ.2: constrained y translation,
	EQ.3: constrained z translation,
	EQ.4: constrained x rotation,
	EQ.5: constrained y rotation,
	EQ.6: constrained z rotation.
	Any combination of local constraints are allowed, e.g., 12, 123.
XC	x-coordinate of center of mass. If nodal point, NODEID, is defined XC, YC, and ZC are ignored and the coordinates of the nodal point, NODEID, are taken as the center of mass. (Required, 0.)
YC	y-coordinate of center of mass (Required, 0.0)
ZC	z-coordinate of center of mass (Required, 0.0)
TM	Translational mass (Required > 0, 0.0)

Field	Contents
NODEID	Optional nodal point defining the CG of the rigid body. If this node is not a member of the set Gi below, its motion will not be updated to correspond with the nodal rigid body after the calculation begins. PNODE and NODEID can be identical if and only if PNODE physically lies at the mass center at time zero. (Integer > 0, 0)
IXX	I_{xx}, xx component of inertia tensor R (Required)
IXY	I_{xy} (set to zero if IRCS=1)(Required, 0.0)
IXZ	I_{xz} (set to zero if IRCS=1) (Required, 0.0)
IYY	I_{yy}, yy component of inertia tensor (Required)
IYZ	I_{yz} (set to zero if IRCS=1)(Required, 0.0)
IZZ	Izz, zz component of inertia tensor (Required, 0.0)
VTX	x-rigid body initial translational velocity in global coordinate system. (Required, 0.0)
VTY	y-rigid body initial translational velocity in global coordinate system. (Required, 0.0)
VTZ	z-rigid body initial translational velocity in global coordinate system. (Required, 0.0)
VRX	x-rigid body initial rotational velocity in global coordinate system. (Required, 0.0)
VRY	y-rigid body initial rotational velocity in global coordinate system. (Required, 0.0)
VRZ	z-rigid body initial rotational velocity in global coordinate system. See Remark 2. (Required, 0.0)
XL	x-coordinate of local x-axis. Origin lies at (0,0,0). (Required, 0.0)
YL	y-coordinate of local x-axis. (Required, 0.0)
ZL	z-coordinate of local x-axis. (Required, 0.0)
XLIP	x-coordinate of local in-plane vector. (Required, 0.0)
YLIP	y-coordinate of local in-plane vector. (Required, 0.0)
ZLIP	z-coordinate of local in-plane vector. (Required, 0.0)
CID2	Local coordinate system ID. See Remark 3. (Integer > 0.0)
Gi	Grid-point numbers. **Thru** indicates a range of grid points. By is the increment to be used within this range. (G2<G3) (Integer > 0, Required)

Remarks:

1. Unlike the RBE2, here the equations of rigid body dynamics are used to update the motion of the nodes and therefore rotations of the nodal sets are admissible. Mass properties are determined from the nodal masses and coordinates. See also Remark 4.
2. The velocities defined above can be overwritten by the TIC, TIC1, TIC2, TIC3, TICGP entries.
3. The local coordinate system is set up in the following way. After the local x-axis is defined, the local z-axis is computed from the cross-product of the local x-axis vector with the given in-plane vector. Finally, the local y-axis is determined from the cross-product of the local z-axis with the local x-axis. The local coordinate system defined by CID has the advantage that the local system can be defined by nodes in the rigid body which makes repositioning of the rigid body in a preprocessor much easier since the local system moves with the nodal points.
4. For modeling a nodal rigid body that undergoes rotations, use RBE2D or RBE2 with the FULLRIG option. Note that the RBE2 without the FULLRIG option can not properly simulate rotations, as shown in the figure below.

Offset nodes a and b are constrained to move together.

RBE2F (SOL 700) Translational Nodal Constrained

Defines nodal constraint sets for translational motion in global coordinates. No rotational coupling. See **Figure 8-154**. Nodal points included in the sets should not be subjected to any other constraints including prescribed motion.

Format:

1	2	3	4	5	6	7	8	9	10
RBE2F	EID	G1	CM	TF	G2	THRU	G3	BY	
	G4	-etc.-							

Example:

RBE2F	22	2205	123		1	THRU	100		

Field	Contents
EID	Number of the nodal rigid-body. (Integer, Required)
CM	Applicable translational degrees-of-freedom (Integer, Required, see Remark 1) Use any combination of 1, 2, 3: = 1: x-translational degree-of-freedom, = 2: y-translational degree-of-freedom, = 3: z-translational degree-of-freedom, = 12: x and y-translational degrees-of-freedom, = 23: y and z-translational degrees-of-freedom, = 13: z and x-translational degrees-of-freedom, = 123: x, y, and z-translational degrees-of-freedom.
TF	Failure time for nodal constraint set. See Remark 2. (Real, Default = 1.0E20)
Gi	Grid-point numbers. Thru indicates a range of grid points. By is the increment to be used within this range. (G2 < G3) (Integer, Required)

Remarks:

1. The masses of the nodes are summed up to determine the total mass of the constrained set. It must be noted that the definition of a nodal rigid body is not possible with this input since no rotation is permitted. For nodal rigid bodies the keyword input: RBE2D or RBE2 (FULLRIG) must be used.

Offset nodes a and b are constrained to move together.

Figure 8-154 FRBE2F can lead to nonphysical responses.

2. When the failure time, *TF*, is reached the nodal constraint becomes inactive and the constrained nodes may move freely.
3. Corresponds to Ls-Dyna entry *****CONSTRAINED_NODE_SET.

RBE2GS Internally Generate an RBE2 Element

Defines an RBE2 connecting the two closest grids to GS.

Format:

1	2	3	4	5	6	7	8	9	10
RBE2GS	EID	GS	TYPE			R	CMB	ALPHA	
	XS	YS	ZS	GNi	GMj				

Example:

RBE2GS	3	17				1.3		6.5-6	
				endl	56	99			

Alternate Formats and Examples:

RBE2GS	EID	GS	TYPE			R	CMB	ALPHA	
	XS	YS	ZS	GNi	THRU	GNj	*endl*		
	GMk	THRU	GM*l*						

RBE2GS	15	35				-.66			
				88	THRU	107	*endl*	15	
	76	88							

RBE2GS	25					.66			
	5.173	0.0	19.3185	88	99	108	*endl*		
	15	THRU	88						

RBE2GS	35	28				-.66			
				56	THRU	102	*endl*	19	
	21	THRU	200						

RBE2GS	45	16				-.66			
				56	THRU	102	1129		
	1146	THRU	1200	*endl*					

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GS	Search POINT or GRID point. (Integer > 0 or blank)
TYPE	Connectivity: (Character) If TYPE='blank' (default), search the complete model. If TYPE='NMFLIP', the independent and dependent DOF's are interchanged.
R	Radius. (Real < 0. or < 0.)
CM	Component number of dependent degrees-of-freedom for grid GM. (Integers 1 through 6 with no embedded blanks. Blank defaults to 1234565.)
ALPHA	Thermal expansion coefficient. (Real $\geq$ 0.0 or blank)
XS, YS, ZS	Location of search point if GS is blank. Used only if GS=0 or blank. (Real or blank)
GNi	List of grids to be excluded from candidate grids for GN. If a GNi list is given it must end with an *endl*. (Integer > 0 or blank or "THRU")
GMj	List of grids to be excluded from candidate grids for GM. If no GNi list is given and a GMj list is present, then GNi must have an *endl* entry. (Integer > 0 or blank or "THRU")

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. This entry will internally define a RBE2 element with the same ID of the RBE2GS entry.
   The grid assigned to GN will always be the independent grid. GM will be the dependent grid.
   If GS is a 'POINT' entry (or GS is blank and XYZ is specified), the first two grids that fall within the search radius $|R|$ about GS (or XYZ) will be chosen as GN and GM. The closest to the search location will be the independent grid GN the next closest will be the dependent grid GM. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in

a GMj list will be excluded from the GM search. After GN and GM have been determined (with or without use of the exclusion lists for GNi and GMj (and if TYPE='NMFLIP', then GN and GM will be reversed.
If GS is 'GRID' entry, and is part of the physical model, i.e., it has physical structural elements attached to it, it will become GN the independent grid for the RBE2 to be generated. If it is in the GNi exclusion list, the next closest grid will be chosen. the closest grid within the search radius about $|R|$ GN will be chosen as GM, the dependent grid. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with our without use of the exclusion lists for GNi and GMj) and if TYPE='NMFLIP', then GN and GM will be reversed.
If GS is a 'GRID' entry, and is determined not to be part of the physical model, the first two grids that fall within the search radius $|R|$ about GS (or XYZ) will be chosen as GN and GM. The closest to the search location will be the independent grid, GN, the next closest will be the dependent grid, GM. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with or without use of the exclusion lists for GNi and GMj) and if TYPE='NMFLIP', then GN and GM will be reversed. The GS grid will remain on the GEOM1 table for post-processing viewing purposes, but will not be part of the MD Nastran solution g-set.
POINT and GRID entries must be unique with respect to all other POINT and GRID entries.

3. If $R < 0$, the two located grids GN and GM will be made coincident to the GS (or XYZ) location.
4. If CM is 0 or blank and GM is touching only solid elements, CM will be internally defaulted to 123. For solid elements, the grids GN and GM should always be coincident.
5. When Module GP4 is run, checks are made to insure that the selected grids, GN and GM, do not violate existing constraint sets. If a violation occurs a fatal message will be issued for the offending grids. These grids can be excluded from further search inclusion by specifying them on the GNi or GMj list.
6. If GS=0 or blank and XS, YX, ZS is not specified or if both GS and XS, YS, ZS are specified a fatal error will occur.
7. The end of a grid exclusion lists is indicated by the existence of "end*l*" in the field following the last entry in the list. In the "thru" option, not all grids in the range need exist.

8. For superelement or part superelement connection the independent grid assigned to GN will be exterior to the superelement. The dependent grid GM will be an interior grid to the superelement. If the resulting RBE2 element, connects two different superelements, the element will be assigned to the superelement with the lower ID and the grid attached to the superelement with the higher ID will be moved to the superelement with the lower ID.
9. If the RBE2GS is listed on a SEELT entry, it will be placed as the SEELT directs. If say grid G1 lies in another superelement and G2 lies in the SEELT superelement, G1 will be moved to the SEELT defined superelement.
10. “THRU” should not be the first nonblank field of a continuation line. Blank fields are allowed in the exclusion lists for readability.
11. Use PARAM,SEP1XOVR,16 to print the grids found by each RBE2GS entry.

RBE3 Interpolation Constraint Element

Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
	G1,3	WT2	C2	G2,1	G2,2	-etc.-	WT3	C3	
	G3,1	G3,2	-etc.-	WT4	C4	G4,1	G4,2	-etc.-	
	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	GM5	CM5	-etc.-			
	“ALPHA”	ALPHA							

Example:

RBE3	14		100	1234	1.0	123	1	3	
	5	4.7	1	2	4	6	5.2	2	
	7	8	9	5.1	1	15	16		
	UM	100	14	5	3	7	2		
	ALPHA	6.5-6							

Field	Contents
EID	Element identification number. Unique with respect to all elements. (0 < Integer < 100,000,000)
REFGRID	Reference grid point identification number. (Integer > 0)
REFC	Component numbers at the reference grid point. (Any of the integers 1 through 6 with no embedded blanks.)
WTi	Weighting factor for components of motion on the following entry at grid points Gi,j. (Real)
Ci	Component numbers with weighting factor WTi at grid points Gi,j. (Any of the integers 1 through 6 with no embedded blanks.)
Gi,j	Grid points with components Ci that have weighting factor WTi in the averaging equations. (Integer > 0)

Field	Contents
“UM”	Indicates the start of the degrees-of-freedom belonging to the dependent degrees-of-freedom. The default action is to assign only the components in REFC to the dependent degrees-of-freedom. (Character)
GMi	Identification numbers of grid points with degrees-of-freedom in the m-set. (Integer > 0)
CMi	Component numbers of GMi to be assigned to the m-set. (Any of the Integers 1 through 6 with no embedded blanks.)
“ALPHA”	Indicates that the next number is the coefficient of thermal expansion. (Character)
ALPHA	Thermal expansion coefficient. See Remark **14.** (Real $\geq$ 0.0 or blank)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, the REFC must be “123”, “456”, or “123456”. No other combination is allowed.
3. For the Lagrange method, MD Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of degrees-of-freedom given by REFC.
4. For the linear method, the dependent degrees-of-freedom indicated by REFC will be made members of the m-set. For Lagrange rigid element, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
5. We recommend that for most applications only the translation components 123 be used for Ci. An exception is the case where the Gi,j are colinear. A rotation component may then be added to one grid point to stabilize its associated rigid body mode for the element.
6. Blank spaces may be left at the end of a Gi,j sequence.

7. For the Lagrange method, the default for “UM” must be used. For the linear method, the default for “UM” should be used except in cases where the user wishes to include some or all REFC components in displacement sets exclusive from the m-set. If the default is not used for “UM”:
   - The total number of components in the m-set (i.e., the total number of dependent degrees-of-freedom defined by the element) must be equal to the number of components in REFC (four components in the example).
   - The components specified after “UM” must be a subset of the components specified under REFC and (Gi,j, Ci).
   - The coefficient matrix $[R_m]$ described in Section 9.4.3 of the *MSC.Nastran Reference Manual* must be nonsingular. PARAM,CHECKOUT in SOLs 101 through 200 may be used to check for this condition.
8. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
9. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
10. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
11. Rigid elements are ignored in heat transfer problems.
12. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.
13. The formulation for the RBE3 element was changed in Version 70.7. This change allowed the element to give consistent answers that are not dependent upon the units of the model. Only models that connected rotation degrees-of-freedom for Ci were affected. Note that these models are ignoring the recommendation in Remark **5.** The formulation prior to Version 70.7 may be obtained by setting SYSTEM(310)=1.

14. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the reference grid point REFGRID and any other grid point Gij are taken as the average temperature of the two connected grid points.
15. For SOL 700, RBE3 are presently converted to MPC's.

RBE3D (SOL 700) Interpolation Constraint Element in MSC.Dytran-Style RBE3

Format:

Defines rigid interpolation constraints used in MD Nastran Explicit Nonlinear (SOL 700) only.

1	2	3	4	5	6	7	8	9	10
RBE3D	EID	GD	CD	CIDD	CIDIall			OPTION	
	GI	CI	WX	WY	WZ	WRX	WRY	WRZ	

Note: Repeat continuation entry as many times as necessary.

Primary Example 1:

RBE3D	51	11	123						
	101	123	0.5	0.5	0.5				
	102	123	0.5	0.5	0.5				

Primary Example 2:

RBE3D	51	11	123	4	8			LOCAL1	
	101	123	0.5	0.5	0.5				
	102	123	0.5	0.5	0.5				

Alternate Format:

RBE3D	EID	GD	CD	CIDD				OPTION	
	CIDI								
	GI	CI	WX	WY	WZ	WRX	WRY	WRZ	

Note: Repeat second and third entries as many times as necessary.

Alternate Example:

RBE3D	51	11	123	4				LOCAL2	
	8								
	101	123	0.5	0.5	0.5				
	25								
	102	123	0.5	0.5	0.5				

Field	Contents
EID	Unique element identification number. (Integer > 0, required. No default.)
GD	Dependent grid ID (Integer > 0, required. No default.)
CD	A list of components corresponding to GD. Applicable values are any or all of the integers 1-6. For example, to specify x,y,z components, enter 123. (Integer > 0, required; Default is 123456)
CIDD	Local coordinate system ID for dependent grids. If OPTION=LOCAL (Integer $\geq$ 0 or blank. Default = 0.)
CIDIall	Local coordinate system ID for all independent grids. If OPTION=LOCAL. (Integer $\geq$ 0 or blank. Leave blank if each independent grid has different local coordinates.)
OPTION	Can be blank (meaning independent grid weighting factors are in the global coordinate system), LOCAL1 meaning that the weighting factors for all independent grids are in local coordinate systems, or LOCAL2 meaning that the weighting factors for the independent grids are in different local coordinate systems. If all local coordinates are the same, use CIDIall to define the coordinate system. If there are differences, use the alternate format to define CIDI for each independent grid. The alternate format requires OPTION=LOCAL2.
CIDI	Local coordinate system ID for individual independent grids. If OPTION=LOCAL. (Integer $\geq$ 0 or blank. Leave blank if each independent grid has different local coordinates.)
GI	Independent grid ID (Integer > 0, required. No default.)
CI	A list of components corresponding to GI -- Applicable values are any or all of the integers 1-6. For example to specify x,y,z components, enter 123. (Integer > 0, required. Default is 123456.)

Field	Contents
WX	Weight factor for grid GI to be applied in x direction as given by coordinate system CID. (Real; Default = 0.0.)
WY	Weight factor for grid GI to be applied in y direction as given by coordinate system CID. (Real; Default = 0.0.).
WZ	Weight factor for grid GI to be applied in z direction as given by coordinate system CID. (Real; Default = 0.0.)
WRX	Rotational weight factor for grid GI to be applied about the x axis as given by coordinate system CID (Real; Default = 0.0.)
WRY	Rotational weight factor for grid GI to be applied about the y axis as given by coordinate system CID (Real; Default = 0.0.)
WRZ	Rotational weight factor for grid GI to be applied about the z axis as given by coordinate system CID (Real; Default = 0.0.)

Remarks:

1. RBE3D should be used instead of RBE3 for SOL 700. If RBE3 entries are entered, they will be mapped to the RBE3D entry. RBE3D can be entered directly in MD Nastran for SOL 700 only. If RBE3 is used, the LOCAL1/LOCAL2 options are not invoked.
2. For SOL 700, RBE3D are presently converted toMPC’s.

RBJOINT (SOL 700) Defines a Joint Between Two Rigid Bodies

Defines a joint between two rigid bodies.

Format:

1	2	3	4	5	6	7	8	9	10
RBJOINT	ID	TYPE	N1	N2	N3	N4	N5	N6	
	ID-STIF	RPS	DAMP	PARM	LCID	MOTTYPE	R1		
	RAID	LST	CID-FAIL	TFAIL	COUPL				
	NXX-FAIL	NYY-FAIL	NZZ-FAIL	MXX-FAIL	MYY-FAIL	MZZ-FAIL			

Examples:

Example 1 - Cylindrical Joint without Failure

RBJOINT	1	CYLIND	101	201	103	204			

Example 2 - Translational Joint with Failure

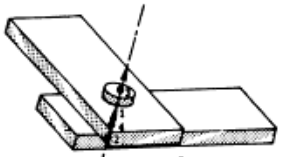
RBJOINT	2	TRANSL	101	201	103	204	105	206	
	1E6	1E6	1E6	1E2	1E2	1E2			

Field	Contents
ID	RBJOIN identification number. (Integer > 0, Required)
TYPE	Type of RBJOIN. (Character, Required) Types available are (node numbers in the images correspond to N1, N2, etc.): TYPE= SPHER (sperical)

Spherical joint

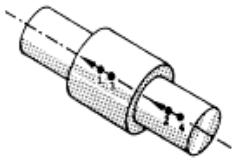
Field	Contents

TYPE=REVOLUTE



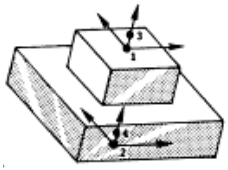
Revolute joint

TYPE=CYLIND (cylindrical)



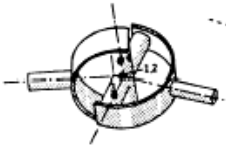
Cylindrical joint

TYPE=PLANAR



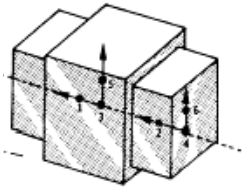
Planar joint

TYPE=UNIVERS



Universal joint

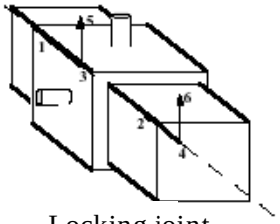
TYPE=TRANSL (translational)



Translational joint

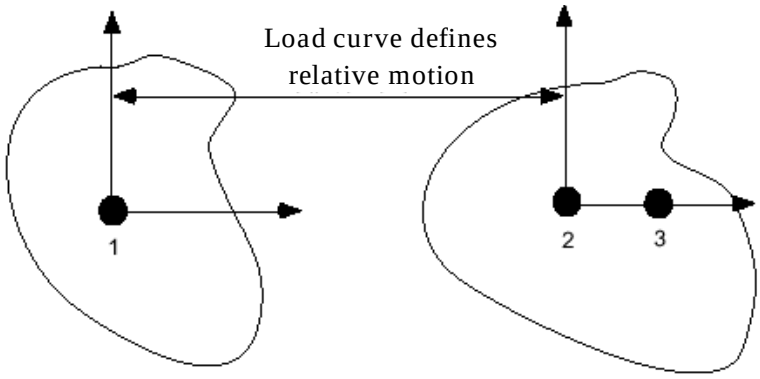
Field	Contents

TYPE=LOCKING



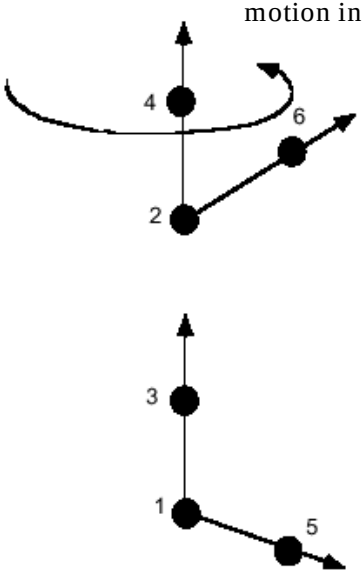
Locking joint

TYPE=MOTORTRA



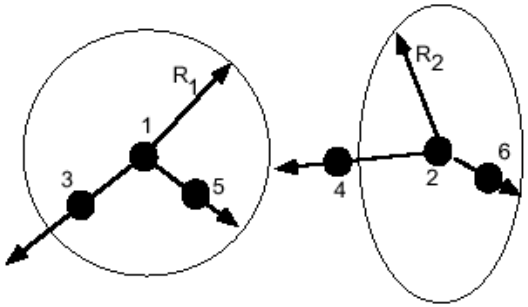
Translational motor joint. This joint can be used in combination with the translational or the cylindrical joit.

Field	Contents
	TYPE=MOTORROT
	Load curve defines relative rotational motion in radians per unit time



Rotational motor joint. This joint can be used in combination with other joints such as the revolute or cylindrical joints.

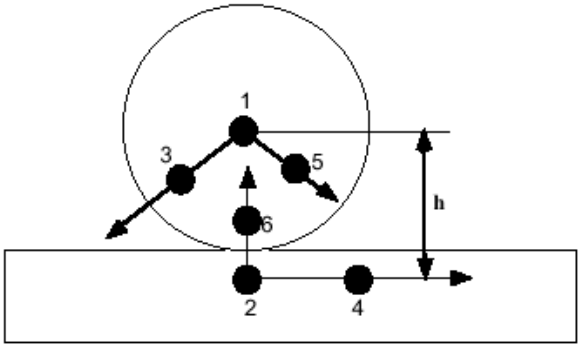
TYPE=GEARS



Gear joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the gears. Nodal pairs (1,5) and (2,6) define vectors in the plane of the gears. The ratio, R_2 / R_1, is specified.

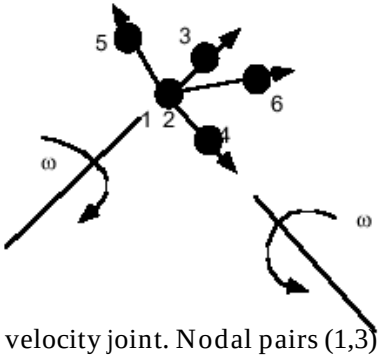
Field	Contents

TYPE=RACK_PIN (rack and pinion)



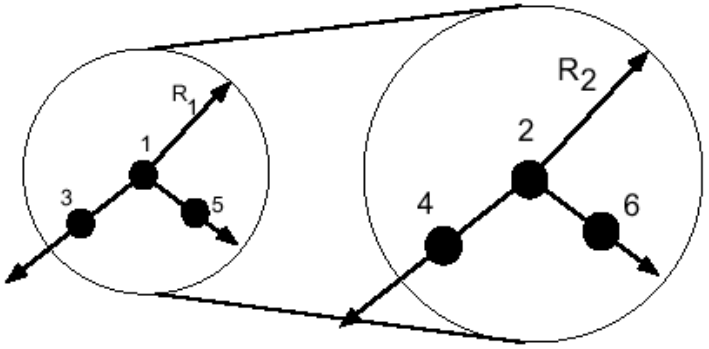
Rack and pinion joint. Nodal pair (1,3) defines a vector that is orthogonal to the plane of the gear. Nodal pair (1,5) is a vector in the plane of the gear. Nodal pair (2,4) defines the direction of travel for the second body. The value h is specified.

TYPE=CONSTVEL



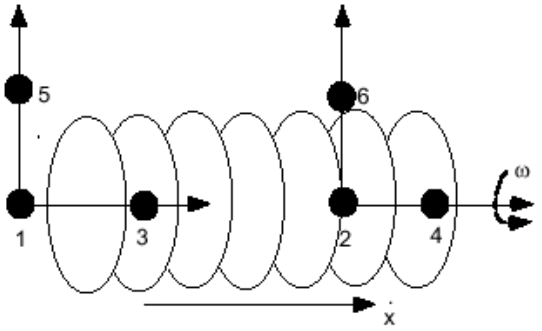
Constant velocity joint. Nodal pairs (1,3) and (2,4) define an axes for the constant angular velocity, and nodal pairs (1,5) are orthogonal vectors. Here nodal pints 1 and 2 must be coincident.

Field	Contents
	TYPE=PULLEY



Pulley joint. Nodal pairs (1,3) and (2,4) define axes that are orthogonal to the pulleys. Nodal pairs (1,5) and (2,6) define vectors in the plane of the pulleys. The ratio, R_2/R_1 is specified.

TYPE=SCREW



Screw joint. The second body translates in response t the spin of the first body. Nodal pairs (1,3) and (2,4) l along the same axis and nodal pairs (1.5) and (2,6) are orthogonal vectors. The helix ratio, $\dot{x}/\omega$, is specified

N1	GRID id of Node 1, in rigid body A. Define for all joint types
N2	GRID id of Node 2, in rigid body B. Define for all joint types
N3	GRID id of Node 3, in rigid body A. Define for all joint types except SPHER
N4	GRID id of Node 4, in rigid body B. Define for all joint types except SPHER

Field	Contents
N5	GRID id of Node 5, in rigid body A. Define only for joint types TRANSL, LOCKING, MOTORROT, CONSTVEL, GEARS, RACK_PIN, PULLEY, and SCREW
N6	GRID id of Node 6, in rigid body B. Define only for joint types TRANSL, LOCKING, MOTORROT, CONSTVEL, GEARS, RACK_PIN, PULLEY, and SCREW
STIFF	ID of a RBJSTIF entry, to define optional rotational and translational joint stiffnesses.
RPS	Relative penalty stiffness (default = 1.0)
DAMP	Damping scale factor on default damping value. (REVOLUTE and SPHERICAL joints): DAMP = 0.0: default is set to 1.0 0.0 < DAMP < 0.01: no damping is used
PARM	Parameter which is a function of the joint type. Leave blank for MOTORS. GEARS: define R2/R1 RACK_PINION: define h PULLEY: define R2/R1 SCREW: define $\dot{x}/\omega$
LCID	TABLED1 ID, to defe the load curve for MOTOR joints
MOTTYPE	Integer flag for MOTOR joints: MOTTYPE=0: velocity MOTTYPE=1: acceleration MOTTYPE=1: displacement
R1	Radius for the GEARS and PULLEY joints. If left undefined, N5 and N6 are assumed to be on the outer radius.
RAID	Rigid body or accelerometer ID. The force resultants are output in the local system of the rigid body or acelerometer.
LST	Integer flag for local system type: LST=0 : rigid body LST=1 : accelerometer
CID-FAIL	Coordinate ID for resultants in the failure criteria.
TFAIL	Time for joint failure. If zero, joint never fails.

Field	Contents
COUPL	Coupling between the force and moment failure criteria. COUPL≥0: The failure criteria is: $\left(\frac{max(N_{xx}, 0)}{N_{xx_F}}\right)^2 + \left(\frac{N_{yy}}{N_{yy_F}}\right)^2 + \left(\frac{N_{zz}}{N_{zz_F}}\right)^2 + \left(\frac{M_{xx}}{M_{xx_F}}\right)^2 + \left(\frac{M_{yy}}{M_{yy_F}}\right)^2 + \left(\frac{M_{zz}}{M_{zz_F}}\right)^2 - 1 = 0$ COUPL>0: The force and moment results are considered independently. The failure criteria is: $\left(\frac{maxN_{xx}, 0)}{N_{xx_F}}\right)^2 + \left(\frac{N_{yy}}{N_{yy_F}}\right)^2 + \left(\frac{N_{zz}}{N_{zz_F}}\right)^2 - 1 = 0 \text{and} \left(\frac{M_{xx}}{M_{xx_F}}\right)^2 + \left(\frac{M_{yy}}{M_{yy_F}}\right)^2 + \left(\frac{M_{zz}}{M_{zz_F}}\right)^2 - 1 = 0$
NXX-FAIL	Axial force resultant at failure. If zero, failure due to this component is not considered.
NYY-FAIL	Shear force (yy) resultant at failure. If zero, failure due to this component is not considered.
NZZ-FAIL	Shear force (zz) resultant at failure. If zero, failure due to this component is not considered.
MXX-FAIL	Torsional moment resultant at failure. If zero, failure due to this component is not considered.
MYY-FAIL	Bending moment (yy) resultant at failure. If zero, failure due to this component is not considered
MZZ-FAIL	Bending moment (zz) resultant at failure. If zero, failure due to this component is not considered

RBJSTIF

Defines optional rotational and traslational joint stiffness for RBJOINT between two rigid bodies. (Multiple forms, i.e., GENERAL, TRANSL, and FLEX-TOR, may be used to define the stiffness.)

Format:

1	2	3	4	5	6	7	8	9	10
RBJSTIF	ID								
	GENERAL	CIDA	CIDB						
		LCIDPPH	LCIDT	LCIDPS	DLCIDPH	DLCIDT	DLCIDPS		
		ESPH	FMPH-TYP	FMPH	EST	FMT-TYPE	FMT		
		ESPS	RFMPS-TYP	FMPS					
		NSAPH	PSAPH	NSAT	PSAT	NSAPS	PSAPS		
	TRANSL	CIDA	CIDB						
		LCIDX	LCIDY	LCIDZ	DLCIDX	DLCIDY	DLCIDZ		
		ESX	FFX-TYPE	FFX	ESY	FFY-TYPE	FFY		
		ESZ	FFZ=TYPE	FFZ					
		NSDX	PSDX	NSDY	PSDY	NSDZ	PSDZ		
	FLEX-TOR	CIDA	CIDB						
		LCIDAL	LCIDG	LCIDBT	DLCIDAL	DLCIDG	DLCIDBT		
		ESAL	FMAL-TYP	FMAL	ESBT	FMBT-TYP	FMBT		
		SAAL	NSABT	PSABT					

Examples:

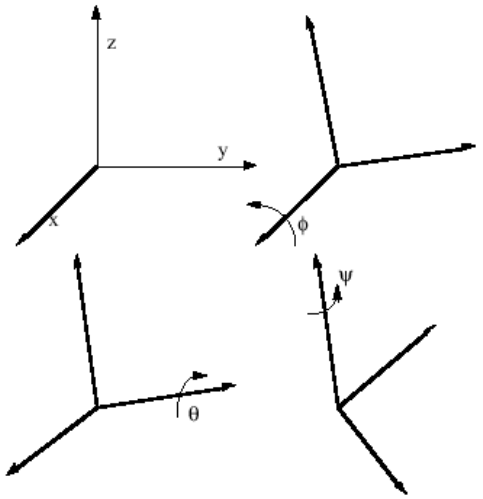
Example 1 - Rotational Stiffness and Stop Angle 30, Cylindrical Joint Without Failure

RBJOINT	1	CYLIND	101	201	103	204			
	101								

A.RBJSTIF	101								
	GENERAL	1	2						
		1							

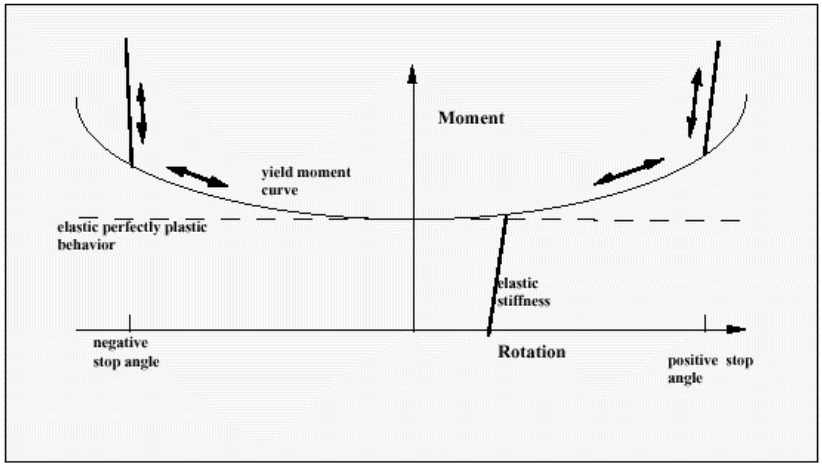
		30	30						

Field	Contents
ID	RBJSTIF identification number. Must be referenced from a RBJOINT entry. (Integer > 0, Required)
GENERAL	Entries for this continuation line describe a generalized joint stiffness.
	After the stop angles are reached the torques increaselinearly to resist further angular motion using the stiffness values specified here. reasonable stiffness values have to be chosen. If the stiffness values are too low or zero, the stop will be violated.



Definition of angles for the generalized joint stiffness. The magnitude of the angular rotations are limited by the stop angles. If the initial local coordinate axes do not coincide, the angles ϕ, θ, and ψ, will be initialized and torques wil develop instantaneously based on the defined load curves

Field	Contents



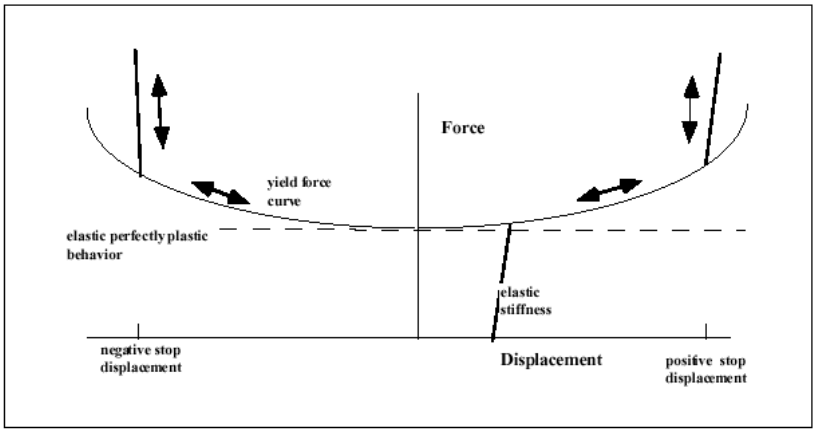
Frictional behavior is modeled by a plasticity model. Elastic behavior is obtained once the stop angles are reached. The same elastic stiffness is used to simulate sticking situations.

CIDA	Coordinate ID for rigid body A
CIDB	Coordinate ID for rigid body B
LCIDPH	TABLED1 id for φ-moment versus rotation in radians. If zero, the applied moment is set to 0.0.
LCIDT	TABLED1 id for θ-moment versus rotation in radians. If zero, the applied moment is set to 0.0.
LCIDPS	TABLED1 id for ψ-moment versus rotation in radians. If zero, the applied moment is set to 0.0
DLCIDPH	TABLED1 id for φ-damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered.
DLCIDT	TABLED1 id for θ-damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered.
DLCIDPS	TABLED1 id for ψ-damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered.
ESPH	Elastic stiffness per unit radian for friction and stop angles for φ-rotation. If zero, friction and stop angles are inactive for φ-rotation.

Field	Contents
FMPH-TYP	Type of friction moment limit, as specifed in FMPH field: FMPH-TYP=CONSTANT FMPH=TP=ROTATION FMPH-TYP=JOINT
FMPH	Limit on the frictional moment. This option may also be thought of as an elastic-plastic spring. The interpretation of the value depends on the value of FMPH-TYPE. If zero, friction is inactive for φ-rotation. FMPH-TYPE=CONSTANT: Limit on the frictional moment for φ-rotation. (Real > 0, Default = 0.) FMPH-TYPE=ROTATION: Value is a TABLED1 id, specifying the yield moment versus φ-rotation. (Integer > 0) FMPH-TYPE=JOINT: Value is a TABLED1 id, specifying the yield moment versus the RBJOINT reaction force. (Integer > 0)
EST	Elastic stiffness per unit radian for friction and stop angles for θ-rotation. If zero, friction and stop angles are inactive for θ-rotation.
FMT-TYPE	Type of friction moment limit, as specified in FMT field: FMT-TYPE=CONSTANT FMT-TYPE=ROTATION FMT-TYPE=JOINT
FMT	Limit on the frictional moment. This option may also be thought of as an elastic-plastic spring. The interpretation of the value depends on the value of FMT-TYPE. If zero, friction is inactive for θ-rotation. FMT-TYPE=CONSTANT: Limit on the frictional moment for θ-rotation. (Real > 0, Default = 0.) FMT-TYPE=ROTATION: Value is a TABLED1 id, specifying the yield moment versus θ-rotation. (Integer > 0) FMT-TYPE=JOINT: Value is a TABLED1 id, specifying the yield moment versus the RBJOINT reaction force. (Integer > 0)
ESPS	Elastic stiffness per unit radian for friction and stop angles for ψ-rotation. If zero, friction and stop angles are inactive for ψ-rotation.

Field	Contents
FMPS-TYP	Type of friction moment limit, as specified in FMPS field: (Character, Default=CONSTANT) FMPS-TYP=CONSTANT FMPS-TYP=ROTATION FMPS-TYP=JOINT
FMPS	Limit on the frictional moment. This option may also be thought of as an elastic-plastic spring. The interpretation of the value depends on the value of FMPS-TYPE. If zero, friction is inactive for ψ-rotation. FMPS-TYPE=CONSTANT: Limit on the frictional moment for ψ-rotation. (Real > 0, Default = 0.) FMPS-TYPE=ROTATION: Value is a TABLED1 id, specifying the yield moment versus ψ-rotation. (Integer > 0) FMPS-TYPE=JOINT: Value is a TABLED1 id, specifying the yield moment versus the RBJOINT reaction force. (Integer > 0)
NSAPH	Stop angle in degrees for negative φ-rotation. Ignored if zero (Real > 0, Default = 0.)
PSAPH	Stop angle in degrees for positive φ-rotation. Ignored if zero (Real > 0, Default = 0.)
NSAT	Stop angle in degrees for negative θ-rotation. Ignored if zero (Real > 0, Default = 0.)
PSAT	Stop angle in degrees for positive θ-rotation. Ignored if zero (Real > 0, Default = 0.)
NSAPS	Stop angle in degrees for negative ψ-rotation. Ignored if zero (Real > 0, Default = 0.)
PSAPS	Stop angle in degrees for positive ψ-rotation. Ignored if zero (Real > 0, Default = 0.)

Field	Contents
TRANSL	Entries for this continuation line describe a translational joint stiffness. After the stop displacements are reached the forces increase linearly to resist further translational motion using the stiffness values specified here. Reasonable stiffness values have to be chosen. If the stiffness values are too low or zero, the stop will be violated.



Frictional behavior is modeled by a plasiticty model. Elastic behavior is obtained once the stop displacements are reached. The same elastic stiffness is used to simulate sticking situations.

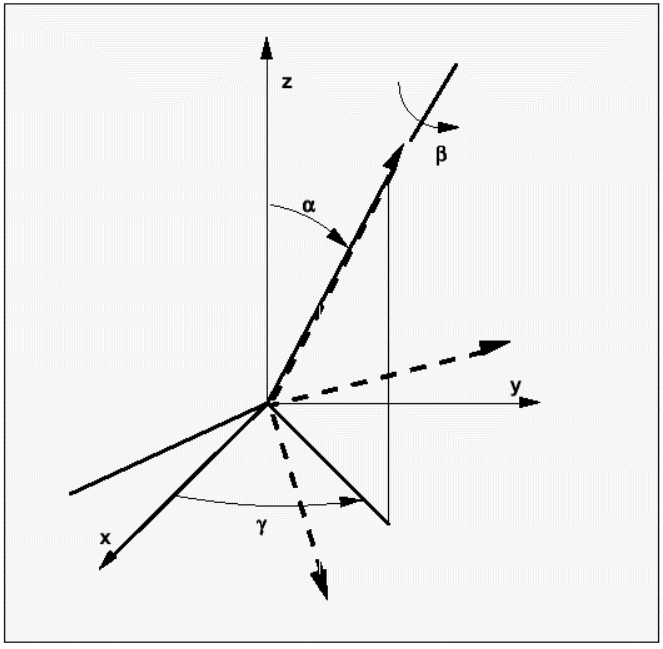
CIDA	Coordinate ID for rigid body A
CIDB	Coordinate ID for rigid body B
LCIDX	TABLED1 id for x-force versus x-translational relative displacement between the origins of CIDA and CIDB, based on the x-direction of CIDB. If zero, the applied force is set to 0.0.
LCIDY	TABLED1 id for y-force versus y-translational relative displacement between the origins of CIDA and CIDB, based on the x-direction of CIDB. If zero, the applied force is set to 0.0.
LCIDZ	TABLED1 id for z-force versus z-translational relative displacement between the origins of CIDA and CIDB, based on the x-direction of CIDB. If zero, the applied force is set to 0.0.
DLCIDX	TABLED1 id for x-damping force versus x-translational relative velocity between the origins of CIDA and CIDB, based on the x-direction of CIDB. If zero, the applied damping force is set to 0.0.

