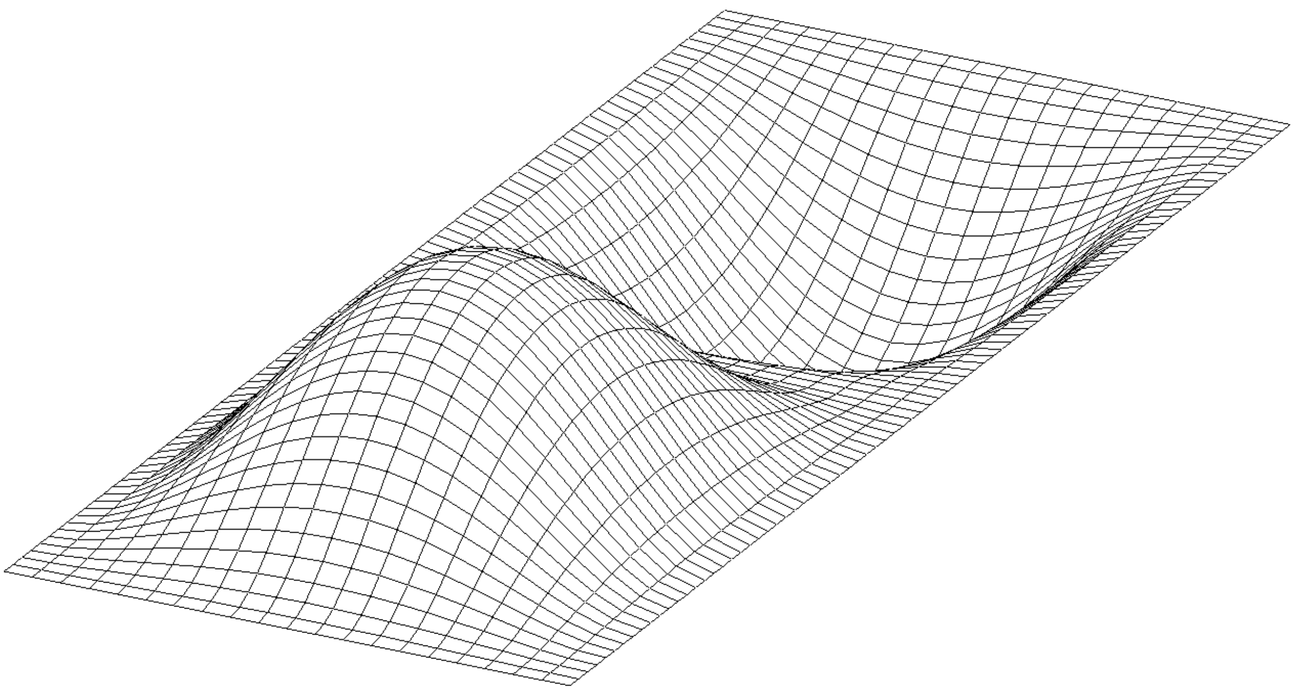
Autodesk Nastran 2024

Reference Manual



AUTODESK

Autodesk® Nastran® 2024

TABLE OF CONTENTS

Contents

NASTRAN COMMAND LINE

Running Autodesk Nastran

Autodesk Nastran is run by executing the file: Nastran.exe. The syntax for this along with the optional command line arguments are shown below:

```
NASTRAN [[d:][path]filename.INI] [[d:][path]filename.NAS] [[d:][path]filename.NDB]
        [directive = option]
```

The command line arguments are defined as follows:

`[d:][path]filename.INI`	Model Initialization File specification. This file contains directives that configure Autodesk Nastran to run on your system. The default filename is *Nastran.INI* and need not be specified unless you plan on using multiple initialization files with different names. This file configures Autodesk Nastran to run on your system and contains primarily file and memory management directives. For details, see Section 2, *Initialization*.
`[d:][path]filename.NAS`	NASTRAN Model Input File specification. This file contains the NASTRAN Case Control commands and Bulk Data entries that define the input model. This file can also be specified in the Model Initialization File using the MODLINFILE directive. For details, see Section 2, *Initialization*.
`[d:][path]filename.NDB`	Model Database Identification File specification. This file contains the model database identification number that locates an existing model database generated by the Model Translator. This file can also be specified in the Model Initialization File using the DATABASE directive. For details, see Section 2, *Initialization*.
`directive = option`	Model Initialization directive or Model Parameter. For details, see Section 2, *Initialization*.

Either a Model Input filename or a Model Database filename or both can be specified for the input model. The Model Input filename and the Database filename can be specified either on the command line or in the Model Initialization File. When a Model Input filename is specified in the Model Initialization File, any extension can be used. In the below example *Nastran.INI* is the Model Initialization File and *filename.NAS* is the NASTRAN Model Input file.

```
NASTRAN filename.NAS
```

File specifications and directives specified on the command line will override ones specified in the Model Initialization File. This allows you to configure the Model Initialization File with your default settings and change specific model dependent settings on the command line. For example, if the Model Initialization directive `RAM` was set to 100 megabytes in the Model Initialization File, it would be set to 200 megabytes using the Nastran command line below.

```
NASTRAN filename.NAS RAM=200
```

Section 2

INITIALIZATION

The Model Initialization File

The Model initialization file performs the following basic functions:

- Defines input and output file specifications.
- Defines model database file locations.
- Defines output format and type.
- Defines memory usage.
- Defines program control settings.
- Defines model parameters.

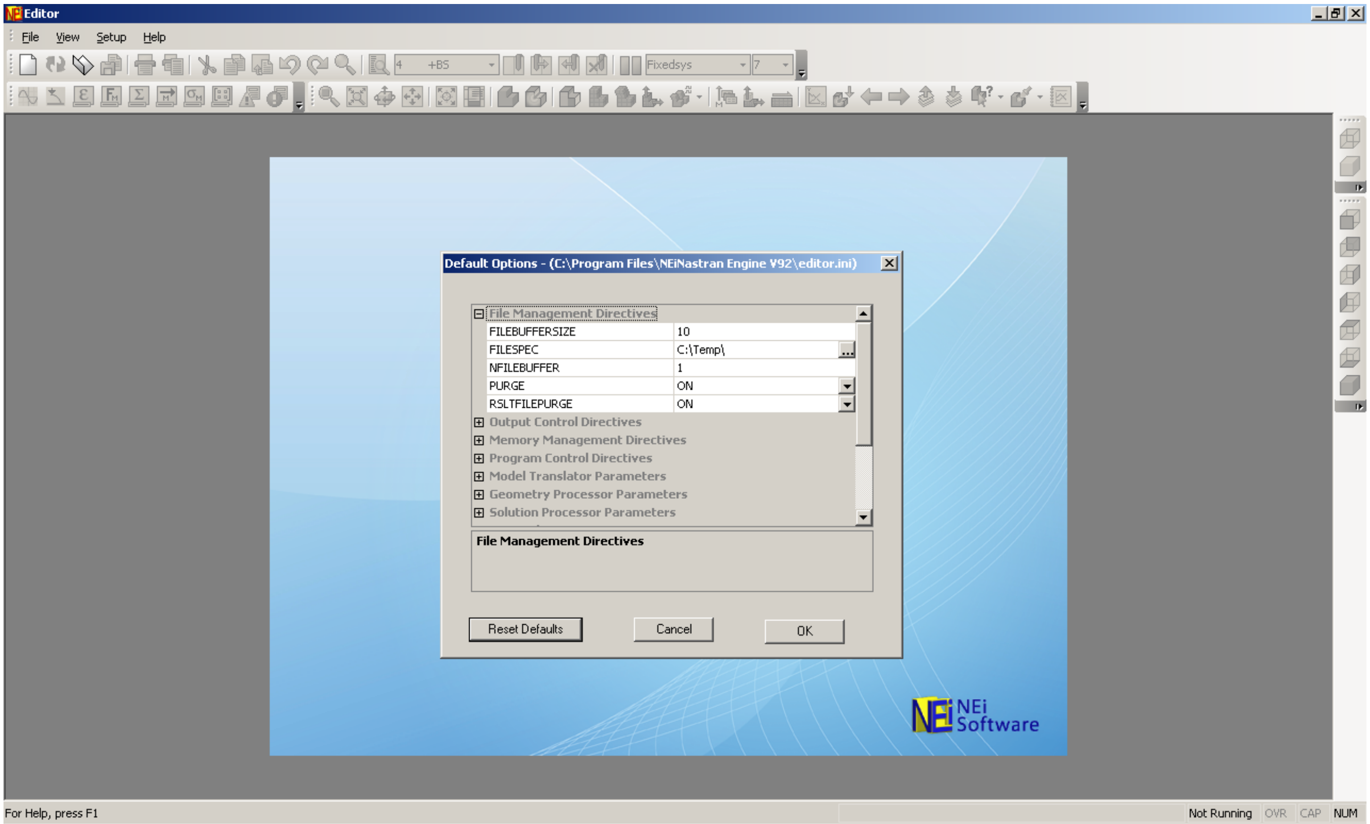
Thus, the Model Initialization File can be divided into the following five sections:

Section	Purpose
[File Management]	File Management directives allow the user to specify the names and locations of input, output, and database files.
[Output Control]	Output Control directives allow the user to control what output files are generated and what they have in them.
[Memory Management]	Memory Management directives allow the user to control what type of memory (virtual or physical) and how much will be used for memory intensive tasks such as matrix assembly and decomposition. By optimizing memory usage the user can optimize performance.
[Program Control]	Program Control directives allow the user to customize program execution by controlling how and what tasks are to be performed.
[Parameters]	Parameter statements that are specified using the PARAMETER command or entry can be specified in this section using the directive format. See Section 5, *Parameters*.

Each section has associated with it a group of related directives and each directive has a default setting (see the Nastran Solver Reference Manual, Section 2, *Initialization Directives,* for directive syntax and default settings).

For most configurations, the default settings in the nastran.ini file will provide optimal performance. There are a few directives you may want to change depending on your configuration. Changes can be made either using a standard text editor or through the Autodesk Nastran Editor Options menu.

The easiest way to modify the Nastran Model Initialization File (nastran.ini) is to open the **Autodesk Nastran Editor** by clicking on the Windows **Start**, **Programs**, **Autodesk Inventor Nastran**, **Autodesk Inventor Nastran Editor Utility** menu item. Then select **Setup**, **Default Analysis Options** and click on the desired section and option. To set model specific options, open the Model Input File and use the options menu displayed to the left.



The first directive you may want to modify is the scratch file folder. Double click on **File Management**, then on the **FILESPEC** directive to change the folder. You will want to select a folder on a disk with a large amount of available space. If you specified a scratch folder during installation it will be displayed here.

The next setting you may want to modify is under **Memory Management**, **RAM**. This setting can greatly affect performance and may not be initially optimized for your particular computer. On ia-32 systems with 2 GB or more of memory, set RAM equal to 1800. For systems with less than 2 GB, set RAM equal to the system memory in MB. On x64 systems set RAM equal to the installed system memory in MB minus 1000 MB (which will be used for the operating system). For example if you have 8 GB of physical memory, set RAM equal to 7000. If you specified a RAM available value during installation it will be displayed here.

Another directive you may want to modify is under **Geometry Processor Parameters**, **SHELLRNODE**. Turning **SHELLRNODE** to **ON** converts all CQUAD4 and TRIA3 elements to CQUADR and CTRIAR. The CQUADR and CTRIAR elements are complete 6 DOF/node elements, which typically give more accurate results.

One last directive you may want to modify is under **Solution Processor Parameters**, **SOLUTIONERROR**. You can avoid getting a fatal error when a non-positive definite caused by a modeling error is encountered by setting **SOLUTIONERROR** to **ON** and **FACTDIAG** to **0.0**. You can also avoid getting a fatal error when a singularity is encountered by setting **SOLUTIONERROR** to **ON** and **FACTDIAG** to **1.0E-10**. Note that while these options are useful for detecting modeling errors, they may lead to solutions of poor quality or fatal messages later in the run. It is recommended that **SOLUTIONERROR** be set to **OFF** for production runs.

Model Initialization Directive Descriptions

Model Initialization directives that listed in **single page format** are described as follows:

Description
A single sentence **Description** is given which states the function of the directive.

Format
The directive syntax is defined under **Format**.

Example
A typical example is given under **Example**.

Remarks
Additional information about the directive is given under **Remarks**.

Model Initialization directives that are listed in **tabular format** are described as follows:

Description
A complete description is given under **Description**, which states the function of the directive along with usage guidelines, any notes and other pertinent information.

Option
Option keyword syntax or allowable data range is given under **Option**. Character keywords are separated by a "/". Only one keyword can be specified.

Default
The default option is given under **Default**.

File Management Directives – Output File Specifications:

The only required file specification is the Model Input filename. All output file specifications will default to the model input filename base with the appropriate extension. The Model Input file can be specified on the Nastran command line (see Section 1, *NASTRAN Command Line*).

Below is a summary of all output file specifications. Detailed descriptions are given later in this section.

Directive	Description
BULKDATAFILE	Bulk Data Output File specification.
DATINFILE1	Data Input File specification 1.
DATINFILE2	Data Input File specification 2.
DISPFILE	Grid Point Displacement Vector Neutral File specification.
FORCFILE	Grid Point Force Vector Neutral File specification.
LOADFILE	Element Internal Load Vector Neutral File specification.
LOGFILE	System Log File specification.
ELEMFILE	Element Results Neutral File specification.
GRIDFILE	Grid Point Results Neutral File specification.
MODALDATFILE	Modal Database File specification.
MODLINFILE	NASTRAN Model Input File specification.
MODLOUTFILE	Model Results Output File specification.
NLINDATFILE	Nonlinear Database File specification.
RSLTDATFILE	Results Database File specification.

File Management Directives – Database File Specifications:

Database file specifications point to the location of permanent and scratch database files used during program execution. When Autodesk Nastran is executed, it generates a database that is located using the FILESPECi directives. A single file located in the same directory as the Model Results Output File is also generated and contains the location of that run's database. The DATABASE directive can be used to specify this file in place of the Model Input File if the database has already been generated by the Model Translator. Database files can become very large and fill up all available storage space. The database file specifications can also be used to break up a very large model database over several storage devices.

Below is a summary of all database file specifications. Detailed descriptions are given later in this section.

Directive	Description
DATABASE	Model Database File specification.
FILESPEC	Model Database File specification 1 – 4.
FILESPEC1	Model Database File specification 1.
FILESPEC2	Model Database File specification 2.
FILESPEC3	Model Database File specification 3.
FILESPEC4	Model Database File specification 4.
OUTFILESPEC	Output file specification.

BULKDATAFILE — Bulk Data Output File Specification

Description: Bulk Data Output File specification.

Format:

BULKDATAFILE = [d:] [path] filename[.ext]

Example:

BULKDATAFILE = c:\bulkhead\BULKHEAD.BDF

Remarks:

1. Maximum file specification length is 256 characters.
2. The default file specification is the Model Output File specification with the ".BDF" extension.

DATABASE

Nastran Database File Specification

Description: Nastran Database File specification.

Format:

DATABASE = [d:] [path] filename[.ext]

Example:

DATABASE = c:\bulkhead\BULKHEAD.NDB

Remarks:

1. Maximum file specification length is 256 characters.
2. The default file specification is the Model Output File specification with the “.NDB” extension.

DATINFILE1 Generic Data Input File Specification 1

Description: Data input file specification used for Modal Assurance Criterion (MAC) analysis.

Format:

DATINFILE1 = [d:] [path] filename[.ext]

Example:

DATINFILE1 = c:\bulkhead\BULKHEAD.MDB

Remarks:

3. Maximum file specification length is 256 characters.
4. To specify an MS Excel compatible Comma Separated Variable file format use a ".CSV" extension. To specify an Autodesk Nastran compatible Modal Database file format use a ".MDB" extension.
5. DATINFILE1 can also be used to reference a DMIG matrix already included in the Model Input File by setting it equal to the DMIG name.
6. DATINFILE1 is defaulted to the current modal database if not specified.

DATINFILE2 — Generic Data Input File Specification 2

Description: Data input file specification used for Modal Assurance Criterion (MAC) analysis.

Format:

DATINFILE2 = [d:] [path] filename[.ext]

Example:

DATINFILE2 = c:\bulkhead\BULKHEAD.CSV

Remarks:

1. Maximum file specification length is 256 characters.
2. To specify an MS Excel compatible Comma Separated Variable file format use a ".CSV" extension. To specify an Autodesk Nastran compatible Modal Database file format use a ".MDB" extension.
3. DATINFILE2 can also be used to reference a DMIG matrix already included in the Model Input File by setting it equal to the DMIG name.

DISPFILE Grid Point Displacement Vector Neutral File Specification

Description: Grid Point Displacement Vector Neutral File specification.

Format:

DISPFILE = [d:] [path] filename[.ext]

Example:

DISPFILE = c:\bulkhead\BULKHEAD.DIS

Remarks:

1. Maximum file specification length is 256 characters.
2. To disable the generation of the Grid Point Displacement Vector Neutral File set DISPFILE = NONE in the Model Initialization File.
3. When the Model Initialization directive RSLTFILETYPE is set to PATRAN ASCII or PATRAN BINARY, multiple file subcases, modes, time steps, etc. are enumerated in the last one to 16 characters of the base filename.
4. The default file specification is the Model Output File specification with the ".DIS" extension.

ELEMFILE — Element Results Neutral File Specification

Description: Element Results Neutral File specification.

Format:

ELEMFILE = [d:] [path] filename[.ext]

Example:

ELEMFILE = c:\bulkhead\BULKHEAD.ELS

Remarks:

1. Maximum file specification length is 256 characters.
2. To disable the generation of the Element Results Neutral File set ELEMFILE = NONE in the Model Initialization File.
3. When the Model Initialization directive RSLTFILETYPE is set to PATRAN ASCII or PATRAN BINARY, multiple file subcases, modes, time steps, etc. are enumerated in the last one to 16 characters of the base filename.
4. The default file specification is the Model Output File specification with the ".ELS" extension.

FILESPEC Model Database File Specification

Description: Model Database path.

Format:

FILESPEC = [d:] path

Example:

FILESPEC = c:\temp

Remarks:

1. This directive sets the default for FILESPEC1 through FILESPEC4.
2. Maximum file specification length is 244 characters.
3. The default directory for storage of database files is the directory where the Nastran command is executed.

FILESPEC1 Model Database File Specification 1

Description: Model Database partition one path.

Format:

FILESPEC1 = [d:] path

Example:

FILESPEC1 = c:\temp

Remarks:

1. Maximum file specification length is 244 characters.
2. The default directory storage of database files is the directory where the Nastran command is executed.