Field	Contents
DLCIDY	TABLED1 id for y-damping force versus y-translational relative velocity between the origins of CIDA and CIDB, based on the x-direction of CIDB. If zero, the applied damping force is set to 0.0.
DLCIDZ	TABLED1 id for z-damping force versus z-translational relative velocity between the origins of CIDA and CIDB, based on the x-direction of CIDB. If zero, the applied damping force is set to 0.0.
ESX	Elastic stiffness for friction and stop displacement for x-translation. If zero, friction and stop angles are inactive for x-translation.
FFX-TYPE	Type of friction force limit, as specified in FFX field: FFX-TYPE=CONSTANT FFX-TYPE=DISP
FFX	Limit on the frictional force. This option may also be thought of as an elastic-plastic spring. The interpretation of the value depends on the value of FFX-TYPE. If zero, friction is inactive for x-translation. FFX-TYPE=CONSTANT: Limit on the frictional force for x-translation. (Real > 0, Default = 0.) FFX-TYPE=DISP: Value is a TABLED1 id, specifying the yield force versus x-translation. (Integer > 0)
ESY	Elastic stiffness for friction and stop displacement for y-translation. If zero, friction and stop angles are inactive for y-translation.
FFY-TYPE	Type of friction force limit, as specified in FFY field: FFY-TYPE=CONSTANT FFY-TYPE=DISP
FFY	Limit on the frictional force. This option may also be thought of as an elastic-plastic spring. The interpretation of the value depends on the value of FFY-TYPE. If zero, friction is inactive for y-translation. FFY-TYPE=CONSTANT: Limit on the frictional force for y-translation. (Real > 0, Default = 0.) FFY-TYPE=DISP: Value is a TABLED1 id, specifying the yield force versus y-translation. (Integer > 0)
ESZ	Elastic stiffness for friction and stop displacement for z-translation. If zero, friction and stop angles are inactive for z-translation.

Field	Contents
FFZ-TYPE	Type of friction force limit, as specified in FFZ field: FFZ-TYPE=CONSTANT FFZ-TYPE=DISP
FFZ	Limit on the frictional force. This option may also be thought of as an elastic-plastic spring. The interpretation of the value depends on the value of FFZ-TYPE. If zero, friction is inactive for z-translation. FFZ-TYPE=CONSTANT: Limit on the frictional force for z-translation. (Real > 0, Default = 0.) FFZ-TYPE=DISP: Value is a TABLED1 id, specifying the yield force versus z-translation. (Integer > 0)
NSDX	Stop displacement for negative x-translation. Ignored if zero. (Real > 0, Default = ignored)
PSDX	Stop displacement for positive x-translation. Ignored if zero. (Real > 0, Default = ignored)
NSDY	Stop displacement for negative y-translation. Ignored if zero. (Real > 0, Default = ignored)
PSDY	Stop displacement for positive y-translation. Ignored if zero. (Real > 0, Default = ignored)
NSDZ	Stop displacement for negative z-translation. Ignored if zero. (Real > 0, Default = ignored)
PSDZ	Stop displacement for positive z-translation. Ignored if zero. (Real > 0, Default = ignored)

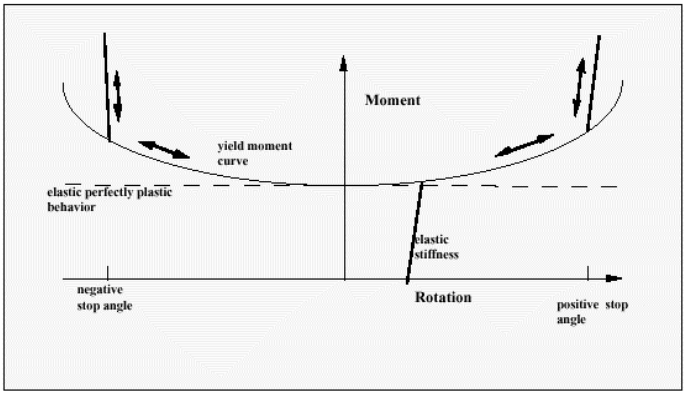
Field	Contents
FLEX-TOR	Entries for this continuation line describe a flexion-torsion joint stiffness.

After the stop angles are reached the torques increase linearly to resist further angular motion using the stiffness values specified here. Reasonable stiffness values have to be chosen. If the stiffness values are too low or zero, the stop will be violated.



Flexion-torsion joint angles. If the initial positions of the local coordinate a the two rigid bodies connected by the joint do not coincide, the angles, α and initialized and torques will develop instantaneously based on the define curves. The angle β is also initialized but no torque will develop about the axis on which β is measured. Rather, β will be measured relative to the con offset.

Field	Contents



Frictional behavior is modeled by a plasticity model. Elastic behavior is obtained once the stop angles are reached. The same elastic stiffness is used to simulate sticking situations.

CIDA	Coordinate ID for rigid body A
CIDB	Coordinate ID for rigid body B
LCIDAL	TABLED1 id for α-moment versus rotation in radians. The load-curve must be defined in the interval $0 \le \alpha \le \pi$. If zero, the applied moment is set to 0.0.
LCIDG	TABLED1 id for a scale factor versus γ-rotation in radians. The factor scales the α-moment. The load-curve must be defined in the interval $-\pi \le \gamma \le \pi$. If blank, the scale defaults to 1.0. (Integer>0)
LCIDBT	TABLED1 id for ß-torsion moment versus twist in radians. If zero, the applied twist is set to 0.0. (Integer > 0)
DLCIDAL	TABLED1 id for α-damping moment versus rate of rotation in radians per unit time. If zero, damping is not considered. (Integer>0)
DLCIDG	TABLED1 id for a scale factor versus γ-rotation rate in radians per unit time. The factor scales the α-damping moment. If blank, the scale defaults to 1.0. (Integer>0)
DLCIDBT	TABLED1 id for ß-dampint torsion moment versus rate of twist in radians per unit time. If zero, damping is not considered. (Integer > 0)
ESAL	Elastic stiffness per unit radian for friction and stop angles for α-rotation. If zero, friction and stop angles are inactive for α-rotation.

Field	Contents
FMAL-TYP	Type of friction moment limit, as specified in FMAL field: FMAL-TYP=CONSTANT FMAL-TYP=ROTATION FMAL-TYP=JOINT
FMAL	Limit on the frictional moment. This option may also be thought of as an elastic-plastic spring. The interpretation of the value depends on the value of FMAL-TYPE. If zero, friction is inactive for α-rotation. FMAL-TYP=CONSTANT: Limit on the frictional moment for α-rotation. (Real > 0, Default = 0.) FMAL-TYP=ROTATION: Value is a TABLED1 id, specifying the yield moment versus α-rotation. (Integer > 0) FMAL-TYP=JOINT: Value is a TABLED1 id, specifying the yield moment versus the RBJOINT reaction force. (Integer > 0)
ESBT	Elastic stiffness per unit radian for friction and stop angles for ß-rotation. If zero, friction and stop angles are inactive for ß-rotation.
FMBT-TYP	Type of friction moment limit, as specified in FMBT field: FMBT-TYP=CONSTANT FMBT-TYP=ROTATION FMBT-TYP=JOINT
FMBT	Limit on the frictional moment. This option may also be thought of as an elastic-plastic spring. The interpretation of the value depends on the value of FMBT-TYP. If zero, friction is inactive for ß-rotation. FMBT-TYP=CONSTANT: Limit on the frictional moment for ß-rotation. (Real > 0, Default = 0.) FMBT-TYP=ROTATION: Value is a TABLED1 id, specifying the yield moment versus ß-rotation. (Integer > 0) FMBT-TYP=JOINT: Value is a TABLED1 id, specifying the yield moment versus the RBJOINT reaction force. (Integer > 0)
SAAL	Stop angle in degrees for α-rotation, where $0 \le \alpha \le \pi$. Ignored if zero
NSABT	Stop angle in degrees for negative ß-rotation. Ignored if zero
PSABT	Stop angle in degrees for positive ß-rotation. Ignored if zero

RCONN (SOL 700) Rigid Connection

Defines a rigid connection between the different parts of Lagrangian meshes (tied surfaces).

Format:

1	2	3	4	5	6	7	8	9	10
RCONN	CID	STYPE	MTYPE	SID	MID	OPTION			
	CLSGAP	GAPDIS	GAPDISV						

Example:

RCONN	7	GRID	SURF	3	7	NORMAL			

Field	Content	
CID	Unique rigid-connection number (Integer, Required)	
STYPE	Type of entity used to define the slave surface (Character, Default = SURF)	
	SURF	A SURFACE entry is used to select the faces of the elements on the slave surface SID is the number of the BSURF entry. See Remark 2.
	GRID	Grid points will be tied to the master surface. SID then refers to a SET or BCGRID entry containing the list of grid points to be used. See Remarks 3. and 4.
MTYPE	Type of entity used to define the master surface (Character, Default = SURF)	
	SURF	A SURFACE entry is used to select the faces of the elements on the master surface. MID is the number of the BSURF entry.
SID	The number of a slave SURFACE entry or the number of a SET1 entry containing the list of grid points (Integer, Required)	
MID	The number of a master SURFACE entry (Integer, Required)	
OPTION	Only used if discrete grid points are tied to a surface (STYPE is equal toGRID). (Character, Default = NORMAL)	

Field	Content	
	NORMAL	The grid points are tied to the master surface. See Remark 3.
	SHELL	The grid points are attached to the edge of shell or beam elements, which are tied to the shell surface. See Remark 4.
CLSGAP	Switch to automatically close any gaps that are present between the master-slave surface (Character, Default = NO)	
	YES	Gaps are automatically closed
	NO	Gaps are not closed. See Remark 3.
GAPDIS	Defines the tolerance used in the search for a master face. If the distance between a slave point and a master face falls within this tolerance, the master face is accepted. If not, the search for a correct master face continues (Character, Default = DISTANCE)	
	DISTANCE	The tolerance has the length as specified at GAPDISV
GAPDISV	The value of the gap tolerance or a factor to calculate this tolerance depending on the value of GAPDIS (Real, Default = 1.0E20)	

Remarks

1. Corresponding LS-Dyna Inputs:

```
OPTION=NORMAL and CLSGAP=NO-> *CONTACT_TIED_NODES_TO_SURFACE
OPTION=NORMAL and CLSGAP=YES -> *CONTACT_TIED_NODES_TO_SURFACE_OFFSET
OPTION=SHELL and CLSGAP=NO -> *CONTACT_TIED_SHELL_EDGE_TO_SURFACE
OPTION=SHELL and CLSGAP=NO -> *CONTACT_TIED_SHELL_EDGE_TO_SURFACE_OFFSET
```

2. The RCONN entry can be used to define three types of connection :

   a. Two Surfaces Tied Together.

      Define slave and master segments representing the two surfaces to be tied together. There should not be a gap between the two sets of segments. The two surfaces are tied together during the analysis.

   b. Grid Points Tied to a Surface.

      If STYPE is set to GRID and OPTION is set to NORMAL, the slave entities comprise discrete grid points that are tied to the master surface during the analysis. The grid points must lie on the surface.

c. Shell Edge Tied to a Shell Surface.

If STYPE is set to GRID and OPTION is set to SHELL, the edges of shell or beams elements can be tied to the faces of other shells. The grid points attached to the edge of the shells/beams must be selected as the slave grid points. The shell surface to which they are tied must be selected as the master surface. The two sets will then be tied together throughout the analysis. All degrees of freedom will be coupled.

3. The CLSGAP entry enables you to define two different meshes that are not coincident over the master/slave interface. If the option is set to YES, the slave surface becomes coincident (according to projections) with the master surface.
4. The search method of the contact algorithm is used to find the closest master face. The tolerance defined with the GAPDIS/GAPDISV fields is similar to the monitoring distance defined on the CONTACT entry with the MONDIS/MONDISV fields.
5. The use of the gap closing CLSGAP can cause an element to collapse. This may happen if the GAPDISV tolerance is set to a value greater than the length of the side of an element.
6. When a solid and a shell mesh are tied together, the rotational degrees of freedom of the shell grid points are not coupled.
7. When OPTION=SHELL and CLSGAP=NO, the time step scale factor will be set to 0.4. This can be overwritten by:

   PARAM*, STEPFCTRCONN, xxx

RCROSS Cross-Power Spectral Density and Cross-Correlation Functions Output

Defines a pair of response quantities for computing the cross-power spectral density and cross-correlation functions in random analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RCROSS	SID	RTYPE1	ID1	COMP1	RTYPE2	ID2	COMP2	CURID	

Example:

RCROSS	10	DISP	100	3	STRESS	200	10	2	

Field	Contents
SID	Case Control RCROSS identification number for cross-power spectral density function and cross-correlation function. (Integer > 0)
RTYPEi	Type of response quantity. At lease one field must be selected. See Remark **2.** (Character or blank)
IDi	Element, grid or scalar point identification number. (Integer > 0)
COMPi	Component code (item) identification number. See Remark **3.** (Integer > 0)
CURID	Curve identification number. See Remark **4.** (Integer ≥ 0 or blank)

Remarks:

1. This entry is required for computing the cross-power spectral density function and cross-correlation function. SID must be selected with the Case Control command (RCROSS = SID). Fields RTYPE1, ID1, and COMP1 represent the first response quantity, and fields RTYPE2, ID2, and COMP2 the second in the correlation.

2. The keywords for field RTYPEi are listed as follows:

Keyword	Meaning
DISP	Displacement Vector
VELO	Velocity Vector
ACCEL	Acceleration Vector
OLOAD	Applied Load Vector
SPCF	Single-point Constraint Force Vector
MPCF	Multi-point Constraint Force Vector
STRESS	Element Stress
STRAIN	Element Strain
FORCE	Element Force

   If anyone of RTYPE1 or RTYPE2 is blank, then the default is the one same as the other field.

3. For elements, the item code COMPi represents a component of the element stress, strain, and force and is described in Tables "**Element Stress-Strain Item Codes**" on page 875 and "**Element Force Item Codes**" on page 917. For an item having both a real and imaginary part, the code of the real part must be selected. This is required for computing both the cross-power spectral density function and cross-correlation function.

   For grid point, the item code is one of 1, 2, 3, 4, 5, and 6, which represent the mnemonics T1, T2, T3, R1, R2, and R3, respectively. For scalar point, always use 1.

4. Field CURID is optional. It is for the user's covenience to identify the output by using a single index.

RELEASE Superelement Boundary Grid Point Release

Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.

Format:

1	2	3	4	5	6	7	8	9	10
RELEASE	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	-etc.-						

Example:

RELEASE	15	456	3	7	11	2	156	9	
	152	162							

Alternate Formats and Examples:

RELEASE	SEID	C	G1	“THRU”	G2				
RELEASE	6	2	15	THRU	127				

RELEASE	SEID	C	“ALL”						
RELEASE	127	156	ALL						

Field	Contents
SEID	Superelement identification number. (Integer > 0)
C	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)
Gi	Grid point identification numbers. (Integer > 0, “THRU”, or “ALL”; For THRU option, G1 < G2.)

Remarks:

1. A grid point referenced on this entry must be an exterior grid point of the superelement referenced on the entry.
2. In the first alternate format, all grid points in the sequence G1 through G2 are not required to be exterior grid points. Such grid points will collectively produce a warning message but will otherwise be ignored.

3. If the “ALL” option is used, all exterior grid points are released for the referenced degrees-of-freedom.
4. The RELEASE entry is applicable to only the superelement solution sequences (SOLs 101 through 200). It may not reference the residual structure (SEID = 0).
5. This entry is not supported for partitioned superelements.

RESTART (SOLs 600/700) Restart Data for MSC.Marc Executed from MD Nastran

Specifies writing or reading of restart data for Nonlinear Analysis when MSC.Marc or dytran-lsdyna is executed from MD Nastran. If this Bulk Data entry is found in the job stream, the type of "restart" specified by KIND and KTYPE will be performed. Only one RESTART entry is allowed.

Format:

1	2	3	4	5	6	7	8	9	10
RESTART	ID	KIND	NINC	NBEGIN	KMATRIX	IPRINT	LAST		
	NAME		TSTEP	ENDTIME	NSTEPS	NDCYC	STEPMAX	PERCENT	
	DTPLOT	DTTH	TSSFAC	DT2MS	RSF	NDUMP	KTYPE		
	RFILE								

SOL 600 Example(s):

RESTART	101	1	1						

The above example writes a restart file. The original run named is abcde.dat.

RESTART	201	3	1	15	0				
	abcde								

The above example restarts the original abcde.dat run. The name of the restart run must not be abcde.dat in this case.

RESTART	151	2		11	0	1	18		
	my_first	_run							

The above example reads the restart file and prints out results not printed in the original run.

SOL 700 Examples:

RESTART	101	1	1000						

The previous above writes restart files on the original run.

RESTART	201	3	2000						
				60.E-3					
	1.E-4	2.E-5				02	1		
	myrestar	t2.dat							

The above example restarts a previous analysis and resets termination time and some output intervals.

Field	Contents
ID	Identification number of the restart entry -- Not presently used (Integer)
KIND (2,1)	Type of restart (Integer > 0; required field, no default) 1 = Write a restart file 2 = Restart a previous analysis (Read an existing restart file) 3 = Restart a previous analysis and write new data on restart file. 11 = Only write restart file for the last converged increment of the run (SOL 600) or time step (SOL 700). 12 = Read a restart file written with KIND=11 13 = Read a restart file written with KIND=11 and write the last increment or time step of the present run on that file as well. KINDs 1-3 and 11-13 are available for SOL 600. KINDs 1-3 are available for SOL 700.
NINC (2,2)	Number of increments between writing of restart data for SOL 600, default=100000000 for SOL700. (Integer > 0; Default = 1)
NBEGIN (2,3)	The “time” increment at which the restart run begins (used only if KIND=2 or 3). (SOL 600 only) (Integer > 0. See Remarks 4., 5.)
KMATRIX (2,7)	Set this field to 1 if the decomposed stiffness matrix is to be saved on the restart file (not recommended due to large disk storage). (SOL 600 only) (Integer ≥ 0; Default = 0)
IPRINT (2,8)	Set this field to 1 if the restart data is to be printed (All data from increment INCBEG to LAST will be printed if IPRINT=1). Use this option if printing on a previous run was suppressed but now is desired. .(SOL 600 only) (Integer ≥ 0; Default = 0)
LAST (2,9)	The last time increment is printed if IPRINT=1, otherwise this field is ignored. (SOL 600 only) (Integer > 0, Default = 0. See Remarks 4., 5.)

Field	Contents
NAME	Name of input file for the original MD Nastran run without extension. NAME is limited to 16 characters and may not contain imbedded blanks. If the small field format is used, NAME may span fields 2 and 3 of the continuation entry. If the large field is used, NAME should be coded in the 2nd field. NAME is required for a restart run, see Remarks 7., 8. If NAME exceeds 8 characters, the continuation line must be coded in small-field fixed format or in large field fixed or free-format. (SOL 600 only)
TSTEP [2.1]	Time step size after restart -- For dynamic problems only. (SOL 600 only) (Real $\geq$ 0.0 or blank; if the value is $\leq$ 0.0 the original step size is used)
ENDTIME [2,2]	Ending time for this restart run (Real $\geq$ 0.0 or blank, if the value is $\leq$ 0.0 the original step size is used)
NSTEPS [2,3]	Total number of time steps for the restart run plus the original run. (SOL 600 only) (Integer $\geq$ 1, Default = 1)
NDCYC [2,6]	Desired number of recycles if AUTO INCREMENT options were specified on the original run. (SOL 600 only) (Integer $\geq$ 0; if the value is $\leq$ 0.0 the original step size is used)
STEPMAX [2,7]	Maximum step size if AUTO INCREMENT options were specified on the original run. (SOL 600 only) (Real $\geq$ 0.0; if the value is $\leq$ 0.0 the original step size is used)
PERCENT [2,8]	Percentage of total load to be applied. (SOL 600 only) (Real $\geq$ 0.0; if the value is $\leq$ 0.0 the original step size is used)
DTPLOT	Dt for complete output states (D3PLOT). (SOL 700 only) (Real $\geq$ 0.0; if the value is $\leq$ 0.0 the original Dt is used)
DTTH	Dt for time history data of element subsets (D3THDT). (SOL 700 only) (Real $\geq$ 0.; if the value is $\leq$ 0.0 the original Dt is used)
TSSFAC	Scale factor for computed time step. EQ:0.0. TSSFAC remains unchanged. (SOL 700 only)
DT2MS	New time step for mass scaled calculations. Mass scaling must be active in the time zero analysis. EQ:0.0. DT2MS remains unchanged. (SOL 700 only)

Field	Contents
RSF	Type of binary output restart file (Integer > 0, Default = 0, see Remark 11.)
	1 = Create or use RUNRSF file (Create or use depends on KIND)
	0 = Create or use D3DUMP file (Create or use depends on KIND)
	2 = Create (in original run) both D3DUMP and RUNRSF (Applicable only if KIND=1)
	(SOL 700 only)
NDUMP	D3DUMPnn file to start with. (Integer, Default = 0)
	dytran-lsdyna generates file named D3DUMP00, D3DUMP01, D3DUMP02, etc. The value of NDUMP selects which of these to use for restart. For example, if NDUMP=25, DEDUMP25 will be used. (SOL 700 only)
KTYPE	Type of restart (Integer $\geq$ 0, Default = 0)
	0 = Simple -- To restart a job which did not fully complete (RFILE not required). See Remark 13.
	1 = Minor -- To restart a job that did not finish or to run longer and change items specified by Bulk Data entries RESTART, DYDELEM, DYRIGSW, DYTERMT, DYRLAX and/or DYCHANG (RFILE must be entered). See Remarks 13. and 14.
	2 = Full -- To restart a job that did not finish or to run longer and to make changes to the input deck for items other than those specified in a minor restart (RFILE must be entered). See Remarks 13. and 14.
	(SOL 700 only)
RFILE	Name of an input data restart file, if any that will be used by dytran-lsdyna as the ASCII text input file for a restart run, (Character; no Default)
	The name of this file can be up to 64 characters long and can use fields 2-9 if necessary. The file extension, if any, should be specified. In the above example, the file is named myrestart2.dat.
	(SOL 700 only)

Remarks:

1. RESTART is available only when MSC.Marc is executed from within MD Nastran Implicit Nonlinear (SOL 600) or from within MD Nastran Explicit Nonlinear (SOL 700)
2. There should only be one RESTART entry in the bulk data. If more than one exists, the first one will be used.
3. The KMATRIX option must be the same on all runs.
4. (i,j) Indicate the field in MSC.Marc's RESTART model definition options. [i,j] Indicate the field in MSC.Marc's REAUTO model definition options.
5. The jid.marc.t16 and jid.marc.t08 files must be saved from the first run when a restart run is executed. Both original and restart runs must be located in the same directory. File extensions are .t08 and t16.
6. A restart run may not have the same input file name (jid) as that of the original run. In other words, if the input file for the original run is named abcd.dat, the input file for a restart run may not be named abcd.dat.
7. For static analyses, normally each load case has a total time of 1.0. The first case goes from 0.0 to 1.0, the second from 1.0 to 2.0, etc. If the first run has two static load cases and terminates at 1.6, it is in the middle of the second load case. The original run should be examined to determine which increment (before 1.6) to begin the restart run.
8. In the second example above, the original run was named my_first_run.dat (or first_run.bdf, etc). This is a small field example. There are 8 characters in the 2nd field of the continuation line and 4 characters in the third field. The name can start anywhere within the two fields. There must not be any blank spaces in the name. If the name exceeds 8 characters, the continuation line must be coded in small-field fixed format or in large-field fixed or free format. NAME is limited to a maximum of 16 characters.
9. Both original and restart run names (jid's) should use only lower case letters except on computer systems that are not case sensitive. "NAME" (continue line, fields 2-3) will be converted automatically to lower case.
10. The restart run must use the same values of MSC.Marc's DIST LOAD parameter as the original run or it may fail. Be sure to use parameters MARCDIS2, MARCDIS3 and MARCDIS4 to set these values to be exactly the same as the original run (examine jid.marc.dat from the original run to determine these values before submitting the new run).

11. The RSF options creates a complete database which is necessary for restarts. When RSF=0 is specified, the same file is overwritten after each interval. When RSF=1 is specified, a new restart file is created after each interval, thus a “family” of files is created numbered sequentially jid.dytr.d3dump10, jid.dytr.d3dump02, etc. These files can take significant disk space but are important if a model might need to be modified prior to the end time.
12. It is not necessary to enter continuation lines if not needed for the particular job to be run.
13. Restarts for SOL 700 are not presently available for Windows PC systems.
14. Minor and Full restarts (KTYPE1 and 2) are not presently available. Simple reastarts require hte use of

    PARAM,MDYRSTNC,2

    and the variables in fields 2 - 6 of the second continuation are not presently available.

RFORCE Rotational Force

Defines a static loading condition due to an angular velocity and/or acceleration.

Format:

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	G	CID	A	R1	R2	R3	METHOD	
	RACC	MB							

Example:

RFORCE	2	5		-6.4	0.0	0.0	1.0	2	
	1.0								

Field	Contents
SID	Load set identification number. (Integer > 0)
G	Grid point identification number through which the rotation vector acts. (Integer $\geq$ 0)
CID	Coordinate system defining the components of the rotation vector. See Remark **16.** (Integer $\geq$ 0, Default = 0)
A	Scale factor of the angular velocity in revolutions per unit time. (Real)
R1, R2, R3	Rectangular components of rotation vector $\vec{R}$. The vector defined will pass through point G. (Real; $R1^2 + R2^2 + R3^2 > 0.0$ unless A and RACC are both zero)
METHOD	Method used to compute centrifugal forces due to angular velocity. For angular acceleration, see Remark **13.** (Integer = 1 or 2; Default = 1)
RACC	Scale factor of the angular acceleration in revolutions per unit time squared. (Real; Default = 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark **15.** (Integer; Default = 0)

Remarks:

1. The forces that are created with the RFORCE entry act on the structure as follows: the forces that are defined with the RFORCE entry for a constant angular velocity (A), act in the positive radial direction. These forces represent the inertia forces on the structure due to a constant angular velocity. The forces that are defined with the RFORCE entry for a constant angular acceleration (RACC), act in the same direction as the angular acceleration. These forces would be opposite to the inertia forces on the structure due to a constant angular acceleration. In **Figure 8-155**, the force vector at grid point Gi is given by

$$\{\vec{F}\}_i = [m]_i[\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)] \qquad \text{Eq. 8-7}$$

where

$$\text{angular velocity} = \vec{\omega} = 2\pi A \cdot \vec{R} \text{ (radians/unit time)}$$

$$\text{angular acceleration} = \vec{\alpha} = 2\pi RACC \cdot \vec{R} \text{ (radians/unit time)}$$

$$[m]_i = 3 \times 3 \text{ translational mass matrix at grid point Gi}$$

Note: The equation for $\vec{F}_i$ will have additional terms if the mass is offset or I^i_{23}, I^i_{13} terms exist relative to the rotation axes and METHOD = 1 is selected.

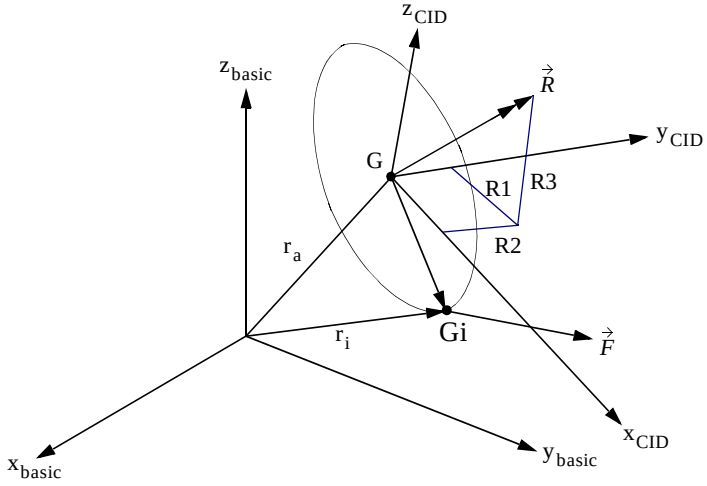


Figure 8-155 RFORCE Vector at Grid Point Gi

2. In the static solution sequences, the load set ID (SID) is selected by the Case Control command LOAD. In the dynamic solution sequences, SID must be referenced in the LID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
3. G = 0 signifies that the rotation vector acts through the origin of the basic coordinate system.
4. CID = 0 (Default) signifies that the rotation vector is defined in the basic coordinate system.
5. The load vector generated by this entry can be printed with an OLOAD command in the Case Control Section.
6. METHOD = 1 yields correct results only when there is no coupling in the mass matrix. This occurs when the lumped mass option is used with or without the ZOFFS option (see the CQUAD4 entry for a description of ZOFFS). METHOD = 2 yields correct results for lumped or consistent mass matrix only if the ZOFFS option is not used. The acceleration terms due to the mass offset (X1, X2, X3) on the CONM2 entry are not computed with METHOD = 2. All the possible combinations of mass matrices and offset and the correct method to be used are shown below.

	No Offset	Offset
Lumped	METHOD = 1 or METHOD = 2	METHOD = 1
Coupled	METHOD = 2	Neither

7. In cyclic symmetry analyses, the T3 axis of the basic coordinate system must be coincident with the axis of symmetry. In the DIH type of cyclic symmetry, the T1 axis also must be parallel to side 1 of segment 1R of the model.
8. For superelement analysis, G should reference a residual structure point that is exterior to all superelements. If it is not exterior to a superelement, then centrifugal loads will not be generated for that superelement. However, in cyclic analysis, User Fatal Message 4347 will be issued.
9. In a geometric nonlinear static analysis (SOL 106 when PARAM LDGISP is set to +1), this type of loading is a follower force type of loading. However, the orientation of coordinate system CID is not updated.

10. In nonlinear static solutions when there is more than one increment (INC) specified on the NLPARM entry for a given subcase, the load vector resulting from the RFORCE input (and not the angular velocity vector) is scaled linearly. This means that loading by increments in the angular velocity can only be achieved by having subcases where the RFORCE loading is applied in a single increment.
11. The continuation entry is optional.
12. Forces due to angular acceleration (RACC) are computed with METHOD = 2 even if METHOD = 1 is specified.
13. Loads derived from this entry do not include effects due to mass specified for scalar points.
14. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter "**FOLLOWK**" in Chapter 5). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, and 159, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106 and 153) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
15. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.
16. If CID is not a rectangular coordinate system, RFORCE will treat it as if it were and unexpected answers may result.
17. Follower force stiffness (param,followk,yes) is supported for method 2 only.

RGYRO Rotordynamic Analysis Parameters

Specifies synchronous or asynchronous analysis, reference rotor, and rotation speed of the reference rotor.

Format:

1	2	3	4	5	6	7	8	9	10
RGYRO	RID	SYNCFLG	REFROTR	SPDUNIT	SPDLOW	SPDHIGH	SPEED		

Example:

RGYRO	100	SYNC	1	RPM	1000.0	5000.0			

Field	Contents
RID	Identification number of RGYRO entry. Selected by Case Control command, RGYRO. (Required, no Default)
SYNCLFG	Specifies whether the analysis is synchronous or asynchronous analysis. Required input for frequency response and complex modes analyses. Not required for static analyses. (Character: 'SYNC', 'ASYNC', or blank)
REFROTR	Specifies the reference rotor ID for the analysis. (Integer > 0; Required, no Default)
SPDUNIT	Specifies whether the entries SPDLOW, SPDHIGH, and SPEED are given in terms of RPM (revolutions / minute) or frequency (revolutions (cycles) / unit time). (Character: 'RPM' or 'FREQ'; no Default)
SPDLOW	Specifies the low speed for synchronous analysis. See Remark **2.** (Real; Default = 0.0)
SPDHIGH	Specifies the high speed for synchronous analysis. See Remark **2.** (Real, Default = 99999.0)
SPEED	Specifies reference rotor speed for asynchronous analysis. Also required for static analyses. See Remark **2.** (Default = 0)

Remarks:

1. Multiple RGYRO entries with the same RID value are not allowed.

2. The required information on the RGYRO entries varies for different analyses. Values for the RID and SPDUNIT fields are always required. Values for SPDLOW, SPDHIGH and SPEED are analysis dependent as shown in the table below:

Solution Sequence	Type of Analysis	PARAM, GYROAVG	Required Entry	COMMENT
Frequency Response	SYNC SYNC ASYNC ASYNC	0 -1 0 -1	None SPDLOW, SPDHIGH SPEED SPEED	-- a, b -- b
Complex Modes	SYNC ASYNC	-- --	SPDLOW, SPDHIGH SPEED	a, b b
Static Analysis	--	--	SPEED	--

a. The relative rotor speeds will be treated as linarly dependent on the reference rotor speed ($\Omega = A0 + A1\Omega_{reference}$). The scale factors A0 and A1 will be determined by a least-mean-square fit of the relative rotor speeds input on the RSPINR entries between SPDLOW and SPDHIGH of the reference rotor. If SPDLOW or SPDHIGH are outside the range specified on the RSPINR entry, the values will be extrapolated from the RSPINR entry values.

b. PARAM, WR3 and PARAM, WR4 are required for rotor damping.

RINGAX Conical Shell Ring

Defines a ring for conical shell problems.

Format:

1	2	3	4	5	6	7	8	9	10
RINGAX	ID		R	Z			PS		

Example:

RINGAX	3		2.0	-10.0			162		

Field	Contents
ID	Ring identification number. See Remark **6.** (Integer > 0)
R	Ring radius. (Real > 0.0)
Z	Ring axial location. (Real)
PS	Permanent single-point constraints. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

Remarks:

1. RINGAX is allowed only if an AXIC entry is also present.
2. The number of degrees-of-freedom defined is $(6 - \text{NPS}) \cdot H$ where H is the harmonic count and NPS is the number of digits in field 8. (See "**AXIC**" on page 1047).
3. RINGAX identification numbers must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.
4. For a discussion of the conical shell problem, see "**Conical Shell Element (RINGAX)**" on page 155 of the *MSC.Nastran Reference Guide.*
5. Constraints may be necessary to avoid matrix singularities. The CONEAX element has no stiffness for rotation about the normal. In addition, there is no stiffness for rotation about V (see **Figure 8-156**) when transverse shear flexibility is not included.

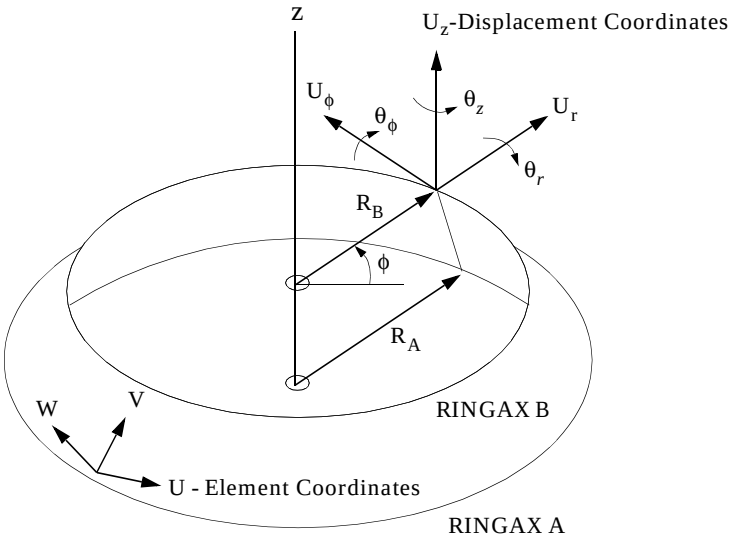


Figure 8-156 RINGAX Coordinate System

6. In order to reference this entry on a SET Case Control command, the ID must be modified by ID(n) = ID + 1000000 · n where n is the harmonic number plus one and ID(n) is the value specified on the SET entry.

RINGFL Axisymmetric Fluid Point

Defines a circle (fluid point) in an axisymmetric fluid model.

Format:

1	2	3	4	5	6	7	8	9	10
RINGFL	IDFA	XA1	XA2	XA3	IDFB	XB1	XB2	XB3	

Example:

RINGFL	3	1.0		30.0					

Field	Contents
IDFA, IDFB	Unique identification number of the fluid points. (0 < Integer < 500000)
XAi, XBi	Coordinates of the point defined in the coordinate system specified on the AXIF entry. (Real; XA1 and XB1 > 0.0)

Remarks:

1. RINGFL is allowed only if an AXIF entry is also present.
2. All fluid point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. X1, X2, X3 are (r, ϕ, z) for a cylindrical coordinate system and (ρ, θ, ϕ) for a spherical coordinate system. θ is in degrees. The value of ϕ must be blank or zero.
4. One or two fluid points may be defined per entry.

RJOINT Rigid Joint

Defines a rigid joint element connecting two coinciding grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RJOINT	EID	GA	GB	CB					

Example:

RJOINT	5	1	2	12345					

Field	Contents
EID	Element identification number. (Integer > 0)
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB. These degrees-of-freedom are constrained to move with the same degrees-of-freedom at GA. See Remarks **4.** and **5.** (Integers 1 through 6 with no embedded or blank.)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, MD Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CB.
3. The length between grid points GA and GB must be zero.
4. When CB = “123456” or blank, the grid point GB is constrained to move with GA and the two grid points moves as a single point. For default CB = “123456”.
5. If any degree-of-freedom is released on CB, RJOINT becomes a mechanical joint element. For example, CB = “12345”, then RJOINT becomes a hinge. CB = “1234”, then RJOINT becomes a universal joint. And CB = “123”, RJOINT becomes a spherical joint.

6. For the Lagrange method, the theory for the RJOINT is formulated such that a consistent mechanical joint is created even if the user requests different global coordinate systems at grid points GA and GB.
7. Thermal expansion effect is not applicable for the RJOINT element, since the distance between grid points GA and GB is zero.
8. Element identification numbers should be unique with respect to all other element identification numbers.

RLOAD1 Frequency Response Dynamic Excitation, Form 1

Defines a frequency-dependent dynamic load of the form

$$\{P(f)\} = \{A\}[C(f) + iD(f)]e^{i\{\theta - 2\pi f\tau\}}$$

for use in frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID	EXCITEID	DELAY	DPHASE	TC	TD	TYPE		

Example:

RLOAD1	5	3			1				

Field	Contents
SID	Set identification number. (Integer > 0)
EXCITEID	Identification number of the DAREA or SPCD entry set that defines $\{A\}$. See Remarks **5.** and **6.** (Integer > 0)
DELAY	Defines time delay τ. (Integer $\geq$ 0, real or blank.) If it is a non-zero integer, it represents the identification number of DELAY Bulk Data entry that defines τ. If it is real, then it directly defines the value of τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See also Remark **2.**
DPHASE	Defines phase angle θ. (Integer $\geq$ 0, real or blank.) If it is a non-zero integer, it represents the identification number of DPHASE Bulk Data entry that defines θ (in degrees). If it is real, then it directly defines the value of θ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See also Remark **2.**
TC	Set identification number of the TABLEDi entry that gives $C(f)$. See Remark **2.** (Integer $\geq$ 0)
TD	Set identification number of the TABLEDi entry that gives $D(f)$. See Remark **2.** (Integer $\geq$ 0)
TYPE	Defines the type of the dynamic excitation. See Remarks **5.** and **6.** (Integer, character or blank; Default = 0)

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.
2. If any of DELAY, DPHASE, TC, or TD fields are blank or zero, the corresponding $\tau, \theta, C(f)$ or $D(f)$ will be zero. Either TC or TD may be blank or zero, but not both.
3. RLOAD1 excitations may be combined with RLOAD2 excitations only by specification on a DLOAD entry. That is, the SID on a RLOAD1 entry must not be the same as that on a RLOAD2 entry.
4. SID must be unique for all RLOAD1, RLOAD2, TLOAD1, TLOAD2, and ACSRCE entries.
5. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS or DISP	Enforced displacement using SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration SPC/SPCD data

6. TYPE (field 8) also determines the manner in which EXCITEID (field 3) is used by the program as described below:

   Excitation specified by TYPE is applied load

   - *There is no LOADSET request in Case Control*

     EXCITEID may also reference DAREA, static and thermal load set entries.