FILESPEC2 — Model Database File Specification 2

Description: Model Database partition two path.

Format:

FILESPEC2 = [d:] path

Example:

FILESPEC2 = c:\temp

Remarks:

1. Maximum file specification length is 244 characters.
2. The default directory for storage of database files is the directory where the Nastran command is executed.

FILESPEC3 Model Database File Specification 3

Description: Model Database partition three path.

Format:

FILESPEC3 = [d:] path

Example:

FILESPEC3 = c:\temp

Remarks:

1. Maximum file specification length is 244 characters.
2. The default directory for storage of database files is the directory where the Nastran command is executed.

FILESPEC4 — Model Database File Specification 4

Description: Model Database partition four path.

Format:

FILESPEC4 = [d:] path

Example:

FILESPEC4 = c:\temp

Remarks:

1. Maximum file specification length is 244 characters.
2. The default directory for storage of database files is the directory where the Nastran command is executed.

FORCFILE — Grid Point Force Neutral File Specification

Description: Grid Point Force Vector Neutral File specification.

Format:

FORCFILE = [d:] [path] filename[.ext]

Example:

FORCFILE = c:\bulkhead\BULKHEAD.GPF

Remarks:

1. Maximum file specification length is 256 characters.
2. To disable the generation of the Grid Point Force Vector Neutral File set FORCFILE = NONE in the Model Initialization File.
3. When the Model Initialization directive RSLTFILETYPE is set to PATRANASCII or PATRANBINARY, multiple file subcases, modes, time steps, etc. are enumerated in the last one to 16 characters of the base filename.
4. The default file specification is the Model Output File specification with the ".GPF" extension.

GRIDFILE Grid Point Results Neutral File Specification

Description: Grid Point Results Neutral File specification.

Format:

GRIDFILE = [d:] [path] filename[.ext]

Example:

GRIDFILE = c:\bulkhead\BULKHEAD.GPS

Remarks:

1. Maximum file specification length is 256 characters.
2. To disable the generation of the Grid Point Results Neutral File set GRIDFILE = NONE in the Model Initialization File.
3. When the Model Initialization directive RSLTFILETYPE is set to PATRANASCII or PATRANBINARY, multiple file subcases, modes, time steps, etc. are enumerated in the last one to 16 characters of the base filename.
4. The default file specification is the Model Output File specification with the ".GPS" extension.

LOADFILE Element Internal Load Vector Neutral File Specification

Description: Element Internal Load Vector Neutral File specification.

Format:

LOADFILE = [d:] [path] filename[.ext]

Example:

LOADFILE = c:\bulkhead\BULKHEAD.ELF

Remarks:

1. Maximum file specification length is 256 characters.
2. To disable the generation of the Element Internal Load Vector Neutral File set LOADFILE = NONE in the Model Initialization File.
3. When the Model Initialization directive RSLTFILETYPE is set to PATRANASCII or PATRANBINARY, multiple file subcases, modes, time steps, etc. are enumerated in the last one to 16 characters of the base filename.
4. The default file specification is the Model Output File specification with the “.GPF” extension.

LOGFILE

System Log File Specification

Description: System Log File specification.

Format:

LOGFILE = [d:] [path] filename[.ext]

Example:

LOGFILE = c:\bulkhead\BULKHEAD.LOG

Remarks:

1. Maximum file specification length is 256 characters.
2. To disable the generation of the System Log File set LOGFILE = NONE in the Model Initialization File.
3. The default file specification is the Model Output File specification with the ".LOG" extension.

MODALDATFILE — Modal Database File Specification

Description: Modal Database File specification.

Format:

MODALDATFILE = [d:] [path] filename[.ext]

Example:

MODALDATFILE = c:\bulkhead\BULKHEAD.MDB

Remarks:

1. Maximum file specification length is 256 characters.
2. The default file specification is the Model Output File specification with the ".MDB" extension.

MODLINFILE — Model Input File Specification

Description: Model Input File specification.

Format:

MODLINFILE = [d:] [path] filename[.ext]

Example:

MODLINFILE = c:\bulkhead\BULKHEAD.NAS

Remarks:

1. Maximum file specification length is 256 characters.
2. The Model Input filename can also be specified on the Nastran command line. See Section 1, *NASTRAN Command Line.*

MODLOUTFILE — Model Results Output File Specification

Description: Model Results Output File specification.

Format:

MODLOUTFILE = [d:] [path] filename[.ext]

Example:

MODLOUTFILE = c:\bulkhead\BULKHEAD.OUT

Remarks:

1. Maximum file specification length is 256 characters.
2. The default file specification is the Model Input File specification with the ".OUT" extension.

NLINDATFILE

Nonlinear Database File Specification

Description: Nonlinear Database File specification.

Format:

NLINDATFILE = [d:] [path] filename[.ext]

Example:

NLINDATFILE = c:\bulkhead\BULKHEADI1L08000.TDB

Remarks:

1. Maximum file specification length is 256 characters.
2. No default file specification is provided.

OUTFILESPEC Output File Specification

Description: Model output path.

Format:

OUTFILESPEC = [d:] path

Example:

OUTFILESPEC = c:\bulkhead

Remarks:

1. Maximum file specification length is 244 characters.
2. The default output file specification is the Model Output File path.

RSLTDATFILE — Results Database File Specification

Description: Results Database File specification.

Format:

RSLTDATFILE = [d:] [path] filename[.ext]

Example:

RSLTDATFILE = c:\bulkhead\BULKHEADI1L08000.RDB

Remarks:

1. Maximum file specification length is 256 characters.
2. No default file specification is provided.

TOPTDATFILE — Topology Design Optimization Database File Specification

Description: Topology Design Optimization Database File Specification.

Format:

TOPTDATFILE = [d:] [path] filename[.ext]

Example:

TOPTDATFILE = c:\bulkhead\BULKHEAD.ODB

Remarks:

1. Maximum file specification length is 256 characters.
2. The default file specification is the Model Output File specification with the “.ODB” extension.

TOPTDEPXFILE — Topology Design Optimization Dependent File Specification

Description: Topology design optimization dependent Model Input File Specification.

Format:

TOPTDEPXFILE = [d:] [path] filename[.ext]

Example:

TOPTDEPXFILE = c:\bulkhead\BULKHEADI1L08000.NAS

Remarks:

1. Maximum file specification length is 256 characters.
2. No default file specification is provided.

TOPTINDXFILE — Topology Design Optimization Independent File Specification

Description: Topology Design Optimization Independent Model Input File Specification.

Format:

TOPTINDXFILE = [d:] [path] filename[.ext]

Example:

TOPTINDXFILE = c:\bulkhead\BULKHEADI2L08000.NAS

Remarks:

1. Maximum file specification length is 256 characters.
2. No default file specification is provided.

File Management Directives – Miscellaneous:

Directive	Description	Option/Type	Default
FILEBUFFERSIZE	File buffer size in kilobytes for all functions. A larger value may increase I/O performance but decreases available physical memory. A value between 1 and 10000 is recommended. **Note:** Overall performance may be significantly affected by this directive. It is recommended that you test a range of values since optimal settings vary with operating system and hardware.	Integer ≥ 0	1000
FILEBUFFERSIZE1	File buffer size in kilobytes for Model Translator functions. A larger value may increase I/O performance but decreases available physical memory. A value between 1 and 10000 is recommended. **Note:** Overall performance may be significantly affected by this directive. It is recommended that you test a range of values since optimal settings vary with operating system and hardware.	Integer ≥ 0	1000
FILEBUFFERSIZE2	File buffer size in kilobytes for Geometry and Results Processor functions. A larger value may increase I/O performance but decreases available physical memory. A value between 1 and 10000 is recommended. **Note:** Overall performance may be significantly affected by this directive. It is recommended that you test a range of values since optimal settings vary with operating system and hardware.	Integer ≥ 0	1000
FILEBUFFERSIZE3	File buffer size in kilobytes for Solution Processor functions. A larger value may increase I/O performance but decreases available physical memory. A value between 1 to 10000 is recommended. **Note:** Overall performance may be significantly affected by this directive. It is recommended that you test a range of values since optimal settings vary with operating system and hardware.	Integer ≥ 0	1000
FILEPFACTOR1	Relative speed index for FILESPEC1 for parallel I/O operations. The fastest device should have an index of 1.0. See also NDISK.	$0.0 <$ Real ≤ 1.0	1.0
FILEPFACTOR2	Relative speed index for FILESPEC2 for parallel I/O operations. The fastest device should have an index of 1.0. See also NDISK.	$0.0 <$ Real ≤ 1.0	1.0
FILEPFACTOR3	Relative speed index for FILESPEC3 for parallel I/O operations. The fastest device should have an index of 1.0. See also NDISK.	$0.0 <$ Real ≤ 1.0	1.0
FILEPFACTOR4	Relative speed index for FILESPEC4 for parallel I/O operations. The fastest device should have an index of 1.0. See also NDISK.	$0.0 <$ Real ≤ 1.0	1.0
NFILEBUFFER	Number of file buffers for all functions. A larger value may increase I/O performance but decreases available physical memory. A value between 1 to 10 is recommended. **Note:** Overall performance may be significantly affected by this directive. It is recommended that you test a range of values since optimal settings vary with operating system and hardware.	Integer > 0	1
NFILEBUFFER1	Number of file buffers for Model Translator functions. A larger value may increase I/O performance but decreases available physical memory. A value between 1 to 10 is recommended. **Note:** Overall performance may be significantly affected by this directive. It is recommended that you test a range of values since optimal settings vary with operating system and hardware.	Integer > 0	1