   - *There is a LOADSET request in Case Control*

     The program may also reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID.

   Excitation specified by TYPE is enforced motion

   - *There is no LOADSET request in Case Control*

     EXCITEID will reference SPCD entries.

- *There is a LOADSET request in Case Control*

  The program will reference SPCD entries specified by the LID field in the selected LSEQ entries corresponding to the EXCITEID.

RLOAD2 Frequency Response Dynamic Excitation, Form 2

Defines a frequency-dependent dynamic excitation of the form.

$$\{P(f)\} = \{A\} \cdot B(f)e^{i\{\phi(f) + \theta - 2\pi f\tau\}}$$

for use in frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID	EXCITEID	DELAY	DPHASE	TB	TP	TYPE		

Example:

RLOAD2	5	3			7				

Field	Contents
SID	Set identification number. (Integer > 0)
EXCITEID	Identification number of the DAREA or SPCD entry set that defines $\{A\}$. See Remarks **5.** and **6.** (Integer > 0)
DELAY	Defines time delay τ. (Integer $\geq$ 0, real or blank.) If it is a non-zero integer, it represents the identification number of DELAY Bulk Data entry that defines τ. If it is real, then it directly defines the value of τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See also Remark **2.**
DPHASE	Defines phase angle θ. (Integer $\geq$ 0, real or blank.) If it is a non-zero integer, it represents the identification number of DPHASE Bulk Data entry that defines θ (in degrees). If it is real, then it directly defines the value of θ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See also Remark **2.**
TB	Set identification number of the TABLEDi entry that gives $B(f)$. (Integer > 0)
TP	Set identification number of the TABLEDi entry that gives $\phi(f)$ in degrees. (Integer $\geq$ 0)
TYPE	Defines the type of the dynamic excitation. See Remarks **5.** and **6.** (Integer, character or blank; Defaults = 0)

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.
2. If any of DELAY, DPHASE, or TP fields are blank or zero, the corresponding τ, θ, or $\phi(f)$ will be zero.
3. RLOAD2 excitations may be combined with RLOAD1 excitations only by specification on a DLOAD entry. That is, the SID on a RLOAD2 entry must not be the same as that on a RLOAD1 entry.
4. SID must be unique for all RLOAD1, RLOAD2, TLOAD1, TLOAD2, and ACSRCE entries.
5. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS or DISP	Enforced displacement using SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration SPC/SPCD data

6. TYPE (field 8) also determines the manner in which EXCITEID (field 3) is used by the program as described below:

   Excitation specified by TYPE is applied load

   - *There is no LOADSET request in Case Control*

     EXCITEID may also reference DAREA, static and thermal load set entries.

   - *There is a LOADSET request in Case Control*

     The program may also reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID.

   Excitation specified by TYPE is enforced motion

   - *There is no LOADSET request in Case Control*

     EXCITEID will reference SPCD entries.

- *There is a LOADSET request in Case Control*

  The program will reference SPCD entries specified by the LID field in the selected LSEQ entries corresponding to the EXCITEID.

ROTORG Rotor Line Model Grids

Specifies grids that compose the rotor line model.

Format:

1	2	3	4	5	6	7	8	9	10
ROTORG	ROTORID	GRID1	GRID2	...	GRIDn				

or

ROTORG	ROTORID	GRID1	THRU	GRID2	BY	INC			

Example:

ROTORG	100	101	1002	103	4001				

ROTORG	200	1001	THRU	1100	BY	2			

Field	Contents
ROTORID	Identification number of rotor. (Integer > 0; Required)
GRIDi	Grids comprising the rotor. (Integer > 0; Required, no Default)
THRU	Specifies a range of identification numbers. (Optional)
BY	Specifies an increment for a THRU specification (Optional)
INC	Increment for THRU range. (Integer > 0; Default = 0)

Remarks:

1. Grid entries must be unique, duplicate entries will produce a fatal error.
2. Multiple ROTORG entries with the same ROTORID are supported.
3. All grids specified on ROTORG entries for a specific ROTORID must be collinear. Collinearity will be checked.

ROTORSE Rotor Superelement Identification

Specifies grids that compose the rotor line model.

Format:

1	2	3	4	5	6	7	8	9	10
ROTORSE	ROTORID	SEID	SEOPT						

Example:

ROTORSE	10	1							

Field	Contents
ROTORID	Identification number of rotor group. (Integer > 0, Required)
SEID	Superelement identification number of rotor superelement. (Integer < 0, Required)
SEOPT	Form of superelement for calculation of gyroscopic terms, see Remark **3.** (Integer = 1, 2, or 3; Default = 1)

Remarks:

1. Multiple ROTORSE entries with the same ROTORID are not supported.
2. ROTORG entries with the same ROTORID are not supported.
3. SEOPT will determine which superelement form will be used for the calculation of the gyroscopic terms. Options are:

   1- Reduced (condensed) superelement, including upstream superelement contributions (A-set).

   2- The specified SEID superelement including upstream superelement contributions (G-set).

   3- The specified SEID superelement, without upstream superelement contributions (J-set).
4. All grids in the SEOPT set must be collinear. Collinearity will be checked.
5. Rotors specified using the ROTORSE entry can be connected directly to the support structure. Using rigid elements to keep the rotor separate in the G-set of the residual structure is not required.
6. Static and component mode reduction of the rotor line moel are supported.

RROD Rigid Pin-Ended Element Connection

Defines a pin-ended element that is rigid in translation.

Format:

1	2	3	4	5	6	7	8	9	10
RROD	EID	GA	GB	CMA	CMB	ALPHA			

Example:

RROD	14	1	2	2		6.5-6			

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification numbers of connection points. (Integer > 0)
CMA, CMB	Component number of one and only one dependent translational degree-of-freedom in the global coordinate system assigned by the user to either GA or GB. (Integer 1, 2, or 3. Either CMA or CMB must contain the integer, and the other must be blank for the linear RROD. For Lagrange RROD, both CMA and CMB can be blank.) See Remark **3.**
ALPHA	Thermal expansion coefficient. See Remark **11.** (Real > 0.0 or blank)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, MD Nastran will create internally one Lagrange multiplier degree-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points.
3. For the Lagrange method, if both CMA and CMB are blanks, MD Nastran will compute the best degree-of-freedom for the dependent degree-of-freedom.
4. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.

5. Element identification numbers should be unique with respect to all other element identification numbers.
6. Rigid elements, unlike MPCs, are not selected through the Case Control command, MPC.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
8. Rigid elements are ignored in heat transfer problems.
9. The degree-of-freedom selected to be dependent must have a nonzero component along the axis of the element. This implies that the element must have finite length.
10. See "**Rigid Elements and Multipoint Constraints (R-type, MPC)**" on page 167 of the *MSC.Nastran Reference Guide* for a discussion of rigid elements.
11. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and Gb.

RSPINR Relative Spin Rates Between Rotors

Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RSPINR	ROTORID	GRIDA	GRIDB	SPDUNT	SPTID				
	GR	ALPHAR1	ALPHAR2	HYBRID					

Example:

RSPINR	100	1001	1002	RPM	100				
	0.02			1001					

Field	Contents
ROTORID	Identification number of rotor. (Integer > 0; Required)
GRIDA / GRIDB	Positive rotor spin direction is defined from GRIDA to GRIDB (Integer > 0; Required. See Remark **2.**)
SPDUNIT	Specifies whether the listing of relative spin rates is given in terms of RPM (revolutions / minute) or frequency (revolutions (cycles) / unit time). (Character; 'RPM' or 'FREQ'; Required)
SPTID	Rotor structural damping factor. See Remark **3.** (Real; Required)
GR	Rotor structural damping factor. See Remark **3.** (Real, Default = 0.0)
ALPHAR1	Scale factor applied to the rotor mass matrix for Rayleigh damping (Real, Default = 0.0. See Remark **5.**)
ALPHAR2	Scale factor applied to the rotor stiffness matrix for Rayleigh damping (Real; Default = 0.0. See Remark **5.**)
HYBRID	Identification number of HYBDMP entry for hybrid damping (Integer $\geq$ 0; Default = 0. See Remark **6.**)

Remarks:

1. A RSPINR entry must be present for each rotor defined by a ROTORG entry.
2. The rotor spin axis is determined from the ROTORG entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG entry.

3. If the entry is a real number, the value is considered constant. If the entry is an integer number, the value references a DDVAL entry that specifies the relative rotor spin rates. The number of spin rates for each rotor must be the same. Relative spin rates are determined by correlation of table entries. The ith entry for each rotor specifies the relative spin rates between rotors at RPMi/FREQi. Spin rates for the reference rotor must be in ascending or descending order.
4. Rotor structural damping specified by the GR entry will be added as equivalent viscous damping. The equivalent damping will be calculated using:

$$[B_{rotor}]_{structural} = \left(\frac{GR}{WR3}\right)[K_{rotor}]$$

where $WR3$ is a user parameter

5. Rayleigh damping for the rotor will be calculated as

$$[B_{rotor}]_{Rayleigh} = \alpha_{R1}(M_{rotor}) + \alpha_{R2}[K_{rotor}]$$

6. For hybrid damping of the rotors, only the rotor mass and stiffness will be used for the modes calculation.

RSPINT Rotor Spin Rates

Specifies rotor spin rates for nonlinear transient analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RSPINT	ROTORID	GRIDA	GRIDB	SPDUNT	SPTID	SPDOUT			
	GR	ALPHAR1	ALPHAR2	HYBRID					

Example:

RSPINT	100	1001	1002	RPM	1001				
	0.01	0.01	0.002						

Field	Contents
ROTORID	Identification number of rotor. (Integer > 0; Required)
GRIDA / GRIDB	Positive rotor spin direction is defined from GRIDA to GRIDB (Integer > 0; Required. See Remark **2.**)
SPDUNIT	Specifies whether the spin rates are given in terms of RPM (revolutions / minute) or frequency (revolutions(cycles) / unit time). (Character; 'RPM' or 'FREQ'; Required)
SPTID	Relative rotor spin rates (Real or Integer; if integer, must be > 0. See Remark 3, Required)
SPDOUT	EPOINT to output the rotor speed vs. time. Output will be in SPDUNITs (Integer > 0 or blank)
GR	Rotor structural damping factor. See Remark **3.** (Real, Default = 0.0)
ALPHAR1	Scale factor applied to the rotor mass matrix for Rayleigh damping (Real, Default = 0.0. See Remark **5.**)
ALPHAR2	Scale factor applied to the rotor stiffness matrix for Rayleigh damping (Real; Default = 0.0. See Remark **5.**)
HYBRID	Identification number of HYBDMP entry for hybrid damping (Integer $\geq$ 0, Default = 0. See Remark **6.**)

Remarks:

1. A RSPINT entry must be present for each rotor defined by a ROTORG entry.

2. The rotor spin axis is determined from the ROTORG entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG entry.
3. If the entry is a real number, the value is considered constant. If the entry is an integer number, the value references a TABLED1 entry that specifies the rotor spin rate history.
4. Rotor structural damping specified by the GR entry will be added as equivalent viscous damping. The equivalent damping will be calculated using:

$$[B_{rotor}]_{structural} = \left(\frac{GR}{WR3}\right)[K_{rotor}]$$

   where WR3 is a user parameter.
5. Rayleigh damping for the rotor will be calculated as

$$[B_{rotor}]_{Rayleigh} = \alpha_{R1}(M_{rotor}) + \alpha_{R2}[K_{rotor}]$$

6. For hybrid damping of the rotors, only the rotor mass and stiffness will be used for the modes calculation.

RSPLINE Interpolation Constraint Element

Defines multipoint constraints for the interpolation of displacements at grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RSPLINE	EID	D/L	G1	G2	C2	G3	C3	G4	
	C4	G5	C5	G6	-etc.-				

Example:

RSPLINE	73	.05	27	28	123456	29		30	
	123	75	123	71					

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
D/L	Ratio of the diameter of the elastic tube to the sum of the lengths of all segments. (Real > 0.0; Default = 0.1)
Gi	Grid point identification number. (Integer > 0)
Ci	Components to be constrained. See Remark **2.** (Blank or any combination of the Integers 1 through 6.)

Remarks:

1. Displacements are interpolated from the equations of an elastic beam passing through the grid points. This is a linear method only element, and not controlled with the Case Control command RIGID.
2. A blank field for Ci indicates that all six degrees-of-freedom at Gi are independent. Since G1 must be independent, no field is provided for C1. Since the last grid point must also be independent, the last field must be a Gi, not a Ci. For the example shown G1, G3, and G6 are independent. G2 has six constrained degrees-of-freedom while G4 and G5 each have three.
3. Dependent (i.e., constrained) degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
4. Degrees-of-freedom declared to be independent by one rigid body element can be made dependent by another rigid body element or by a multipoint constraint.

5. EIDs must be unique.
6. Rigid elements (including RSPLINE), unlike MPCs, are not selected through the Case Control Section.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
8. Rigid elements are ignored in heat transfer problems.
9. See "**Rigid Elements and Multipoint Constraints (R-type, MPC)**" on page 167 of the *MSC.Nastran Reference Guide* for a discussion of rigid elements.
10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
11. The constraint coefficient matrix is affected by the order of the Gi Ci pairs on the RSPLINE entry. The order of the pairs should be specified in the same order that they appear along the line that joins the two regions. If this order is not followed then the RSPLINE will have folds in it that may yield some unexpected interpolation results.
12. The independent degrees-of-freedom that are the rotation components most nearly parallel to the line joining the regions should not normally be constrained.
13. The RSPLINE has a limit of 100 grid points.

RSSCON Shell-to-Solid Element Connector

Defines multipoint constraints to model clamped connections of shell-to-solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
RSSCON	RBID	TYPE	ES1	EA1	EB1	ES2	EA2	EB2	

Examples:

RSSCON	110	GRID	11	12	13	14	15	16	
RSSCON	111	GRID	31	74	75				
RSSCON	115	ELEM	311	741					

Field	Contents
RBID	Element identification number. (0 < Integer < 100,000,000)
TYPE	Type of connectivity TYPE = “ELEM” connection is decscribed with element identification numbers. TYPE = “GRID” connection is described with grid point identification numbers. (Character: “GRID” or “ELEM”; Default = “ELEM”)
ES1	Shell element identification number if TYPE = “ELEM”. Shell grid point identification number if TYPE = “GRID”. See **Figure 8-157**. (Integer > 0)
EA1	Solid element identification number if TYPE = “ELEM”. Solid grid point identification number if TYPE = “GRID”. (Integer > 0)
EB1	Solid grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)
ES2	Shell grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)
EA2	Solid grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)
EB2	Solid grid-point identification number for TYPE = “GRID” only. (Integer > 0 or blank)

Remarks:

1. RSSCON generates a multipoint constraint that models a clamped connection between a shell and a solid element. The shell degrees-of-freedom are put in the dependent set (m-set). The translational degrees-of-freedom of the shell edge are connected to the translational degrees-of-freedom of the upper and lower solid edge. The two rotational degrees-of-freedom of the shell are connected to the translational degrees-of-freedom of the lower and upper edges of the solid element face. Poisson's ratio effects are considered in the translational degrees-of-freedom.
2. The shell grid point must lie on the line connecting the two solid grid points. It can have an offset from this line, which can not be more than 5% of the distance between the two solid grid points. The shell grid points that are out of the tolerance will not be constrained, and a fatal message will be issued. This tolerance is adjustable. Please see PARAM,TOLRSC and PARAM,SEPIXOVR.
3. When using the TYPE = "ELEM" option
    - The elements may be p-elements or h-elements. The solid elements are CHEXA, CPENTA, and CTETRA with and without midside nodes. The shell elements are CQUAD4, CTRIA3, CQUADR, CTRIAR, CQUAD8, or CTRIA6.
    - In case of p-elements, the p-value of the shell element edge is adjusted to the higher of the p-value of the upper or lower solid p-element edge. If one of the elements is an h-element, then the p-value of the adjacent edge is lowered to 1.
    - Both the shell and solid elements have to belong to the same superelement. This restriction can be bypassed using SEELT entry to reassign the downstream boundary element to an upstream superelement.
    - When a straight shell p-element edge and a solid p-element are connected, the geometry of the shell edge is not changed to fit the solid face. When a curved shell p-element edge and a solid p-element are connected, the two solid edges and solid face are not changed to match the shell edge.
    - It is not recommended to connect more than one shell element to the same solid using the ELEM option. If attempted, conflicts in the multipoint constraint relations may lead to UFM 6692.

4. When using TYPE = “GRID” option
   - The GRID option does not verify that the grids used are valid shell and / or solid grids.
   - The hierarchical degrees-of-freedom of p-element edges are not constrained. The GRID option is therefore not recommended for p-elements.
   - The grids in the GRID option can be in different superelements. The shell grid must be in the upstream superelement.
5. It is recommended that the height of the solid element’s face is approximately equal to the shell element’s thickness of the shell. The shell edge should then be placed in the middle of the solid face.
6. The shell edge may coincide with the upper or lower edge of the solid face.
7. The RSSCON entry, unlike MPCs, cannot be selected through the Case Control Section.
8. Forces of multipoint constraints may be recovered with the MPCFORCE Case Control command.
9. The RSSCON is ignored in heat-transfer problems.
10. The m-set coordinates (shell degrees-of-freedom) may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.

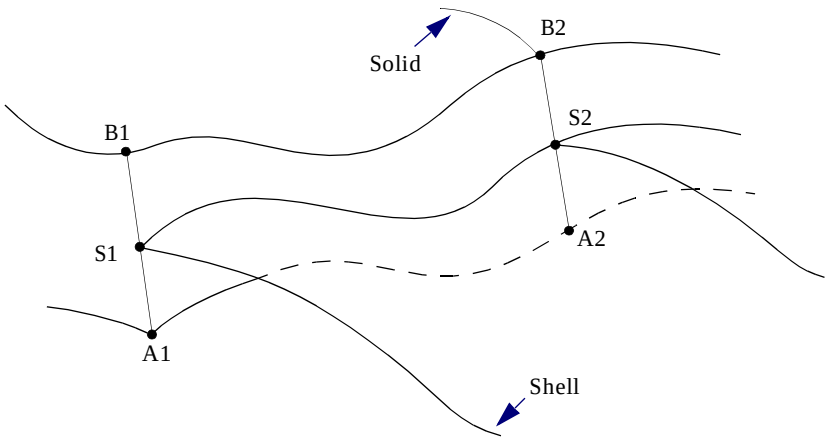


Figure 8-157 Shell Elements Connected to the Faces of Solid Elements

RTRPLT Rigid Triangular Plate

Defines a rigid triangular plate.

Format:

1	2	3	4	5	6	7	8	9	10
RTRPLT	EID	GA	GB	GC	CNA	CNB	CNC		
	CMA	CMB	CMC	ALPHA					

Example:

RTRPLT	7	1	2	3	1236	3	3		

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB, GC	Grid point identification number of connection points.
CNA, CNB, CNC	Independent degrees-of-freedom in the global coordinate system for the element at grid points GA, GB, and GC, indicated by any of the Integers 1 through 6 with no embedded blanks. See Remark **3.** (Integer $\geq$ 0 or blank)
CMA, CMB, CMC	Component numbers of dependent degrees-of-freedom in the global coordinate system. (Any of the Integers 1 through 6 with no embedded blanks, or 0 or blank.)
ALPHA	Thermal expansion coefficient. See Remark **12.** (Real)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, MD Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 18 displacement degrees-of-freedom given by grid points GA, GB, and GC. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.

3. For the linear method, the total number of components in CNA, CNB, and CNC must equal six; for example, CNA = 1236, CNB = 3, CNC = 3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. For the Lagrange method, the total number of components must also be six. However, only CNA= 123456 or CNB = 123456 or CNC = 123456 is allowed. For this type of element, RTRPLT1 gives a simpler input format.
4. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
5. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
6. Element identification numbers should be unique with respect to all other element identification numbers.
7. Rigid elements, unlike MPCs, are not selected through the Case Control command, MPC.
8. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
9. Rigid elements are ignored in heat transfer problems.
10. See "**Rigid Elements and Multipoint Constraints (R-type, MPC)**" on page 167 of the *MSC.Nastran Reference Guide* for a discussion of rigid elements.
11. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
12. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is defined as the following. the bar GA-GB will have the average temperature of grid points GA and GB. The bar GA-GC will have the average temperature of the grid points GA and GC.

RTRPLT1 Rigid Triangular Plate (Alternative Format)

Alternative format to define a rigid triangular plate element connecting three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RTRPLT1	EID	GA	GB	GC	CMB	CMC	ALPHA		

Example:

RTRPLT1	7	1	2	3	1236	3	6.0-6		

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB, GC	Grid point identification number of connection points. (Integer > 0)
CMB, CMC	Component numbers at GB and GC in the global coordinate systems, which are constrained to move with the rigid body. See Remark **4.** (Integers 1 through 6 with no embedded blanks or blank.)
ALPHA	Thermal expansion coefficient. See Remark **9.** (Real > 0.0 or blank)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
2. For the Lagrange method, MD Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 18 displacement degrees-of-freedom given by grid points GA, GB, and GC. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
3. RTRPLT1 is a preferred input format for the Lagrange method.
4. When CMB = “123456” or blank, CMC = “123456” or blank, the grid points GB and BC are constrained to move with GA as a rigid triangular plate. For default, CMB = “123456” and CMC = “123456”. Any number of degrees-of-freedom at grid points GB and GC can be released not to move with the rigid body.
5. The length of any two connected grid points must be greater than zero.

6. For the Lagrange method, the theory is formulated such that a consistent rigid body motion for grid points GA, GB, and GC will be computed even if these three points have different global coordinate systems.
7. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The bar GA-GB will have the average temperature of grid points GA and GB. The bar GA-GC will have the average temperature of the grid points GA and GC.
8. Element identification numbers should be unique with respect to all other element identification numbers.
9. Rigid elements are ignored in heat transfer problems.

RVDOF Degrees-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are to be applied to obtain static solutions for use in residual vector computations.

Format:

1	2	3	4	5	6	7	8	9	10
RVDOF	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

RVDOF	800	1	850	2					

Field	Contents
IDi	Grid or scalar identification number. (Integer > 0)
Ci	Component numbers. (Any one of the integers 1 through 6 for grid points and integer zero or blank for scalar points)

Remarks:

1. In multiple superelement analysis, the IDi points may be interior to any superelement. The program automatically partitions the data for allocation to the appropriate superelements. Separate entries for separate superelements are not required as in the case of USETi,U6 and SEUSETi,U6 entries.
2. The unit loads applied to the interior points of a superelement due to the RVDOF/RVDOF1 entries are passed downstream all the way down to the residual for the purpose of residual vector processing by all superelements in its downstream path, resulting in more accurate results. This is in contrast to the way residual vector processing is performed when USETi,U6 or SEUSETi,U6 entries are employed. In the latter case, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

RVDOF1 Degrees-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are to be applied to obtain static solutions for use in residual vector computations.

Format:

1	2	3	4	5	6	7	8	9	10
RVDOF1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	

Example:

RVDOF1	3	100	210	450					

Field	Contents
C	Component numbers. (Any one of the integers 1 through 6 for grid points and integer zero or blank for scalar points)
IDi	Grid or scalar identification number. (Integer > 0)

Remarks:

1. In multiple superelement analysis, the IDi points may be interior to any superelement. The program automatically partitions the data for allocation to the appropriate superelements. Separate entries for separate superelements are not required as in the case of USETi,U6 and SEUSETi,U6 entries.
2. The unit loads applied to the interior points of a superelement due to the RVDOF/RVDOF1 entries are passed downstream all the way down to the residual for the purpose of residual vector processing by all superelements in its downstream path, resulting in more accurate results. This is in contrast to the way residual vector processing is performed when USETi,U6 or SEUSETi,U6 entries are employed. In the latter case, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

SBPRET (SOL 700) Seat Belt Pretensioner

Defines a seat belt pretensioner. A combination with sensors and retractors is also possible.

Format:

1	2	3	4	5	6	7	8	9	10
SBPRET	SBPRID	SBPRTY	SBSID1	SBSID2	SSBSID3	SBSID4			
	SBRID	TIME	PTLCID	LMTFRC					

Example:

SBPRET	12	64	54	53	65	74			
	0	0.0	0	0					

Field	Contents
SBPRID	Pretensioner ID. A unique number has to be used. (Integer > 0, Required)
SBPRTY	Pretensioner type (see Remark 2 below): (Integer > 0, Required) = 1: pyrotechnic retractor with force limits, = 2: pre-loaded spring becomes active, = 3: lock spring removed, = 4: force versus time retractor. = 5: pyrotechnic retractor (old type) but with optional force limiter, LMTFRC.
SBSID1	Sensor 1, see SBSENSR. (Integer $\geq$ 0, Default = 0)
SBSID2	Sensor 2, see SBSENSR. (Integer $\geq$ 0, Default = 0)
SBSID3	Sensor 3, SBSENSR. (Integer $\geq$ 0, Default = 0)
SBSID4	Sensor 4, see SBSENSR. (Integer $\geq$ 0, Default = 0)
SBRID	Retractor number (SBPRTY = 1) or spring element number (SBPRTY = 2 or 3). (Integer $\geq$ 0, Default = 0)
TIME	Time between sensor triggering and pretensioner acting. (Real $\geq$ 0.0, Default = 0.0)

Field	Contents
PTLCID	TABLED ID for pretensioner. (Time after activation, Pull-in) (SBPRTY = 1). (Integer $\geq$ 0, Default = 0)
LMTFRC	Optional limiting force for retractor type 5. If zero, this option is ignored. (Real $\geq$ 0.0, Default = 0.0)

Remarks:

1. Corresponds to Ls-Dyna entry * ELEMENT_SEATBELT_PTRETENSIONER
2. At least one sensor should be defined.

   Pretensioners allow modeling of five types of active devices which tighten the belt during the initial stages of a crash. Types 1 and 5 represent a pyrotechnic device which spins the spool of a retractor, causing the belt to be reeled in. The user defines a pull-in versus time curve which applies once the pretensioner activates. Types 2 and 3 represent preloaded springs or torsion bars which move the buckle when released. The pretensioner is associated with any type of spring element including rotational. Note that the preloaded spring, locking spring and any restraints on the motion of the associated nodes are defined in the normal way; the action of the pretensioner is merely to cancel the force in one spring until (or after) it fires. With the second type, the force in the spring element is canceled out until the pretensioner is activated. In this case the spring in question is normally a stiff, linear spring which acts as a locking mechanism, preventing motion of the seat belt buckle relative to the vehicle. A preloaded spring is defined in parallel with the locking spring. This type avoids the problem of the buckle being free to 'drift' before the pretensioner is activated. Type 4, a force type, is described below.

   To activate the pretensioner, the following sequence of events must occur:

   a. Any one of up to four sensors must be triggered.

   b. Then a user-defined time delay occurs.

   c. Then the pretensioner acts.

3. There are three types of seat belt pretensioners that can be simulated. Types 2 and 3 are simple triggers for activating or deactivating springs, which then pull on the buckle. No changes have been made to these, and they are not discussed here. The type 1 pretensioner is intended to simulate a pyrotechnic retractor. The user inputs a load curve describing the pull-in of the pretensioner as a function of time. This pretensioner type interacts with the retractor, forcing it to pull in the amount of belt indicated. It works well,

and does exactly what it says it will do, but it can be difficult to use in practice. The reason for this is that it has no regard for the forces being exerted on the belt. If a pull-in of 20mm is specified at a particular time, then 20mm of belt will be pulled in, even if this results in unrealistic forces in the seat belt. Furthermore, there was no explicit way to turn this pretensioner off. Once defined, it overrode the retractor completely, and the amount of belt passing into or out of the retractor depended solely on the load curve specified.

In the 970 version of LS-DYNA, the behavior of the type 1 pretensioner was changed due to user feedback regarding these shortcomings. The behavior now is fundamentally simpler, though a bit confusing to explain. Each retractor has a loading (and optional unloading) curve that describes the force on the belt element as a function of the amount of belt that has been pulled out of the retractor since the retractor locked. The new type 1 pretensioner acts as a shift of this retractor load curve. An example will make this clear. Suppose at a particular time that 5mm of belt material has left the retractor. The retractor will respond with a force corresponding to 5mm pull-out on it's loading curve. But suppose this retractor has a type 1 pretensioner defined, and at this instant of time the pretensioner specifies a pull-in of 20mm. The retractor will then respond with a force that corresponds to (5mm + 20mm) on it's loading curve. This results in a much larger force. The effect can be that belt material will be pulled in, but unlike in the 950 version, there is no guarantee. The benefit of this implementation is that the force vs. pull-in load curve for the retractor is followed and no unrealistic forces are generated. Still, it may be difficult to produce realistic models using this option, so two new types of pretensioners have been added. These are available in 970 versions 1300 and later.

The type 4 pretensioner takes a force vs. time curve, See **Figure 8-158**. Each time step, the retractor computes the desired force without regard to the pretensioner. If the resulting force is less than that specified by the pretensioner load curve, then the pretensioner value is used instead. As time goes on, the pretensioner load curve should drop below the forces generated by the retractor, and the pretensioner is then essentially inactive. This provides for good control of the actual forces, so no unrealistic values are generated. The actual direction and amount of belt movement is unspecified, and will depend on the other forces being exerted on the belt. This is suitable when the force the pretensioner exerts over time is known.

The type 5 pretensioner is essentially the same as the old type 1 pretensioner, but with the addition of a force limiting value. The pull-in is given as a function of time, and the belt is drawn into the retractor exactly as desired. However, if at any point the forces generated in the belt exceed the pretensioner force limit, then the pretensioner is deactivated and the retractor takes over. In order to prevent a large discontinuity in the force at this point, the loading curve for the retractor is shifted (in the abscissa) by the amount required to put the current (pull-out, force) on the load curve. For example, suppose the current force is 1000, and the current pull-out is -10 (10mm of belt has been pulled IN by the pretensioner). If the retractor would normally generate a force of 1000 after 25mm of belt had been pulled OUT, then the load curve is shifted to the left by 35, and remains that way for the duration of the calculation. So that at the current pull in of 10, it will generate the force normally associated with a pull out of 25. If the belt reaches a pull out of 5, the force will be as if it were pulled out 40 (5 + the shift of 35), and so on. This option is included for those who liked the general behavior of the old type 1 pretensioner, but has the added feature of the force limit to prevent unrealistic behavior.

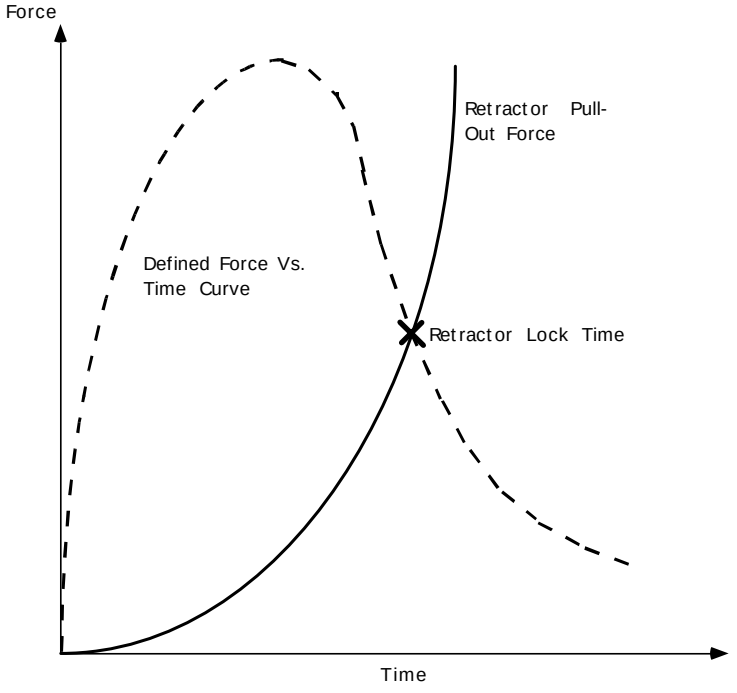


Figure 8-158 Force versus time pretensioner. At the intersection, the retractor locks.

SBRETR (SOL 700) Seat Belt Retractor

Defines a seat belt retractor.

Format:

1	2	3	4	5	6	7	8	9	10
SBRETR	SBRID	SBRNID	SBID	SID1	SID2	SID3	SID4		
	TDEL	PULL	LLCID	ULCID	LFED				

Example:

SBRETR	12	64	54	53	65	74	75		
	0.0	0.0	12						

Field	Contents
SBRID	Retractor ID. A unique number has to be used. (Integer > 0, required)
SBRNID	Retractor node ID (Integer > 0, Required)
SBSID1	Sensor 1, see SBSENSR. (Integer $\geq$ 0, Default = 0)
SBSID2	Sensor 2, see SBSENSR. (Integer $\geq$ 0, Default = 0)
SBSID3	Sensor 3, SBSENSR. (Integer $\geq$ 0, Default = 0)
SBSID4	Sensor 4, see SBSENSR. (Integer $\geq$ 0, Default = 0)
TDEL	Time delay after sensor triggers. (Real $\geq$ 0.0, Default = 0.0)
PULL	Amount of pull-out between time delay ending and retractor locking, a length value. (Real $\geq$ 0.0, Default = 0.0)
LLCID	Tabled ID for loading (Pull-out, Force), see **Figure 8-160**. (Integer $\geq$ 0, Default = 0)
ULCID	Tabled ID for unloading (Pull-out, Force), see **Figure 8-160**. (Integer $\geq$ 0, Default = 0)
LFED	Fed length, see explanation below. (Real $\geq$ 0.0, Default = 0.0)

Remarks:

1. Corresponds to Ls-Dyna entry *ELEMENT_SEATBELT_RETRACTOR.

2. The retractor node should not be on any belt elements. The element defined should have one node coincident with the retractor node but should not be inside the retractor.
3. At least one sensor should be defined.
4. The first point of the load curve should be (0, T_{min}). T_{min} is the minimum tension. All subsequent tension values should be greater than T_{min}.
5. The unloading curve should start at zero tension and increase monotonically (i.e., no segments of negative or zero slope).

   Retractors allow belt material to be paid out into a belt element. Retractors operate in one of two regimes: unlocked when the belt material is paid out, or reeled in under constant tension and locked when a user defined force-pullout relationship applies.

   The retractor is initially unlocked, and the following sequence of events must occur for it to become locked:

   a. Any one of up to four sensors must be triggered. (The sensors are described below.)
   b. Then a user-defined time delay occurs.
   c. Then a user-defined length of belt must be paid out (optional).
   d. Then the retractor locks and once locked, it remains locked.

   In the unlocked regime, the retractor attempts to apply a constant tension to the belt. This feature allows an initial tightening of the belt and takes up any slack whenever it occurs. The tension value is taken from the first point on the force-pullout load curve. The maximum rate of pull out or pull in is given by $0.01 \times$ fed length per time step. Because of this, the constant tension value is not always achieved.

   In the locked regime, a user-defined curve describes the relationship between the force in the attached element and the amount of belt material paid out. If the tension in the belt subsequently relaxes, a different user-defined curve applies for unloading. The unloading curve is followed until the minimum tension is reached.

   The curves are defined in terms of initial length of belt. For example, if a belt is marked at 10mm intervals and then wound onto a retractor, and the force required to make each mark emerge from the (locked) retractor is recorded, the curves used for input would be as follows:

0	Minimum tension (should be > zero)
10mm	Force to emergence of first mark

20mm Force to emergence of second mark

Pyrotechnic pretensions may be defined which cause the retractor to pull in the belt at a predetermined rate. This overrides the retractor force-pullout relationship from the moment when the pretensioner activates.

If desired, belt elements may be defined which are initially inside the retractor. These will emerge as belt material is paid out, and may return into the retractor if sufficient material is reeled in during unloading.

Elements e2, e3 and e4 are initially inside the retractor, which is paying out material into element e1. When the retractor has fed L_{crit} into e1, where

L_{crit} = fed length - 1.1 × minimum length

minimum length defined on belt material input)

(fed length defined on retractor input)

element e2 emerges with an unstretched length of 1.1 x minimum length; the unstretched length of element e1 is reduced by the same amount. The force and strain in e1 are unchanged; in e2, they are set equal to those in e1. The retractor now pays out material into e2.

If no elements are inside the retractor, e2 can continue to extend as more material is fed into it.

As the retractor pulls in the belt (for example, during initial tightening), if the unstretched length of the mouth element becomes less than the minimum length, the element is taken into the retractor.

To define a retractor, the user enters the retractor node, the ‘mouth’ element (into which belt material will be fed), e1 in **Figure 8-159**, up to 4 sensors which can trigger unlocking, a time delay, a payout delay (optional), load and unload curve numbers, and the fed length. The retractor node is typically part of the vehicle structure; belt elements should not be connected to this node directly, but any other feature can be attached including rigid bodies. The mouth element should have a node coincident with the retractor but should not be inside the retractor. The fed length would typically be set either to a typical element initial length, for the distance between painted marks on a real belt for comparisons with high speed film. The fed length should be at least three times the minimum length.

If there are elements initially inside the retractor (e2, e3 and e4 in the Figure) they should not be referred to on the retractor input, but the retractor should be identified on the element input for these elements. Their nodes should all be coincident with the retractor node and should not be restrained or constrained. Initial slack will automatically be set to 1.1 × minimum length for these elements; this overrides any user-defined value.

Weblockers can be included within the retractor representation simply by entering a 'locking up' characteristic in the force pullout curve, see **Figure 8-160**. The final section can be very steep (but must have a finite slope).

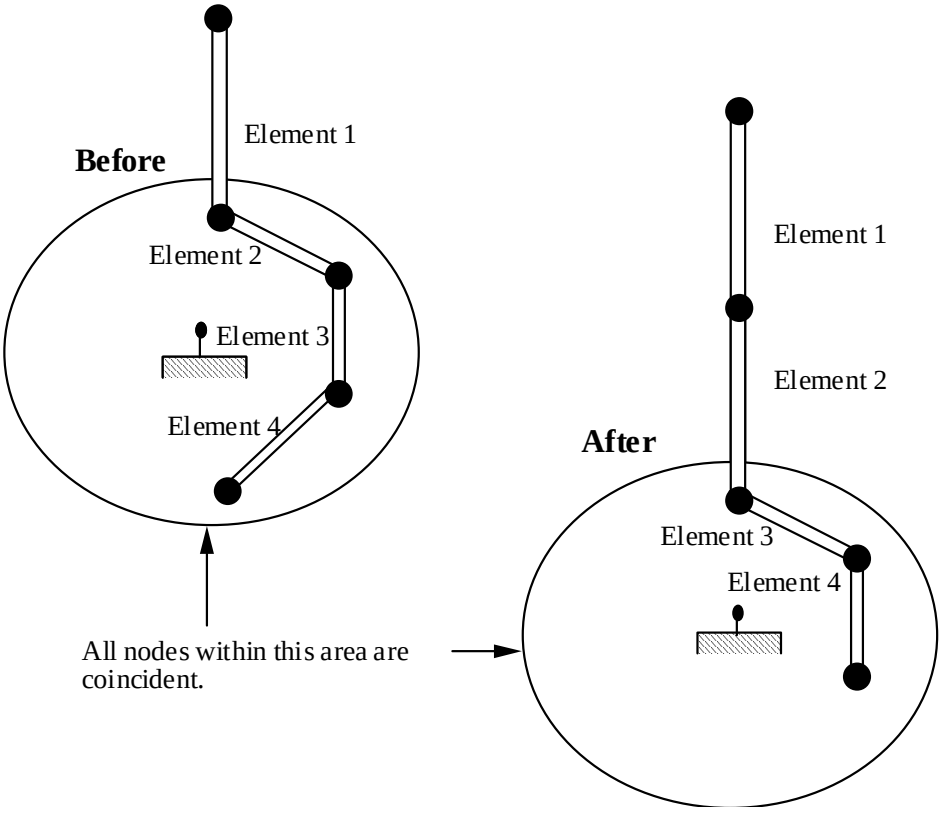


Figure 8-159 Elements in a retractor.

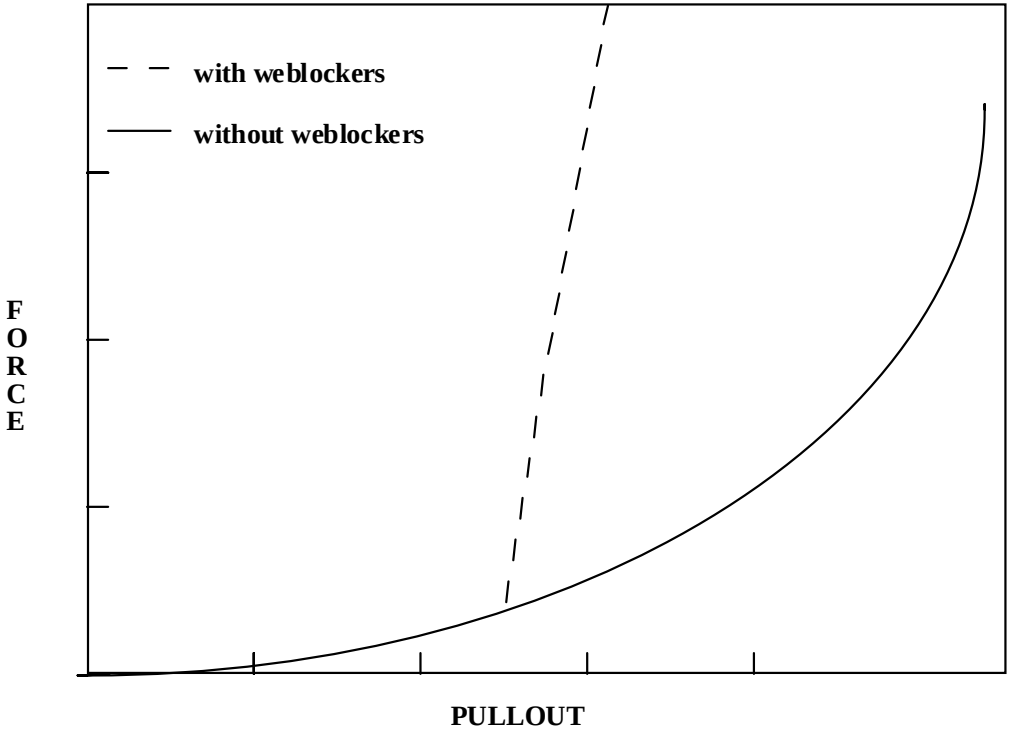


Figure 8-160 Retractor force pull characteristics.

SBSENSR (SOL 700) Seat Belt Sensor

Defines a seat belt sensor.

Format:

1	2	3	4	5	6	7	8	9	10
SBSENSR	SBSID	SBSTYP	SBSFL						
	NID	DOF	ACC	ATIME					
	SBRID	PULRAT	PULTIM						
	TIME								
	NID1	NID2	DMX	DMN					

Example:

SBSENSR	12	3	0						
	15.e-3								

Field	Contents
SBSID	Sensor ID. A unique number has to be used. (Integer > 0, Required)
SBSTYP	Sensor type: (Integer > 0, Required)
	=1: acceleration of node,
	=2: retractor pull-out rate,
	=3: time,
	=4: distance between nodes.
SBSFL	Sensor flag: (Integer > 0, required)
	=0: sensor active during dynamic relaxation,
	=1: sensor can be triggered during dynamic relaxation.
NID	Node ID of sensor. Only use when SBSTYP=1. (Integer > 0, Required)
DOF	Degree of freedom Only use when SBSTYP=1. (Integer > 0, Required)
	=1: x,
	=2: y,

Field	Contents
	=3: z.
ACC	Activating acceleration. Only use when SBSTYP=1. (Real ≥ 0.0, Default = 0.0)
ATIME	Time over which acceleration must be exceeded. Only use when SBSTYP=1. (Real ≥ 0.0, Default = 0.0)
SBRID	Retractor ID, see SBRETR. Only use when SBSTYP=2. (Integer > 0, Required)
PULRAT	Rate of pull-out (length / time units) Only use when SBSTYP=2. (Real ≥ 0.0, Default = 0.0)
PULTIM	Time over which rate of pull-out must be exceeded Only use when SBSTYP=2. (Real ≥ 0.0, Default = 0.0)
TIME	Time at which sensor triggers Only use when SBSTYP=3. (Real ≥ 0.0, Default = 0.0)
NID1	Node 1 ID Only use when SBSTYP=4. (Integer > 0, Required)
NID2	Node 2 ID Only use when SBSTYP=4. (Integer > 0, Required)
DMX	Maximum distance Only use when SBSTYP=4. (Real ≥ 0.0, Default = 0.0)
DMN	Minimum distance Only use when SBSTYP=4. (Real ≥ 0.0, Default = 0.0)

Remarks:

1. Corresponds to Ls-Dyna entry *ELEMENT_SEATBELT_SENSOR
2. Node should not be on rigid body, velocity boundary condition, or other 'imposed motion' feature.
3. Sensor triggers when the distance between the two nodes is $d \geq d_{dmax}$ or $d \leq d_{min}$.

   Sensors are used to trigger locking of retractors and activate pretensioners. Four types of sensors are available which trigger according to the following criteria:

   Type 1–When the magnitude of x-, y-, or z- acceleration of a given node has remained above a given level continuously for a given time, the sensor triggers. This does not work with nodes on rigid bodies.

   Type 2–When the rate of belt payout from a given retractor has remained above a given level continuously for a given time, the sensor triggers.

   Type 3–The sensor triggers at a given time.

Type 4–The sensor triggers when the distance between two nodes exceeds a given maximum or becomes less than a given minimum. This type of sensor is intended for use with an explicit mass / spring representation of the sensor mechanism.

By default, the sensors are inactive during dynamic relaxation. This allows initial tightening of the belt and positioning of the occupant on the seat without locking the retractor or firing any pretensioners. However, a flag can be set in the sensor input to make the sensors active during the dynamic relaxation phase.

SBSLPR (SOL 700) Seat Belt Slipring

Defines seat belt slipring.

Format:

1	2	3	4	5	6	7	8	9	10
SBSLPR	SBSRID	SBID1	SBID2	FC	SBRNID	LTIME	FCS		

Example:

SBSLPR	21	1	2	0.3	23				

Field	Contents
SBSRID	Slipring ID. A unique number has to be used. (Integer, Required)
SBID1	Seat belt element 1 Id (Integer, Required)
SBID2	Seat belt element 2 Id (Integer, Required)
FC	Coulomb dynamic friction coefficient (Real, Required)
SBRNID	Slip ring node, NID (Integer >0, Required)
LTIME	Slip ring lockup time. After this time no material is moved from one side of the slip ring to the other. This option is not active during dynamic relaxation. (Real, Default = 1.0E20)
FCS	Optional static Coulomb friction coefficient. (Real, Default = 0.0)

Remarks:

Corresponds to Ls-Dyna input *ELEMENT_SEATBELT_SLIPRING

Elements 1 and 2 should share a node which is coincident with the slip ring node. The slip ring node should not be on any belt elements.

Sliprings allow continuous sliding of a belt through a sharp change of angle. Two elements (1 and 2 in **Figure 8-161**) meet at the slipring. Node B in the belt material remains attached to the slipring node, but belt material (in the form of unstretched length) is passed from element 1 to element 2 to achieve slip. The amount of slip at each time step is calculated from the ratio of forces in elements 1 and 2. The ratio of forces is determined by the relative angle between elements 1 and 2 and the coefficient of friction, μ. The tension in the belts are taken as T_1 and T_2, where T_2 is on the high tension side and T_1 is the force on the low tension side. Thus, if T_2 is sufficiently close

to T_1, no slip occurs; otherwise, slip is just sufficient to reduce the ratio $T_2 T_1$ to $e^{\mu\Theta}$. No slip occurs if both elements are slack. The out-of-balance force at node B is reacted on the slipring node; the motion of node B follows that of slipring node.

If, due to slip through the slipring, the unstretched length of an element becomes less than the minimum length (as entered on the belt material card), the belt is remeshed locally: the short element passes through the slipring and reappears on the other side (see **Figure 8-161**). The new unstretched length of e1 is 1.1 × minimum length. Force and strain in e2 and e3 are unchanged; force and strain in e1 are now equal to those in e2. Subsequent slip will pass material from e3 to e1. This process can continue with several elements passing in turn through the slipring.

To define a slipring, the user identifies the two belt elements which meet at the slipring, the friction coefficient, and the slipring node. The two elements must have a common node coincident with the slipring node. No attempt should be made to restrain or constrain the common node for its motion will automatically be constrained to follow the slipring node. Typically, the slipring node is part of the vehicle body structure and, therefore, belt elements should not be connected to this node directly, but any other feature can be attached, including rigid bodies.

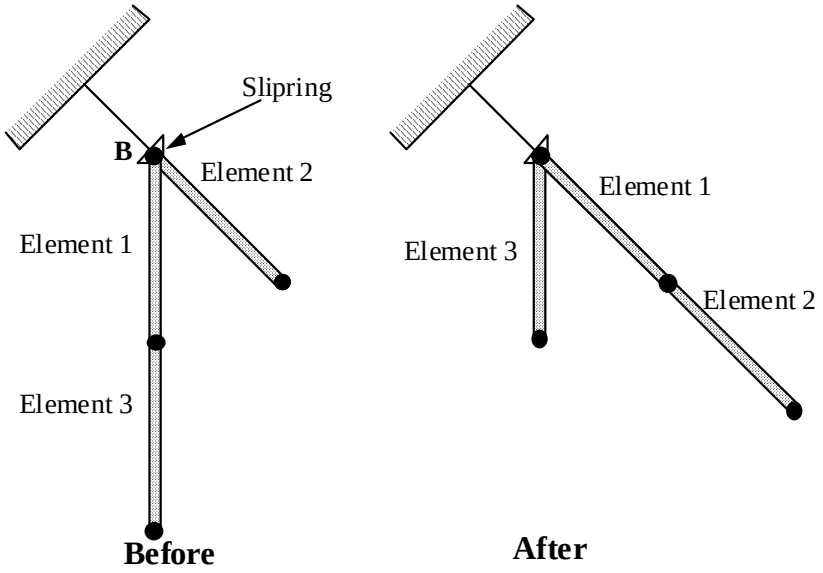


Figure 8-161 Elements passing through slipring.

SEBNDRY Superelement Boundary-Point Definition

Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.

Format:

1	2	3	4	5	6	7	8	9	10
SEBNDRY	SEIDA	SEIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	-etc.-						

Example 1:

SEBNDRY	400	4	10	20	30	40			

Example 2:

SEBNDRY	400	ALL	10	20	30	THRU	40		

Field	Contents
SEIDA	Partitioned superelement identification number. (Integer > 0)
SEIDB	Superelement identification. See Remark 2. (Integer > 0 or Character "ALL"; Default = "ALL")
GIDAi	Identification number of a boundary grid point in superelement SEIDA. (Integer > 0 or "THRU"; For "THRU" option, G1 < G2.)

Remarks:

1. SEBNDRY may only be specified in the main Bulk Data Section and is not recognized after the BEGIN SUPER=n.
2. SEIDB may reference partitioned superelements or superelements in the main Bulk Data Section.

SEBSET Fixed Boundary Degree-of-Freedom

Defines boundary degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
SEBSET	SEID	ID1	C1	ID2	C2	ID3	C3		

Example:

SEBSET	5	2	135	14	6				

Field	Contents
SEID	Superelement identification number. (Integer > 0)
Ci	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points)
IDi	Grid or scalar point identification numbers. (Integer > 0)

Remarks:

1. If there are no SECSETi or SEBSETi entries present, all boundary points are, by default, fixed during component mode analysis. If only SEBSETi are entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If both SEBSETi and SECSETi entries are present, the c-set degrees-of-freedom are defined by the SECSETi entries and any remaining boundary points are placed in the b-set.
2. Degrees-of-freedom listed on SEBSETi entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSETi entries.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
   - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

- If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SEBSET1 Fixed Boundary Degree-of-Freedom, Alternate Form of SEBSET

Defines boundary degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode calculations.

Format:

1	2	3	4	5	6	7	8	9	10
SEBSET1	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

SEBSET1	5	2	135	14	6	23	24	25	
	122	127							

Alternate Format and Example:

SEBSET1	SEID	C	G1	“THRU”	G2				
SEBSET1	5	3	6	THRU	32				

Field	Contents
SEID	Superelement identification number. (Integer > 0)
C	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points, 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or “THRU”; for THRU option G1 < G2.)

Remarks:

1. If there are no SECSETi or SEBSETi entries present, all boundary points are, by default, fixed during component mode analysis. If there are only SEBSETi entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both SEBSETi and SECSETi entries present, the c-set degrees-of-freedom are defined by the SECSETi entries, and any remaining boundary points are placed in the b-set.
2. Degrees-of-freedom listed on SEBSETi entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSETi entries.

3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
    - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
    - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SEBULK Partitional Superelement Connection

Defines superelement boundary search options and a repeated, mirrored, or collector superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEBULK	SEID	TYPE	RSEID	METHOD	TOL	LOC	UNITNO		

Example:

SEBULK	14	REPEAT	4	AUTO	1.0E-3				

Field	Contents	
SEID	Partitioned superelement identification number. (Integer > 0)	
TYPE	Superelement type. (Character; no Default)	
	PRIMARY	Primary
	REPEAT	Identical
	MIRROR	Mirror
	COLLCTR	Collector
	EXTERNAL	External
	EXTOP2	External using an OUTPUT2 file created in an earlier run.
RSEID	Identification number of the reference superelement, used if TYPE = “REPEAT” and “MIRROR”. (Integer $\geq$ 0; Default = 0)	
METHOD	Method to be used when searching for boundary grid points. (Character: “AUTO” or “MANUAL”; Default = “AUTO”)	
TOL	Location tolerance to be used when searching for boundary grid points. (Real; Default = 10E-5)	
LOC	Coincident location check option for manual connection option. (Character: “YES” or “NO”; Default = “YES”)	
UNITNO	FORTRAN unit number for the OUTPUT2 file (applicable and meaningful only when TYPE=”EXTOP2”).	

Remarks:

1. The TYPE = “REPEAT” or “MIRROR” does not include superelements upstream of the reference superelement. A repeated or mirrored superelement can have boundaries, loads, constraints, and reduction procedures that are different than the reference superelement.
2. METHOD = “MANUAL” requires SECONCT entries. SEBNDRY and SEEXCLD, which reference SEID, will produce a fatal message.
3. SECONCT, SEBNDRY, and SEEXCLD entries can be used to augment the search procedure and / or override the global tolerance.
4. For combined automatic and manual boundary search, the METHOD = “AUTO” should be specified and connections should be specified on a SECONCT entry.
5. TOL and LOC are the default values that can be modified between two superelements by providing the required tolerance on the SECONCT entry.
6. TYPE = “MIRROR” also requires specification of a SEMPLN entry.
7. TYPE = “COLLCTR” indicates a collector superelement, which does not contain any grids or scalar points.
8. For TYPE = “EXTERNAL” or “EXTOP2,” see discussion under the description of the EXTSEOUT Case Control command for employing external superelements using the new two-step procedure. For employing external superelements using the old three-step procedure, see discussion under the description of “**EXTDRUNT**” in Chapter 5.
9. This entry will only work if PART superelements (BEGIN SUPER) or external superelements created by employing the EXTSEOUT Case Control command exist.

SECONCT Partitioned Superelement Boundary-Point Connection

Explicitly defines grid and scalar point connection procedures for a partitioned superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SECONCT	SEIDA	SEIDB	TOL	LOC					
	GIDA1	GIDB1	GIDA2	GIDB2	GIDA3	GIDB3	-etc.-		

Example:

SECONCT	10	20	1.0E-4	YES					
	1001	4001			2222	4444			

Alternate Format and Example:

SECONCT	SEIDA	SEIDB	TOL	LOC					
	GIDA1	‘THRU’	GIDA2	GIDB1	‘THRU’	GIDB2			

SECONCT	10	20							
	101	‘THRU’	110	201	‘THRU’	210			

Field	Contents
SEIDA	Partitioned superelement identification number. (Integer > 0)
SEIDB	Identification number of superelement for connection to SEIDA. (Integer ≥ 0)
TOL	Location tolerance to be used when searching for or checking boundary grid points. (Real; Default = 10E-5)
LOC	Coincident location check option for manual connection. (Character; “YES” or “NO”; Default = “YES”)
GIDAi	Identification number of a grid or scalar point in superelement SEIDA, which will be connected to GIDBi.
GIDBi	Identification number of a grid or scalar point in superelement SEIDB, which will be connected to GIDAi.

Remarks:

1. SECONCT can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER = n command.
2. TOL and LOC can be used to override the default values specified on the SEBULK entries.
3. The continuation entry is optional.
4. The (GIAi, GIBi) pair must both be grids or scalar points.
5. All six degrees-of-freedom of grid points will be defined as boundary degrees-of-freedom.
6. This entry will only work if PART superelements (BEGIN SUPER) exist.
7. Blank fields are allowed after the first GIDA1-GIDB1 pair. Blank fields must also occur in pairs. This remark does not apply to the alternate format.
8. For Alternate Format 1, the thru ranges must be closed sets. That is, all IDs listed between 101 and 110 in the example must exist in the model.

SECSET Free Boundary Degree-of-Freedom

Defines boundary degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
SECSET	SEID	ID1	C1	ID2	C2	ID3	C3		

Example:

SECSET	3	124	1	5	23	6	15		

Field	Contents
SEID	Superelement identification number. (Integer > 0)
Ci	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
IDi	Grid or scalar point identification number. (Integer > 0)

Remarks:

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSETi or SESUP entries. Coordinates listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined by the program and listed in the SEMAP table output.
2. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.
3. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
    - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.

- If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SECSET1 Free Boundary Degree-of-Freedom, Alternate Form of SECSET

Defines boundary degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
SECSET1	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

SECSET1	5	2	135	14	6	23	24	25	
	122	127							

Alternate Formats and Example:

SECSET1	SEID	C	G1	“THRU”	G2				
SECSET1	5	3	6	THRU	32				

SECSET1	SEID		“ALL”						
SECSET1	SEID		ALL						

Field	Contents
SEID	Superelement identification number. (Integer > 0)
C	Component numbers of degree-of-freedoms. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification number. (Integer > 0)

Remarks:

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSETi or SESUP entries. Degrees-of-freedom listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined automatically and listed in the SEMAP table output.

2. If the alternate formats are used, the grid points Gi are not required to exist or to be exterior degrees-of-freedom and may be listed on SECSET1 entries. Points of this type will cause one warning message but will otherwise be ignored.
3. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
4. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.
5. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
    - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
    - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

SECTAX Conical Shell Sector

Defines a sector of a conical shell.

Format:

1	2	3	4	5	6	7	8	9	10
SECTAX	ID	RID	R	PHI1	PHI2				

Example:

SECTAX	1	2	3.0	30.0	40.0				

Field	Contents
ID	Sector identification number. (Unique Integer > 0)
RID	Ring identification number. See RINGAX entry. (Integer > 0)
R	Effective radius. (Real)
PHI1, PHI2	Azimuthal limits of sector in degrees. (Real)

Remarks:

1. SECTAX is allowed only if an AXIC entry is also present.
2. SECTAX identification numbers must be unique with respect to all other POINTAX, RINGAX and SECTAX identification numbers.
3. For a discussion of the conical shell problem, see "**Conical Shell Element (RINGAX)**" on page 155 of the *MSC.Nastran Reference Guide*.

SEELT Superelement Boundary Element Reassignment

Reassigns superelement boundary elements to an upstream superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEELT	SEID	EID1	EID2	EID3	EID4	EID5	EID6	EID7	

Example:

SEELT	2	147	562	937					

Alternate Format and Example:

SEELT	SEID	EID1	“THRU”	EID2					
SEELT	5	12006	THRU	12050					

Field	Contents
SEID	Superelement identification number. (Integer > 0)
EIDi	Element identification numbers. (Integer > 0 or “THRU”; for “THRU” option EID1 < EID2.)

Remarks:

1. Elements connected entirely to the exterior points of an upstream superelement are called boundary elements and are assigned to the downstream superelement. The SEELT entry provides the means of reassigning the element to the upstream superelement. This entry may be applied to boundary elements only.
2. Open sets are allowed with the “THRU” option.
3. Elements processed with primary superelements will also be contained in any referencing secondary superelement.
4. EIDi may refer to plot elements, general elements, and structural elements.
5. This entry does not change the exterior grid point set of the superelement.
6. SEELT can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.

SEEXCLD Partitioned Superelement Exclusion

Defines grids that will be excluded during the attachment of a partitioned superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEEXCLD	SEIDA	SEIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	-etc.-						

Example1:

SEEXCLD	110	10	45	678	396				

Example 2:

SEEXCLD	400	ALL	10	20	30	THRU	40		

Field	Contents
SEIDA	Partitioned superelement identification number. (Integer $\geq$ 0)
SEIDB	Superelement identification. (Integer $\geq$ 0 or Character = “ALL”, Default = “ALL”)
GIDAi	Identification number of a grid in superelement SEIDA to be excluded from connection to superelement SEIDB. (Integer > 0 or “THRU”; for “THRU” option GIDA1 < GIDA2.)

Remarks:

1. SEEXCLD can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. SEIDA and SEIDB may reference only substructures or the residual structure, that is, parts defined after a BEGIN SUPER = entry.
3. This entry will only work if PART superelements (BEGIN SUPER) exist.

SELABEL Superelement Output Label

Defines a label or name to be printed in the superelement output headings.

Format:

1	2	3	4	5	6	7	8	9	10
SELABEL	SEID	LABEL							

Example:

SELABEL	10	LEFT REAR FENDER, MODEL XYZ2000							

Field	Contents
SEID	Partitioned superelement identification number. (Integer > 0)
LABEL	Label associated with superelement SEID for output headings. (Character)

Remarks:

1. SELABEL can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. Only one SELABEL per superelement may be specified.
3. The label will appear in all superelement output headings. However, in some headings the label may be truncated.
4. This entry will only work if PART superelements (BEGIN SUPER) exist.

SELOC Partitioned Superelement Location

Defines a partitioned superelement relocation by listing three noncolinear points in the superelement and three corresponding points not belonging to the superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SELOC	SEID	PA1	PA2	PA3	PB1	PB2	PB3		

Example:

SELOC	110	10	100	111	1010	112	30		

Field	Contents
SEID	Partitioned identification number of the partitioned superelement. (Integer > 0)
PAi	Identification numbers of three noncolinear grids (GRID entry) or points (POINT entry) which are in the partitioned superelement. (Integer > 0)
PBi	Identification numbers of three grids (GRID entry) or points (POINT entry) defined in the main Bulk Data Section to which PAi will be aligned. (Integer > 0)

Remarks:

1. SELOC can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. The superelement will be rotated and translated for alignment of the GAi and GBi locations.
3. The PAi and PBi can be either GRIDs or POINTs.
4. PA1, PA2, and PA3 must be contained in superelement SEID.
5. PB1, PB2, and PB3 must be specified in the main Bulk Data Section. If they belong to a superelement that is also relocated, then the original (unmoved) positions of PB1, PB2, and PB3 are used.
6. PB1, PB2, and PB3 must have the same relative locations as PA1, PA2, and PA3.
7. Three grids or points are required even if the superelement connects to only one or two exterior grids.

8. Coordinate systems, global displacement directions, and element coordinate systems for the superelement will rotated and translated.
9. The global coordinate directions of the boundary grid points of the upstream superelement will be transformed internally to the global coordinate directions of the attachment grid points in the downstream superelement. For displacement data recovery, the output will be in the original global coordinate system.
10. The translation and rotation of the superelement to the new position is accomplished by defining local rectangular coordinate systems based on the specified grid locations:
    - The local systems have their origin at PX1 and the x-axis points from PX1 to PX2.
    - The y-axis lies in the plane containing PX1, PX2, and PX3, is perpendicular to the x-axis, and points toward PX3.
    - The z-axis is defined by the cross product of the x-axis into the y-axis.
    - The rotation and translation transformation aligns the local system defined by the superelement grids with the local system defined by the main Bulk Data Section grids.
11. This entry will only work if PART superelements (BEGIN SUPER) exist.

SEMPLN Superelement Mirror Plane

Defines a mirror plane for mirroring a partitioned superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEMPLN	SEID	“PLANE”	P1	P2	P3				

Example:

SEMPLN	110	PLANE	12	45	1125				

Field	Contents
SEID	Partitioned superelement identification number. (Integer > 0)
“PLANE”	Flag indicating that the plane is defined by three noncolinear points.
Pi	GRID or POINT entry identification numbers of three noncolinear points. (Integer > 0)

Remarks:

1. SEMPLN can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. Grids or points referenced on this entry must be defined in the main Bulk Data Section.

SENQSET Superelement Internal Generalized Degree-of-Freedom

Defines number of internally generated scalar points for superelement dynamic reduction.

Format:

1	2	3	4	5	6	7	8	9	10
SENQSET	SEID	N							

Example:

SENQSET	110	45							

Field	Contents
SEID	Partitioned superelement identification number. See Remark 3. (Integer > 0 or Character = “ALL”)
N	Number of internally generated scalar points for dynamic reduction generalized coordinates. (Integer > 0; Default = 0)

Remarks:

1. SENQSET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER = n command.
2. SENQSET is only required if the user wants to internally generated scalar points used for dynamic reduction.
3. SEID = “ALL” will automatically generate N q-set degrees-of-freedom for all superelements, except the residual structure (SEID = 0). Specifying additional SENQSET entries for specific superelements will override the value of N specified on this entry.
4. If the user manually specifies q-set degrees-of-freedom using a SEQSETi or QSETi entries, then the internally generated scalar points will not be generated.
5. See PARAM,NQSET for an alternate method of specifying QSET degree-of-freedoms.
6. This entry will only work if PART superelements (BEGIN SUPER) exist.

SEQGP Grid and Scalar Point Resequencing

Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

Format:

1	2	3	4	5	6	7	8	9	10
SEQGP	ID1	SEQID1	ID2	SEQID2	ID3	SEQID3	ID4	SEQID4	

Example:

SEQGP	5392	15.6	596	0.2	2	1.9	3	2	

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
SEQIDi	Sequenced identification number. (Real > 0.0 or Integer > 0)

Remarks:

1. The real format is used to insert a point ID between two consecutively numbered and existing point IDs. In the example above, point ID 5392 is inserted between IDs 15 and 16 by specifying 15.6 for SEQID.
2. The SEQIDi numbers must be unique and may not be the same as a point IDi which is not being changed. No grid point IDi may be referenced more than once.
3. From one to four grid or scalar points may be resequenced on a single entry.
4. If a point IDi is referenced more than once, the last reference will determine its sequence.
5. Automatic resequencing is also available. See “**OLDSEQ**” on page 801.

SEQSEP Superelement Sequences

Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEQSEP	SSID	PSID	GP1	GP2	GP3	GP4	GP5	GP6	
	GP7	GP8	-etc.-						

Example:

SEQSEP	121	21	109	114	124	131			

Field	Contents
SSID	Identification number for secondary superelement. (Integer > 0)
PSID	Identification number for the primary superelement. (Integer $\geq$ 0)
GPi	Exterior grid point identification numbers for the primary superelement. (Integer > 0)

Remarks:

1. This entry is not needed if the grid points listed on the CSUPER entry with the same SSID are in the order of the corresponding exterior grid points of the primary superelement.
2. In **Figure 8-162**, the exterior grid points of 10, 20, and 30 of SEID = 1 correspond to the points 13, 12, and 11, respectively, of image SEID = 2. The CSUPER entry may be defined alone or with a SEQSEP entry as shown in **Figure 8-162**.

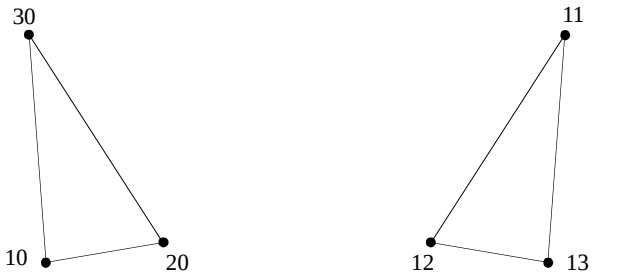


Figure 8-162 Grid Point Correspondence Between Primary and Secondary Superelements

CSUPER Entry Only:

1	2	3	4	5	6	7	8	9	10
CSUPER	2	1	13	12	11				

CSUPER and SEQSEP Entries:

CSUPER	2	1	11	12	13				
SEQSEP	2	1	30	20	10				

SEQSET Superelement Generalized Degree-of-Freedom

Defines the generalized degrees-of-freedom of the superelement to be used in generalized dynamic reduction or component mode synthesis.

Format:

1	2	3	4	5	6	7	8	9	10
SEQSET	SEID	ID1	C1	ID2	C2	ID3	C3		

Example:

SEQSET	15	1	123456	7	5	22	3		

Field	Contents
SEID	Superelement identification number. Must be a primary superelement. (Integer > 0)
Ci	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points.)
IDi	Grid or scalar point identification numbers. Must be an exterior point. (Integer > 0)

Remarks:

1. Degrees-of-freedom specified on this entry may not be specified for another superelement.
2. Generalized degrees-of-freedom are interior to the residual structure.
3. Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.
4. Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.

5. This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user's responsibility to place them in a downstream superelement.

   Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set to allow the boundary points to participate in the system mode shapes.

   Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user's responsibility to place these variables in the proper set in all downstream superelements of which they are members.

6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these degrees-of-freedom for secondary superelements.

SEQSET1 Superelement Generalized Degree-of-Freedom, Alternate Form

Defines the generalized degrees-of-freedom of the superelement to be used in generalized dynamic reduction or component mode synthesis.

Format:

1	2	3	4	5	6	7	8	9	10
SEQSET1	SEID	C	G1	G2	G3	G4	GS	G6	
	G7	G8	-etc.-						

Example:

SEQSET1	15	123456	1	7	9	22	105	6	
	52	53							

Alternate Format and Example:

SEQSET1	SEID	C	G1	“THRU”	G2				
SEQSET1	16	0	101	THRU	110				

Field	Contents
SEID	Superelement identification number. Must be a primary superelement. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. Must be exterior points. (Integer > 0 or “THRU”; for THRU option G1 < G2.)

Remarks:

1. Degrees-of-freedom specified on this entry may not be specified for another superelement.
2. Generalized degrees-of-freedom are interior to the residual structure.
3. Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.

4. Degrees-of-freedom specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
5. This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user's responsibility to place them in a downstream superelement.

   Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set, to allow the boundary points to participate in the system mode shapes.

   Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user's responsibility to place these variables in the proper set in all downstream superelements of which they are members.
6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these entries for secondary superelements.

SESET Superelement Interior Point Definition

Defines interior grid points for a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SESET	SEID	G1	G2	G3	G4	G5	G6	G7	

Example:

SESET	5	2	17	24	25	165			

Alternate Format and Example:

SESET	SEID	G1	“THRU”	G2					
SESET	2	17	THRU	165					

Field	Contents
SEID	Superelement identification number. Must be a primary superelement. (Integer ≥ 0)
Gi	Grid or scalar point identification number. (0 < Integer < 1000000; G1 < G2)

Remarks:

1. Interior grid points may also be defined via field 9 of the GRID and GRIDG Bulk Data entries. The SESET entry takes precedence over the SEID field on the GRID on GRIDG entries. SESET defines grid and scalar points to be included as interior to a superelement. SESET may be used as the primary means of defining superelements or it may be used in combination with SEELT entries which define elements interior to a superelement.
2. Gi may appear on an SESET entry only once.
3. Scalar points are ignored.
4. Open sets are allowed with the “THRU” option. Missing grid points (whether in “THRU” range or mentioned explicitly) are not identified.
5. All degrees-of-freedom for Gi are placed in the o-set of the superelement. See “**Degree-of-Freedom Sets**” on page 939.

6. SESET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER = n command.

SESUP Fictitious Support

Defines determinate reaction superelement degrees-of-freedom in a free-body analysis.

Format:

1	2	3	4	5	6	7	8	9	10
SESUP	SEID	ID1	C1	ID2	C2	ID3	C3		

Example:

SESUP	5	16	215						

Field	Contents
SEID	Superelement identification number. Must a primary superelement. (Integer > 0)
IDi	Grid or scalar point identification number. Must be exterior points. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points; Any unique combination of the Integers 1 through 6 for grid points.)

Remarks:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
2. The Ci degrees-of-freedom must be exterior degrees-of-freedom of the SEID superelement.
3. See "**Rigid Body Supports**" on page 357 of the *MSC.Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
4. There must be a sufficient number of degrees-of-freedom on SESUP entries to discard any free body modes of the superelement.
5. SESUP Bulk Data entries are not allowed for part (partitioned bulk data) superelements. Use the SUPORT Bulk Data records to identify component rigid body modes.

SET1 Set Definition

Defines a list of structural grid points or element ID's.

Format:

1	2	3	4	5	6	7	8	9	10
SET1	SID	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	-etc.-							

Example:

SET1	3	31	62	93	124	16	17	18	
	19								

Alternate Format and Example:

SET1	SID	ID1	"THRU"	ID2					
SET1	6	32	THRU	50					

Field	Contents
SID	Unique identification number. (Integer > 0)
IDi	List of structural grid point identification numbers or element ID's. (Integer > 0 or "THRU"; for the "THRU" option, ID1 < ID2 or "SKIN"; in field 3)

Remarks:

1. When using the "THRU" option for SPLINEi or PANEL data entries, all intermediate grid points must exist.
2. When using the "THRU" option for XYOUTPUT requests, missing grid points are ignored.
3. When using the "SKIN" option, MD Nastran will generate a panel consisting of the structural portion of the fluid-structural boundary.

SET2 Grid Point List

Defines a list of structural grid points in terms of aerodynamic macro elements.

Format:

1	2	3	4	5	6	7	8	9	10
SET2	SID	MACRO	SP1	SP2	CH1	CH2	ZMAX	ZMIN	

Example:

SET2	3	111	0.0	0.75	0.0	0.667	3.51		

Field	Contents
SID	Unique identification number. (Integer > 0)
MACRO	Element identification number of an aerodynamic macro element. (Integer > 0)
SP1, SP2	Lower and higher span division points defining the prism containing the set. (Real)
CH1, CH2	Lower and higher chord division points defining the prism containing the set. (Real)
ZMAX, ZMIN	Z-coordinates of top and bottom (using right-hand rule with the order of the corners as listed on a CAEROi entry) of the prism containing set. (Real)

Remarks:

1. The SET2 entry is referenced by the SPLINEi entry.
2. Every grid point within the defined prism and within the height range will be in the list. For example:

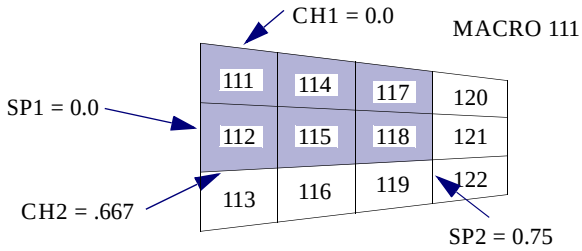


Figure 8-163 SET2 Entry Example.

The shaded area in **Figure 8-163** defines the cross section of the prism for the sample data given above. Points exactly on the boundary may be missed; therefore, to get all the grid points within the area of the macro element, SP1=-.01, SP2=1.01, etc. should be used.

3. A zero value for ZMAX or ZMIN implies a value of infinity. Usually, ZMAX $\geq$ 0.0 and ZMIN $\leq$ 0.0.
4. To print the (internal) grid IDs found, use DIAG 18.

SET3 Labeled Set Definition

Defines a list of grids, elements or points.

Format:

1	2	3	4	5	6	7	8	9	10
SET3	SID	DES	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	ID8	-etc-						

Example:

SET3	1	POINT	11	12	13	15	18	21	

Alternate Format and Example:

SET3	SID	DES	ID1	“THRU”	ID2				
SET3	33	POINT	20	THRU	60				

Field	Contents
SID	Unique identification number. (Integer>0)
DES	Set description (Character). Valid options are “GRID”, “ELEM”, “POINT” and “PROP”.
IDi	Identifiers of grids points, elements, points or properties. (Integer > 0)

Remarks:

1. The SET3 entry is referenced on a PBMSECT or PBRSECT entry the POINTs must lie in the (xy) plane of the basic coordinate system, and be in the order when traversing the boundary or the profile.
2. When the SET3 entry is referenced by a panel, describers can be “GRID”, “ELEM” or “PROP”.

SETREE Superelement Tree Definition (Alternate Form of DTI,SETREE)

Specifies superelement reduction order.

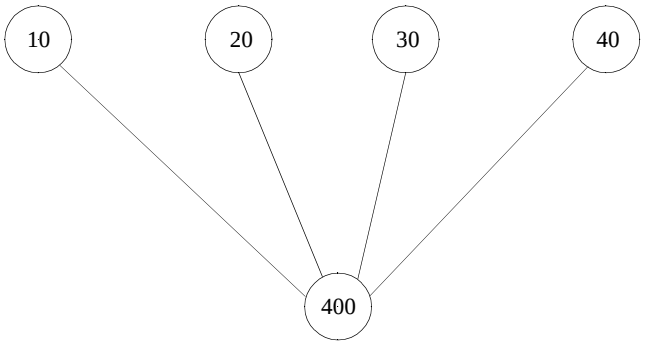
Format:

1	2	3	4	5	6	7	8	9	10
SETREE	SEID	SEUP1	SEUP2	SEUP3	SEUP4	SEUP5	SEUP6	SEUP7	
	SEUP8	SEUP9	-etc.-						

Example:

SETREE	400	10	20	30	40				

Field	Contents
SEID	Partitioned superelement identification number of a downstream superelement. (Integer $\geq$ 0)
SEUPi	Identification number of superelements that are upstream of SEID. (Integer > 0)



Remarks:

1. SETREE entries or DTI,SETREE entry are required for multilevel superelement configurations.
2. At least one SETREE entry is required for each nontip superelement, including the residual structure (SEID = 0). Multiple SETREE entries with the same SEID are allowed.
3. A superelement may appear only once in an SEUPi field on all SETREE entries.

4. If an DTI,SETREE entry is provided, then SETREE entries are not required.
5. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.
6. If a superelement is not referenced on the DTI,SETREE or SETREE entry, then the manner in which it is handled depends on the type of that superelement. If it is a PART superelement, then the residual will be regarded as its downstream superelement and the undefined superelement will therefore be placed immediately above the residual in the tree. If it is a Main Bulk Data superelement, then it will also be handled like an undefined PART superelement as above *if all of its exterior points belong to the residual.* However, if one or more of its exterior points do not belong to the residual, then the program will terminate with a user fatal error complaining that one of more of the superelements are not in the same path.
7. The SETREE entry will only work if PART (BEGIN SUPER) superelements exist in the model. If there are no PARTs in the model, the SETREE entries will be ignored.

SEUSET Superelement Degree-of-Freedom Set Definition

Defines a degree-of-freedom set for a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEUSET	SEID	SNAME	ID1	C1	ID2	C2	ID3	C3	

Example:

SEUSET	15	U4	1	123456	7	5	22	3	

Field	Contents
SEID	Superelement identification number. (Integer > 0)
SNAME	Set name. (One to four characters or string “ZERO”, followed by the set name.)
IDi	Grid or scalar point identification numbers. (Integer > 0)
Ci	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blank for grid points; Integer 0 or blank for scalar points.)

Remarks:

1. SNAME may refer to any of the set names given in “**Degree-of-Freedom Sets**” on page 939 or their new names on the DEFUSET entry. However, in the Solution Sequences 0 through 200, it is recommended that SNAME refer only to the set names “U1" through “U6" or their new names on the DEFUSET entry.
2. If SNAME = “ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.

SEUSET1 Superelement Degree-of-Freedom Set Definition, Alternate Form

Defines a degree-of-freedom set for a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEUSET1	SEID	SNAME	C	G1	G2	G3	G4	G5	
	G6	G7	-etc.-						

Example:

SEUSET1	15	U4	1	12	15	17	22	25	
	52	53							

Alternate Format and Example:

SEUSET1	SEID	SNAME	C	G1	“THRU”	G2			
SEUSET1	15	U4	1	12	THRU	27			

Field	Contents
SEID	Superelement identification number. (Integer > 0)
SNAME	Set name. (One to four characters or string “ZERO”, followed by the set name.)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification number. (Integer > 0)

Remarks:

1. SNAME may refer to any of the set names given in “**Degree-of-Freedom Sets**” on page 939 or their new names on the DEFUSET entry. However, in the Solution Sequences 0 through 200, it is recommended that SNAME refer only to the set names “U1" through “U6" or their new names on the DEFUSET entry.

2. If SNAME = “ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.
3. If the alternate format is used, all of the points G1 through G2 are assigned to the set.

SLBDY Slot Boundary List

Defines a list of slot points that lie on an interface between an axisymmetric fluid and a set of evenly spaced radial slots.

Format:

1	2	3	4	5	6	7	8	9	10
SLBDY	RHO	M	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	-etc.-							

Example:

SLBDY	0.002	6	16	17	18	25	20	21	
	22								

Field	Contents
RHO	Density of fluid at boundary. (Real > 0.0 or blank)
M	Number of slots. (Integer $\geq$ 0 or blank)
IDj	Identification numbers of GRIDS slot points at boundary with axisymmetric fluid cavity, j = 1, 2, ..., J. (Integer > 0)

Remarks:

1. SLDBY is allowed only if an AXSLOT entry is also present.
2. If RHO or M is blank, the default value on the AXSLOT entry is used. The effective value must not be zero for RHO. If the effective value of M is zero, no matrices at the boundary will be generated.
3. The order of the list of points determines the topology of the boundary. The points are listed sequentially as one travels along the boundary in either direction. At least two points must be defined.

SLOAD Static Scalar Load

Defines concentrated static loads on scalar or grid points.