File Management Directives – Miscellaneous: (Continued)

Directive	Description	Option/Type	Default
NFILEBUFFER2	Number of file buffers for Geometry and Results Processor functions. A larger value may increase I/O performance but decreases available physical memory. A value between 1 to 10 is recommended. **Note:** Overall performance may be significantly affected by this directive. It is recommended that you test a range of values since optimal settings vary with operating system and hardware.	Integer $>$ 0	1
NFILEBUFFER3	Number of file buffers for Solution Processor functions. A larger value may increase I/O performance but decreases available physical memory. A value between 1 to 10 is recommended. **Note:** Overall performance may be significantly affected by this directive. It is recommended that you test a range of values since optimal settings vary with operating system and hardware.	Integer $>$ 0	1
PURGE	Deletes all output and data files that match the current output filename. Only temporary data files are deleted when a model database is specified for the Model Input File.	ON/OFF	ON
RSLTFILEPURGE	Deletes the Femap Binary Neutral File and Model Data Output File after the Nastran Binary Results File is generated.	ON/OFF	ON
TOPTDATFILE	Topology Design Optimization Database File specification		
TOPTDEPXFILE	Topology Design Optimization Dependent Model Input File specification		
TOPTINDXFILE	Topology Design Optimization Independent Model Input File specification.		

Output Control Directives:

<table>
<tr><th>Directive</th><th>Description</th><th>Option/Type</th><th>Default</th></tr>
<tr><td>BULKDATAOUT</td><td>Case Control and Bulk Data echo in Model Results Output File.</td><td>ON/OFF</td><td>OFF</td></tr>
<tr><td>BULKDATASORT</td><td>Output Bulk Data sorting.</td><td>ON/OFF</td><td>ON</td></tr>
<tr><td>DISKSTATUS</td><td>Disk space status during critical phases of program execution.</td><td>ON/OFF</td><td>ON</td></tr>
<tr><td>ELAPSEDTIME</td><td>System Log File elapsed time output.</td><td>ON/OFF</td><td>OFF</td></tr>
<tr><td>FEMAPRSLTVECTID</td><td>Femap result vector identification numbers in Femap binary results neutral file. For full results post processing support with Femap this value should set to ON.</td><td>ON/OFF</td><td>ON</td></tr>
<tr><td>INCRRSLTOUT</td><td>Incremental results neutral file output during nonlinear analysis. When set to ON, a separate Femap binary results neutral file will be generated for each load increment or time step. At the end of the analysis a single neutral file with all steps will be generated.</td><td>ON/OFF</td><td>OFF</td></tr>
<tr><td>LEFTMARGIN</td><td>Model Results Output File left margin size in characters.</td><td>1 – 80</td><td>1</td></tr>
<tr><td>LINE</td><td>Model Results Output File lines per page. This value should correspond to the number of printed lines per page of your printer.</td><td>Integer > 0</td><td>75</td></tr>
<tr><td>MEMORYSTATUS</td><td>Available physical and virtual memory status during critical phases of program execution.</td><td>ON/OFF</td><td>ON</td></tr>
<tr><td>MODLDATAFORMAT</td><td>Expanded model data output format in Model Results Output File. See below table.
<table>
<tr><th></th><th colspan="9">MODLDATAFORMAT Setting</th></tr>
<tr><th>Data Type</th><th>0</th><th>1</th><th>2</th><th>3</th><th>4</th><th>5</th><th>6</th><th>7</th><th>8</th></tr>
<tr><td>Subcase</td><td></td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td></td><td></td><td></td></tr>
<tr><td>Coordinate Systems</td><td></td><td>✓</td><td>✓</td><td>✓</td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>Grid Definitions</td><td></td><td>✓</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>Element Definitions</td><td></td><td>✓</td><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>Element Properties</td><td></td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td></td><td>✓</td><td>✓</td><td></td></tr>
<tr><td>Material Properties</td><td></td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td>✓</td></tr>
<tr><td>Tables</td><td></td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td>✓</td><td></td><td></td></tr>
<tr><td>Loads</td><td></td><td>✓</td><td>✓</td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
<tr><td>Constraints</td><td></td><td>✓</td><td>✓</td><td></td><td></td><td></td><td></td><td></td><td></td></tr>
</table>
</td><td>1 – 8</td><td>3</td></tr>
<tr><td>MODLDATAOUT</td><td>Expanded model data output in Model Results Output File.</td><td>ON/OFF</td><td>ON</td></tr>
<tr><td>MODLINITOUT</td><td>Model Initialization File directives echo in Model Results Output File.</td><td>ON/OFF</td><td>ON</td></tr>
<tr><td>MODLSTATUS</td><td>Destination of program status information.</td><td>DISPLAY/FILE/
BOTH/NONE</td><td>DISPLAY</td></tr>
<tr><td>OUTCONTSYMBOL</td><td>Bulk Data Output File continuation symbol option. When set to ON, a continuation symbol will be used whenever a continuation entry is present.</td><td>ON/OFF</td><td>OFF</td></tr>
</table>

Output Control Directives: (Continued)

Directive	Description	Option/Type	Default
OUTDISPGEOMMODE	Specifies the subcase, mode number, or time step to be used in generating translated deformed geometry. See also TRSLDFGMDATA below.	Integer > 0	1
OUTDISPSETID	Translated enforced displacement set identification number. See also TRSLDISPDATA below.	Integer ≥ 0	100
OUTGRIDOFFSET	Specifies the starting grid point id associated with generated PLOADG entries. See also TRSLPRESDATA below.	Integer ≥ 0	100000
OUTLOADSETID	Translated force and moment set identification number. See also TRSLLOADDATA below.	Integer ≥ 0	100
OUTPAGEFORMAT	Model Results Output File page format option. When set to ON, blank lines will be added as required to position page headings correctly at the top of the page.	ON/OFF	OFF
OUTSPCSETID	Translated single point constraint set identification number. See also TRSLSPCDATA below.	Integer ≥ 0	100
OUTSTRNSETID	Translated element strain set identification number. See also TRSLSTRNDATA below.	Integer ≥ 0	100
OUTTEMPSETID	Translated grid point temperature set identification number. See also TRSLTEMPDATA below.	Integer ≥ 0	100
OUTWIDEFIELD	Option for wide field output in Bulk Data Output File generation. When set to ON, translated GRID and CORD2i Bulk Data entries will be translated in wide field format. When set to OFF, entries will be in narrow field format.	ON/OFF	ON
OUTZEROVECT	Output a zero global vector at a grid point. When set to ON, a zero vector at a grid point will be output.	ON/OFF	OFF
PCHFILEDBLEPRCS	Double precision option for Nastran ASCII Result File (Nastran punch file format). When set to ON, extends the data precision from 6 decimal places to 15.	ON/OFF	OFF
PCHFILETYPE	Punch file compatibility option. When set to NASTRAN will provide compatibility with MSC.Nastran element type codes and labels.	NASTRAN/ NORAN	NASTRAN
RSLTFILEDBLEPRCS	Double precision option for the Femap Binary Results Neutral File. When set to OFF, will use single precision data storage with extended length titles and labels. When set to ON, will use double precision data storage with standard length titles and labels. The OFF option is only compatible with Femap versions 9.3 and higher and will provide better performance and more informative results labels.	ON/OFF	OFF
RSLTFILECOMP	Results Neutral File compression option. When set to ON, will use sparse storage formatting which typically reduces disk space requirements and increases results processing performance. The AUTO setting will use sparse storage when the model contains composite laminates or SUBCOM Case Control commands.	ON/OFF AUTO	AUTO

Output Control Directives: (Continued)

Directive	Description	Option/Type	Default
RSLTFILETYPE	Results neutral file type and format. For compatibility with the Femap use FEMAPBINARY. For compatibility with Patran, Hypermesh, and I-Deas use NASTRANBINARY. For compatibility with Pro-E use NASTRANXDB. For CADAS compatibility use CADAS. **Note:** The FEMAPBINARY setting will produce a single binary results neutral file of the form *filename*.FNO generated from the NORANBINARY formatted displacement, element, and grid point results neutral files. The FEMAPASCII setting will produce a single ASCII results neutral file of the form *filename*.NEU generated from the NORANBINARY formatted displacement, element, and grid point results neutral files. The NASTRANBINARY setting will produce a single binary NASTRAN Output 2 formatted results file. The NASTRANXDB setting will produce a single binary NASTRAN XDB results database file which will permit the selective importing of results.	NORANBINARY/ NORANASCII/ PATRANBINARY/ PATRANASCII/ FEMAPBINARY/ FEMAPASCII/ NASTRANBINARY/ NASTRANXDB/ CADAS	FEMAPBINARY
RSLTLABEL	Specifies the format and location of the subcase or step label in the results neutral file system. For Femap compatibility this value should be set to 1.	1 or 4	1
SECONDS	Process time output in seconds.	ON/OFF	ON
SYSTEMSTATUS	System status at the start of program execution. The operating system, CPU type, CPU speed, and installed physical memory will be output to the System Log File.	ON/OFF	OFF
TRSLDDAMDATA	DDAM data translation option for Bulk Data Output File generation. When set to ON, will translate DDAM coefficient data into equivalent response/shock spectrum tables and output scaled mode shapes.	ON/OFF	OFF
TRSLDFGMDATA	Deformed grid point translation option for Bulk Data Output File generation. See also the Results Processor parameter, DISPGEOMSFACT, in Section 5, *Parameters*, for more information.	ON/OFF	OFF
TRSLDISPDATA	Enforced displacement translation option for Bulk Data Output File generation. When set to ON, will translate the global displacement vector into equivalent SPC Bulk Data entries. See also OUTDISPSETID above.	ON/OFF	OFF
TRSLDMIDATA	Direct matrix input data translation option for Bulk Data Output File generation.	ON/OFF	OFF
TRSLLOADDATA	Applied load translation option for Bulk Data Output File generation. When set to ON, will translate the global applied load vector into equivalent FORCE and MOMENT Bulk Data entries. See also OUTLOADSETID above.	ON/OFF	OFF
TRSLMODLDATA	Model data translation option for Bulk Data Output File generation.	ON/OFF	OFF
TRSLPRESDATA	Applied pressure load translation option for Bulk Data Output File generation. When set to ON, will translate applied surface element pressure loads (PLOAD2 and PLOAD4) on shell elements to grid point PLOADG Bulk Data entries. The OUTGRIDOFFSET directive is used to specify the starting grid point id associated with the generated PLOADG entries.	ON/OFF	OFF