Format:

1	2	3	4	5	6	7	8	9	10
SLOAD	SID	S1	F1	S2	F2	S3	F3		

Example:

SLOAD	16	2	5.9	17	-6.3	14	-2.93		

Field	Contents
SID	Load set identification number. (Integer > 0)
Si	Scalar or grid point identification number. (Integer > 0)
Fi	Load magnitude. (Real)

Remarks:

1. In the static solution sequences, SID is selected by the LOAD Case Control command.
2. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.
3. Up to three loads may be defined on a single entry.
4. If Si refers to a grid point, the load is applied to component T1 of the displacement coordinate system (see the CD field on the GRID entry).

SNORM Surface Normal Vector at Grid Point

Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
SNORM	GID	CID	N1	N2	N3				

Example:

SNORM	3	2	0.	-1.	0.				

Field	Contents
GID	Unique grid point identification number. (Integer > 0)
CID	Identification number of coordinate system in which the components of the normal vector are defined. See Remark **3.** (Integer $\geq$ 0; Default = 0 for the basic coordinate system)
Ni	Components of normal vector. The three components of the normal need not define a unit vector. (Real; Default = 0.0)

Remarks:

1. The SNORM Bulk Data entry overrides any unique, internally-generated grid point normals that may have been requested with the user parameter SNORM, described in Chapter 6 of this guide.
2. The normal is used in CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements. For all other elements, the normal is ignored.

3. If CID is a cylindrical or spherical coordinate system, the components Ni are in the local tangent system at grid GID. For example, if CID=10 is a spherical coordinate system and normals must be defined pointing outwards in the radial direction of the sphere, see **Figure 8-164**, then the SNORM entries for all grids GID on the sphere are simply

   SNORM, GID, 10, 1., 0., 0.

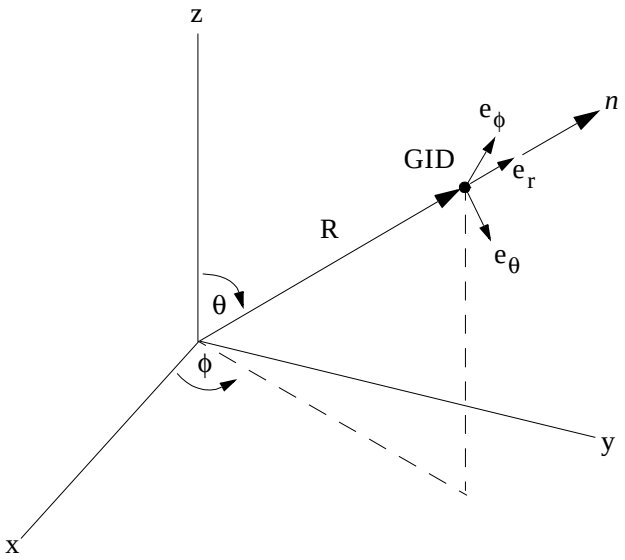


Figure 8-164

SPC Single-Point Constraint

Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis).

Format:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1	D1	G2	C2	D2		

Example:

SPC	2	32	3	-2.6	5				

Field	Contents
SID	Identification number of the single-point constraint set. (Integer > 0)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (0 ≤ Integer ≤ 6; up to six Unique Integers, 1 through 6, may be placed in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
Di	Value of enforced motion for all degrees-of-freedom designated by Gi and Ci. (Real)

Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
3. Single-point forces of constraint are recovered during stress data recovery.
4. From 1 to 12 degrees-of-freedom may be specified on a single entry.
5. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.
6. For reasons of efficiency, the SPCD entry is the preferred method for applying enforced motion rather than the Di field described here.

SPC1 Single-Point Constraint, Alternate Form

Defines a set of single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

SPC1	3	2	1	3	10	9	6	5	
	2	8							

Alternate Format and Example:

SPC1	SID	C	G1	“THRU”	G2				
SPC1	313	12456	6	THRU	32				

Field	Contents
SID	Identification number of single-point constraint set. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or “THRU”; For “THRU” option, G1 < G2.)

Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. Enforced displacements are available via this entry when used with the recommended SPCD entry.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.

4. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.
5. If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will collectively produce a warning message but will otherwise be ignored.

SPCADD Single-Point Constraint Set Combination

Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.

Format:

1	2	3	4	5	6	7	8	9	10
SPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc.-						

Example:

SPCADD	101	3	2	9	1				

Field	Contents
SID	Single-point constraint set identification number. (Integer > 0)
Si	Identification numbers of single-point constraint sets defined via SPC or by SPC1 entries. (Integer > 0; SID ≠ Si)

Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. No Si may be the identification number of a single-point constraint set defined by another SPCADD entry.
3. The Si values must be unique.
4. SPCADD entries take precedence over SPC or SPC1 entries. If both have the same set ID, only the SPCADD entry will be used.

SPCAX Conical Shell Single-Point Constraint

Defines a set of single-point constraints or enforced displacements for conical shell coordinates.

Format:

1	2	3	4	5	6	7	8	9	10
SPCAX	SID	RID	HID	C	D				

Example:

SPCAX	2	3	4	13	6.0				

Field	Contents
SID	Identification number of a single-point constraint set. (Integer > 0)
RID	Ring identification number. See RINGAX entry. (Integer $\geq$ 0)
HID	Harmonic identification number. (Integer $\geq$ 0)
C	Component identification number. (Any unique combination of the Integers 1 through 6.)
D	Enforced displacement value. (Real)

Remarks:

1. SPCAX is allowed only if an AXIC entry is also present.
2. Single-point constraint sets must be selected with the Case Control command SPC = SID.
3. Coordinates appearing on SPCAX entries may not appear on MPCAX, SUPAX, or OMITAX entries.
4. For a discussion of the conical shell problem, see "**Conical Shell Element (RINGAX)**" on page 155 of the *MSC.Nastran Reference Guide*.

SPCD Enforced Motion Value

Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
SPCD	SID	G1	C1	D1	G2	C2	D2		

Example:

SPCD	100	32	436	-2.6	5		2.9		

Field	Contents
SID	Set identification number of the SPCD entry. (Integer > 0)
Gi	Grid or scalar point identification number. (integer > 0)
Ci	Component numbers. ($0 \leq$ Integer ≤ 6; up to six unique Integers may be placed in the field with no embedded blanks, a blank or 0 is treated the same as 1.)
Di	Value of enforced motion for at Gi and Ci. (Real)

Remarks:

1. In the static solution sequences, the set ID of the SPCD entry (SID) is selected by the LOAD Case Control command.
2. In dynamic analysis, the selection of SID is determined by the presence of the LOADSET request in Case Control as follows:
   - *There is no LOADSET request in Case Control*

     SID is selected by the EXCITEID ID of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field
   - *There is a LOADSET request in Case Control*

     SID is selected by LID in the selected LSEQ entries that correspond to the EXCITEID entry of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field.

3. A global coordinate (Gi and CI) referenced on this entry must also be referenced on a SPC or SPC1 Bulk Data entry and selected by the SPC Case Control command.
4. Values of Di will override the values specified on an SPC Bulk Data entry, if the SID is selected as indicated above.
5. The LOAD Bulk Data entry will not combine an SPCD load entry.
6. In static analysis, this method of applying enforced displacements is more efficient than the SPC entry when more than one enforced displacement condition is applied. It provides equivalent answers.
7. In dynamic analysis, this direct method of specifying enforced motion is more accurate, efficient and elegant than the large mass and Lagrange multiplier techniques.

SPCD2 (SOL 700) Prescribed Boundary Motion

Defines an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes. Also velocities and displacements can be imposed on rigid bodies. If the local option is active the motion is prescribed with respect to the local coordinate system for the rigid body (See variable LCO for MATD020 or MATRIG). Translational nodal velocity and acceleration specifications for rigid body nodes are allowed and are applied as described at the end of this section. For nodes on rigid bodies use the NODE option. Do not use the NODE option in r-adaptive problems since the node ID's may change during the adaptive step.

Format:

1	2	3	4	5	6	7	8	9	10
SPCD2	SID	TYPE	SETID	DOF	VAD	LCID	SF	COORD	
	START	END							
	XT	YT	ZT	XH	YH	ZH			
	OFFSET1	OFFSET2	MRB	NODE1	NODE2				

Example:

SPCD2	12	RIGID	15	3	2	10	1	LOCAL	
	0.0	.015							
	0.	0.	0.	0.	1.	0.	CID		
			10						

Field	Contents
SID	ID of a matching SPC Case Control command. (Integer > 0, Required)
TYPE	GRID or RIGID (Character, Required)
SETID	When TYPE=GRID: ID of SET which will hold the nodes that will be constrained (Integer, Required)
	When TYPE=RIGID: ID of a Rigid Body (Integer, Required)
DOF	Applicable degrees-of-freedom: (Integer > 0, Required)
	=1: x-translational degree-of-freedom,
	=2: y-translational degree-of-freedom,
	=3: z-translational degree-of-freedom,

Field	Contents
	=4: translational motion in direction given by the VID. Movement on plane normal to the vector is permitted.
	=-4: translational motion in direction given by the VID. Movement on plane normal to the vector is not permitted. This option does not apply to rigid bodies.
	=5: x-rotational degree-of-freedom,
	=6: y-rotational degree-of-freedom,
	=7: z-rotational degree-of-freedom,
	=8: rotational motion about the vector given by the VID. Rotation about the normal axes is permitted.
	=-8: rotational motion about the vector given by the VID. Rotation about the normal axes is not permitted. This option does not apply to rigid bodies.
	=9: y / z degrees-of-freedom for node rotating about the x-axis at location (OFFSET1,OFFSET2) in the yz-plane, *point (y,z)*. Radial motion is NOT permitted. Not applicable to rigid bodies.
	=-9: y / z degrees-of-freedom for node rotating about the x-axis at location (OFFSET1,OFFSET2) in the yz-plane, *point (y,z)*. Radial motion is permitted. Not applicable to rigid bodies.
	=10: z / x degrees-of-freedom for node rotating about the y-axis at location (OFFSET1,OFFSET2) in the zx-plane, *point (z,x)*. Radial motion is NOT permitted. Not applicable to rigid bodies.
	=-10: z / x degrees-of-freedom for node rotating about the y-axis at location (OFFSET1,OFFSET2) in the zx-plane, *point (z,x)*. Radial motion is permitted. Not applicable to rigid bodies.
	=11: x / y degrees-of-freedom for node rotating about the z-axis at location (OFFSET1,OFFSET2) in the xy-plane, *point (x,y)*. Radial motion is NOT permitted. Not applicable to rigid bodies.
	=-11: x / y degrees-of-freedom for node rotating about the z-axis at location (OFFSET1,OFFSET2) in the xy-plane, *point (x,y)*. Radial motion is permitted. Not applicable to rigid bodies.
VAD	Velocity / Acceleration / Displacement flag: (Integer $\geq$ 0, Required)
	=0: velocity (rigid bodies and nodes),
	=1: acceleration (rigid bodies and nodes only),

Field	Contents
	=2: displacement (rigid bodies and nodes).
	=3: velocity versus displacement (rigid bodies and nodes)
	=4: relative displacement (rigid bodies only)
LCID	TABLED1 ID to describe motion value versus time. (Integer > 0, Required)
SF	Load curve scale factor. (Real > 0.0, Default = 1.0)
	(Note if SF=0.0 it will be reset to 10^{28})
COORD	LOCAL or Blank. Only allowed when TYPE=RIGID. When COORD=LOCAL, the orientation of the local coordinate system rotates with time in accordance with rotation of the rigid body. (Character, Default = Blank)
START	Time imposed motion/constraint is activated. (Real $\geq$ 0.0, Default = 0.0)
END	Time imposed motion/constraint is removed: (Real $\geq$ 0.0, Default = 0.0)
XT	X-coordinate of tail of vector (Real, Default = 0.0)
YT	Y-coordinate of tail of vector (Real, Default = 0.0)
ZT	Z-coordinate of tail of vector (Real, Default = 0.0)
XH	X-coordinate of head of vector (Real, Default = 0.0)
YH	Y-coordinate of head of vector (Real, Default = 0.0)
ZH	Z-coordinate of head of vector (Real, Default = 0.0)
CID	Coordinate system ID to define vector in local coordinate system. All coordinates, XT, YT, ZT, XH, YH, and ZH are in respect to CID. (Integer, no Default)
OFFSET1	Offset for DOF types 9-11 (y, z, x direction) (Real $\geq$ 0.0, Default = 0.0)
OFFSET2	Offset for DOF types 9-11 (z, x, y direction) (Real $\geq$ 0.0, Default = 0.0)
MRB	Master rigid body for measuring the relative displacement. (Integer > 0, Required)
NODE1	Optional orientation node, n1, for relative displacement (Integer > 0, Default = 0)
NODE2	Optional orientation node, n2, for relative displacement (Integer > 0, Default = 0)

Remarks:

Corresponds to Ls-Dyna entry * BOUNDARY_PRESCRIBED_MOTION.

Arbitrary translations and rotations are possible. Rotations around local axis can be defined either by setting DOF = 8 or by using the offset option of DOF > 8. The load curve scale factor can be used for simple modifications or unit adjustments.

The relative displacement can be measured in either of two ways:

1. Along a straight line between the mass centers of the rigid bodies,
2. Along a vector beginning at node n1 and terminating at node n2.

With option 1, a positive displacement will move the rigid bodies further apart, and, likewise a negative motion will move the rigid bodies closer together. The mass centers of the rigid bodies must not be coincident when this option is used. With option 2 the relative displacement is measured along the vector, and the rigid bodies may be coincident. Note that the motion of the master rigid body is not directly affected by this option, i.e., no forces are generated on the master rigid body.

The activation time, BIRTH, is the time during the solution that the constraint begins to act. Until this time, the prescribed motion card is ignored. The function value of the load curves will be evaluated at the offset time given by the difference of the solution time and BIRTH, i.e., (solution time-BIRTH). Relative displacements that occur prior to reaching BIRTH are ignored. Only relative displacements that occur after BIRTH are prescribed.

When the constrained node is on a rigid body, the translational motion is imposed without altering the angular velocity of the rigid body by calculating the appropriate translational velocity for the center of mass of the rigid body using the equation:

$$v_{cm} = v_{node} - \omega \cdot (x_{cm} - x_{node})$$

where v_{cm} is the velocity of the center of mass, v_{node} is the specified nodal velocity, ω is the angular velocity of the rigid body, x_{cm} is the current coordinate of the mass center, and x_{node} is the current coordinate of the nodal point. Extreme care must be used when prescribing motion of a rigid body node. Typically, for nodes on a given rigid body, the motion of no more than one node should be prescribed or unexpected results may be obtained.

When the RIGID option is used to prescribe rotation of a rigid body, the axis of rotation will always be shifted such that it passes through the center-of-mass of the rigid body. By using *PART_INERTIA or *CONSTRAINED_NODAL_RIGID_BODY_INERTIA, one can override the internally-calculated location of the center-of-mass.

When the RIGID_LOCAL option is invoked, the orientation of the local coordinate system rotates with time in accordance with rotation of the rigid body.

SPCOFF Excludes Degrees-of-Freedom from the AUTOSPC Operation

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See "**Constraint and Mechanism Problem Identification in SubDMAP SEKR**" on page 409 of the *MSC.Nastran Reference Guide* for a description of the AUTOSPC operation.

Format:

1	2	3	4	5	6	7	8	9	10
SPCOFF	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

SPCOFF	32	436	5	1					

Field	Contents
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points; Integers 1 through 6 with no embedded blanks for grid points.)

Remarks:

1. Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.
2. Degrees-of-freedom that are specified as both SPC and SPCOFF will be considered as SPC.

SPCOFF1 Excludes DOF's from AUTOSPC Processing, Alternate Form

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See "**Constraint and Mechanism Problem Identification in SubDMAP SEKR**" on page 409 of the *MSC.Nastran Reference Guide* for a description of the AUTOSPC operation.

Format:

1	2	3	4	5	6	7	8	9	10
SPCOFF1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	-etc.-						

Example:

SPCOFF1	2	1	3	10	9	6	5	4	
	8								

Alternate Format and Example:

SPCOFF1	C	G1	"THRU"	G2					
SPCOFF1	12456	6	THRU	32					

Field	Contents
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or "THRU"; for "THRU" option, G1 < G2.)

Remarks:

1. Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.
2. Degrees-of-freedom that are both specified as SPC and SPCOFF will be considered as SPC.
3. If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points which do not exist will collectively produce a warning message but will otherwise be ignored.

SPLINE1 Surface Spline Methods

Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE1	EID	CAERO	BOX1	BOX2	SETG	DZ	METH	USAGE	
	NELEM	MELEM							

Example:

SPLINE1	3	111	115	122	14	0.			

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero-element (CAEROi entry ID) that defines the plane of the spline. (Integer > 0)
BOX1, BOX2	First and last box with motions that are interpolated using this spline; see Remark **3.** when using Mach Box method. (Integer > 0, BOX2 $\geq$ BOX1)
SETG	Refers to the SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real $\geq$ 0.0)
METH	Method for the spline fit. IPS,TPS or FPS. See Remark **1.** (Character, Default = IPS)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark **5.** (Character, Default = BOTH)
NELEM	The number of FE elements along the local spline x-axis if using the FPS option. (Integer > 0; Default = 10)
MELEM	The number of FE elements along the local spline y-axis if using the FPS option. (Integer > 0; Default = 10)

Remarks:

1. The default METHOD will result in the use of the Harder-Desmarais Infinite Plate Spline (IPS). The other options are the Thin Plate Spline (TPS) and the Finite Plate Spline (FPS). The continuation applies only to the FPS option and is required only if the defaults are not adequate.
2. The interpolated points (k-set) will be defined by aero boxes. **Figure 8-165** shows the cells for which u_k is interpolated if BOX1 = 115 and BOX2 = 118.

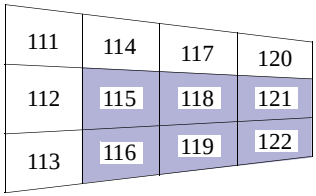


Figure 8-165 SPLINE1 Entry Example

3. The attachment flexibility (units of area) is used for smoothing the interpolation. If DZ = 0.0, the spline will pass through all deflected grid points. If DZ is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.
4. When using the Mach Box method, BOX1 and BOX2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to structural grids. BOX1 and BOX2 do not refer to Mach Boxes.
5. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

   $Fg = [GPkg]^T\{Pk\}$ (FORCE/BOTH splines are in the transform)

   $Uk = [GDkg]\{Ug\}$ (DISP/BOTH splines are in the transform)

   In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship $[GPgk]^T = [GDkg]$ satisfied. The default behavior (BOTH for all splines) is compatible with versions of MSC.Nastran prior to Version 70.5.

   In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the

aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from tis effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCEs from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). MD Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform "rigid aerodynamic" analyses).

SPLINE2 Linear Spline

Defines a beam spline for interpolating motion and / or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE2	EID	CAERO	ID1	ID2	SETG	DZ	DTOR	CID	
	DTHX	DTHY		USAGE					

Example:

SPLINE2	5	8	12	24	60	0.	1.0	3	
	1.								

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero panel or body (CAEROi entry ID) that is to be interpolated. (Integer > 0)
ID1, ID2	First and last box or body element whose motions are interpolated using this spline. See Remark **6.** when using the Mach Box method. (Integer > 0, ID2 ≥ ID1)
SETG	Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real ≥ 0.0)
DTOR	Torsional flexibility ratio (*EI* / *GJ*). (Real > 0.0; Default = 1.0; use 1.0 for bodies.)
CID	Rectangular coordinate system for which the y-axis defines the axis of the spline. Not used for bodies, CAERO2. (Integer ≥ 0)
DTHX, DTHY	Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (in-plane bending rotations); however, it is not used for bodies. DTHY is for rotation about the spline's y-axis (torsion); however, it is used for slope of bodies. (Real)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark **9.** (Character, Default = BOTH)

Remarks:

1. The interpolated points (k-set) will be defined by aero boxes.
2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the spline axis is parallel to the x-axis of the aerodynamic coordinate system.
3. The flexibilities DZ, DTHX, and DTHY are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing, whereas negative values of DTHX or DTHY will imply infinity, therefore, no attachment). See the ***MSC.Nastran Aeroelastic Analysis User's Guide*** for a discussion of special cases.
4. The continuation entry is required.
5. The SPLINE2 EID must be unique with respect to all SPLINEi entries.
6. When using the Mach Box method, ID1 and ID2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to the structural grids. ID1 and ID2 do not refer to Mach Boxes.
7. DTOR is the ratio of rotational to linear deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur.
8. If a SPLINE2 element only references one grid point, the job will fail without a message in the GI Module.
9. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark **5.** of the SPLINE1 Bulk Data entry.

SPLINE3 Aeroelastic Constraint Equation

Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE3	EID	CAERO	BOXID	COMP	G1	C1	A1	USAGE	
	G2	C2	A2		-etc.				

Example:

SPLINE3	7000	107	109	6	5	3	1.0		
	43	5	-1.0						

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Identification number of the macro-element on which the element to be interpolated lies. (Integer > 0)
BOXID	Identification number of the aerodynamic element; i.e., the box number. (Integer > 0)
COMP	The component of motion to be interpolated. See Remark **4.** (One of the Integers 1, 2, 3, 4, 5, or 6.)
Gi	Grid point identification number of the independent grid point. (Integer > 0)
Ci	Component numbers in the displacement coordinate system. (One of the Integers 1 through 6 for grid points, or 0 for scalar points.)
Ai	Coefficient of the constraint relationship. (Real)
USAGE	Spline uage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark **6.** (Character, Default = BOTH).

Remarks:

1. The independent grid points and components must refer to degrees-of-freedom in the g-set.
2. The constraint is given by

$$u_d = \sum A_i u_i$$

where

u_d = value of the dependent component of the aerodynamic element

u_i = displacement at grid Gi, component Ci.

3. The SPLINE3 EID must be unique with respect to all SPLINEi entries.
4. The allowable components by CAEROi entry type are indicated by an “X” in the table below:

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X	X	
CAERO2		X	X	X	X
CAERO3			X		
CAERO4			X	X	X
CAERO5			X	X	X
3D Geometry	X	X	X	X	X

COMP = 2: lateral displacement

COMP = 3 transverse displacement

COMP = 5: pitch angle

COMP = 6: relative control angle for CAERO4 and CAERO5 yaw angle for CAERO2.

For general 3D aerodynamic geometries the components numbers refer to axes of the Aerodynamic Coordinate System $(u_x, u_y, u_z, \theta_x, \theta_y, \theta_z)$.

5. For Strip theory and Piston theory, the COMP = 6 control surface relative angle is positive when the trailing edge has a negative z-deflection in the element coordinate system (see the ***MSC.Nastran Aeroelastic Analysis User’s Guide***).

6. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark **5.** of the SPLINE1 Bulk Data entry.

SPLINE4 Surface Spline Methods

Defines a curved surface spline for interpolating motion and / or forces for aeroelastic problems on general aerodynamic geometries using either the Infinite Plate, Thin Plate or Finite Plate splining method.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE4	EID	CAERO	AELIST		SETG	DZ	METH	USAGE	
	NELEM	MELEM							

Example:

SPLINE4	3	111	115		14	0.	IPS		

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero panel ID that defines the interpolation surface. (Integer > 0)
AELIST	Identification of an AELIST entry listing boxes with motions that are interpolated using this spline; see Remark **3.** when using Mach Box method. (Integer > 0)
SETG	Refers to the SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real $\geq$ 0.0)
METH	Method for the spline fit. One of IPS, TPS or FPS. See Remark **1.** (Character, Default = IPS)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark **4.** (Character, Default = BOTH)
NELEM	The number of FE elements along the local spline x-axis if using the FPS option. (Integer > 0; Default = 10)
MELEM	The number of FE elements along the local spline y-axis if using the FPS option. (Integer > 0; Default = 10)

Remarks:

1. The default METHOD will result in the use of the Harder-Desmarais Infinite Plate Spline (IPS). The other options are the Thin Plate Spline (TPS) and the Finite Plate Spline (FPS). The continuation applies only to the FPS option and is required only if the defaults are not adequate.
2. The attachment flexibility (units of area) is used for smoothing the interpolation. If DZ = 0.0, the spline will pass through all deflected grid points. If DZ is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.
3. When using the Mach Box method, the AELIST boxes refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to structural grids. They do not refer to Mach Boxes.
4. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark **5.** of the SPLINE1 Bulk Data entry.

SPLINE5 Linear Spline

Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by irregular arrays of aerodynamic points. The interpolating beam supports axial rotation and bending in the yz-plane.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE5	EID	CAERO	AELIST		SETG	DZ	DTOR	CID	
	DTHX	DTHY		USAGE					

Example:

SPLINE5	5	8	12		60			3	
	1.			BOTH					

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero panel or body (CAEROi entry ID) that is to be interpolated. (Integer > 0)
AELIST	Identification number of an AELIST entry that identifies the aerodynamic boxes whose motions are interpolated using this spline. See Remark **6.** when using the Mach Box method. (Integer > 0)
SETG	Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark **8.** (Character, Default = BOTH)
CID	Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz-planes for bending. Not used for bodies, CAERO2. (Integer ≥ 0)
DTHX, DTHY	Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (the bending rotations). DTHY is for rotation about the spline's y-axis (torsion); however, it is used for bending of bodies. (Real)

Field	Contents
DZ	Linear attachment flexibility. (Real ≥ 0.0, Default = 0.0)
DTOR	Torsional flexibility ratio (EI/GJ) for the bending in the zy-plane. This value is ignored for slender bodies since they have no torsion; see Remark 7. (Real > 0.0; Default = 1.0; ignored for CAERO2 bodies.)

Remarks:

1. The interpolated points (k-set) will be defined by aero boxes.
2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the interpolating beam (y-axis) is parallel to the x-axis of the aerodynamic coordinate system; the z-axis is taken from the referenced CID and x is made orthogonal.
3. The flexibilities DZ, DTHX and DTHY are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing, whereas negative values of DTHX or DTHY will imply infinity, therefore, no attachment.) See the ***MSC.Nastran Aeroelastic Analysis User's Guide*** for a discussion of special cases.
4. The continuation entry is required.
5. The SPLINE5 EID must be unique with respect to all SPLINEi entries.
6. When using the Mach Box method, the AELIST entries refer to the ID numbers of aerodynamic grids (x,y pairs on the AEFACT entry) which will be used for interpolation to the structural grids. They do not refer to Mach Boxes.
7. DTOR is the ratio of axial rotational to bending deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur. The values will affect the results only if the structural grids overconstrain the motion of the interpolating beam. Slender bodies have no torsional motion, so these values will not be used for CAERO2 entries.
8. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.

SPLINE6 3D Finite Surface Spline

Defines a 6DOF or 3DOF finite surface spline for interpolating motion and / or forces between two meshes.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE6	EID	CAERO	AELIST		SETG	DZ	METHOD	USAGE	
	VSTYPE	VSLIST	I2VNUM	D2VNUM	METHVS	DZR	METHCON	NGRID	
	ELTOL	NCYCLE	AUGWEI						

Example:

SPLINE6	5	8	12		60		FBS6	DISP	
	AERO	2	4	4	VS6				

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero panel or body (CAEROi entry ID) that is to be interpolated. See Remark 5. (Integer > 0 or blank)
AELIST	Identification number of an AELIST entry that identifies the aerodynamic boxes whose motions are interpolated using this spline. (Integer > 0)
SETG	Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real $\geq$ 0.0, Default = 1.0)
METHOD	Method for the spline fit. Either FPS3 or FPS6. See Remark 6..(Character, Default = FPS6)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 4..(Character, Default = BOTH)
VSTYPE	Virtual surface connectivity type. Either AERO or STRUC. (Character, Default = AERO)
VSLIST	Identification number of an AELIST entry listing quadrilateral and / or triangular shell elements of the VSTYPE mesh which define the connectivity of the virtual surface mesh. (Integer > 0 or blank)

Field	Contents
I2VNUM	The minimum number of structural mesh points to connect to each virtual mesh point. See Remarks **8.**, **9.** and **10.** (0 < Integer, Default = 3)
D2VNUM	The minimum number of aero mesh points to connect to each virtual mesh point. See Remarks **8.**, **9.** and **10.** (0 < Integer, Default = 3)
METHVS	Similar to METHOD, this field chooses whether or not to include the rotational degrees-of-freedom of virtual surface. Either VS6 or VS3. See Remark **5.** (Character, Default VS6)
DZR	Rotational attachment flexibility. (Real $\geq$ 0.0, Default = 1.0)
METHCON	Method used to determine RBE3 connecting points between the meshes. Either NODEPROX or CIRCBIAS. See Remarks **8.**, **9.** and **10.** (Character, Default = NODEPROX)
NGRID	Number of closest grids that are used to determine the element list that is used to define the RBE3 elements. Only valid for METHCON=CIRCBIAS. See Remarks **8.** and **10.** (Integer > 0, Default = 1)
ELTOL	Tolerance used to determine whether or not a node projects onto an element of the mesh. Specified as % of element size. Only valid for METHCON=CIRCBIAS. See Remarks **8.** and **10.** (REAL, Default = 100.0)
NCYCLE	Maximum number of cycles used to find elements onto which the nodes project. Only valid for METHCON=CIRCBIAS. See Remark **8.** and **10.** (Integer > 0. Default = 3)
AUGWEI	RBE3 weighting factor augmentation parameter. Only valid for METHCON=CIRCBIAS. See Remarks **8.** and **10.** (Real $\geq$ 0.0, Default = 0.0)

Remarks:

1. The flexibilities DZ and DZR are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing). The DZ and DZR values are used to derive stiffness of the translational and rotational (respectively) bushing stiffnesses. Bushing elements are placed between the interpolating surface and the connections to the dependent and independent grids.
2. The SPLINE6 EID must be unique with respect to all SPLINEi entries.

3. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5 of the SPLINE1 Bulk Data entry.
4. The FPS3 method will map only the three translational degrees of freedom of the independent grids. The FPS6 method will map all six degrees of freedom.
5. The CAERO field may be blank, in which case the swarm of points in the dependent grid list can span aero panels.
6. The METHOD option provides a choice in using all 6 degrees of freedom (FPS6) on the independent points or only the translational degrees of freedom (FPS3) in connecting between the virtual surface and the independent points. Similarly, there is a choice in connecting the virtual surface to the dependent points (METHVS).
7. The connection between the independent points (structural) and the dependent points (aero) is made through a virtual surface whose mesh is defined by elements listed in the VSLIST (these are either AERO box id's or STRUCtural shell elements (QUAD4,R,TRIA3,R).
8. To bind the points to the virtual surface, a connection is made between the points and the surface using automatically generated virtual RBE3 elements. Two methods exist to choose which independent mesh points are connected to each dependent mesh point: nodal-proximity (NODEPROX) and circular bias (CIRCBIAS).
9. The nodal proximity method selects the closest independent mesh points to each dependent mesh point. The actual number of points will depend on the user inputs I2VNUM and D2VNUM as well as collinearity checks. Larger values will spread the connectivity (smearing). Smaller values allow for more concentration (with additional points added as necessary for collinearity).
10. The circular bias method uses elements of the virtual mesh in an attempt to select independent mesh points that encircle each dependent mesh point. This method will be restricted to the case where the virtual mesh is the target mesh.
    This method will do the following:
    - For each splined dependent mesh node, find the closest NGRID splined independent mesh node(s).
    - Assemble the list of virtual mesh elements that use the closest node(s).

- Check each of these elements to see if the dependent node projects onto the element in the element's mean plane normal direction. Note that this check may be computationally expensive, so it is performed only to the "possible" elements, not the entire virtual mesh. The projection check will contain a user-defined tolerance, ELTOL, to expand the area of the element that is acceptable for a match.
- If the dependent node does not project onto any element, use the candidate element's nodes to expand the list of elements to check. Repeat the projection check (the original elements will not be rechecked). Repeat this process up to NCYCLE times.
- All elements that are found to encompass the dependent node (and there may be more than one due to curvature) will be selected to move forward.
- Assemble the list of all splined nodes that connect the selected elements.
- Generate RBE3 elements based on this node list. An optional user-defined input parameter, AUGWEI, will be used to augment the RBE3 weighting factors with the following formula:

  $$\text{weight} = (\text{NE} - 1) \cdot \text{AUGWEI} + 1$$

  where NE is the number of elements that are connected with the RBE3 node.

SPLINE7 Finite Beam Spline

Defines a 6DOF finite beam spline for interpolating motion and / or forces between two meshes.

Format:

1	2	3	4	5	6	7	8	9	10
SPLINE7	EID	CAERO	AELIST		SETG	DZ	DTOR	CID	
				USAGE	METHOD	DZR	IA2		

Example:

SPLINE7	5	8	12		60			3	
				BOTH	FBS6				

Field	Contents
EID	Element identification number. (Integer > 0)
CAERO	Aero panel or body (CAEROi entry ID) that is to be interpolated. See Remark 5. (Integer > 0 or blank)
AELIST	Identification number of an AELIST entry that identifies the aerodynamic boxes whose motions are interpolated using this spline. (Integer > 0)
SETG	Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)
DZ	Linear attachment flexibility. (Real $\geq$ 0.0, Default = 1.0)
DTOR	Ratio of the beam bending stiffness to the beam torsional stiffness. See Remark 3. (Real > 0.0; Default = 1.0)
USAGE	Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 4..(Character, Default = BOTH)
CID	Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz-planes for bending. Not used for bodies, CAERO2. (Integer $\geq$ 0)
METHOD	Method for the spline fit. Either FPS3 or FPS6. See Remark 6..(Character, Default = FPS6)

Field	Contents
DZR	Rotational attachment flexibility. (Real $\geq$ 0.0, Default = 1.0)
IA2	Ratio of the beam bending stiffness to the beam extensional stiffness. (Real > 0.0; Default = xxxx)

Remarks:

1. The flexibilities DZ and DZR are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing). The DZ and DZR values are used to derive stiffness of the translational and rotational (respectively) bushing stiffnesses. Bushing elements are placed between the interpolating beam and the connections to the dependent and independent grids.
2. The SPLINE7 EID must be unique with respect to all SPLINEi entries.
3. DTOR is the ratio of axial rotational to bending deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily translational deflection will occur. The values will affect the results only if the structural grids over constrain the motion of the interpolating beam. Slender bodies have no torsional motion, so these values will not be used for CAERO2 entries.
4. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark **5.** of the SPLINE1 Bulk Data entry.
5. The FPS3 method will map only the three translational degrees of freedom. The FPS6 method will map all six degrees of freedom.
6. The CAERO field may be blank, in which case the swarm of points in the dependent grid list can span aero panels.

SPOINT Scalar Point Definition

Defines scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
SPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	

Example:

SPOINT	3	18	1	4	16	2			

Alternate Format and Example:

SPOINT	ID1	“THRU”	ID2						
SPOINT	5	THRU	649						

Field	Contents
IDi	Scalar point identification number. (0 < Integer < 100000000; For “THRU” option, ID1 < ID2)

Remarks:

1. A scalar point defined by its appearance on the connection entry for a scalar element (see the CELASi, CMASSi, and CDAMPi entries) need not appear on an SPOINT entry.
2. All scalar point identification numbers must be unique with respect to all other structural, scalar, and fluid points. However, duplicate scalar point identification numbers are allowed in the input.
3. This entry is used primarily to define scalar points appearing in single-point or multipoint constraint equations to which no scalar elements are connected.
4. If the alternate format is used, all scalar points ID1 through ID2 are defined.
5. For a discussion of scalar points, see “**Scalar Elements (CELASi, CMASSi, CDAMPi)**” on page 193 of the *MSC.Nastran Reference Guide*.

SUPAX Conical Shell Fictitious Support

Defines determinate reaction degrees-of-freedom in free bodies for conical shell analysis.

Format:

1	2	3	4	5	6	7	8	9	10
SUPAX	RID1	HID1	C1	RID2	HID2	C2			

Example:

SUPAX	4	3	2						

Field	Contents
RIDi	Ring identification number. (Integer > 0)
HIDi	Harmonic identification number. (Integer $\geq$ 0)
Ci	Conical shell degree-of-freedom numbers. (Any unique combination of the Integers 1 through 6.)

Remarks:

1. SUPAX is allowed only if an AXIC entry is also present.
2. Up to 12 degrees-of-freedom may appear on a single entry.
3. Degrees-of-freedom appearing on SUPAX entries may not appear on MPCAX, SPCAX, or OMITAX entries.
4. For a discussion of conical shell analysis, see “**Conical Shell Element (RINGAX)**” on page 155 of the *MSC.Nastran Reference Guide.*

SUPORT Fictitious Support

Defines determinate reaction degrees-of-freedom in a free body.

Format:

1	2	3	4	5	6	7	8	9	10
SUPORT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

SUPORT	16	215							

Field	Contents
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. The SUPORT entry specifies reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint (i.e., SPCi entry or PS on the GRID entry).
2. SUPORT and / or SUPORT1 entries are required to perform inertia relief in static analysis (SOL 101) if PARAM,INREL,-1 is specified. But if PARAM,INREL,-2 is specified, then SUPORT and / or SUPORT1 entries are not required.
3. Be careful not to spell SUPORT with two Ps.
4. Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See “**Degree-of-Freedom Sets**” on page 939 for a list of these entries.
5. From 1 to 24 support degrees-of-freedom may be defined on a single entry.
6. See “**Rigid Body Supports**” on page 357 of the *MSC.Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
7. An alternative to SUPORT is the SUPORT1 entry, which is requested by the SUPORT1 Case Control command.

SUPORT1 Fictitious Support, Alternate Form

Defines determinate reaction degrees-of-freedom (r-set) in a free body-analysis. SUPORT1 must be requested by the SUPORT1 Case Control command.

Format:

1	2	3	4	5	6	7	8	9	10
SUPORT1	SID	ID1	C1	ID2	C2	ID3	C3		

Example:

SUPORT1	5	16	215						

Field	Contents
SID	Identification number of the support set. See Remark 1. (Integer > 0)
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. The SUPORT entry specifies reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint; (i.e., SPCi entry or PS on the GRID entry).
2. SUPORT and / or SUPORT1 entries are required to perform inertia relief in static analysis (SOL 101) if PARAM,INREL,-1 is specified. But if PARAM,INREL,-2 is specified, then SUPORT and / or SUPORT1 entries are not required.

   In SOL 101, PARAM,INREL,-1 must also be specified or the SUPORTi entries will be treated as constraints.
3. SUPORT1 must be requested by the SUPORT1 Case Control command. The degrees-of-freedom specified on SUPORT1 will be combined with those on the SUPORT entry.
4. Be careful not to spell SUPORT with two Ps.

5. Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See "**Degree-of-Freedom Sets**" on page 939 for a list of these entries.
6. From 1 to 18 support degrees-of-freedom may be defined on a single entry.
7. See "**Rigid Body Supports**" on page 357 of the *MSC.Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
8. In superelement analysis, SUPORT1 may be specified for points belonging to the residual structure only.

SUPORT6 (SOL 600) Inertia Relief for SOL 600

Inertia relief used in Nastran Implicit Nonlinear (SOL 600 only)

Format:

1	2	3	4	5	6	7	8	9	10
SUPORT6	SID	METH	IREMOV	GID			IDS1		

Example:

SUPORT6	0	3	1				101		
SUPORT6	4	3	-2						

Field	Contents
SID	Set ID corresponding to a Case Control SUPORT1 entry or zero. (Integer; Default = 0) 0 = If this is the only SUPORT6 entry, use this SUPORT6 entry for all subcases. If there are multiple SUPORT6 entries, use the one with SID=0 for Marc increment zero. N = Use this SUPORT6 entry for the subcase specified by Case Control SUPORT1=N. Different SUPORT6 entries can be used for each subcase if desired and different subcases can use different methods. If there is only one SUPORT6 entry (with SID=0), no Case Control SUPORT1 entries are necessary.
METH	Method to use (Integer, Default = 0) 0 = Inertia relief is not active for this subcase. 3 = Use the “Support Method”, usually specified using param,inrel,-1 for other solution sequences. (See Remark 1.). Do not enter the continuation line. Input will come from all SUPORT entries and those SUPORT1 entries with ID=SID.

Field	Contents
IREMOV	Method to retain or remove inertia relief from a previous subcase (Integer; Default = 1) 1 = Retain inertia relief conditions from previous subcase. -1 = Remove inertia relief loads immediately. -2 = Remove inertia relief loads gradually. IREMOV should be blank or 1 unless METH is 0.
GID	Reference Grid ID for kinematic method (Integer; Default = 0) =0 Use the origin. =N Use grid ID N. (Used for METH=1 ONLY).
IDS1	ID of SUPORT1 entries to be used if METH=3 and SID=0 (Integer, no Default) For METH=3, only SUPORT1 entries with ID=IDS1 will be used in Marc increment zero. All SUPORT entries will be used (Used for METH=3 when SID=0 ONLY)

Remark:

1. The parameter INREL is ignored by SOL 600.

SWLDPRM Parameters for CFAST, CWELD, and CWSEAM Connectors

Overrides default values of parameters for CFAST, CWELD, and CWSEAM connectivity search.