Output Control Directives: (Continued)

Directive	Description	Option/Type	Default
TRSLRBSEDATA	Automatic spring element and associated grid point translation option for Bulk Data Output File generation. Applicable when the AUTOFIXRIGIDSPC model parameter is set to ON and CELAS1 elements are generated to correct improperly constrained rigid elements.	ON/OFF	OFF
TRSLSPCDATA	Automatic single point constraint translation option for Bulk Data Output File generation. See also OUTSPCSETID above.	ON/OFF	OFF
TRSLSTRNDATA	Solid and shell element strain translation option for Bulk Data Output File generation. See also OUTSTRNSETID above.	ON/OFF	OFF
TRSLTEMPDATA	Temperature data translation option for Bulk Data Output File generation. See also OUTTEMPSETID above.	ON/OFF	OFF
TRSLTOPTDATA	Topology Optimization translation option for Bulk Data Output File. This option will output all elements with a SOLID STATUS density over TOPTBTHRESHOLD	ON/OFF	OFF
TRSLTOQEDATA	Reverted tension-only quad element translation option for Bulk Data Output File generation. When set to ON, CQUAD4/CQUADR and CSHEAR element Bulk Data entries will be written out for each subcase.	ON/OFF	OFF
XYPLOTCSVOUT	MS Excel Comma Separated Variable file (.CSV) generation option when an x-y plot is requested.	ON/OFF	OFF

Memory Management Directives:

Directive	Description	Option/Type	Default
MAXRAM	Maximum amount of system memory in megabytes. This value is used to provide an upper bound when RAM is set to zero and all available physical memory is used. See also MINRAM and RAM below.	Integer ≥ 0	0
MINRAM	Minimum amount of system memory in megabytes. This value is used to provide a lower bound when RAM is set to zero and all available physical memory is used. See also MAXRAM above and RAM below.	Integer ≥ 0	0
RAM	Amount of system memory available for program operations. If an integer value is specified that value will be used and the units are megabytes. If a real value is specified then that fraction will be used based on the physical memory available at the start of program execution. The AUTO setting uses a value equal to 0.8 times the available physical memory. **Note:** If RAM is set to integer zero or real 1.0, all available physical memory will be used. This may result in either improved or degraded performance depending on the model size and available physical memory. The MAXRAM and MINRAM settings will override the RAM value specified. See also MAXRAM and MINRAM above.	0.0 < Real ≤ 1.0 Integer ≥ 0 AUTO	AUTO
RESERVEDRAM	Amount of reserved system memory in megabytes. This directive is used mostly when running in a multitasking environment such as Microsoft Windows. It directs the program memory manager to reserve the specified amount of system memory in megabytes for use by other programs. AUTO selects the value based on the amount of memory available at the start of program execution. A maximum of 500 megabytes is reserved depending on the RAM value specified and available physical memory.	Integer ≥ 0 AUTO	AUTO

Program Control Directives:

Directive	Description	Option/Type	Default
CNRMETHOD	Conditional Numerical Reproducibility (CNR) method. When set to ON program results will be more consistent and repeatable regardless of the number of processors used (NPROCESSORS directive) and how many times the same model is run repeatedly. Enabling CNR may degrade performance. The default AUTO setting enables CNR for topology optimization and nonlinear solution sequences which tend to display slight differences in results when many iterations occur in the analysis sequence.	ON/OFF/AUTO	AUTO
DECOMPMETHOD	Decomposition method: PCGLSS – Selects the parallel sparse iterative solver available in all linear and nonlinear static solutions. This solver is recommended for large problems and will generally be faster than the VSS solver. VSS – Selects the sparse direct solver available in all solutions. This solver is recommended for most problems. Significant performance degradation can occur if the RAM directive is set too low and an out of core solution is performed and/or physical memory is limited. The PCGLSS solver should be faster for these types of problems. VIS – Selects the sparse iterative solver available in all except eigenvalue solutions. If VIS solver is selected for an eigenvalue solution, the VSS solver will be used. This solver is recommended for static solutions of models consisting mostly of solid elements. It can be significantly faster that the VSS solver in some cases and uses less resources (memory and disk space). PSS – Selects the parallel sparse direct solver available in all solutions. This solver will be generally faster than the VSS solver especially on multiple CPU machines, but may require more memory. AUTO – The program picks the best method based on the RAM directive setting, material properties, model size, and solution selected in the model. See also DECOMPAUTOSIZE.	PCGLSS/ VSS/VIS/PSS/ AUTO	AUTO
DECOMPAUTOSIZE	DECOMPAUTOSIZE is the threshold model size in degrees of freedom used to select the PCGLSS over the PSS solver. DECOMPAUTOSIZE is only used when DECOMPMETHOD is set to AUTO. For very large models the PCGLSS solver is usually faster than the PSS solver, especially if there is not enough physical memory available for an in-core solution.	Integer $>$ 0	50,000
DYNRSLTMETHOD	Dynamic results calculation method. Two methods are available for the calculation of element results during modal transient and frequency response: MATRIX and DISP. Both methods will give the same results. Typically when a large number of time/frequency steps are specified versus the number of modes requested, MATRIX works best. AUTO selects the most efficient method based on the number of modes requested and the number of time/frequency steps specified.	MATRIX/ DISP/AUTO	AUTO

Program Control Directives: (Continued)

Directive	Description	Option/Type	Default
EXTRACTAUTOSIZE	EXTRACTAUTOSIZE is the threshold model size in degrees of freedom used to select the Lanczos eigensolver over the subspace eigensolver. EXTRACTAUTOSIZE is only used when EXTRACTMETHOD is set to AUTO. For very large models the PCGLSS Lanczos eigensolver is usually faster than the subspace solver, especially if there is not enough physical memory available for an in-core solution.	Integer > 0	10,000
EXTRACTMETHOD	Eigenvalue extraction method: LANCZOS – Selects the high performance PCGLSS block Lanczos eigensolver. This eigensolver is recommended for large problems and will generally be faster than the subspace eigensolver. SUBSPACE – Selects the subspace eigensolver. FEAST – Selects the FEAST eigensolver. AUTO – The program picks the best method based on the RAM directive setting and model size. See also EXTRACTAUTOSIZE.	LANCZOS/ SUBSPACE/ FEAST/ AUTO	AUTO
FEATURECODE	Updates license information by supplying a coded 20 character string to the security processor.	Character	Blank
GPWEIGHTMETHOD	Mass properties calculation method. Two methods are available for the calculation of mass properties: MATRIX and VECTOR. The MATRIX method is the most accurate, but is more time consuming and not efficient unless a coupled mass matrix formulation is requested (see the Geometry Processor parameter, COUPMASS, in Section 5, *Parameters*, for more information). AUTO selects the most efficient method based on the type of mass matrix formulated.	MATRIX/ VECTOR/ AUTO	AUTO
HEXEGRID	Hex element edge grid generation option. When HEXEGRID is set to ON, all hex elements are converted from an eight node to a 20 node configuration. This results in the generation of an additional grid point at each common element edge node. If a single point constraint exists at both adjacent corner grid points, a similar constraint will be generated for the edge grid point using the corner with the most constraint.	ON/OFF	OFF
HPFAFILESPEC	Helius PFA path specification to set the Autodesk Helius PFA binary resources location for Autodesk Nastran (helius.dll). Autodesk Helius PFA installer sets the latest Helius PFA installed location automatically during its installation	Character	
INCRGEOMOUT	Writes a separate geometry file based on the OUTGEOMFILETYPE directive setting for each topology optimization design iteration and nonlinear solution increment. When OUTGEOMFILETYPE is set to a value greater than 0 for nonlinear solutions, a non-smoothed geometry file based on the deformed geometry is generated.	ON/OFF	OFF
INMEMORYIO	Directive to store temporary files in memory or on disk. When INMEMORYIO is set to ON, the program will store the temporary files created during the analysis in memory. When INMEMORYIO is set to OFF, the temporary files will be stored on disk. Setting the directive to ON improves the performance. INMEMORYIO is set to OFF when either RESTART is set to ON or DATABASEACCEL is set to OFF.	ON/OFF AUTO	ON