Format:

1	2	3	4	5	6	7	8	9	10
SWLDPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	PARAM4	VAL4	
	PARAM5	VAL5	-etc.-						

Example:

SWLDPRM	GSPROJ	15.0	GSMOVE	2	PRTSW	1			

Field	Contents
PARAMi	Name of the spot weld parameter. Allowable names are listed in **Table 8-33**. (Character)
VALi	Value of the parameter. (Real or Integer; see **Table 8-33**.)

Table 8-33 PARAMi Names and Descriptions

Name	Type	Default	Description
CHKRUN	Integer 0, 1, 2	0	Stop the program after the connectivity of CWELD elements are generated. 0=abort on first error or run to completion; 1=stop after weld connectivity has been checked; 2=continue run if no errors are found or stop run after all weld connectivity errors have been found.

Table 8-33 PARAMi Names and Descriptions

Name	Type	Default	Description
GMCHK	Integer 0, 1, 2	0	For CWELD with PARTPAT format and CWSEAM only. 0=no geometry error checks; 1=check errors of CWELD elements with patch A and patch B tilting toward each other or check errors of the CWSEAM across a cutout or over a corner with patch elements in plane or out of plane; 2=check errors and output all candidate shell elements if an error is encountered. If GMCHK=1 or 2 and an error is detected, the program will loop back to search for next candidate element until a good pair of connection is found or all adjacent elements have been checked. In the latter case, a user fatal message 7595, 7638, or 7667 will be issued.
			A UFM 7595 is issued if the angle between the normal vectors of the patches at end GS or between the normal vectors of the patches at end GE exceeds the value of GSPROJ; a UFM 7638 is issued if either the length of the seam spans more than three elements or the seam spans a cutout; a UFM 7667 is issued if the angle between the normal vectors of the top patches at GS and GE or between the normal vectors of the bottom patches at GS and GE exceeds GSPROJ or if the angle between the free edges of the shell elements onto which GS and GE are projected is less than 160^{o} (if GSPROJ $\leq$ 20.0) or (180^{o}-GSPROJ) (if GSPROJ > 20.0).
GSMOVE	Integer $\geq$ 0	0	Maximum number of times GS for the CFAST or CWELD (PARTPAT or ELPAT options only) or GS/GE for the CWSEAM is moved in case a complete projection of all auxiliary points has not been found.

Table 8-33 PARAMi Names and Descriptions

Name	Type	Default	Description
GSPROJ	Real	20.0	Maximum angle allowed between the normal vectors of shell A and shell B. The connector element will not be generated if the angle between these two normal vectors is greater than the value of GSPROJ. For CWSEAM, see also GMCHK for additional error checks using GSPROJ. If GSPROJ is set to -1.0, the program will skip the checking of GSPROJ.
GSTOL	Real $\geq$ 0.0	0.0	For CFAST or CWELD (PARTPAT and ELPAT only), if GSTOL > 0.0 and the distance between GS and the projected point GA or GB is greater the GSTOL, a UFM 7549 is issued and the CFAST or CWELD is rejected. For CWSEAM, if GSTOL > 0.0 and the distance between GS and the projected point GSA or GSB or the distance between GE and the projected point GEA or GEB is greater than the GSTOL, a UFM 7549 is issued and the CWSEAM is rejected.
NREDIA	0 $\leq$ Integer $\leq$ 4	0	CFAST or CWELD (PARTPAT and ELPAT) only. Maximum number of times the diameter D is reduced in half in case a complete projection of all points has not been found.

Table 8-33 PARAMi Names and Descriptions

Name	Type	Default	Description
PROJTOL	0.0 ≤ Real ≤ 0.2	0.02	For CFAST or CWELD, tolerance to accept the projected point GA or GB if the computed coordinates of the projection point lie outside the shell element but is located within PROJTOL*(dimension of the shell element forming the patch). For the PARTPAT option for the CWELD or the PROP option for the CFAST it is recommended that PROJTOL=0.0. For the CWSEAM, a projection from GS/GE will always be attempted as if PROJTOL=0.0 and if one cannot be found then the non-zero value of PROJTOL will be used.
PRTSW	Integer 0, 1, 2	0	Print diagnostic output. 0=no diagnostic output; 1=print diagnostic output to f06 file; 2=punch diagnostic output.

Remarks:

1. This entry changes the default settings of control variables for the CFAST, CWELD, and CWSEAM connector elements. None of the parameters of this entry are required. Only one SWLDPRM entry is allowed in the Bulk Data Section.
2. Connectivity information is generated for the CFAST and CWSEAM elements. For the CWELD, connectivity information is only generated for the PARTPAT, ELPAT, ELEMID, and GRIDID options.
3. The user defined grid point GS is projected on shell A and B to create the connector end points GA and GB, respectively (see **Figure 8-166**). If GS lies in between the shells A and B, the grids GA and GB are created through a normal projection of GS on shell A and B, respectively. If GS lies outside of the shells A and B, the grids GA' and GB' are created through a normal projection of GS on shell A and B, respectively. Next, the point GC' is created in the middle between GA' and GB', A second normal projection of GC' on shells A and B gives the final points GA and GB. Then GA and GB are connected to the shell grids GAi and GBi, respectively.

4. For the formats PARTPAT and ELPAT, starting with the axis GA,GB, four pairs of auxiliary points GAHi, GBHi, i=1,2,3,4 must be found on shell A and B, respectively (see **Figure 8-167**). The auxiliary points form a hexahedral of length L and cross sectional area equivalent to a circle of diameter D (see **Figure 8-168**). Two subsequent sets of internal constraint equations are used to connect GA and GB to the shell grids GAi and GBi respectively. First, the points GA and GB are connected to the auxiliary points GAHi and GBHi, respectively. Second, the auxiliary points GAHi and GBHi are connected to existing shell grids GAi and GBi, respectively.
5. For the formats ELEMID and GRIDID, the shell grids GAi and GBi are user specified, either through shell element ids SHIDA and SHIDB or directly. The end points GA and GB are connected to the shell grids GAi and GBi regardless of the cross sectional area of the CWELD connector.
6. The parameters GSMOVE and NREDIA are used only for the formats ELPAT and PARTPAT. If GSMOVE > 0 and the number of pairs of GAHi and GBHi found is one, two, or three, then GS will be moved to a new location following the rules listed below.

    - If one pair of GAHi and GBHi is found, GS is moved to the middle position between that pair and the original GS.
    - If two pairs of GAHi and GBHi are found, GS is moved to the middle position between those two pairs and the original GS.
    - If three pairs of GAHi and GBHi are found, GS is moved closer to the second pair found. The move is one quarter of the distance from GS to the second pair.

    The search for pairs GAHi and GBHi resumes after GS is moved. This process will be repeated until either four pairs of GAHi and GBHi are found or the number of times GS is moved is equal to the value of GSMOVE. For the latter case, the search diameter D to find GAHi and GBHi is cut in half if NREDIA > 0 is specified. The iteration process repeats until a complete projection is found or the number of times the diameter D is reduced is equal to NREDIA.

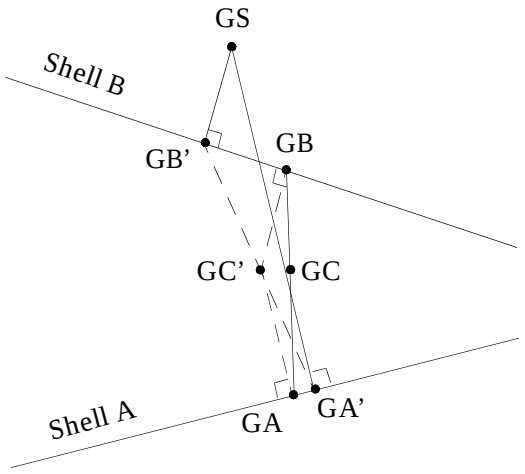


Figure 8-166 Projection of GS to Shells A and B

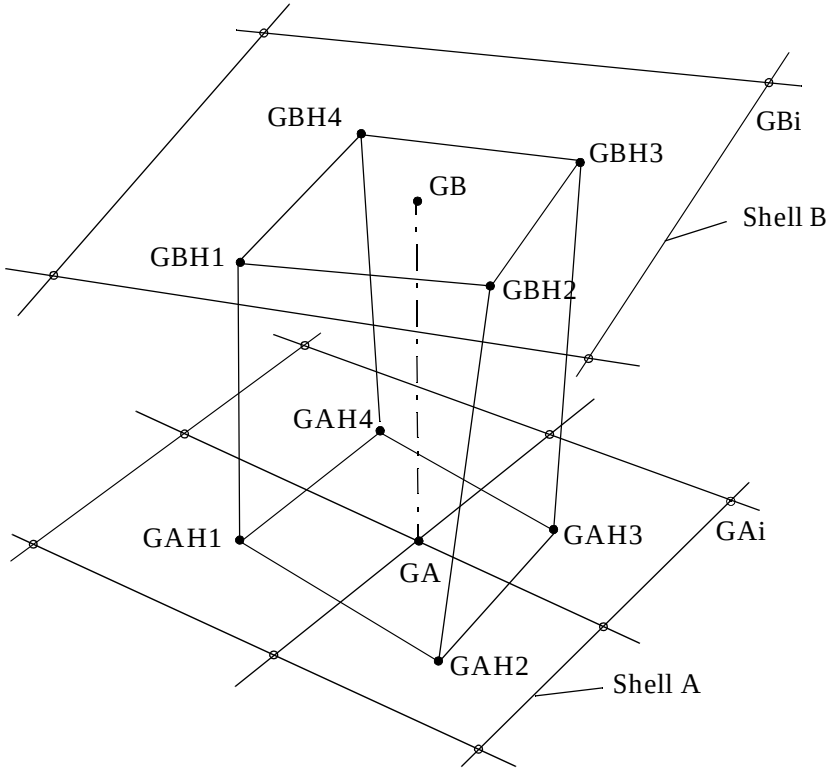


Figure 8-167 Eight Auxiliary Points Forming a Hexahedral for Formats ELPAT and PARTPAT

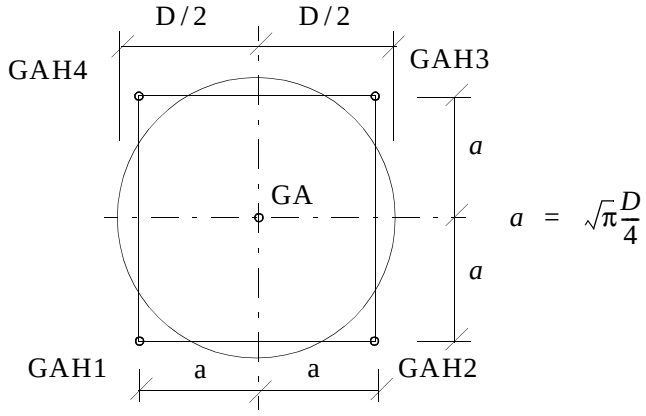


Figure 8-168 Equivalent Cross Sectional Area for Formats ELPAT and PARTPAT

TABDMP1 Modal Damping Table

Defines modal damping as a tabular function of natural frequency.

Format:

1	2	3	4	5	6	7	8	9	10
TABDMP1	TID	TYPE							
	f1	g1	f2	g2	f3	g3	-etc.-		

Example:

TABDMP1	2								
	2.5	.01057	2.6	.01362	ENDT				

Field	Contents
TID	Table identification number. (Integer > 0)
TYPE	Type of damping units. (Character: “G”, “CRIT”, or “Q”; Default is “G”)
fi	Natural frequency value in cycles per unit time. (Real $\geq$ 0.0)
gi	Damping value. (Real)

Remarks:

1. Modal damping tables must be selected with the Case Control command SDAMPING = TID.
2. The frequency values, fi, must be specified in either ascending or descending order, but not both.
3. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in **Figure 8-169** discontinuities are allowed only between points f2 through f7. Also, if g is evaluated at a discontinuity, then the average value of g is used. In **Figure 8-169**, the value of g at f = f3 is $g = (g3 + g4)/2$.
4. At least one continuation entry must be specified.
5. Any fi or gi entry may be ignored by placing “SKIP” in either of the two fields used for that entry.

6. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

7. The TABDMP1 uses the algorithm

$$g = g_T(f)$$

where f is input to the table and g is returned. The table look-up $g_T(f)$ is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points. See **Figure 8-169**. No warning messages are issued if table data is input incorrectly. See Remark **11.**

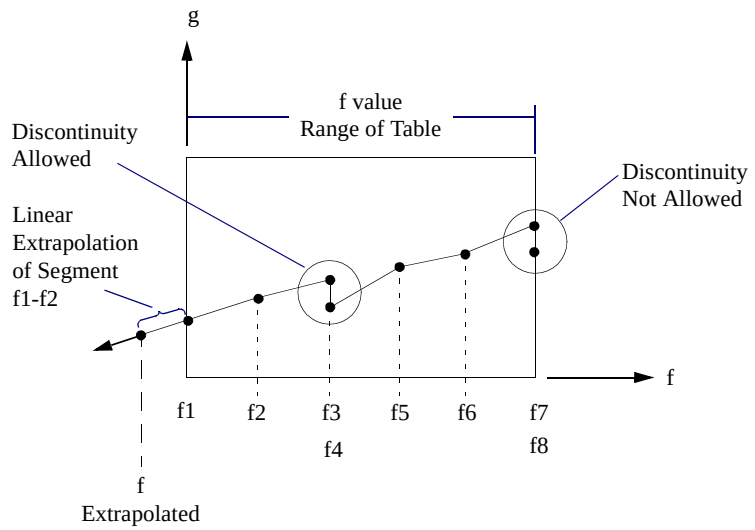


Figure 8-169 Example of Table Extrapolation and Discontinuity

8. This form of damping is used only in modal formulations of complex eigenvalue analysis, frequency response analysis, or transient response analysis. The type of damping used depends on the solution sequence (structural damping is displacement-dependent, and viscous damping is velocity-dependent). See “**Formulation of Dynamic Equations in SubDMAP GMA**” on page 429 of the *MSC.Nastran Reference Guide* for the equations used.

9. PARAM,KDAMP may be used in solution sequences that perform modal frequency and modal complex analysis, to select the type of damping.

KDAMP	Result
1 (Default)	*B* Matrix
-1	$(1 + ig)K$

See "**Formulation of Dynamic Equations in SubDMAP GMA**" on page 429 of the *MSC.Nastran Reference Guide* for a full explanation.

10. If TYPE is "G" or blank, the damping values gi, etc., are in units of equivalent viscous dampers, as follows:

$$b_i = \frac{\text{gi}}{\omega_i} K_i$$

(See "**Formulation of Dynamic Equations in SubDMAP GMA**" on page 429 of the *MSC.Nastran Reference Guide*) If TYPE is "CRIT", the damping values gi, etc., are in the units of fraction of critical damping C/C_0. If TYPE is "Q", the damping values gi are in the units of the amplification or quality factor, *Q*. These constants are related by the following equations:

$$C/C_0 = g/2$$

$$Q = \begin{cases} 1/(2C/C_0) \\ 1/g \end{cases}$$

11. A user warning message is used if either of the following conditions is satisfied:

    a. The modal damping value is computed as a result of extrapolation.

    b. The computed modal damping value is negative.

    For any modal damping value that satisfies condition a or b, the program lists the cyclic frequency and the corresponding modal damping value and indicates whether this value was computed as a result of interpolation or extrapolation. For the latter case, it also indicates whether the extrapolation was beyond the left end of the table or beyond the right end of the table.

    If a modal damping value satisfies ***both*** of the conditions, a and b above (that is, the modal damping value is computed as a result of extrapolation ***and*** it is negative), the program terminates the job with a user fatal message.

    The user can prevent the program from terminating the job as above by specifying MDAMPEXT=1 [or SYSTEM(426)=1] on the NASTRAN

statement. The user fatal message mentioned above does inform the user of this avoidance scheme.

TABLE3D Tabular Function with Three Variables

Specify a function of three variables for the GMBC, GMLOAD, and TEMPF entries only.

Format:

1	2	3	4	5	6	7	8	9	10
TABLE3D	TID	X0	Y0	Z0	F0				
	X1	Y1	Z1	F1	X2	Y2	Z2	F2	
	X3	Y3	Z3	F3	X4	Y4	Z4	F4	
	-etc.-	ENDT							

Example:

TABLE3D	128	0.	0.	1.					
	7.	8.	9.	100.	12.	14.	11.	200.	
	17.	18.	19.	1100.	112.	114.	111.	1200.	
	ENDT								

Field	Contents	Type	Default
TID	Table identification number.	Integer > 0	Required
X0,Y0,Z0	Offset of the independent variables.	Real	0.0
F0	Offset of the dependent variables.	Real	0.0
Xi,Yi,Zi	Independent variables.	Real	0.0
Fi	Dependent variable.	Real	0.0

Remarks:

1. At least two continuation entries must be specified.
2. The value of the function at (x,y,z) is calculated as

$$f = \frac{\sum_{i=1}^{4} \frac{Fi - F0}{d_i}}{\sum_{i=1}^{4} \frac{1}{d_i}}$$

where *f* are the function values at the four points with the lowest value of

$$d_i^2 = (x - X0 - Xi)^2 + (y - Y0 - Yi)^2 + (z - Z0 - Zi)^2$$

TABLED1 Dynamic Load Tabular Function, Form 1

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED1	TID	XAXIS	YAXIS						
	x1	y1	x2	y2	x3	y3	-etc.-	“ENDT”	

Example:

TABLED1	32								
	-3.0	6.9	2.0	5.6	3.0	5.6	ENDT		

Field	Contents
TID	Table identification number. (Integer > 0)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. See Remarks **6.** and **10.** (Character: “LINEAR” or “LOG”; Default = “LINEAR”)
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. See Remarks **6.** and **10.** (Character: “LINEAR” or “LOG”; Default = “LINEAR”)
xi, yi	Tabular values. (Real)
“ENDT”	Flag indicating the end of the table.

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in **Figure 8-170** discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In **Figure 8-170**, the value of y at $x = x3$ is $y = (y3 + y4)/2$. If the y-axis is a LOG axis then the jump at the discontinuity is evaluated as $y = \sqrt{y3y4}$.
3. At least one continuation must be specified.

4. Any xi-yi pair may be ignored by placing the character string “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of the character string “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
6. TABLED1 uses the algorithm

$$y = y_T(x)$$

where x is input to the table and y is returned. The table look-up is performed using interpolation within the table and extrapolation outside the table using the two starting or end points. See **Figure 8-170**. The algorithms used for interpolation or extrapolation are:

XAXIS	YAXIS	$y_T(x)$
LINEAR	LINEAR	$\frac{xj - x}{xj - xi} yi + \frac{x - xi}{xj - xi} yj$
LOG	LINEAR	$\frac{ln(xj/x)}{ln(xj/xi)} yi + \frac{ln(x/xi)}{ln(xj/xi)} yj$
LINEAR	LOG	$\exp\left[\frac{xj - x}{xj - xi} ln\ yi + \frac{x - xi}{xj - xi} ln\ yj\right]$
LOG	LOG	$\exp\left[\frac{ln(xj/x)}{ln(xj/xi)} ln\ yi + \frac{ln(x/xi)}{ln(xj/xi)} ln\ yj\right]$

where xj and yj follow xi and yi.

No warning messages are issued if table data is input incorrectly.

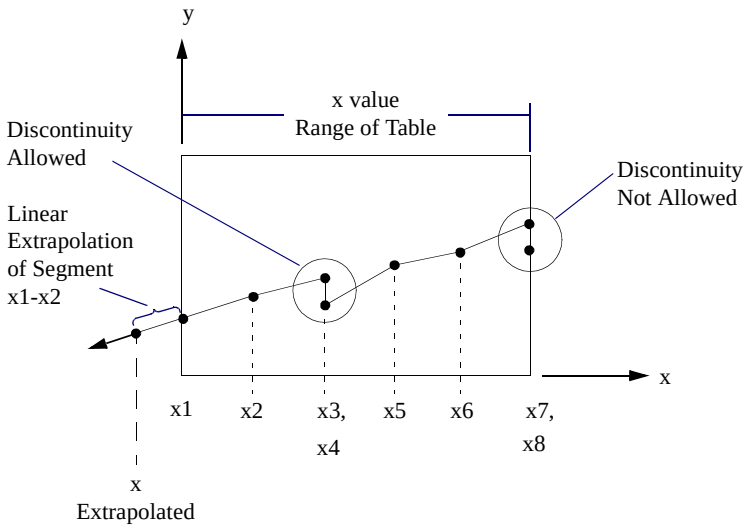


Figure 8-170 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads, xi is measured in cycles per unit time.
9. Tabular values on an axis if XAXIS or YAXIS = LOG must be positive. A fatal message will be issued if an axis has a tabular value ≤ 0.
10. LOG is not supported for SOLs 600 or 700. Fields 3 and 4 should be blank.

TABLED2 Dynamic Load Tabular Function, Form 2

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED2	TID	X1							
	x1	y1	x2	y2	x3	y3	-etc.-		

Example:

TABLED2	15	-10.5							
	1.0	-4.5	2.0	-4.2	2.0	2.8	7.0	6.5	
	SKIP	SKIP	9.0	6.5	ENDT				

Field	Contents
TID	Table identification number. (Integer > 0)
X1	Table parameter. See Remark 6. (Real)
xi, yi	Tabular values. (Real)

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in **Figure 8-171** discontinuities are allowed only between points $x2$ and $x7$. Also if y is evaluated at a discontinuity, then the average value of y is used. In **Figure 8-171**, the value of y at $x = x3$ is $y = (y3 + y4)/2$.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. TABLED2 uses the algorithm

$$y = y_T(x - X1)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See **Figure 8-171**. No warning messages are issued if table data is input incorrectly.

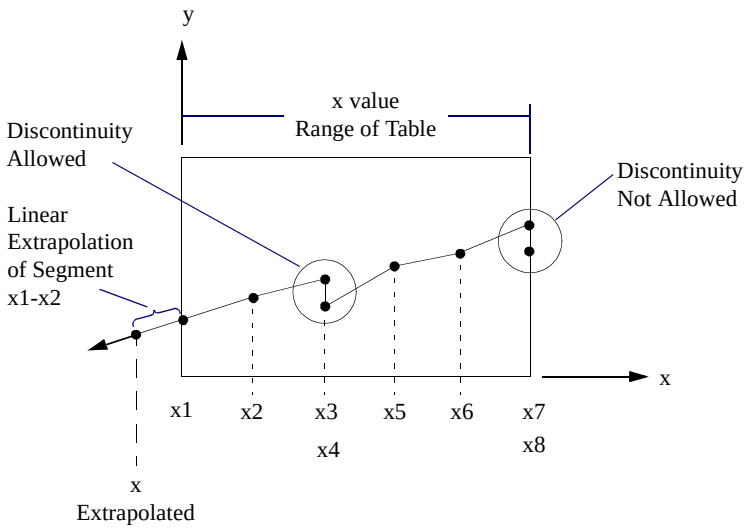


Figure 8-171 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads, $X1$ and xi are measured in cycles per unit time.

TABLED3 Dynamic Load Tabular Function, Form 3

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	-etc.-		

Example:

TABLED3	62	126.9	30.0						
	2.9	2.9	3.6	4.7	5.2	5.7	ENDT		

Field	Contents
TID	Table identification number. (Integer > 0)
X1, X2	Table parameters. (Real; X2 ≠ 0.0)
xi, yi	Tabular values. (Real)

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in **Figure 8-172** discontinuities are allowed only between points $x2$ and $x7$. Also if y is evaluated at a discontinuity, then the average value of y is used. In **Figure 8-172**, the value of y at $x = x3$ is $y = (y3 + y4)/2$.
3. At least one continuation entry must be present.
4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. TABLED3 uses the algorithm

$$y = y_T\left(\frac{x - X1}{X2}\right)$$

where *x* is input to the table and *y* is returned. The table look-up is performed using interpolation within the table and linear extrapolation outside the table using the two starting or end points. See **Figure 8-172**. No warning messages are issued if table data is input incorrectly.

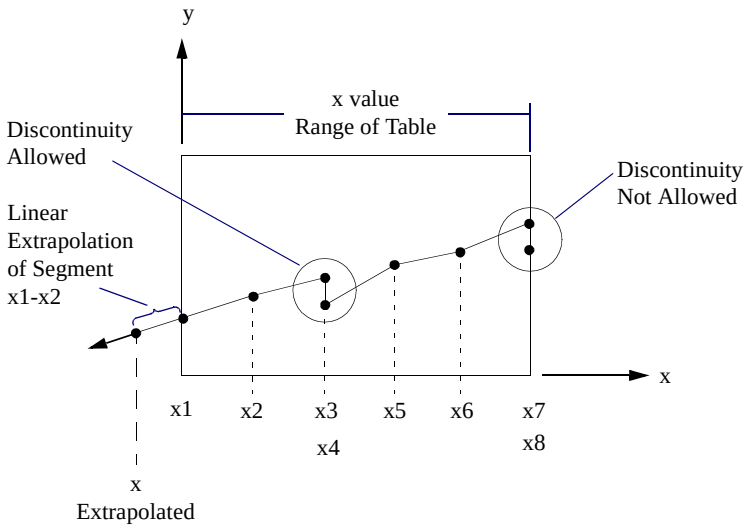


Figure 8-172 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads, X1, X2, and xi are measured in cycles per unit time.

TABLED4 Dynamic Load Tabular Function, Form 4

Defines the coefficients of a power series for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED4	TID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	A5	-etc.-		

Example:

TABLED4	28	0.0	1.0	0.0	100.				
	2.91	-0.0329	6.51-5	0.0	-3.4-7	ENDT			

Field	Contents
TID	Table identification number. (Integer > 0)
Xi	Table parameters. (Real; X2 ≠ 0.0; X3<X4)
Ai	Coefficients. (Real)

Remarks:

1. At least one continuation entry must be specified.
2. The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
3. TABLED4 uses the algorithm

$$y = \sum_{i=0}^{N} \mathrm{Ai}\left(\frac{x - \mathrm{X1}}{\mathrm{X2}}\right)^{i}$$

where x is input to the table, y is returned, and N is the number of pairs. Whenever x < X3, use X3 for x; whenever x > X4, use X4 for x. There are N + 1 entries in the table. There are no error returns from this table look-up procedure.

4. For frequency-dependent loads, xi is measured in cycles per unit time.

TABLEHT Heat Transfer Coefficient Table with Two Variables

Specifies a function of two variables for convection heat transfer coefficient.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEHT	TID								
	x1	TID1	x2	TID2	x3	-etc.			

Example:

TABLEHT	85								
	10.0	101	25.0	102	40.0	110	ENDT		

Field	Contents
TID	Table identification number. (Integer > 0)
xi	Independent variables. (Real)
TIDi	Table identification numbers of TABLEH1 entries. (Integer > 0)

Remarks:

1. xi must be listed in ascending order.
2. At least one continuation entry must be present.
3. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
4. This table is referenced only by PCONV entries that define free convection boundary condition properties.

TABLEDR (SOL 700)

Defines a table to reference other tables. A table contains a number of values that are related to TABLED1 entries. The values have to be in ascending order. For example, to define strain rate dependency where it is desired to provide a stress versus strain curve for each strain rate, n strain rates would be defined. Each TABLED1 may have unique spacing and an arbitrary number of points in their definition. (TABLED1's defined for the TABLEDR may be referenced elsewhere in the input.) *However, the curves must not cross except at the origin and the curves must share the same origin and end point.* This rather awkward input is done for efficiency reasons related to the desire to avoid indirect addressing in the inner loops used in the constitutive model stress evaluation.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEDR	TIB								
	VALUE1	TBID1							
	VALUEi	TBIDi							

Examples:

TABLEDR	101								
	1.0	102							
	1.5	110							

Field	Contents
TIB	Table ID. (Integer, No Default)
VALUEi	Scale factor applied to TBIODi. (Real, No Default)
TBIDi	ith Referenced table ID.

Remark:

1. If for example, 10 stress-strain curves for 10 different strain rates are given, 10 cards with the ascending values of strain rate then follow the first card. Elsewhere, 10 corresponding TABLED1 entries have to be present.

TABLEH1 Heat Transfer Coefficient Table, Form 1

Defines a tabular function referenced by TABLEHT for convection heat transfer coefficient.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEH1	TID								
	y1	f1	y2	f2	y3	-etc.=			

Example:

TABLEH1	123								
	50.0	5.23	75.0	3.76	110.0	0.97	ENDT		

Field	Contents
TID	Table identification number. (Integer > 0)
yi	Independent variables. (Real)
fi	Dependent variable. (Real)

Remarks:

1. yi must be listed in ascending order.
2. At least one continuation entry must be present.
3. Any yi-fi pair may be ignored by placing “SKIP” in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
5. TABLEH1 is used to input a curve in the form of

   $$f = f(y)$$

   where y is input to the table and f is returned. The table look-up is performed using linear interpolation within the table and is evaluated at the starting or end point outside the table. No warning messages are issued if table data is input incorrectly.
6. Discontinuities are not recommended and may lead to unstable results.

TABLEM1 Material Property Table, Form 1

Defines a tabular function for use in generating temperature-dependent material properties.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM1	TID								
	xI	y1	x2	y2	x3	y3	-etc.-	“ENDT”	

Example:

TABLEM1	32								
	-3.0	6.9	2.0	5.6	3.0	5.6	ENDT		

Field	Contents
TID	Table identification number. (Integer > 0)
xi, yi	Tabular values. (Real)
“ENDT”	Flag indicating the end of the table.

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in **Figure 8-173** discontinuities are allowed only between points $x2$ through $x7$. Also, if y is evaluated at a discontinuity, then the average value of y is used. In **Figure 8-173**, the value of y at $x = x3$ is $y = (y3 + y4)/2$.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
6. TABLEM1 uses the algorithm

$$y = y_T(x)$$

where *x* is input to the table and *y* is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See **Figure 8-173**. No warning messages are issued if table data is input incorrectly.

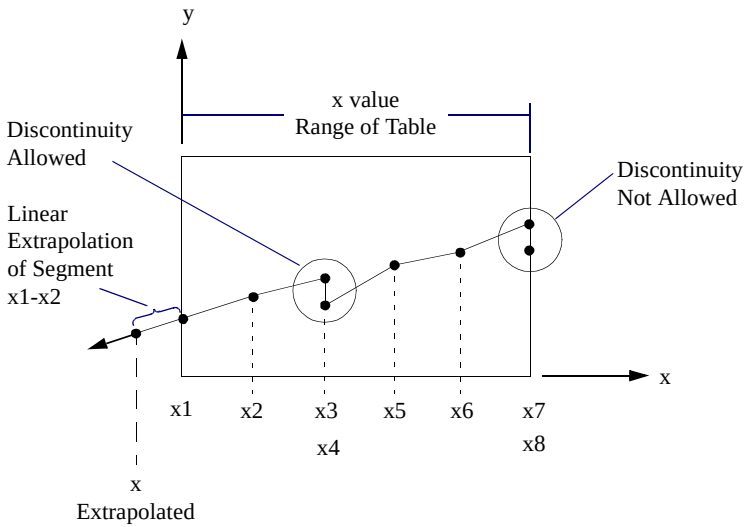


Figure 8-173 Example of Table Extrapolation and Discontinuity

TABLEM2 Material Property Table, Form 2

Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM2	TID	X1							
	x1	yI	x2	y2	x3	y3	-etc.-		

Example:

TABLEM2	15	-10.5							
	1.0	-4.5	2.0	-4.5	2.0	2.8	7.0	6.5	
	SKIP	SKIP	9.0	6.5	ENDT				

Field	Contents
TID	Table identification number. (Integer > 0)
X1	Table parameter. (Real)
xi, yi	Tabular values. (Real)

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in **Figure 8-174**, discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In **Figure 8-174**, the value of y at x = x3 is $y = (y3 + y4)/2$.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. TABLEM2 uses the algorithm

$$y = zy_T(x - \text{X1})$$

where x is input to the table, y is returned and z is supplied from the MATi entry. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See **Figure 8-174**. No warning messages are issued if table data is input incorrectly.

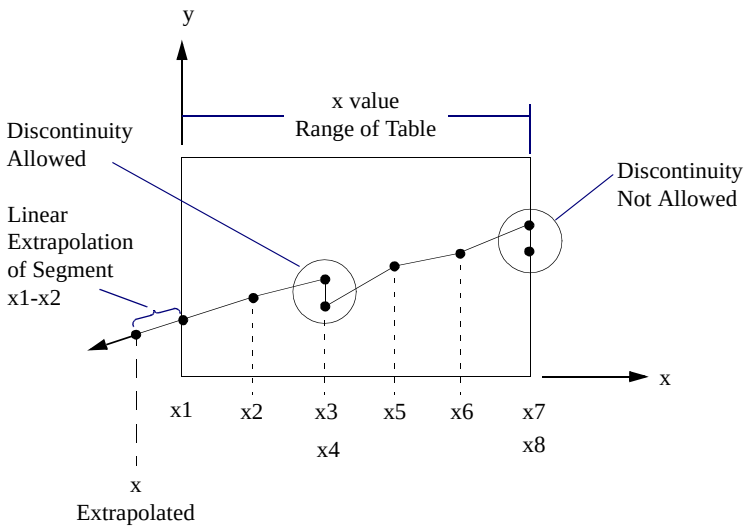


Figure 8-174 Example of Table Extrapolation and Discontinuity

TABLEM3 Material Property Table, Form 3

Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	-etc.-		

Example:

TABLEM3	62	126.9	30.0						
	2.9	2.9	3.6	4.7	5.2	5.7	ENDT		

Field	Contents
TID	Table identification number. (Integer > 0)
X1, X2	Table parameters. See Remark 6. (Real; X2 ≠ 0.0)
xi, yi	Tabular values. (Real)

Remarks:

1. Tabular values for xi must be specified in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in **Figure 8-175** discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In **Figure 8-175**, the value of y at x = x3 is $y = (y3 + y4) / 2$.
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. TABLEM3 uses the algorithm

$$y = zy_T\left(\frac{x - X1}{X2}\right)$$

where x is input to the table, y is returned and z is supplied from the MATi entry. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See **Figure 8-175**. No warning messages are issued if table data is input incorrectly.

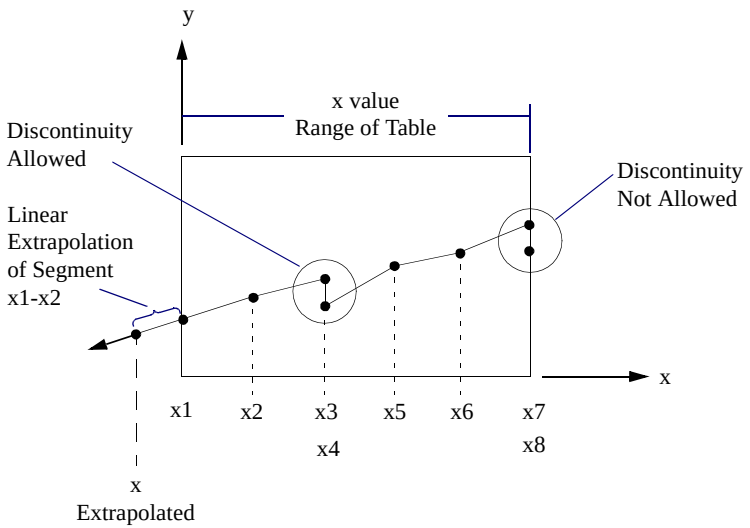


Figure 8-175 Example of Table Extrapolation and Discontinuity

TABLEM4 Material Property Table, Form 4

Defines coefficients of a power series for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM4	TID	X1	X2	X3	X4				
	A0	A1	A2	A3	A4	A5	-etc.-		

Example:

TABLEM4	28	0.0	1.0	0.0	100.				
	2.91	-0.0329	6.51-5	0.0	-3.4-7	ENDT			

Field	Contents
TID	Table identification number. (Integer > 0)
Xi	Table parameters. (Real; X2 $\neq$ 0.0; X3 < X4)
Ai	Coefficients. (Real)

Remarks:

1. At least one continuation entry must be specified.
2. The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
3. TABLEM4 uses the algorithm

$$y = z \sum_{i=0}^{N} \mathrm{Ai}\left(\frac{x - \mathrm{X1}}{\mathrm{X2}}\right)^{i}$$

where x is input to the table, y is returned and z is supplied from the MATi entry. Whenever x < X3, use X3 for x; whenever x > X4, use X4 for x. There are $N + 1$ entries in the table. There are no error returns from this table look-up procedure.

TABLES1 Material Property Table, Form 1

Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

Format:

1	2	3	4	5	6	7	8	9	10
TABLES1	TID								
	x1	y1	x2	y2	x3	y3	-etc.-	“ENDT”	

Example:

TABLES1	32								
	0.0	0.0	.01	10000.	.02	15000.	ENDT		

Field	Contents
TID	Table identification number. (Integer > 0)
xi, yi	Tabular values. (Real)
“ENDT”	Flag indicating the end of the table.

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in **Figure 8-176** discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In **Figure 8-176**, the value of y at x = x3 is $y = (y3 + y4)/2$.
3. At least one continuation entry must be present.
4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields used for that entry.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of table flag ENDT.

6. TABLES1 is used to input a curve in the form of

$$y = y_T(x)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See **Figure 8-176**. No warning messages are issued if table data is input incorrectly.

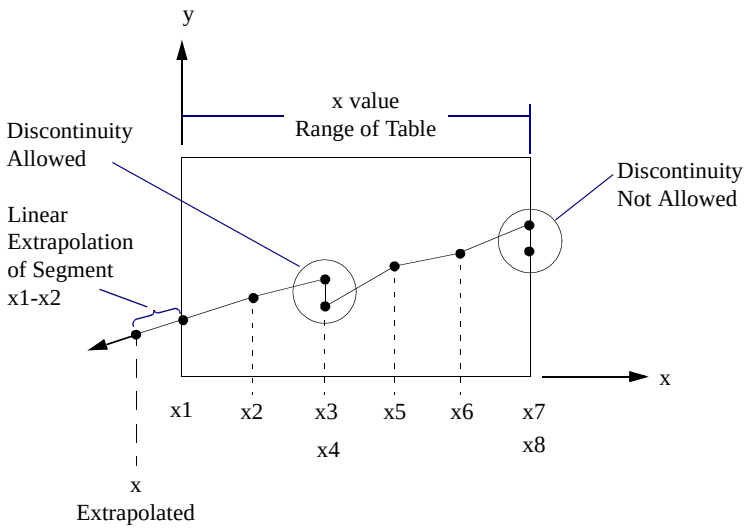


Figure 8-176 Example of Table Extrapolation and Discontinuity

7. Discontinuities are not recommended and may lead to unstable results. Discontinuities are not allowed in nonlinear solution sequences.

8. For SOL 600, general temperature-dependent stress vs. plastic strain curves may be entered using a combination of TABLEST and TABLES1 entries. Each TABLES1 entry is at a constant temperature. All entries must be in the form of stress vs. plastic strain using the stress and strain measures to be incomporated into the analysis. All sets of stress-strain values for a particular TABLES1 entry must be at the same temperature. One set is required for the lowest temperature in the model and another at or above the highest temperature in the model.

9. For SOL 600, the stress and strain values entered here depend on the stress and strain measures selected for the analysis. In addition, the strain is controlled using PARAM,MRTABLS1 which provides several methods of converting an engineering stress-strain curve to a stress vs. plastic strain curve (see MRTABLS1 in the Parameters Section)

TABLEST Material Property Temperature-Dependence Table

Specifies the material property tables for nonlinear elastic temperature-dependent materials.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEST	TID								
	T1	TID1	T2	TID2	T3	-etc.-			

Example:

TABLEST	101								
	150.0	10	175.0	20	ENDT				

Field	Contents
TID	Table identification number. (Integer > 0)
Ti	Temperature values. (Real)
TIDi	Table identification numbers of TABLES1 entries. (Integer > 0)

Remarks:

1. TIDi must be unique with respect to all TABLES1 and TABLEST table identification numbers.
2. Temperature values must be listed in ascending order.
3. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
4. This table is referenced only by MATS1 entries that define nonlinear elastic (TYPE = “NLELAST”) materials.
5. For SOL 600, this entry provides IDs of TABLES1 curves as a function of temperature for use with Marc’s AF_flowmat. The strains are plastic strain for all curves entered. The first curve must be entered at the lowest temperature encountered in the analysis run. Curves must be defined that equal or exceed the maximum temperature encountered in the run.

TABRND1 Power Spectral Density Table

Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

Format:

1	2	3	4	5	6	7	8	9	10
TABRND1	TID	XAXIS	YAXIS						
	f1	g1	f2	g2	f3	g3	-etc.-		

Example:

TABRND1	3								
	2.5	.01057	2.6	.01362	ENDT				

Field	Contents
TID	Table identification number. (Integer > 0)
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis. (Character: “LINEAR” or “LOG”; Default = “LINEAR”)
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis. (Character: “LINEAR” or “LOG”; Default = “LINEAR”)
fi	Frequency value in cycles per unit time. (Real $\geq$ 0.0)
gi	Power spectral density. (Real)

Remarks:

1. The fi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in **Figure 8-177** discontinuities are allowed only between points f2 through f7. Also, if g is evaluated at a discontinuity, then the average value of g is used. In **Figure 8-177**, the value of g at f = f3 is $g = (g3 + g4)/2$ If the y-axis is a LOG axis then the jump at the discontinuity is evaluated as $y = \sqrt{y3y4}$.
3. At least two entries must be present.
4. Any fi-gi pair may be ignored by placing “SKIP” in either of the two fields used for that entry.