Program Control Directives: (Continued)

Directive	Description	Option/Type	Default
LICENSECODE	License manager feature code string that contains a series of alphanumeric pairs defining which analysis sequence, results translation, and additional features are available. The license code is provided with the license file by customer service. Both the AUTO and DIAGNOSTIC options will locate the required license code and features for the specified model by searching all license types including a security device, external licensing, and FlexLM network and token licensing. When set to AUTO, the first valid license is selected. When set to DIAGNOSTIC, all licenses are checked and reported on with the last valid license checked being selected.	Character AUTO/DIAGNOSTIC	AUTO
LICENSEMANAGER	License manager type. The license manager type is provided with the license file by customer service.	ADLM/ FLEXLM/ DOMINO	ADLM
NDISKS	Number of physical disk drives for parallel I/O operations. A value greater than one enables parallel I/O for PCGLSS solver operations. The number of disks specified should correspond to physical devices defined using the FILESPECi and FILEPFACTORi directives. See also FILESPEC1 – FILESPEC4 and FILEPFACTOR1 – FILEPFACTOR4 above.	$0 < \text{Integer} \leq 64$	1
NPROCESSORS	Number of processors for parallel processing operations. A value greater than one enables parallel processing for PCGLSS solver operations.	$\text{Integer} > 0$	1
OPTIMIZESETTINGS	Option for modifying all Model Initialization directives to optimize SPEED, ACCURACY, or BOTH speed and accuracy. When SPEED is selected, directives are set to give the best possible performance at the cost of accuracy. When ACCURACY is selected, directives are set to give the best accuracy at the cost of speed. When BOTH is selected, directives are a compromise between speed and accuracy. When NASTRAN is selected, directives are set to provide similar accuracy and performance to other Nastran versions. Note that several initialization directives and model parameters will be reset by this single directive. See the OPTIMIZESETTINGS Directive Function Matrix below for more information.	NONE/ SPEED/ ACCURACY/ BOTH/ NASTRAN	NONE
OUTGEOMDATFILE	Output Geometry File Specification		
OUTGEOMFILETYPE	Outputs stereo lithography (.STL) geometry in topology optimization solutions. For option 0, no file is generated. For option 1, the generated STL geometry represents the outer element faces of the final design as determined by PARAM, TOPTBTHRESHOLD. For option 2, smoothed geometry is generated in the STL file.	0, 1, 2	2

Program Control Directives: (Continued)

Directive	Description	Option/Type	Default
PCGLSSDMI	When set to ON, enables DMIG support for the PCGLSS solver and LANCZOS eigensolver. The ON setting also forces six degrees of freedom per node regardless of connected element types if the Model Input File references DMIG Bulk Data entries. Typically, solid elements only require three degrees of freedom per node. When set to OFF, the PCGLSS solver and Lanczos eigensolver will not be used if the Model Input File references DMIG Bulk Data entries regardless of the DECOMPMETHOD and EXTRACTMETHOD settings.	ON/OFF	ON
PENTEGRID	Pent element edge grid generation option. When PENTEGRID is set to ON, all pent elements are converted from a six node to a 15 node configuration. This results in the generation of an additional grid point at each common element edge node. If a single point constraint exists on both adjacent corner grid points, a similar constraint will be generated for the edge grid point using the corner with the most constraint.	ON/OFF	OFF
PYREGRID	Pyr element edge grid generation option. When PYREGRID is set to ON, all pyr elements are converted from a five node to a 13 node configuration. This results in the generation of an additional grid point at each common element edge node. If a single point constraint exists on both adjacent corner grid points, a similar constraint will be generated for the edge grid point using the corner with the most constraint.	ON/OFF	OFF
QUADEGRID	Quad element edge grid generation option. When QUADEGRID is set to ON, all quad elements are converted from a four node to an eight node configuration. This results in the generation of an additional grid point at each common element edge node. If a single point constraint exists on both adjacent corner grid points, a similar constraint will be generated for the edge grid point using the corner with the most constraint.	ON/OFF	OFF
RESTART	Database restart option when a database is specified for an input file name. When RESTART is set to ON and a DATABASE file name is specified as the input file, the program will restart execution at the end of the last completed module. When RESTART is set to OFF and a DATABASE file name is specified as the input file, the program will load the DATABASE and perform a complete process control sequence.	ON/OFF	ON
RSPECDISPMETHOD	Modal summation vector results method used in response spectrum analysis for calculating vector results. . Note that the NODAL setting is required for NAVSEA 0908-LP-000-3010 conformance in DDAM analysis.	NODAL/ GLOBAL	NODAL

Program Control Directives: (Continued)

Directive	Description	Option/Type	Default
RSPECVECTMETHOD	Modal summation vector method option used in response spectrum analysis for calculating element results. When set to OFF, modal direct stresses, strains, and forces are summed and other result measures such as von Mises stress are derived from these summed values. When set to ON, all modal results measures are calculated and then summed. The ON setting may result in higher resource requirements and solution times. Note that the ON setting is required for NAVSEA 0908-LP-000-3010 conformance in DDAM analysis.	ON/OFF	ON
SHELLEGRID	Shell element edge grid generation option. When SHELLEGRID is set to ON, all shell elements are augmented with midside edge nodes. This results in the generation of an additional grid point at each common element edge node. If a single point constraint exists on both adjacent corner grid points, a similar constraint will be generated for the edge grid point using the corner with the most constraint.	ON/OFF	OFF
SOLIDEGRID	Solid element edge grid generation option. When SOLIDEGRID is set to ON, all solid elements are augmented with midside edge nodes. This results in the generation of an additional grid point at each common element edge node. If a single point constraint exists on both adjacent corner grid points, a similar constraint will be generated for the edge grid point using the corner with the most constraint.	ON/OFF	OFF

Program Control Directives: (Continued)

Directive	Description	Option/Type	Default
SOLUTION	Type of solution sequence. Available solution types depend on the license purchased. The following solution types are supported: LINEAR STATIC PRESTRESS STATIC NONLINEAR STATIC MODAL MODAL COMPLEX EIGENVALUE LINEAR PRESTRESS MODAL NONLINEAR PRESTRESS MODAL LINEAR PRESTRESS COMPLEX EIGENVALUE NONLINEAR PRESTRESS COMPLEX EIGENVALUE LINEAR BUCKLING NONLINEAR BUCKLING DIRECT FREQUENCY RESPONSE MODAL FREQUENCY RESPONSE LINEAR PRESTRESS FREQUENCY RESPONSE NONLINEAR PRESTRESS FREQUENCY RESPONSE DIRECT TRANSIENT RESPONSE MODAL TRANSIENT RESPONSE NONLINEAR TRANSIENT RESPONSE LINEAR PRESTRESS TRANSIENT RESPONSE NONLINEAR PRESTRESS TRANSIENT RESPONSE LINEAR STEADY STATE HEAT TRANSFER NONLINEAR STEADY STATE HEAT TRANSFER NONLINEAR TRANSIENT HEAT TRANSFER This directive may also be specified on the first line of the Model Input File.	License Dependent	LINEAR STATIC
TETEGRID	Tet element edge grid generation option. When TETEGRID is set to ON, all tet elements are converted from a four node to a ten node configuration. This results in the generation of an additional grid point at each common element edge node. If a single point constraint exists on both adjacent corner grid points, a similar constraint will be generated for the edge grid point using the corner with the most constraint.	ON/OFF	OFF

Program Control Directives: (Continued)

Directive	Description	Option/Type	Default
TOPTENGINE	Determines the appropriate Topology Optimization Engine to use. When set to OCM, the Optimality Criteria Method is used. When set to GOCM, the Generalized Optimality Criteria Method is used. When set to MAM, the Moving Asymptote Method is used. When set to AUTO, the solver determines the best optimization approach, based on the objective and constraint applied. The Optimality Criteria Method can provide improved performance when the objective of the topology optimization is to minimize compliance with a fixed volume fraction. The Moving Asymptote Method is a general purpose topology optimization engine that will work with any objective or constraint. It may require more design iterations and result in an increased solution time. The Generalized Optimality Criteria Method can offer improved performance over MAM for large models.	OCM, GOCM MAM, AUTO	AUTO
TOPTXPRCSMODE	When set to 1 or INTERPOLATE activates the Independent Process As a Non-Worker (IPANW) mode. The independent and dependent meshes must be voxelized with the same origin and overall dimensions but do not need to be the same mesh density. The advantage of IPANW is that the worker or FEA mesh can be a coarser size improving performance and the non-worker or optimization design space mesh can be finer improving the design quality.	ALIGN/ INTERPOLATE	ALIGN
TRIEGRID	Tri element edge grid generation option. When TRIEGRID is set to ON, all tri elements are converted from a three node to a six node configuration. This results in the generation of an additional grid point at each common element edge node. If a single point constraint exists on both adjacent corner grid points, a similar constraint will be generated for the edge grid point using the corner with the most constraint.	ON/OFF	OFF
WAITFORLICENSE	WAITFORLICENSE specifies how long to wait in seconds for a license to become available before generating a fatal error.	Integer ≥ 0	100

PROCESSCONTROL — Program Sequence Control

Description: Controls program operation by allowing selective execution of specific modules and functions.