5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.

6. TABRND1 uses the algorithm

$$g = g_T(f)$$

where f is input to the table and g is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See **Figure 8-177**. No warning messages are issued if table data is input incorrectly.

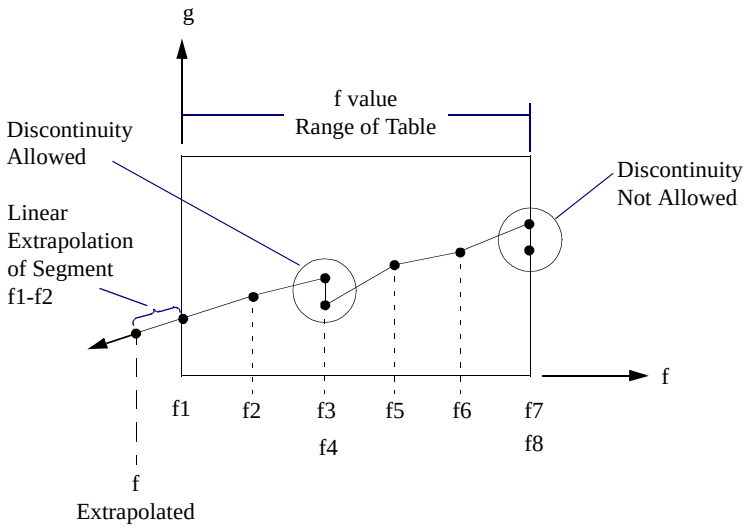


Figure 8-177 Example of Table Extrapolation and Discontinuity

7. For auto spectral density, the value of g returned must be greater than or equal to zero, as shown in Remark **6.**

8. Tabular values on an axis if XAXIS or YAXIS = LOG must be positive. A fatal message will be issued if an axis has a tabular value ≤ 0.

9. The algorithms used are:

XAXIS	YAXIS	f(x)
LINEAR	LINEAR	$\frac{f_{i+1}-f}{f_{i+1}-f_i}g_i + \frac{f-f_i}{f_{i+1}-f_i}g_{i+1}$
LOG	LINEAR	$\frac{\ln(f_{i+1}/f)}{\ln(f_{i+1}/f_i)}g_i + \frac{\ln(f/f_i)}{\ln(f_{i+1}/f_i)}g_{i+1}$
LINEAR	LOG	$\exp\left[\frac{f_{i+1}-f}{f_{i+1}-f_i}\ln g_i + \frac{f-f_i}{f_{i+1}-f_i}\ln g_{i+1}\right]$
LOG	LOG	$\exp\left[\frac{\ln(f_{i+1}/f)}{\ln(f_{i+1}/f_i)}\ln g_i + \frac{\ln(f/f_i)}{\ln(f_{i+1}/f_i)}\ln g_{i+1}\right]$

where $f_i < f < f_{i+1}$

TABRNDG Gust Power Spectral Density

Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TABRNDG	TID	TYPE	L/U	WG					

Example:

TABRNDG	1020	1	1.3	.1					

Field	Contents
TID	Table identification number. (Integer > 0)
TYPE	PSD type: von Karman (TYPE = 1) or Dryden model (TYPE = 2). (Integer = 1 or 2)
L/U	Scale of turbulence divided by velocity (units of time). See L/U in Remark 2. (Real)
WG	Root-mean-square gust velocity. (Real)

Remarks:

1. This entry must be referenced by a RANDPS entry.
2. The power spectral density is given by

$$S_q(\omega) = 2(\mathrm{WG})^2(\mathrm{L}/\mathrm{U})\frac{1 + 2(p+1)k^2(\mathrm{L}/\mathrm{U})^2\omega^2}{[1 + k^2(\mathrm{L}/\mathrm{U})^2\omega^2]^{p+3/2}}$$

   where

Type	p	k
1=von Karman	1/3	1.339
2=Dryden	1/2	1.0

   and $\omega = 2\pi f$. The units of $S_q(\omega)$ are velocity squared per frequency (f).
3. Other power spectral density functions may be defined using the TABRND1 entry.

TEMP Grid Point Temperature Field

Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Format:

1	2	3	4	5	6	7	8	9	10
TEMP	SID	G1	T1	G2	T2	G3	T3		

Example:

TEMP	3	94	316.2	49	219.8				

Field	Contents
SID	Temperature set identification number. (Integer > 0)
Gi	Grid point identification number. (Integer > 0)
Ti	Temperature. (Real)

Remarks:

1. In the static solution sequences, the temperature set ID(SID) is selected by the Case Control command TEMP. In the dynamic solution sequences, SID must be referenced in the TID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. From one to three grid point temperatures may be defined on a single entry.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. If the element material is temperature dependent, its properties are evaluated at the average temperature.
6. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data are defined. Gauss point temperatures are averaged for solid elements instead of grid point temperature.

7. For steady state heat transfer analysis, this entry together with the TEMPD entry supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT) = SID requests selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.
8. For transient heat transfer analysis, this entry together with the TEMPD entry supplies the initial condition temperatures. The Case Control command IC = SID requests selections of this entry. The temperature values specified here must be coincident with any temperature boundary condition specified.
9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See "**Buckling Analysis in SubDMAP MODERS**" on page 468 and "**Nonlinear Static Analysis**" on page 650 of the *MSC.Nastran Reference Manual.*

TEMPAX Conical Shell Temperature

Defines temperature sets for conical shell problems.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPAX	SID1	RID1	PHI1	T1	SID2	RID2	PHI2	T2	

Example:

TEMPAX	4	7	30.0	105.3					

Field	Contents
SIDi	Temperature set identification number. (Integer > 0)
RIDi	Ring identification number (see RINGAX entry). (Integer > 0)
PHIi	Azimuthal angle in degrees. (Real)
Ti	Temperature. (Real)

Remarks:

1. TEMPAX is allowed only if an AXIC entry is also present.
2. SIDi must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. Temperature sets must be selected with the Case Control command TEMP=SID.
4. One or two temperatures may be defined on each entry.
5. For a discussion of the conical shell problem, see “**Restart Procedures**” on page 398 of the *MSC.Nastran Reference Guide*.
6. TEMP(INIT) is not used with this entry.

TEMPB3 CBEAM3 Element Temperature Field

Defines a temperature field for the three-node beam element (CBEAM3 entry).

Format:

1	2	3	4	5	6	7	8	9	10
TEMPB3	SID	EID	T(A)	T(B)	T(C)	TPY(A)	TPZ(A)	TPY(B)	
	TPZ(B)	TPY(C)	TPZ(C)	TC(A)	TD(A)	TE(A)	TF(A)	TC(B)	
	TD(B)	TE(B)	TF(B)	TC(C)	TD(C)	TE(C)	TF(C)		
		Element	ID	List					

Example:

TEMPB3	101	23.0	45.9	10.0	0.0	1.3	23.9	3.8	
		2.5	68.0	91.0	45.0		48.0	80.0	
	20.0		33.9			45.6			
	9	10	THRU	30	41	51	67	78	
	THRU	110	BY	2					

Field	Contents
SID	Temperature set identification number. (Integer>0; Required)
EID	Element identification number. (Integer>0, Required)
T(*j*)	Temperature at *j* (*j*=A,B,C) on the neutral axis. (Real. Default=0.0)
TP*i*(*j*)	Effective linear gradient in local direction *i* (*i* = y, z) at *j* (*j* = A, B, C).(Real. Default=0.0)
T*i*(*j*)	Temperature at stress recovery point *i* (*i*=C, D, E, F) defined in PBEAM3 at location *j* (*j*=A, B, C). (Real. Default=0.0. See Remark 3.)
Element ID List	List of CBEAM3 element identification numbers. Character strings “THRU” and “BY” may be used in the list. (Integer>0, “THRU” or”BY”. At least one element ID is required.)

Remarks:

1. In the static solution sequences, the temperature set ID (SID) is selected by the Case Control command TEMP. In the dynamic solution sequences, SID must be referenced in the TIP field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.

2. SID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. If all T*i*(*j*) fields are blank, linear temperature gradients are assumed for stress recovery.
4. Temperature field defined by TEMPB3 entry always takes precedence over the grid point temperatures given by TEMP and TEMPD entries.
5. The effective thermal gradients are defined in the local coordinate system. For their definitions, see Remark 6 of Bulk Data entry TEMPRB for the details.

TEMPBC Grid Point Temperatures

Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPBC	SID	TYPE	TEMP1	GID1	TEMP2	GID2	TEMP3	GID3	

Example:

TEMPBC	10	STAT	100.0	1	100.0	2	100.0	3	

Alternate Format and Example:

TEMPBC	SID	TYPE	TEMP1	GID1	“THRU”	GID2	“BY”	INC	

TEMPBC	20	STAT	100.0	4	THRU	50	BY	2	

Field	Contents
SID	Temperature set identification number. (Integer > 0)
TYPE	Type of temperature boundary condition. See Remarks. (Character; Default = “STAT”): STAT - Constant temperature boundary condition TRAN - Time-varying temperature boundary condition
TEMPi	Temperature (Real)
GIDi	Grid point identification number. (Integer>0 or “THRU” or “BY”)
INC	Grid point number increment. (Integer)

Remarks:

1. For a constant Boundary Condition (TYPE = “STAT”), the temperature boundary load set (SID) is selected in the Case Control Section (SPC = SID). TYPE = “STAT” may be used in both steady-state (SOL 153) and transient analysis (SOL 159).
2. For transient analysis (SOL 159), a constant boundary condition should be specified using the SPC Bulk Data entry.

3. For a time-varying boundary condition (TYPE = “TRAN”), SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. TYPE=“TRAN” is permitted only in transient analysis (SOL 159). A function of time $F(t - \tau)$ defined on the TLOADi entry multiplies the general load. τ provides any required time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis.
4. In the alternate format, TEMP1 is the nodal temperature for the grid points GID1,GID1+INC,...,GID2. If “BY” and INC are not specified, then the grid point number increment is unity.
5. If TYPE = “STAT”, then no SPCi Bulk Data entries may be specified.
6. If TYPE = “TRAN”, then no CELAS2 or DAREA Bulk Data entries may be specified. Also, “U” must be specified in the CONV field on the TSTEPNL entry to obtain accurate results.
7. All TEMPBC entries in the Bulk Data Section must indicate either TYPE = “STAT” or TYPE = “TRAN” but not both.
8. In transient thermal analysis, the TEMPBC option is used to set a grid, known temperature as a function of time. Internally NASTRAN uses SLOAD and CELAS2 entries to enforce the temperature as a function of time. The u=P/K or temperature is equal to SLOAD divided by CELAS2. The default stiffness for the CELAS2 entry is 1.0E10. This value is fine most of the time. However, if the user desired to run the model using thermal conductivity in the following unit (Btu/sec.inch.F), then it may run into a numerically convergence issue. This is because the thermal conductivity for this unit has conductivity value in the 1.0E-6 range. The avoidance is to set a NASTRAN system cell, TBCMAG to 1.0E2.

TEMPD Grid Point Temperature Field Default

Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPD	SID1	T1	SID2	T2	SID3	T3	SID4	T4	

Example:

TEMPD	1	216.3							

Field	Contents
SIDi	Temperature set identification number. (Integer > 0)
Ti	Default temperature value. (Real)

Remarks:

1. For structural analysis in the static solution sequences, the temperature set ID (SID) is selected by the Case Control command TEMP. In the dynamic solution sequences, SID must be referenced in the TID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. SIDi must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. From one to four default temperatures may be defined on a single entry.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. If the element material is temperature dependent, its properties are evaluated at the average temperature.
6. Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data is defined.

7. For steady-state heat transfer analysis, this entry together with the TEMP entry supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT) = SID requests selection of this entry. The temperature values specified here must be coincident with any temperatures boundary conditions that are specified.
8. For transient heat transfer analysis, this entry together with the TEMP entry supplies the initial condition temperatures. The Case Control command IC=SID request selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.
9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See "**Buckling Analysis in SubDMAP MODERS**" on page 468 and "**Nonlinear Static Analysis**" on page 650 of the *MSC.Nastran Reference Manual.*
10. For partitioned Bulk Data superelements and auxiliary models, TEMPD must be specified in all partitioned Bulk Data Sections.

TEMPF p-Element Temperature Field with Function Definition

Defines the thermal loading to be applied to a group of elements.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPF	SID	EID1	FTEMP	FTABID					
	EID2	EID3	-etc.-						

Example:

TEMPF	127	12	111						

Alternate Format:

TEMPF	SID	EID1	FTEMP	FTABID					
	EID2	“THRU”	EIDn						

Field	Contents	Type	Default
SID	Temperature set identification number.	Integer > 0	Required
FTEMP	ID of a DEQATN entry describing the temperature field as a function of x,y,z. See Remark 1.	Integer > 0	
FTABID	ID of a TABLE3D entry describing the temperature field. See Remark 1.	Integer > 0	
EIDi	Identification numbers of the p-elements to which this thermal load is applied.	Integer > 0	Required

Remarks:

1. Either FTEMP or FTABID must be specified but not both.
2. The TEMPF entry overrides the temperature at the element vertices specified on the TEMP or TEMPD entries.

TEMPP1 Plate Element Temperature Field, Form 1

Defines a temperature field for plate, membrane, and combination elements (by an average temperature and a thermal gradient through the thickness) for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPP1	SID	EID1	TBAR	TPRIME	T1	T2			
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

Example:

TEMPP1	2	24	62.0	10.0	57.0	67.0			
	26	21	19	30					

Alternate Format and Example of Continuation Entry:

	EID2	“THRU”	EIDi	EIDj	“THRU”	EIDk			
	1	THRU	10	30	THRU	61			

Field	Contents
SID	Temperature set identification number. (Integer > 0)
EIDi, EIDj, EIDk	Unique element identification number(s). (Integer > 0 or the continuation entries may have “THRU” in fields 3 and/or 6, in which case EID2 < EIDi and EIDj < EIDk.)
TBAR	Temperature at the element’s reference plane as defined by ZOFFS on the connection entry. (Real)
TPRIME	Effective linear thermal gradient. Not used for membranes. (Real)
T1, T2	Temperatures for stress calculation at points defined on the element property entry. ($Z1$ and $Z2$ field on PSHELL entry.) T1 may be specified on the lower surface and T2 on the upper surface for the CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR elements. These data are not used for membrane elements. See Remark 9. If both T1 and T2 are blank, they are computed from the equation T = TBAR + z · TPRIME, where z is the distance from the center fiber. The program replaces T1 with a flag, and z is computed in a later operation. (Real)

Remarks:

1. In the static solution sequences, the temperature set ID (SID) is selected by the Case Control command TEMP. In the dynamic solution sequences, SID must be referenced in the TID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. If continuation entries are present, ElD1 and elements specified on the continuation entry are used. Elements must not be specified more than once.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. For temperature field other than a constant gradient, the "effective gradient" for a homogeneous plate is

$$\text{TPRIME} = \frac{1}{I}\int_z T(z)z\,dz$$

where I is the bending inertia and z is the distance from the neutral surface in the positive normal direction.

6. The "average" temperature for a homogeneous plate is

$$\text{TBAR} = \frac{1}{Volume}\int_{Volume} T\,dVolume$$

7. If the element material is temperature dependent, its properties are evaluated at the average temperature TBAR.
8. Large "THRU" ranges will lead to System Fatal Message 3008 ("Insufficient Core") and should be avoided, particularly for open sets.
9. If the element material is nonlinear then T1 and T2 should be left blank (see the MATS1 entry).
10. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See "**Buckling Analysis in SubDMAP MODERS**" on page 468 and "**Nonlinear Static Analysis**" on page 650 of the *MSC.Nastran Reference Manual*.

11. The bending and twisting moments can be reduced to outer fiber stresses and combined with membrane stresses in the composite plate elements. If, in addition, the temperature is specified by the user at a point where outer fiber stresses are calculated, the thermal expansion due to the difference between the specified temperature and the temperature that would be produced by a uniform gradient, T', is assumed to be completely restrained. Stated differently, the second and higher order moments of the thermal expansion are assumed to be completely restrained by elastic stiffness. The resulting stress increment is

$$\{\Delta\sigma\} = -[G_e]\{\alpha_e\}(T - T_o - T'z)$$

T_o: temperature at reference plane

where $[G_e]$ and $\{\alpha_e\}$ are evaluated for the average temperature of the element $\bar{T}$.

TEMPP3 Plate Element Temperature Field, Form 3

TEMPP3 is no longer available. Use TEMPP1.

TEMPRB One-Dimensional Element Temperature Field

Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPRB	SID	EID1	TA	TB	TP1A	TP1B	TP2A	TP2B	
	TCA	TDA	TEA	TFA	TCB	TDB	TEB	TFB	
	EID2	EID3	EID4	EID5	EID6	EID7	-etc.-		

Example:

TEMPRB	200	1	68.0	23.0	0.0	28.0		2.5	
	68.0	91.0	45.0		48.0	80.0	20.0		
	9	10							

Alternate Format and Example of Continuation Entry:

	EID2	“THRU”	EIDi	EIDj	“THRU”	EIDk			
	2	THRU	4	10	THRU	14			

Field	Contents
SID	Temperature set identification number. (Integer > 0)
EIDi, EIDj, EIDk	Unique element identification number(s). (Integer>0 or the second continuation entry may have “THRU” in fields 3 and / or 6 in which case EID2 < EIDi and EIDj < EIDk.)
TA, TB	Temperature at end A and end B on the neutral axis. (Real)
TPij	Effective linear gradient in direction i on end j; used with CBAR, CBEAM, and CBEND only. (Real)
Tij	Temperature at point i as defined on the PBAR, PBEAM, and PBEND entries at end j. This data is used for stress recovery only with CBAR, CBEAM, and CBEND exclusively. (Real)

Remarks:

1. In the static solution sequences, the temperature set ID (SID) is selected by the Case Control command TEMP. In the dynamic solution sequences, SID must be referenced in the TID field of an LSEQ entry, which in turn must be selected by the Case Control command LOADSET.
2. SID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. If at least one nonzero or nonblank Tij is present, the point temperatures given are used for stress recovery. If no Tij values are given, linear temperature gradients are assumed for stress recovery. The Tij values are not used in the calculation of differential stiffness.
4. If the second (and succeeding) continuation is present, EID1 and elements specified on the second (and succeeding) continuations are used. Elements must not be specified more than once.
5. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
6. The effective thermal gradients in the element coordinate system for the CBAR element are defined by the following integrals over the cross section. For end “A” (end “B” is similar),

$$\text{TA} = \frac{1}{A}\int TA(y, z)\,dA$$

$$\text{TP1A} = \frac{I_2}{\Delta}\int_A (y - y_n)TA(y, z)\,dA - \frac{I_{12}}{\Delta}\int_A (z - z_n)TA(y, z)\,dA$$

$$\text{TP2A} = \frac{I_1}{\Delta}\int_A (z - z_n)TA(y, z) - \frac{I_{12}}{\Delta}\int_A (y - y_n)TA(y, z)\,dA$$

$$\Delta = I_1 I_2 - I_{12}^2$$

if $I_{12} = 0$

$$\text{TP1A} = \frac{1}{I_1}\int_A (y - y_n)TA(y, z)\,dA$$

$$\text{TP2A} = \frac{1}{I_2}\int_A (z - z_n)TA(y, z)\,dA$$

where $TA(y, z)$ is the temperature at point y,z (in the element coordinate system) at end "A" of the bar. See the CBAR entry description for the element coordinate system: $I1, I_2$ and I_{12} are the moments of inertia about the z and y axes, respectively. The temperatures are assumed to vary linearly along the length (x-axis). Note that if the temperature varies linearly over the cross section, then TP1A, TP1B, TP2A and TP2B are the actual gradients.

7. If the element material is temperature-dependent, the material properties are evaluated at the average temperature $(TA + TB)/2$.
8. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See "**Buckling Analysis in SubDMAP MODERS**" on page 468 and "**Nonlinear Static Analysis**" on page 650 of the *MSC.Nastran Reference Manual*.
9. If any T_y is specified the stresses computed by the effective gradient are corrected by $\Delta\sigma$ such that:

$$\sigma = \sigma\Big|_{T_A + y^{TPIA} + z^{TP2A}} + \Delta\sigma$$

where $\Delta\sigma$ is in the form

$$\Delta\sigma = -\alpha E[T_{CA} - T_o - C_1 \cdot TPIA - C_2 \cdot TP2A] \text{ etc}$$

for CBAR and CBEAM

$$\Delta\sigma = -\alpha E[T_{CA} - T_o - (C_1 + \Delta N) \cdot TPIA - C_2 \cdot TP2A] \text{ etc}$$

for CBEND.

TF Dynamic Transfer Function

Defines a dynamic transfer function of the form

$$(\mathrm{B0} + \mathrm{B1} \cdot p + \mathrm{B2} \cdot p^2)u_d + \sum_i (\mathrm{A0}(i) + \mathrm{A1}(i)p + \mathrm{A2}(i)p^2)u_i = 0 \qquad \text{Eq. 8-8}$$

Can also be used as a means of direct matrix input. See Remark 4.

Format:

1	2	3	4	5	6	7	8	9	10
TF	SID	GD	CD	B0	B1	B2			
	G(1)	C(1)	A0(1)	A1(1)	A2(1)	-etc.-			

Example:

TF	1	2	3	4.0	5.0	6.0			
	3	4	5.0	6.0	7.0				

Field	Contents
SID	Set identification number. (Integer > 0)
GD, G(i)	Grid, scalar, or extra point identification numbers. (Integer > 0)
CD, C(i)	Component numbers. (Integer zero or blank for scalar or extra points, any one of the Integers 1 through 6 for a grid point.)
B0, B1, B2 A0(i), A1(i), A2(i)	Transfer function coefficients. (Real)

Remarks:

1. Transfer function sets must be selected with the Case Control command TFL = SID.
2. Continuation entries are optional.
3. The matrix elements defined by this entry are added to the dynamic matrices for the problem.

4. The constraint relation given in **Eq. 8-8** will hold only if no structural elements or other matrix elements are connected to the dependent coordinate u_d. In fact, the terms on the left side of **Eq. 8-8** are simply added to the terms from all other sources in the row for u_d.
5. See the ***MSC.Nastran Dynamics Users Guide*** for a discussion of transfer functions.
6. For each SID, only one logical entry is allowed for each GD, CD combination.
7. For heat transfer analysis, the initial conditions must satisfy **Eq. 8-8**.

TIC Transient Analysis Initial Condition

Defines values for the initial conditions of variables used in structural transient analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom. This entry may not be used for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TIC	SID	G	C	U0	V0				

Example:

TIC	100	10	3	0.1	0.5				

Field	Contents
SID	Set identification number. (Integer > 0)
G	Grid, scalar, or extra point or modal coordinate identification number. (Integer > 0). See Remark **4.**
C	Component numbers. (Any one of the integers 1 through 6 for grid points, integer zero or blank for scalar or extra points and -1 for modal coordinates.)
U0	Initial displacement. (Real)
V0	Initial velocity. (Real)

Remarks:

1. Transient analysis initial condition sets must be selected with the IC Case Control command. Note the use of IC in the Case Control command versus TIC on the Bulk Data entry. For heat transfer, the IC Case Control command selects TEMP or TEMPD entries for initial conditions and not the TIC entry.
2. If no TIC set is selected in the Case Control Section, all initial conditions are assumed to be zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed to be zero.

4. In direct transient analysis (SOL 109 and 129) as well as in modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC or IC(PHYSICAL) Case Control command, G may reference only grid, scalar or extra points. In modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC(MODAL) Case Control command, G may reference only modal coordinates or extra points.
5. The initial conditions for the independent degrees-of-freedom specified by this Bulk Data entry are distinct from, and may be used in conjunction with, the initial conditions for the enforced degrees-of-freedom specified by TLOAD1 and / or TLOAD2 Bulk Data entries.

TICD (SOL 700) Transient Analysis Initial Conditions with Increment Options

Defines values for the initial conditions of variables used in structural transient analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom. This entry may not be used for heat transfer analysis. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
TIC	SID	G	C	U0	V0	G2	INC		

Example:

TIC	100	10	3	0.1	0.5				

Field	Contents
SID	Set identification number. (Integer > 0)
G	Grid, scalar, or extra point or modal coordinate identification number. See Remark **4.** (Integer > 0)
C	Component numbers. (Any one of the integers 1 through 6 for grid points, integer zero or blank for scalar or extra points and -1 for modal coordinates.)
U0	Initial displacement. (Real)
V0	Initial velocity. (Real)
G2	A second grid. If entered, all grids from G to G2 by INC will have initial conditions U0 and V0. (Integer $\geq$ 0 or blank; Default is blank)
INC	See description for G2. (Integer $\geq$ 0 or blank; Default is blank)

TIC3 (SOL 700) Transient Analysis Initial Velocity with Increment Options

Allows for the definition of a velocity field of grid points consisting of a rotation and a translation specification. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
TIC3	SID	G		SCALE					+
TIC3	7	5		10.					+

Alternate Format:

+	XVEL	YVEL	ZVEL	XROT	YROT	ZROT			+
+	100.0			5.0		-7.5			+

Example:

+	G1	G2	THRU	G3	BY	G4	-etc.-		
+	1	2	THRU	1000	BY	23			

Field	Contents
SID	Number of a set of loads. (Integer > 0; Required)
G	Number of a grid point at the center of rotation. (Integer > 0; Required)
XVEL, YVEL, ZVEL	Initial translational velocity components. (Real; Default = 0.0)
XROT, YROT, ZROT	Initial rotational velocity components. (Real; Default = 0.0)
G1, G2, ...	Grid points to be initialized. THRU indicates a range of grid points. BY is the increment to be used within this range. (Integer > 0; Required)

Remarks:

1. Any number of TIC3 entries can be used.
2. The rotational velocity components are defined in radians per unit time.
3. For six degree of freedom grid points, the angular velocity components are also initialized.

4. Initial conditions for grid points that are not specified on TICn or TICGP entries are assumed to be zero.
5. If the THRU specification is used, the grid points in the range definition are not required to exist. Grid points that do not exist are ignored. The first grid point in the THRU specification must be a valid grid point. The BY option enables grid points to be ignored in this range.
6. None of the fields in the list of grid points can be blank or zero, since this indicates the end of the list.
7. The initial conditions to be used in SOL 700 must be selected in the Case Control Section (TIC = SID).

TLOAD1 Transient Response Dynamic Excitation, Form 1

Defines a time-dependent dynamic load or enforced motion of the form

$$\{P(t)\} = \{A\} \cdot F(t - \tau)$$

for use in transient response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	EXCITEID	DELAY	TYPE	TID	US0	VS0		

Example:

TLOAD1	5	7		LOAD	13				

Field	Contents
SID	Set identification number. (Integer > 0)
EXCITEID	Identification number of DAREA or SPCD entry set or a thermal load set (in heat transfer analysis) that defines $\{A\}$. See Remarks **2.** and **3.** (Integer > 0)
DELAY	Defines time delay τ. If it is a non-zero integer, it represents the identification number of DELAY Bulk Data entry that defines τ. If it is real, then it directly defines the value of τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See also Remark **9.** (Integer $\geq$ 0, real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks **2.** and **3.** (Integer, character or blank; Default = 0)
TID	Identification number of TABLEDi entry that gives $F(t)$. (Integer > 0)
US0	Factor for initial displacements of the enforced degrees-of-freedom. See Remarks **10.** and **12.** (Real; Default = 0.0)
VS0	Factor for initial velocities of the enforced degrees-of-freedom. See Remarks **11.** and **12.** (Real; Default = 0.0)

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.

2. The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS, or DISP	Enforced displacement using large mass or SPC / SPCD data
2, V, VE, VEL or VELO	Enforced velocity using large mass or SPC / SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration using large mass or SPC / SPCD data

3. TYPE (field 5) also determines the manner in which EXCITEID (field 3) is used by the program as described below

   Excitation specified by TYPE is applied load

   - *There is no LOADSET request in Case Control*

     EXCITEID may also reference DAREA, static, and thermal load set entries

   - *There is a LOADSET request in Case Control*

     The program may also reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID.

   Excitation specified by TYPE is enforced motion

   - *There is no LOADSET request in Case Control*

     EXCITEID will reference SPCD entries. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference DAREA and static and thermal load set entries just as in the case of applied load excitation.

- *There is a LOADSET request in Case Control*

  The program will reference SPCD entries specified by the LID field in the selected LSEQ entries corresponding to the EXCITEID. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries, just as in the case of applied load excitation.

4. EXCITEID may reference sets containing QHBDY, QBDYi, QVECT, QVOL and TEMPBC entries when using the heat transfer option.
5. TLOAD1 loads may be combined with TLOAD2 loads only by specification on a DLOAD entry. That is, the SID on a TLOAD1 entry may not be the same as that on a TLOAD2 entry.
6. SID must be unique for all TLOAD1, TLOAD2, RLOAD1, RLOAD2, and ACSRCE entries.
7. If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore A may be a function of time.
8. If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and then combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. Please refer to "**Fourier Transform**" on page 206 of the *MSC.Nastran 2006 Dynamics Users Guide.*
9. If DELAY is blank or zero, τ will be zero.
10. The USO field is used only when the dynamic excitation defined by the TYPE field is enforced velocity or enforced acceleration using SPC/SPCD specification. The initial displacements for the enforced degrees-of-freedom in this case are given by the product $\{A\}(\mathrm{US0})$ where $\{A\}$ is defined by the EXCITEID field.
11. The VS0 field is used only when the dynamic excitation defined by the TYPE field is enforced acceleration using SPC/SPCD specification. The initial velocities for the enforced degrees-of-freedom in this case are given by the product $\{A\}(\mathrm{VS0})$ where $\{A\}$ is defined by the EXCITEID field.

12. The initial conditions for the enforced degrees-of-freedom implied by the US0 and VS0 fields are distinct from, and may be used in conjunction with, the initial conditions for the independent degrees-of-freedom specified by a TIC Bulk Data entry (which, in turn, is selected by an IC Case Control command).

TLOAD2 Transient Response Dynamic Excitation, Form 2

Defines a time-dependent dynamic excitation or enforced motion of the form

$$\{P(t)\} = \begin{cases} 0 & , \quad t < (\mathrm{T1} + \tau) \text{ or } t > (\mathrm{T2} + \tau) \\ \{A\}\tilde{t}^{B} e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) & , \quad (\mathrm{T1} + \tau) \le t \le (\mathrm{T2} + \tau) \end{cases}$$

for use in a transient response problem, where $\tilde{t} = t - \mathrm{T1} - \tau$

Format:

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	EXCITEID	DELAY	TYPE	T1	T2	F	P	
	C	B	US0	VS0					

Example:

TLOAD2	4	10			2.1	4.7	12.0		
	2.0								

Field	Contents
SID	Set identification number. (Integer > 0)
EXCITEID	Identification number of DAREA or SPCD entry set or a thermal load set (in heat transfer analysis) that defines $\{A\}$. See Remarks **2.** and **3.** (Integer > 0)
DELAY	Defines time delay τ. (Integer $\ge$ 0, real or blank). If it is a non-zero integer, it represents the identification number of DELAY Bulk Data entry that defines τ. If it is real, then it directly defines the value of τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See also Remark **5.**
TYPE	Defines the type of the dynamic excitation. See Remarks **2.** and **3.** (Integer, character or blank; Default = 0)
T1	Time constant. (Real $\ge$ 0.0)
T2	Time constant. (Real; T2 > T1)
F	Frequency in cycles per unit time. (Real $\ge$ 0.0; Default = 0.0)
P	Phase angle in degrees. (Real; Default = 0.0)

Field	Contents
C	Exponential coefficient. (Real; Default = 0.0)
B	Growth coefficient. (Real; Default = 0.0)
US0	Factor for initial displacements of the enforced degrees-of-freedom. See Remarks **10.** and **12.** (Real; Default = 0.0)
VSO	Factor for intial velocities of the enforced degrees-of-freedom. See Remarks **11.** and **12.** (Real; Default = 0.0)

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command with DLOAD=SID.
2. The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS, or DISP	Enforced displacement using large mass or SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using large mass or SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration using large mass or SPC/SPCD data

3. TYPE (field 5) also determines the manner in which EXCITEID (field 3) is used by the program as described below

   Excitation specified by TYPE is applied load

   - *There is no LOADSET request in Case Control*

     EXCITEID may also reference DAREA, static and thermal load set entries

   - *There is a LOADSET request in Case Control*

     The program may also reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID.

   Excitation specified by TYPE is enforced motion

   - *There is no LOADSET request in Case Control*

EXCITEID will reference SPCD entries. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference DAREA and static and thermal load set entries just as in the case of applied load excitation.

- *There is a LOADSET request in Case Control*

  The program will reference SPCD entries specified by the LID field in the selected LSEQ entries corresponding to the EXCITEID. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID, just as in the case of applied load excitation.

4. EXCITEID (field 3) may reference sets containing QHBDY, QBDYi, QVECT, and QVOL and TEMPBC entries when using the heat transfer option.
5. If DELAY is blank or zero, τ will be zero.
6. TLOAD1 loads may be combined with TLOAD2 loads only by specification on a DLOAD entry. That is, the SID on a TLOAD1 entry may not be the same as that on a TLOAD2 entry.
7. SID must be unique for all TLOAD1, TLOAD2, RLOAD1, RLOAD2, and ACSRCE entries.
8. If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore A may be a function of time.
9. If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and them combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. In this case, B will be rounded to the nearest integer. Please refer to “**Fourier Transform**” on page 206 of the *MSC.Nastran 2006 Dynamics Users Guide.*
10. The USO field is used only when the dynamic excitation defined by the TYPE field is enforced velocity or enforced acceleration using SPC/SPCD specification. The initial displacements for the enforced degrees-of-freedom in this case are given by the product $\{A\}(\text{US0})$ where $\{A\}$ is defined by the EXCITEID field.

11. The VS0 field is used only when the dynamic excitation defined by the TYPE field is enforced acceleration using SPC/SPCD specification. The initial velocities for the enforced degrees-of-freedom in this case are given by the product $\{A\}(\text{VS0})$ where $\{A\}$ is defined by the EXCITEID field.
12. The initial conditions for the enforced degrees-of-freedom implied by the US0 and VS0 fields are distinct from, and may be used in conjunction with, the initial conditions for the independent degrees-of-freedom specified by a TIC Bulk Data entry (which, in turn, is selected by an IC Case Control command).
13. The continuation entry is optional.

TMPSET Temperature Group Set Definition

Define a time-dependent dynamic thermal load group for use in TTEMP Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
TMPSET	ID	G1	G2	G3	G4	G5	G6	G7	

Alternate Format:

TMPSET	ID	G1	“THRU”	G2	“BY”	INC			

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	G8	G9	G10	G11	-etc.-				

Continuation Entry Format 2:

	G8	“THRU”	G9	“BY”	INC				

Example:

TMPSET	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Field	Contents
ID	Temperature group identification number. (Integer >0)
Gi	Grid point Identification numbers in the group (Integer >0)

Remarks:

1. This entry is used in SOL 400 only when ANALYSIS=NLTRAN (nonlinear transient analysis) and the temperature load is applied. It only applies to the nonlinear elements in the Residual (SEID=0).

2. GROUP_ID determines the group of a specified the time-dependent distribution of temperatures. It is used by the TTEMP Bulk Data entry to define the corresponding TABLEDi entry. GROUP_ID must be unique for all of the other TMPSET entries.
3. The temperature of grid point Gi must be defined using TEMP, TEMPD, TEMPP1, or TEMPRB Bulk Data entry. These bulk data entries must have the same SID as that referenced on the associated TTEMP Bulk Data entry.

TODYNA (SOL 700) Defines the Start of Direct Text to Dytran-lsdyna.

All entries between TODYNA and ENDDYNA will be passed directly to MD Nastran to Dytran-lsdyna. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
TODYNA	KIND								

Example:

TODYNA	0								
MAT1	345	29.0E6		0.285	0.0004				
ENDDYN									

Field	Contents
KIND	Type of text input. Iinteger; default=0)
	0 Bulk Data only (see remark 1.)
	1 Case Control commands and Bulk Data entries (see remark 2.).
	2 Executive Control statements, Case Control commands, and Bulk Data entries (see remark 3.) .

Remarks:

1. If KIND=0, all entries between TODYNA and ENDDYNA will be passed directly without any checking to the Bulk Data portion of Dytran-lsdyna. These entries are placed immediately after BEGIN BULK in the Dytran-lsdyna input file. Additional entries (which are not between TODYNA and ENDDYNA) are placed after these direct text entries. The headers TODYNA and ENDDYNA are not passed to Dytran-lsdyna.
2. If KIND=1, all entries between TODYNA and ENDDYNA will be passed directly without any checking to the Case Control and Bulk Data portions of Dytran-lsdyna. These entries are placed immediately after CEND in the Dytran-lsdyna input file and must be arranged so that all Case Control commands come before _BEGIN_BULK (see remark 5.) and all Bulk Data entries come after it. A _BEGIN_BULK entry (see remark 5.) must be included. Additional entries (which are not between TODYNA and

ENDDYNA) are placed after these the direct text entries in the Bulk Data. Any Case Control commands normally translated from the Nastran file will be skipped. The headers TODYNA and ENDDYNA are not passed to Dytran-lsdyna.

3. If KIND=2, all entries between TODYNA and ENDDYNA will be passed directly without any checking to the Executive Control, Case Control and Bulk Data portions of Dytran-lsdyna. These entries will form the entire Dytran-lsdyna input file except that other Bulk Data entries in the Nastran file (which are not between TODYNA and ENDDYNA) are placed after these the direct text entries in the Bulk Data. The user must place all direct text Executive Control statements first followed by _CEND_, followed by all Case Control commands, followed by _BEGIN_BULK_, followed by whatever direct text Bulk Data entries are desired. MD Nastran Executive Control statements and Case Control commands normally translated will be ignored using this option.
4. For KIND=1 or KIND=2, Case Control commands must start in Column 1. If spaces are desired, they must be replaced by the underscore (_) character, for example TIMES(D3PLOT)=0.0 THRU END BY 1.00e-03 should be entered as

   TIMES(D3PLOT)=0.0_THRU_END_BY_1.00e-03

   The underscore character will be removed when these entries are transferred to Dytran-lsdyna. If this is not done, the standard Nastran Case Control routines will change these entries in unanticipated ways. An underscore in column 1 will be ignored rather than treated as a blank.
5. For KIND=1 and KIND=2 certain entries must have underscores before and at the end of the string to keep standard Nastran Case Control routines from deleting the strings. Those identified to date include the following:

```
_ECHO=NONE_
_ENDTIME=1.00E-02_  (or some other time)
_BEGIN_BULK_
```

   In the above, all strings begin with an underscore in column 1. These strings will appear in the Dytran-lsdyna file as shown below:

```
ECHO=NONE
ENDTIME=1.00E-02
BEGIN BULK
```

   Fatal messages may be produced by some of theses entries, however the analysis will usually proceed to completion.

6. Due to the items described in remarks **4.** and **5.**, use of KIND=2 or KIND=3 is not recommended.

TOPVAR Topological Design Variable

Define a topology design region for topology optimization.

Format:

1	2	3	4	5	6	7	8	9	10
TOPVAR	ID	LABEL	PTYPE	XINIT	XLB	DELXV	POWER	PID	
	“SYM”	CID	MSi	MSi	MSi				
	“CAST”	CID	DDi	DIE					
	“EXT”	CID	EDi						
	“TDMIN”	TV							

Example:

1	2	3	4	5	6	7	8	9	10
TOPVAR	2	PS1	PSOLID	0.3				10	
	SYM	5	XY	ZX					
	CAST	5	X	2					
	TDMIN	0.6							

Field	Contents
ID	Unique topology design region identification number. (Integer > 0)
LABEL	User-supplied name for printing purpose. (Character)
PTYPE	Property entry name. Used with PID to identify the elements to be designed. (Character: “PBAR”, “PSHELL”, ‘PSOLID”, etc.)
XINIT	Initial value. (Real, XLB $\leq$ XINIT). Typically, XINIT is defined to match the mass target constraint, so the initial design does not have violated constraints. For example, if the mass target is 30% on DRESP1=FRMASS, then it is recommended XINIT=0.3.
XLB	Lower bound to prevent the singularity of the stiffness matrix. (Real; Default = 0.001)
DELXV	Fractional change allowed for the design variable during approximate optimization. (Real > 0.0; Default = 0.2. See Remark 3.)

Field	Contents
POWER	A penalty factor used in the relation between topology design variables and element Young's modulus. (Real > 1.0; Default = 3.0). 2.0 ≤ POWER ≤ 5.0 is recommended.
PID	Property entry identifier (Integer > 0)
"SYM"	Indicates that this line defines symmetry constraints.
CID	Rectangular coordinate system ID used for specifying manufacturing constraints. See Remark **4.** (Blank or Integer > 0; Default = 0 = the basic coordinate system)
MSi	Mirror symmetry plane. See Remark **5.** & **7.** (Character, 'XY', 'YZ', or 'ZX')
"CAST"	Indicates that this line defines casting constraints (i.e., die draw direction constraints). See Remarks **6.**, **7.**, and **13.**
DDi	Draw Direction. DDi=X, Y, Z or X-, Y-, Z- for a single die option (DIE=1) where X-, Y-, Z- indicates the opposite direction of X, Y, and Z respectively. DDi=X, Y, and Z for two die option (DIE =2) (Character)
DIE	Die Options. (Blank or integer 1 or 2; Default = 1) = 1 (or blank). A single die will be used and the die slides in the given draw direction (i.e., material grows from the bottom in the draw direction) = 2. Two dies will be used and the dies split apart along the draw direction (i.e., material grows from the splitting plane in opposite direction along the axis specified by the draw direction DDi. The splitting plane is determined by optimization)
"EXT"	Indicates that this line defines extrusion constraints (i.e., enforce constant cross-section) See Remark **6.** and **7.**
EDi	Extrusion direction. (Character, X, Y, or Z)
"TDMIN"	Indicates that this line defines a minimum member size, See Remarks **9.** and **10.**
TV	Minimum member size. See Remark **9.** (Real ≥ 0.0)

Remarks:

1. The topologically designable element properties include PROD, PBAR, PBARL, PBEND, PBEAM, PBEAML, PSHELL, PSHEAR, PSOLID, and PWELD. Multiple TOPVAR's are allowed in a single file.
2. All designed element properties must refer to a MAT1 entry; therefore, a PCOMP cannot be used as designed property in topology optimization. PCOMP's can be used as non-designed properties in a topology optimization job.
3. If DELXV is blank, the default is taken from the specification of DELX parameter on the DOPTPRM entry.
4. Only CORD1R and CORD2R can be used as a referenced coordinate system to specify topology manufacturing constraints. Only one reference coordinate system CID is allowed for each TOPVAR entry.
5. One, two or three different mirror symmetry planes can present (such as MS1=XY, MS2=YZ, and MS3=ZX).
6. Casting ("CAST") and Extrusion ("EXT") manufacturability constraints can be applied to PTYPE="PSOLID" only. Casting constraints cannot be combined with extrusion constraints for the same TOPVAR entry.
7. Some symmetry constraint types can be combined with casting or extrusion constraints. The referenced coordinate system CID must be the same for the combined constraints. Some possible combinations are:
8. For "EXT" constraints, possible combinations are (EDi=X, MSi=XY and / or ZX), (EDi=Y, MSi=YZ and / or XY), (EDi=Z, MSi=ZX and / or YZ).
9. For "CAST" constraints, possible combinations are (DDi=X or -X, MSi=XY and / or ZX), (DDi=Y or -Y, MSi=YZ and / or XY), (DDi=Z or -Z, MSi=ZX and / or YZ).
10. For two dies option (DIE=2), the splitting plane is optimized. For a single die DIE=1, the parting plane is the bottom surface of the designed part in the draw direction.
11. TDMIN is a dimensional quantity with a guideline that it be set to at least three times a representative element dimension.
12. Without a TDMIN continuation line, the minimum member size constraint is taken from the specification of TDMIN parameter on the DOPTPRM entry. This option is applied on 2 and 3 D elements only. Minimum member size constraints can be used with "SYM", "CAST", and "EXT" constraints.

13. It is recommended to use a smooth top surface in the draw direction for one die casting constraints, and smooth top and bottom surfaces in the draw direction for two die casting constraints.

TRIM Trim Variable Constraint

Specifies constraints for aeroelastic trim variables.

Format:

1	2	3	4	5	6	7	8	9	10
TRIM	ID	MACH	Q	LABEL1	UX1	LABEL2	UX2	AEQR	
	LABEL3	UX3	-etc.-						

Example:

TRIM	1	0.9	100.	URDD3	1.0	ANGLEA	7.0	0.0	
	ELEV	0.2							

Field	Contents
SID	Trim set identification number. (Integer > 0)
MACH	Mach number. (Real $\geq$ 0.0 and $\neq$ 1.0)
Q	Dynamic pressure. (Real > 0.0)
LABELi	The label identifying aerodynamic trim variables defined on an AESTAT or AESURF entry. (Character)
UXi	The magnitude of the aerodynamic extra point degree-of-freedom. (Real)
AEQR	Flag to request a rigid trim analysis (Real $\geq$ 0.0 and $\leq$ 1.0, Default =1.0. A value of 0.0 provides a rigid trim analysis, see Remark 4.

Remarks:

1. The TRIM entry must be selected with the Case Control command TRIM=SID.
2. The selected TRIM entry specifies the constrained values of the listed extra point degrees-of-freedom (“trim variables”) for a particular loading condition. These variables are listed on AESTAT and/or AESURF entries.
3. If MACH is less than 1.0, then the Doublet-Lattice theory is used. If MACH is greater than 1.0, then the ZONA51 theory is used.

4. AEQR=0.0 can be used to perform a rigid trim analysis (ignoring the effects of structural deformation on the loading). AEQR=1.0 provides standard aeroelastic trim analysis. Intermediate values are permissible, but have no physical interpretation (they may be useful for model checkout).

TSTEP Transient Time Step

Defines time step intervals at which a solution will be generated and output in transient analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TSTEP	SID	N1	DT1	NO1					
		N2	DT2	NO2					
		-etc.-							

Example:

TSTEP	2	10	.001	5					
		9	0.01	1					

Field	Contents
SID	Set identification number. (Integer > 0)
Ni	Number of time steps of value DTi. (Integer ≥ 1)
DTi	Time increment. (Real > 0.0)
NOi	Skip factor for output. Every NOi-th step will be saved for output. (Integer > 0; Default = 1)

Remarks:

1. TSTEP entries must be selected with the Case Control command TSTEP = SID.
2. Note that the entry permits changes in the size of the time step during the course of the solution. Thus, in the example shown, there are 10 time steps of value .001 followed by 9 time steps of value .01. Also, the user has requested that output be recorded for t = 0.0, .005, .01, .02, .03, etc.
3. See "**Guidelines for Effective Dynamic Analysis**" on page 287 of the *MSC.Nastran 2006 Basic Dynamics* for a discussion of considerations leading to the selection of time steps.
4. In modal frequency response analysis (SOLs 111 and 146), this entry is required only when TLOADi is requested; i.e., when Fourier methods are selected.

5. The maximum and minimum displacement at each time step and the SIL numbers of these variables can be printed by altering DIAGON(30) before the transient module TRD1 and by altering DIAGOFF(30) after the module. This is useful for runs that terminate due to overflow or excessive run times.
6. For heat transfer analysis in SOL 159, use the TSTEPNL entry.

TSTEPNL Parameters for Nonlinear Transient Analysis

Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. TSTEPNL is intended for SOLs 129, 159, 400, 600 and 700.

Format:

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO	METHOD	KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS		
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB		

Example:

TSTEPNL	250	100	.01	1	ADAPT	2	10	PW	
	1.E-2	1.E-3	1.E-6	2	10	2	.02		
	5	5	0	0.75	16.0	0.1	20.		

Field	Contents
ID	Identification number. (Integer > 0)
NDT	Number of time steps of value DT. See Remark **2.** (Integer $\geq$ 4)
DT	Time increment. See Remark **2.** (Real > 0.0)
NO	Time step interval for output. Every NO-th step will be saved for output. See Remark **3.** (Integer $\neq$ 0; Default = 1)
METHOD	Method for stiffness matrix update and the transient time integration schemes. For SOLs 129 and 159, only METHOD=ADAPT is allowed and it is also the default. For SOL 400, all options are allowed and the default is AUTO. See Remark **4.** (Character: “AUTO”, “ITER”, “ADAPT”, or “SEMI”.)
KSTEP	KSTEP is the number of converged bisection solutions between stiffness updates for ADAPT method. (Default = 2). It is the number of iterations before stiffness updates for ITER method (Default = 10). See Remark **16.** (Integer > 0).
MAXITER	Limit on number of iterations for each time step. See Remark **16.** (Integer > 0; Default = 10 for ADAPT method and 25 for non-ADAPT methods)

Field	Contents
CONV	Flags to select convergence criteria. See Remark **6.** (Character: Default = "PW" for SOLs 129 and 159, "UPW" for SOL 400)
EPSU	Error tolerance for displacement (U) criterion. (Real > 0.0; Default = 1.0E-2)
EPSP	Error tolerance for load (P) criterion. (Real > 0.0; Default = 1.0E-3 for SOLs 129 and 159, 1.0E-2 for SOL 400)
EPSW	Error tolerance for work (W) criterion. (Real > 0.0; Default = 1.0E-6 for SOLs 129 and 159, 1.0E-2 for SOL 400)
MAXDIV	Limit on the number of diverging conditions for a time step before the solution is assumed to diverge. See Remark **7.** (Integer > 0; Default = 2)
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark **8.** (Integer $\geq$ 0; Default = 10)
MAXLS	Maximum number of line searches allowed per iteration. See Remark **8.** (Integer $\geq$ 0; Default = 2)
FSTRESS	Fraction of effective stress ($\bar{\sigma}$) used to limit the subincrement size in the material routines. See Remark **9.** (0.0 < Real < 1.0; Default = 0.2)
MAXBIS	Maximum number of bisections allowed for each time step. See Remark **10.** (-9 $\leq$ Integer $\leq$ 9; Default = 5)
ADJUST	Time step skip factor for automatic time step adjustment. See Remark **11.** (Integer $\geq$ 0; Default = 5)
MSTEP	Number of steps to obtain the dominant period response. See Remark **12.** (10 $\leq$ Integer $\leq$ 200; Default = variable between 20 and 40.)
RB	Define bounds for maintaining the same time step for the stepping function during the adaptive process. See Remark **12.** (0.1 $\leq$ Real $\leq$ 1.0; Default = 0.6)
MAXR	Maximum ratio for the adjusted incremental time relative to DT allowed for time step adjustment. See Remark **13.** (Real $\geq$ 1.0; Default = 32.0)
UTOL	Tolerance on displacement or temperature increment below which a special provision is made for numerical stability. See Remark **14.** (0.001 < Real $\leq$ 1.0; Default = 0.1)
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark **15.** (Real > 2.0; Default = 20.0)

Remarks:

1. The TSTEPNL Bulk Data entry is selected by the Case Control command TSTEPNL = ID. Each residual structure subcase requires a TSTEPNL entry and either applied loads via TLOADi data or initial values from a previous subcase. Multiple subcases are assumed to occur sequentially in time with the initial values of time and displacement conditions of each subcase defined by the end conditions of the previous subcase.
2. NDT is used to define the total duration for analysis, which is NDT * DT. Since DT is adjusted during the analysis , the actual number of time steps, in general, will not be equal to NDT). Also, DT is used only as an initial value for the time increment.
3. For printing and plotting the solution with SOLs 129 and 159, data recovery is performed at time 0, NO*DT, 2*NO*DT, …, and the last converged step, where DT is internally computed time increment and may change at every time step. For SOL 400 and NO>0, data recovery is performed at time 0, NO*DTI, 2*NO*DTI, …, and the last converged step, where DTI is the user input initial time increment and it is a constant. Also, if ADJUST≠0, the time step size computed by the automatic time adjustment (see remark 15) can never be greater than DTI. For SOL 400 and NO<0, the SOL 129 scheme is used for SOL 400, i.e., data recovery is performed at time 0, |NO|*DT, 2*|NO|*DT, …, and the converged step. Also, the time step size computed by automatic time adjustment can be either greater or less than the user input DTI.
4. The stiffness update strategy as well as the transient integration method is selected in the METHOD field.
   a. METHOD=”ADAPT”: The program automatically adjusts the incremental time and uses bisection. During the bisection process, the stiffness matrix is updated every KSTEPth converged bisection solution. This is the **only** method available for SOLs 129 and 159 and is also their **default**.
   b. METHOD=”AUTO”: The stiffness matrix is automatically updated to improve the convergence. Also, the program automatically adjusts the incremental time and uses bisection. The automatic time adjustment can be deselected by using ADJUST=0. KSTEP value is ignored during iteration. This is the **default** method for SOL 400.

c. The stiffness is updated at every KSTEPth iterations. Also, the program automatically adjusts the incremental time and uses bisection. The automatic time adjustment can be deselected by using ADJUST=0. If KSTEP=1, this is the full Newton iteration method.

d. METHOD="SEMI": Same as the AUTO method except that the stiffness updated at the first iteration, and then start the AUTO iteration scheme.

5. The number of iterations for a time step is limited to MAXITER. If MAXITER is negative, the analysis is terminated when the divergence condition is encountered twice during the same time step or the solution diverges for five consecutive time steps. If MAXITER is positive, the program computes the best solution and continues the analysis until divergence occurs again. If the solution does not converge in MAXITER iterations, the process is treated as a divergent process. See Remark **7.**

6. The convergence test flags (U = displacement error test, P = load equilibrium error test, W = work error test) and the error tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All requested criteria (combination of U, P, and / or W) are satisfied upon convergence. Note that at least two iterations are necessary to check the displacement convergence criterion.

7. MAXDIV provides control over diverging solutions. Depending on the rate of divergence, the number of diverging solutions (NDIV) is incremented by 1 or 2. The solution is assumed to diverge when NDIV reaches MAXDIV during the iteration. If the bisection option is used (allowed MAXBIS times) the time step is bisected upon divergence. Otherwise, the solution for the time step is repeated with a new stiffness based on the converged state at the beginning of the time step. If NDIV reaches MAXDIV again within the same time step, the analysis is terminated.

8. Nonzero values of MAXQN and MAXLS will activate the quasi-Newton update and the line search process, respectively.

9. The number of subincrements in the material routines is determined such that the subincrement size is approximately FSTRESS $\cdot \bar{\sigma}$. FSTRESS is also used to establish a tolerance for error correction in elastoplastic material, i.e.,

$$\text{error in yield function} < \text{FSTRESS} \cdot \text{yield stress}$$

If the limit is exceeded at the converging state, the program will terminate with a fatal error message. Otherwise, the stress state is adjusted to the current yield surface, resulting in $\delta = 0$.

10. The bisection process is activated when divergence occurs and MAXBIS $\neq$ 0. The number of bisections for a time increment is limited to |MAXBIS|. If MAXBIS is positive and the solution does not converge after MAXBIS bisections, the best solution is computed and the analysis is continued to the next time step. If MAXBIS is negative and the solution does not converge in |MAXBIS| bisection, the analysis is terminated.

11. ADJUST controls the automatic time stepping. Since the automatic time step adjustment is based on the mode of response and not on the loading pattern, it may be necessary to limit the adjustable step size when the period of the forcing function is much shorter than the period of dominant response frequency of the structure. It is the user's responsibility to ensure that the loading history is properly traced with the ADJUST option. The ADJUST option should be suppressed for the duration of short pulse loading. If unsure, start with a value for DT that is much smaller than the pulse duration in order to properly represent the loading pattern.

    - If ADJUST = 0, then the automatic adjustment is deactivated. This is recommended when the loading consists of short duration pulses.
    - If ADJUST > 0, the time increment is continually adjusted for the first few steps until a good value of Δt is obtained. After this initial adjustment, the time increment is adjusted every ADJUST-th time step only.
    - If ADJUST is one order greater than NDT, then automatic adjustment is deactivated after the initial adjustment.

12. MSTEP and RB are used to adjust the time increment during analysis. The recommended value of MSTEP for nearly linear problems is 20. A larger value (e.g., 40) is required for highly nonlinear problems. By default, the program automatically computes the value of MSTEP based on the changes in the stiffness.

    The time increment adjustment is based on the number of time steps desired to capture the dominant frequency response accurately. The time increment is adjusted as follows:

$$\Delta t_{n+1} = f(r)\Delta t_n$$

where

$$r = \frac{1}{\text{MSTEP}}\left(\frac{2\pi}{\omega_n}\right)\left(\frac{1}{\Delta t_n}\right)$$

with

$f = 0.25$ for $r < 0.5 \cdot RB$

$f = 0.5$ for $0.5 \cdot RB \le r < RB$

$f = 1.0$ for $RB \le r < 2.0$

$f = 2.0$ for $2.0 \le r < 3.0/RB$

$f = 4.0$ for $r \ge 3.0/RB$

13. MAXR is used to define the upper and lower bounds for adjusted time step size, i.e.,

$$\text{MIN}\left(\frac{DT}{2^{MAXBIS}}, \frac{DT}{MAXR}\right) \le \Delta t \le MAXR \cdot DT$$

14. UTOL is a tolerance used to filter undesirable time step adjustments; i.e.,

$$\frac{\left\|\dot{U}_n\right\|}{\left\|\dot{U}\right\|_{max}} < UTOL$$

Under this condition no time step adjustment is performed in a structural analysis (SOL 129). In a heat transfer analysis (SOL 159) the time step is doubled.

15. The bisection is activated if the incremental rotation for any degree-of-freedom ($\Delta\theta_x$, $\Delta\theta_y$, $\Delta\theta_z$) exceeds the value specified by RTOLB. This bisection strategy is based on the incremental rotation and controlled by MAXBIS.

16. For non-ADAPT (AUTO, ITER, and SEMI) methods, the stiffness will be updated at convergence if the number of iterations since last iteration is equal or greater than KSTEP. For SOL 400 and ADAPT method, the stiffness will be updated if 3*MAXITER iterations are performed. For SOL 400 and non-ADAPT methods, the stiffness will be updated if MAXITER iterations are performed.

TTEMP Temperature Distribution of Transient Response

Defines a time-dependent temperature distribution for the nonlinear transient analysis.

$$\{T(t)\} = \{A(T) \cdot F(t)\}$$

where $A(T)$ defines a spatial temperature distribution and $F(t)$ a time function. $T(t)$ is the time dependent temperature distribution for use in the nonlinear elements in nonlinear transient analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TTEMP	SID	GROUP_ID	TID						

Example:

TTEMP	11	101	31						

Field	Contents
SID	Temperature set identification number. (Integer > 0)
GROUP_ID	Temperature group identification number (Integer > 0 or =-1)
TID	Identification number of TABLEDi entry that gives $F(t)$. (Integer > 0)

Remarks:

1. The temperature distribution TTEMP must be selected by a Case Control command TEMP(LOAD)=SID in order to be used in the nonlinear transient analysis.
2. This command is used in SOL 400 only when ANALYSIS=NLTRAN (nonlinear transient analysis) and the temperature load is applied. It only applies to the nonlinear elements in the Residual (SEID=0). There should be only one temperature set for each STEP.
3. GROUP_ID determines the distribution of temperatures. It references the TMPSET Bulk Data entry to define all grid points, which reference the same TABLEDi entry. Each grid point can have its own GROUP_ID if necessary. GROUP_ID=-1 means all grid points are in one group and reference the same TTEMP Bulk Data entry.

4. If the TEMP(INIT) Case Control command references a TTEMP entry, then only the spatial distribution of the TTEMP will be used as the initial temperature distribution for the TEMP(INIT) command.

UNBALNC Specifies an Unbalanced Load

Specifies an unbalanced load for transient analysis in terms of a cylindrical system with the rotor rotation axis as the z-axis.

Format:

1	2	3	4	5	6	7	8	9	10
UNBALNC	RID	MASS	GRID	X1	X2	X3			
	ROFFSET	THETA	ZOFFSET	T_{on}	T_{OFF}	CFLAG			
	UFT1	UFT2	UFT3	UFR1	UFR2	UFR3			
	MCT1	MCT2	MCT3	MCR1	MCR2	MCR3			
	SCR1	SCR2	SCR3						

Example:

UNBALNC	100	0.1	1001	0.0	1.0	0.0			
	0.02	30.0	0.5			-1			

Field	Contents
RID	Identification number of UNBALNC entry. Selected by Case Control command, RGYRO. (Integer; Required; no Default)
MASS	Mass of imbalance (real or integer; if integer must be > 0; see Remarks **3.** and **6.**; Required, no Default)
GRID	Grid identification number of applying imbalance. The grid must appear on a ROTORG entry. (Integer .0; Required, no Default)
X1, X2, X3	Components of the vector, from GRID, in the displacement coordinate system of GRID, which is used to define a cylindrical coordinate system centered at GRID; see Remark **4.** (Real, Required, no Default)
ROFFSET	Offset of mass in the radial direction of the unbalance coordinate system (real or integer; if integer, must be > 0; see Remark **3.**; Default = 1.0)
THETA	Angular position of the mass in the unbalance coordinate system. (Real; Default = 0.0)
ZOFFSET	Offset of mass in the z-direction of the unbalance coordinate system. (Real or Integer; if integer, must be > 0; see Remark **3.**; Default = 0.0)
T_{ON}	Start time for applying imbalance load. (Real $\geq$ 0.0; Default = 0.0)

Field	Contents
T_{OFF}	Time for terminating imbalance load. (Real > 0.0; Default = 999999.0)
CFLAG	Correct flag to specify whether 1) the mass will be used to modify the total mass in the transient response calculations, 2) the effect of the rotor spin rate change will be included in the transient response calculation, or 3) both; see Remark 5. (Character: NONE, MASS, SPEED, or BOTH, Default = NONE)
UFT1-3	EPOINTs to output the unbalanced forces in the T1, T2, and T3 directions. (Integer > 0)
UFR1-3	EPOINTs to output the unbalanced forces in the R1, R2, and R3 directions. (Integer > 0)
MCT1-3	EPOINTs to output the mass-correction forces in the T1, T2, and T3 directions. (Integer > 0)
MCR1-3	EPOINTs to output the mass-correction forces in the R1, T2, and R3 directions. (Integer > 0)
SCR1-3	EPOINTs to output the speed-correction forces in the R1, R2, and R3 directions. (Integer > 0)

Remarks:

1. Multiple UNBALNC entries with the same RID value may be used.
2. The imbalance load will be generated based on the mass value, offset values, and the rotor spin speed.
3. If the entry is a real number, the value is considered constant. If the entry is an integer number, the value references a table entry that specifies the value as a function of time.
4. The cylindrical coordinate system used for the initial position of the mass unbalance has the positive z-axis direction from GRIDA to GRIDB specified on the RSPINT entry. Theta is measured from the plane defined by the z-axis and the user specified vector (X1, X2, X3). Theta = 0.0 is in the direction of the user-specified vector.
5. If the mass loss (increase) is relatively small, the correction may safely be ignored (CFLAG = NONE). If CFLAG is set to MASS, SPEED, or BOTH; all six degrees-of-freedom of the GRID must be in the A-set, otherwise a FATAL error message will be issued.
6. PARAM,WTMASS will scale the mass value for load generation.

7. Combined forces can be output using the NLLOAD Case Control request.

USET Degree-of-Freedom Set Definition

Defines a degree-of-freedom set.

Format:

1	2	3	4	5	6	7	8	9	10
USET	SNAME	ID1	C1	ID2	C2	ID3	C3		

Example:

USET	U4	333	26	17	0				

Field	Contents
SNAME	Set name. (One to four characters, or the string “ZERO” followed by the set name.)
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. SNAME may refer to any of the set names given in “**Degree-of-Freedom Sets**” on page 939 or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U6 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.
2. If SNAME = “ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.
3. A maximum of 18 degrees-of-freedom may be designated on a single entry.
4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.

5. The USET entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.
6. If a USETi Bulk Data entry lists a standard MD Nastran set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET Bulk Data entries.

USET1 Degree-of-Freedom Set Definition, Alternate Form

Defines a degrees-of-freedom set.

Format:

1	2	3	4	5	6	7	8	9	10
USET1	SNAME	C	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	ID8	-etc.-						

Example:

USET1	SB	345	2	1	36	5	9	7	
	40								

Alternate Format and Example:

USET1	SNAME	C	ID1	“THRU”	ID2				
USET1	SB	123	170	THRU	180				

Field	Contents
SNAME	Set name. (One to four characters or the word “ZERO” followed by the set name.)
C	Component numbers. (Integer zero or blank for scalar points or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification number. (Integer > 0; for “THRU” option, ID1 < ID2.)

Remarks:

1. SNAME may refer to any of the set names given in “**Degree-of-Freedom Sets**” on page 939 or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U6 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.
2. If SNAME=“ZEROi“, where i is a set name, then the degrees-of-freedom are omitted from set i.

3. If the alternate format is used, all of the points ID1 through ID2 are assigned to the set.
4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.
5. The USET1 entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.
6. If a USETi Bulk Data entry lists a standard MD Nastran set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET Bulk Data entries.

USRSUB6 (SOL 600) Defines User Subroutines for SOL 600

Defines user subroutines used in MD Nastran Implicit Nonlinear (SOL 600) only.

Format:

1	2	3	4	5	6	7	8	9	10
USRSUB6	U1	U2	U3	U4	U5	U6	U7	U8	
	U9	U10							

Examples:

USRSUB6	UDAMAG	UVOID	TENSOF						

USRSUB6*	SEPFORBBC								

Field	Contents
Ui	Name of user subroutine to be included. See MSC.Marc Volume D for list of available user subroutines. (Character, no default)

Notes:

1. All user subroutines must reside in the directory where the MD Nastran input file resides.
2. All names must be in lower case and have the extension .f
3. SOL 600 combines all user subroutines into one large subroutine named u600.f and u600.f is passed to the MSC.Marc command line when spawned from MD Nastran.
4. If only one user subroutine is required, an alternate is to use PARAM,MARCUSUB,name.

UXVEC Control Parameter State

Specification of a vector of aerodynamic control point (extra point) values. These data define the control positions corresponding to user defined nonlinear control forces that have been defined by AEDW, AEPRESS and AEFORCE entries. Only nonzero values need to be defined.

Format:

1	2	3	4	5	6	7	8	9	10
UXVEC	ID								
	LABEL1	UX1	LABEL2	UX2	-etc.-				

Example:

UXVEC	1001								
	THRUST	1.E4	ANGLEA	.015					

Field	Contents
ID	Control vector identification number, see Remark **1.** (Integer > 0)
LABELi	Controller name. This must correspond to an existing AESURF, AESTAT or AEPARM label (Character).
UXi	The magnitude of the aerodynamic extra point degree-of-freedom (Real)

Remarks:

1. The ID is referenced by the AEUXREF Case Control command and / or by AEDW, AEPRESS, and / or AEFORCE entries.
2. The units of the user defined AEPARM controllers are implied by their use on this entry and the corresponding values on the force vector definition. The user must be self-consistent in all uses of the same controller. AESURF controllers are expressed in radians as are the rigid body angles ANGLEA and BETA. The rigid body rates, ROLL, PITCH and YAW are nondimensional rates pb / 2V, qc / 2V, rb / 2V; respectively. V is the velocity and b and c are the reference span and chord lengths, respectively.
3. LABELs that are part of the UX vector that are omitted in the UXVEC specification are assigned a value of 0.0.

VIEW View Factor Definition

Defines radiation cavity and shadowing for radiation view factor calculations.

Format:

1	2	3	4	5	6	7	8	9	10
VIEW	IVIEW	ICAVITY	SHADE	NB	NG	DISLIN			

Example:

VIEW	1	1	BOTH	2	3	0.25			

Field	Contents
IVIEW	Identification number. (Integer > 0)
ICAVITY	Cavity identification number for grouping the radiant exchange faces of CHBDYi elements. (Integer > 0)
SHADE	Shadowing flag for the face of CHBDYi element. (Character, Default = “BOTH”) NONE means the face can neither shade nor be shaded by other faces. KSHD means the face can shade other faces. KBSHD means the face can be shaded by other faces. BOTH means the face can both shade and be shaded by other faces. (Default)
NB	Subelement mesh size in the beta direction. (Integer > 0; Default = 1)
NG	Subelement mesh size in the gamma direction. (Integer > 0; Default = 1)
DISLIN	The displacement of a surface perpendicular to the surface. See **Figure 8-178**. (Real; Default = 0.0)

Remarks:

1. VIEW must be referenced by CHBDYE, CHBDYG, or CHBDYP elements to be used.
2. ICAVITY references the cavity to which the face of the CHBDYi element belongs; a zero or blank value indicates this face does not participate in a cavity.

3. NB, NG, and DISLIN are used in the calculation of view factors by finite difference or contour integration techniques. They are not used with the VIEW3D entry.
4. A summary of the shadowing conditions can be requested by the PARAM,MESH,YES Bulk Data entry.
5. SHADE references shadowing for CHBDYi elements participating in a radiation cavity, the VIEW calculation can involve shadowing.
6. DISLIN should only be used with LINE type CHBDYE and CHBDYP surface elements. DISLIN > 0.0 means into the cavity. See **Figure 8-178**.

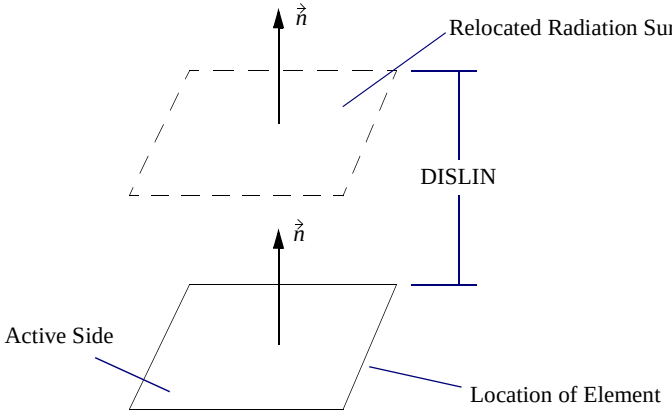


Figure 8-178 DISLIN Convention

7. NB and NG define the subelement mesh refinement when using the VIEW module (as opposed to the VIEW3D module) for the calculation of view factors.

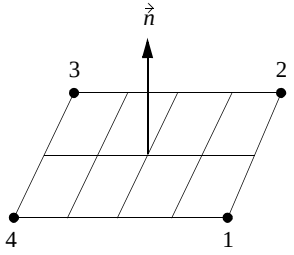


Figure 8-179 Typical AREA4 surface element where NB=2 and NG=4

VIEW3D View Factor Definition - Gaussian Integration Method

Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

Format:

1	2	3	4	5	6	7	8	9	10
VIEW3D	ICAVITY	GITB	GIPS	CIER	ETOL	ZTOL	WTOL	RADCHK	

Example:

VIEW3D	1	2	2	4		1.0E-6			

Field	Contents
ICAVITY	Radiant cavity identification number on RADCAV entry. (Integer > 0)
GITB	Gaussian integration order to be implemented in calculating net effective view factors in the presence of third-body shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)
GIPS	Gaussian integration order to be implemented in calculating net effective view factors in the presence of self-shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)
CIER	Discretization level used in the semi-analytic contour integration method. ($1 \leq$ Integer ≤ 20; Default = 4)
ETOL	Error estimate above which a corrected view factor is calculated using the semi-analytic contour integration method. (Real ≥ 0.0; Default = 0.1)
ZTOL	Assumed level of calculation below which the numbers are considered to be zero. (Real ≥ 0.0; Default = 1.E-10)
WTOL	Assumed degree of warpage above which the actual value of F_{ii} will be calculated. ($0.0 \leq$ Real ≤ 1.0; Default = 0.01)

Field	Contents
RADCHK	Type of diagnostic output desired for the radiation exchange surfaces. (Integer; Default = 3) RADCHK = -1, No diagnostic output requested RADCHK = 1, Grid table and element connectivity RADCHK = 2, Surface Diagnostics - Surface type, area, skewness, taper, warpage, grid point sequencing, aspect ratio, and shading flags. RADCHK = 3, Area, view factor, area-view factor product with error estimate, existence flags for partial self-shadowing, third-body shadowing with error estimate, and enclosure summations for view factor. (Default) RADCHK = 0, Same as RADCHK = 1, 2, and 3 RADCHK = 12, Same as RADCHK = 1 and 2 RADCHK = 13, Same as RADCHK = 1 and 3 RADCHK = 23, Same as RADCHK = 2 and 3

Remarks:

1. For ETOL, when the error estimate exceeds the value input for the ETOL entry, the contour method is employed to develop an improved view factor.
2. For ZTOL, the use of a geometry scale that results in small numerical values of $A_i F_{ij}$ should be avoided.
3. When WTOL is exceeded, the actual value of F_{ii} will be calculated when using the adaptive view module. Warpage will not be considered in the calculation of F_{ij}.
4. For axisymmetric analysis, RADCHK = -1 or 3 only.

WALL (SOL 700) Rigid Wall

Defines a rigid plane through which specified Lagrangian grid points cannot penetrate. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
WALL	ID	XP	YP	ZP	NXA	NY	NZ	SET	
WALL	17						1.0	21	

Example:

	METHOD	FS	FK	EXP					
	PENALTY	0.2							

Field	Contents	
ID	Unique rigid-wall number. (Integer > 0; Required)	
XP, YP, ZP	Coordinates of the origin of the wall. (Real; Default = 0.0)	
NX, NY, NZ	A vector normal to the wall pointing towards the model. (Real; Default = 0.0)	
SET	Number of a SET1 entry listing the points that cannot penetrate the wall. (Integer > 0; Required)	
METHOD	Algorithm for contact processing.	
	PENALTY	Penalty method, allowing for extra boundary conditions, friction and output.
	KINMATIC	Kinematic method, only included for compatibility reasons with older Dytran versions. This method allows no extra boundary conditions, no friction and no output.
FS	Static coefficient of friction. See Remark 5. (Real $\geq$ 0; Default = 0.0)	
FK	Kinetic coefficient of friction. See Remark 5. (Real $\geq$ 0; Default = 0.0)	
EXP	Exponential decay coefficient. See Remark 5. (Real $\geq$ 0; Default = 0.0)	

Remarks

1. A rigid plane of infinite size is generated that the grid points cannot penetrate. The plane is fixed in space.
2. The grid points can slide on the wall and separate from it.

3. A (moving) rigid plane of finite size can be modeled by using a rigid surface and a master-slave contact.
4. For the wall definition using penalty method, output can be requested by referencing it in a SET command in the Case Control Section. The keywords for output are WALLS and WALLOUT.
5. The coefficient of friction is given by:

$$\mu = \mu_k + (\mu_s - \mu_k)e^{-\beta v}$$

where:

μ_s = Static coefficient of friction FS.

μ_k = Kinetic coefficient of friction FK.

β = Exponential decay coefficient EXP.

v = Relative sliding velocity at the point of contact.

WALLGEO (SOL 700) Defines a Geometric Rigid Wall with an Analytically Described Form

Defines a geometric rigid wall with an analytically described form. Four forms are possible. A prescribed motion is optional. For general rigid bodies with arbitrary surfaces and motion, refer to the CONTACT definition. This option is for treating contact between rigid and deformable surfaces only. Used in MD Nastran Explicit Nonlinear (SOL 700) only.

Format:

1	2	3	4	5	6	7	8	9	10
WALLGEO	WID	NSID	NSIDEX		TYPE				+
	XT	YT	ZT	XH	YH	ZH	FRIC		
	A1	A2	A3	A4	A5	A6			
	LCID	OPT	VX	VY	VZ				

Example:

WALLGEO	33	101			FLAT				
	0.	0.	0.	0.	0.	1.	1.0		
	0.	1.	0.	5.	10.				
	101	1	0.	0.	1.				

Field	Contents
WID	Rigid Wall ID. Unique numbers have to be used. (Integer, no default, Required)
NID	SET1 ID containing slave nodes. (Integer, no default, Required)
NSIDEX	SET1 ID containing nodes that exempted as slave nodes. (Integer, default = blank, meaning not used)
Type	FLATR (Character, no default, required) PRISM CYLINDER SPERE
XY	x-coordinate of tail of any outward drawn normal vector, n, originating on wall (tail) and terminating in space (head), see **Figure 8-180**. (Real, no default, required)

Field	Contents
YT	y-coordinate of tail of normal vector n. (Real, no default, required)
ZY	z-coordinate of tail of normal vector n. (Real, no default, required)
XH	x-coordinate of head of normal vector n. (Real, no default, required)
YH	y-coordinate of head of normal vector n. (Real, no default, required)
ZH	z-coordinate of head of normal vector n. (Real, no default, required)
FRIC	Interface friction: (Real, default = 0.0) = 0.0: frictionless sliding after contact, = 1.0: stick condition after contact, 0.<FRIC<1.: Coulomb friction coefficient.
Ai	Depending on TYPE: (Real, no default, required depending on TYPE) FLAT A1x-coordinate of head of edge vector 1, see **Figure 8-180**. A2y-coordinate of head of edge vector 1 A3z-coordinate of head of edge vector 1 A4Length of 1 edge. A zero value defines an infinite size plane. A5Length of m edge. A zero value defines an infinite size plane. PRISM A1x-coordinate of head of edge vector 1, see **Figure 8-180**. A2y-coordinate of head of edge vector 1 A3z-coordinate of head of edge vector 1 A4Length of 1 edge. A zero value defines an infinite size plane. A5Length of m edge. A zero value defines an infinite size plane. A6Length of prism in the direction negative to n, see **Figure 8-180**. CYLINDER A1Radius of cylinder A2Length of cylinder, see **Figure 8-180**. Only if a value larger than zero is specified is a finite length assumed. SHERE A1Radius of sphere
LCID	TABLED ID which describes stonewall motion. (Integer, no defaults, required)
OPT	Type of motion: 0= velocity specified 1= displacement specified

Field	Contents
VX	x-direction cosine of velocity / displacement vector. (Real, Default = 0.0)
VY	y-direction cosine of velocity / displacement vector. (Real, Default = 0.0)
VZ	z-direction cosine of velocity / displacement vector. (Real, Default = 0.0)

Remarks:

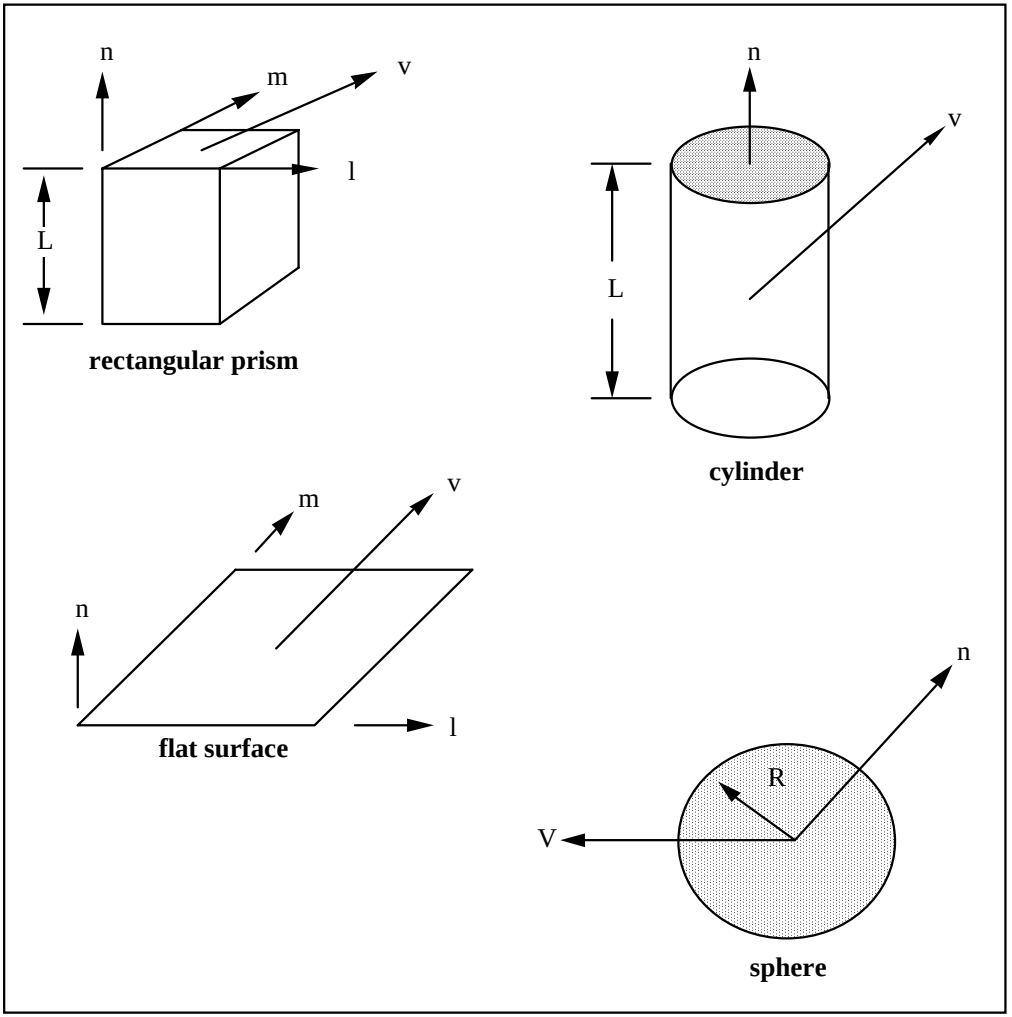


Figure 8-180 Vector n determines the orientation of the generalized stonewalls. For the prescribed motion options, the wall can be moved in the direction V as shown.