Format:

PROCESSCONTROL(module/function) = control command

Control Command	Definition
EXECUTE	Execute module or function and continue program execution.
TERMINATE	Execute module or function and terminate program execution.
SKIP	Skip module or function and continue program execution.
HALT	Skip module or function and terminate program execution.

Module/Function	Type	Definition
CPASPRCS	Module	Component Assembly Processor Module.
DFRSPRCS	Module	Direct Frequency Response Processor Module.
DTRSPRCS	Module	Direct Transient Response Processor Module.
EIGVPRCS	Module	Eigenvalue Processor Module.
GEOMPRCS	Module	Geometry Processor Module.
INSTPRCS	Module	Initial Stress Processor Module.
MCEGPRCS	Module	Modal Complex Eigenvalue Processor Module.
MFRSPRCS	Module	Modal Frequency Response Processor Module.
MTRDPRCS	Module	Matrix Reduction Processor Module.
MTRSPRCS	Module	Modal Transient Response Processor Module.
NLINPRCS	Module	Nonlinear Static Solution Processor Module.
NLTHPRCS	Module	Nonlinear Transient Heat Solution Processor Module.
NLTRPRCS	Module	Nonlinear Transient Response Solution Processor Module.
RSLTPRCS	Module	Results Processor Module.
SEASPRCS	Module	Superelement Assembly Processor Module.
SOLNPRCS	Module	Solution Processor Module.
TRSLMODL	Module	Model Translator Module.
AASETMOD	Function	Matrix ASET reduction.
AEPSILON	Function	Solution error estimation calculation.
AFACTOR	Function	Stiffness matrix factorization.
AMLS	Function	AMLS eigenvalue extraction.
AMPCMOD	Function	Matrix multipoint constraint modification.

Module/Function	Type	Definition
AQSETMOD	Function	Matrix QSET reduction.
ASOLUTN	Function	Solution for displacement vector.
ASSEMBLA	Function	Global stiffness matrix assembly.
ASSEMBLB	Function	Global mass matrix assembly.
ASSEMBLC	Function	Global differential stiffness matrix assembly.
ASSEMBLD	Function	Prescribed non-zero SPC vector assembly.
ASSEMBLF	Function	Transient load vector assembly.
ASSEMBLG	Function	Frequency load vector assembly.
ASSEMBLH	Function	Global capacitance matrix assembly.
ASSEMBLN	Function	Nonlinear transient load vector assembly.
ASSEMBLQ	Function	Modal damping matrix assembly.
ASSEMBLR	Function	Static load vector assembly.
ASSEMBLT	Function	Global tangent stiffness matrix assembly.
ASSEMBLU	Function	Direct enforced motion transient load vector assembly.
ASSEMBLV	Function	Direct enforced motion frequency load vector assembly.
ASSEMBLW	Function	Global damping matrix assembly.
AUTOBPDB	Function	Automated global mass matrix SPC.
AUTOSPCA	Function	Automated global stiffness matrix SPC.
CNE2FASCI	Function	Femap complex ASCII results file translator.
CNE2FBIN	Function	Femap complex binary results file translator.
DASETMOD	Function	Displacement vector ASET expansion.
DMPCMOD	Function	Multipoint constraint displacement calculation.
DSOLUTN	Function	Dynamic differential equation solution.
ELEMRSLT	Function	Element and grid point results generation.
GPFRSLT	Function	Element grid point force generation.
INITEIGV	Function	Eigenvalue extraction initialization parameter determination.
INITNLND	Function	Nonlinear solution initialization parameter determination.
LANCZOS	Function	Lanczos eigenvalue extraction.
NE2FASCI	Function	Femap ASCII results file translator.
NE2FBIN	Function	Femap binary results file translator.
NE2OP2	Function	Nastran binary results file translator.
NE2XDB	Function	Nastran XDB results file translator.
PASETMOD	Function	Load vector ASET reduction.
PMPCMOD	Function	Load vector multipoint constraint modification.
RESEQ	Function	Stiffness matrix profile minimization.

Module/Function	Type	Definition
RESVECT	Function	Residual vector generator.
RMPCMOD	Function	Multipoint constraint force calculation.
RSLTLIM	Function	Results limits generation.
RSOLUTN	Function	Single point constraint force calculation.
SPCA	Function	User defined global stiffness matrix SPC.
SUBSPACE	Function	Subspace eigenvalue extraction.
UNRESEQ	Function	Unresequences model database grid point labels.
UPDTDISP	Function	Nonlinear incremental global displacement vector update.
UPDTGEOM	Function	Nonlinear geometry update.
UPDTSTRN	Function	Nonlinear stress, strain, and internal force update.

Remarks:

1. Incorrect use of this directive may produce unpredictable and erroneous results.

OPTIMIZESETTINGS Directive Function Matrix:

The matrix below depicts initialization directive and model parameter settings based on the value of the OPTIMIZESETTINGS directive.

	OPTIMIZESETTINGS Value				
Parameter/Directive	**SPEED**	**ACCURACY**	**BOTH**	**FUSION**	**NASTRAN**
ALIGNEDGENODE	OFF	ON	ON	OFF	OFF
AUTOFIXRIGIDELEM	ON	ON	ON	ON	ON
AUTOFIXRIGIDSPC	OFF	OFF	OFF	OFF	ON
BAREQVLOAD	ON	ON	ON	ON	ON
BISECT	OFF	ON	ON	OFF	ON
COUPMASS	OFF	ON	AUTO	OFF	OFF
DATABASEACCEL	ON	AUTO	AUTO	ON	AUTO
DECOMPMETHOD	AUTO	AUTO	AUTO	AUTO	AUTO
ELEMGEOMCHECKS	OFF	ON	ON	OFF	ON
ENHCBARRSLT	OFF	ON	ON	OFF	OFF
ENHCQUADRSLT	OFF	ON	ON	OFF	OFF
EXTRACTMETHOD	LANCZOS	AUTO	AUTO	LANCZOS	LANCZOS
FREQRESPRSLTOUT	OFF	ON	ON	OFF	OFF
GPFORCEMETHOD	NASTRAN	NASTRAN	NASTRAN	NASTRAN	NASTRAN
HEXINODE	AUTO	ON	AUTO	AUTO	AUTO
MAXSPARSEITER	1000	AUTO	AUTO	1000	AUTO
MODLDATAOUT	OFF	ON	ON	OFF	OFF
NBEAMINTNODE	2	4	2	2	2
NLAYERS	6	12	9	6	6
NLINSOLACCEL	AUTO	AUTO	AUTO	AUTO	AUTO
NLTOL	3	1	2	3	2
PCHFILEDBLEPRCS	OFF	ON	OFF	OFF	OFF
PCHFILETYPE	NASTRAN	NASTRAN	NASTRAN	NASTRAN	NASTRAN
QUADINODE	AUTO	ON	AUTO	AUTO	AUTO
QUADSECT	OFF	ON	ON	OFF	OFF
RANDRESPRSLTOUT	OFF	ON	ON	OFF	ON
ROTINERTIA	ON	ON	ON	ON	OFF
SHEARELEMTYPE	NASTRAN	NASTRAN	NASTRAN	NASTRAN	NASTRAN
SHELLEQVLOAD	OFF	ON	OFF	OFF	OFF
SHELLRNODE	OFF	ON	ON	OFF	OFF
SHELLTVSMATTYPE	FLEXIBLE	FLEXIBLE	FLEXIBLE	FLEXIBLE	RIGID
SKINGEN	DISABLE	SURFACE	SURFACE	DISABLE	DISABLE
SLINEMAXACTDIST	AUTO	1.0E+30	1.0E+30	AUTO	AUTO

OPTIMIZESETTINGS Directive Function Matrix (Continued):

	OPTIMIZESETTINGS Value				
Parameter/Directive	**SPEED**	**ACCURACY**	**BOTH**	**FUSION**	**NASTRAN**
SPARSEITERMETHOD	AUTO	AUTO	AUTO	AUTO	AUTO
SPARSEITERMODE	3	AUTO	AUTO	3	AUTO
SPARSEITERTOL	1.0E-4	1.0E-6	1.0E-5	1.0E-4 for Modal solutions 1.0E-5 for other solutions	1.0E-5
TETINODE	OFF	AUTO	AUTO	OFF	OFF
XYCSVPLOT	OFF	OFF	OFF	OFF	ON

DECOMPMETHOD Directive Applicability Matrix:

The matrix below depicts which initialization directives are applicable to the four linear equation solvers available. The DECOMPMETHOD directive is used to choose a particular solver.

	Solver (DECOMPMETHOD)				
Directive/Parameter	**PCGLSS (Sparse Iterative)**	**PSS (Sparse Direct)**	**PIS (Sparse Direct)**	**VSS (Sparse Direct)**	**VIS (Sparse Iterative)**
DECOMPAUTOSIZE	✓	✓	✓	✓	✓
DECOMPMETHOD	✓	✓	✓	✓	✓
MAXSPARSEITER	✓				✓
RESEQGRIDMETHOD		✓	✓	✓	
SPARSEITERTOL	✓		✓		✓
SPARSEITERMETHOD	✓				
SPARSEITERMODE	✓				
SPARSEMETHOD				✓	

Section 3

CASE CONTROL

The Case Control Section

The Case Control Section performs the following basic functions:

- Selects loads and constraints.
- Defines the contents of the Model Results Output File.
- Defines the output coordinate system for element and grid point results.
- Defines the subcase structure for the analysis.

Case Control Command Descriptions

Case Control commands may be abbreviated down to the first four characters provided the abbreviation is unique relative to all other commands. Each command is described as follows:

Description

A single sentence **Description** is given which states the function of the Case Control command.

Format

The command syntax is defined under **Format**. Listed options are further described under **Option**. The following conventions are used:

- Options in uppercase are keywords that must be specified as shown.
- Options in lowercase indicate that the user must provide a value.
- Parentheses ( ) must be included if an option requiring them is specified.
- Brackets [ ] indicate that specifying an option is not required.
- Braces { } indicate that specifying an option is required.
- If the command line is longer than 80 columns, then it may be continued to the next line with a comma. For example:

```
SET 12 =15, 16, 17, 28, 39,
        100 THRU 556
```

Example

A typical example is given under **Example**.

Option, Definition, and Type

Each option is listed under **Option** and briefly discussed under **Definition**. The option's type (e.g., Integer, Real, or Character) and allowable range are specified under **Type**. The default option is annotated with a ✓ symbol.

Remarks

Additional information about the command is given under **Remarks**.

$ Comment

Description: Used to add comments to the Model Input File.

Format:

$ followed by any characters out to column 80.

Example:

```
$ Nitrogen Tank Model Version 8.4, 17 Feb 2000
```

Remarks:

1. Comments are ignored by the program and may appear anywhere within the Model Input File.
2. Comments will not appear in either the sorted or unsorted echo of the Bulk Data.

ACCELERATION — Acceleration Output Request

Description: Requests acceleration vector output.

Format:

$$\text{ACCELERATION}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{ABS}\\ \text{REL}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

Example:

ACCELERATION = 25

Option	Definition	Type	Default
PRINT	Grid point accelerations will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point accelerations will be output only to the results neutral file system.	Character	
PUNCH	Grid point accelerations will be output additionally to the Model Results Punch File.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
ABS	Requests output as absolute displacement (see Remark 2).	Character	✓
REL	Requests output as relative displacement (see Remark 2).	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
ALL	Accelerations for all grid points will be output.	Character	✓
n	Set identification of previously appearing SET command. Only accelerations of grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point accelerations will not be output.	Character	

Remarks:

1. ACCELERATION results are output in the global coordinate system. (See CD field on the GRID Bulk Data entry in Section 4, *Bulk Data.*)

2. Relative acceleration output is only applicable to modal transient and linear direct transient response solutions. The reference point for relative motion is defaulted to the direct enforced motion input point. When direct enforced motion is not specified the point with the largest mass in the model is used. The reference point may be specified explicitly using the DYNSOLRELGRID model parameter. See Section 5, *Parameters*, for more information on DYNSOLRELGRID.

ANALYSIS — Analysis Type

Description: Specifies the type of analysis being performed.

Format:

ANALYSIS = *type*

Example:

ANALYSIS = HEAT

Option	Definition	Type	Default
STRU	Structural Analysis.	Character	✓
HEAT	Heat Transfer Analysis.	Character	
BUCK	Buckling Analysis.	Character	

Remarks:

1. ANALYSIS = HEAT must be specified for linear heat transfer solutions.

B2GG Direct Input Damping Matrix Selection

Description: Selects a direct input damping matrix.

Format:

B2GG = *name*

Example:

B2GG = BDMIG

Option	Definition	Type
name	Name of the $\left[B_{gg}^{2}\right]$ matrix that is defined on the DMIG Bulk Data entry.	Character

Remarks:

1. Direct input matrices will not be used unless selected.
2. Terms are added to the global damping matrix before any constraints are applied.
3. The matrix must be symmetric in form (field 4 on DMIG Bulk Data entry must contain the integer 6).
4. A scale factor may be applied to this input using PARAM, CB2.

B2PP — Direct Input Damping Matrix Selection

Description: Selects a direct input damping matrix.

Format:

B2PP = *name*

Example:

B2PP = BDMIG

Option	Definition	Type
name	Name of the $\left[B_{pp}^{2}\right]$ matrix that is defined on the DMIG Bulk Data entry.	Character

Remarks:

1. Direct input matrices will not be used unless selected.
2. Terms are added to the global damping matrix after constraints are applied.
3. The matrix must be square or symmetric in form (field 4 on DMIG Bulk Data entry must contain the integer 1 or 6).
4. This command is only supported in complex eigenvalue solutions.

BEGIN BULK
Case Control Delimiter

Description: Designates the end of the Case Control Section and the beginning of the Bulk Data Section.

Format:

BEGIN BULK

Remarks:

1. BEGIN BULK and ENDDATA are required even if there are no Bulk Data entries.
2. Only one occurrence of BEGIN BULK is allowed.

BOLTLD

Bolt Load Set Selection

Description: Selects the BOLTFOR Bulk Data entry for bolt preload processing.

Format:

BOLTLD = *n*

Example:

BOLTLD = 10

Option	Definition	Type
n	Set identification of BOLTFOR Bulk Data entries.	Integer > 0

Remarks:

1. BOLTFOR Bulk Data entries will not be used unless selected in the Case Control Section.
2. Bolt preloads are supported in the following solutions:

Solution Character Variable	Solution Number
LINEAR STATIC	101
LINEAR BUCKLING	105
NONLINEAR STATIC	106
DIRECT FREQUENCY RESPONSE	108
DIRECT TRANSIENT RESPONSE	109
NONLINEAR TRANSIENT RESPONSE	129
NONLINEAR BUCKLING	180
PRESTRESS STATIC	181
LINEAR PRESTRESS MODAL	182
LINEAR PRESTRESS FREQUENCY RESPONSE	183
LINEAR PRESTRESS TRANSIENT RESPONSE	184
LINEAR PRESTRESS COMPLEX EIGENVALUE	188
NONLINEAR PRESTRESS COMPLEX EIGENVALUE	189

CMETHOD — Complex Eigenvalue Extraction Method Selection

Description: Selects the complex eigenvalue extraction parameters.

Format:

CMETHOD = *n*

Example:

CMETHOD = 45

Option	Definition	Type
n	Set identification of an EIGC Bulk Data entry.	Integer > 0

Remarks:

1. The CMETHOD command must be specified in order to compute complex eigenvalues.

CONTACTGENERATE — Automated Surface Contact Generation

Description: Automated Surface Contact Generation (ASCG) and Automated Edge Contact Generation (AECG). Automatically generates surface contact/weld elements between solid or shell elements near or in contact with other solid or shell elements.

Format:

CONTACTGENERATE, *ptype, esid, sfact, fstif, mu, maxad, w0, tmax, eid*

Example:

CONTACTGENERATE, 1, , , , 0.1

Option	Definition	Type	Default
ptype	Penetration type. See Remark 1. 1 = Symmetric general contact 2 = Symmetric welded contact 3 = Symmetric bi-directional sliding contact 4 = Symmetric rough contact 5 = Offset welded contact.	$1 \le$ Integer ≤ 5	2
esid	Element set identification number. Set identification of previously appearing SET command. Only elements whose identification numbers appear on this SET command will be used.	Integer > 0	All
sfact	Stiffness scaling factor used to scale the penalty values determined automatically. See Remark 2.	Real ≥ 0.0	1.0
fstif	Frictional stiffness for stick. See Remark 3.	Real ≥ 0.0	Model dependent
mu	Coefficient of static friction.	Real ≥ 0.0	0.0
maxad	Maximum normal and radial activation distance. See Remark 4.	Real ≥ 0.0	See Remark 4
w0	Penetration surface offset. See Remark 5.	Real	0.0
tmax	Maximum allowable penetration used in the adjustment of penalty values normal to the contact plane. A positive value activates the penalty value adjustment. See Remark 6.	Real ≥ 0.0	0.0
eid	Element identification number.	Integer > 0 or blank	See Remark 8

Remarks:

1. Welded contact behavior is accomplished by selecting the welded contact setting (2). With this setting the element will behave the same in tension as in compression and will not slide. Note that for linear solutions with the LINEARCONTACT model parameter set to OFF, general contact will default to welded behavior (see Section 5, *Parameters*, for more information on LINEARCONTACT). Bi-directional sliding contact behavior is accomplished by selecting the bi-directional contact setting (3). With this setting the element will act similar to a welded contact element in tension and compression, but will slide in-plane. Bi-directional sliding contact is intended for use on planar surfaces and is available in all solutions. Rough contact behavior is accomplished by selecting the rough contact setting (4). With this setting the element will act similar to a general contact element in tension and compression, but will not permit sliding in-plane. The offset weld setting (5) is intended for welded connections with significant separation between contact surfaces. Welded contact with a separation less than the value defined by the SLINEOFFSETTOL model parameter is automatically converted to an offset weld (see Section 5, *Parameters*, for more information on SLINEOFFSETTOL).
2. *sfact* may be used to scale the penalty values that are determined automatically based on adjacent diagonal stiffness matrix coefficients. Additionally, penalty values calculated may be further scaled by the SLINEKSFACT model parameter (see Section 5, *Parameters*, for more information on SLINEKSFACT). The penalty value is then equal to $k * sfact * |\text{SLINEKSFACT}|$, where k is a value selected for each slave node based on the diagonal stiffness matrix coefficient and *sfact* is specified in the *sfact* field above. Note that the SLINEKSFACT value applies to all contact regions in the model. The use of a scale factor (*sfact* or SLINEKSFACT) less than one is recommended when convergence problems arise and a value greater than one when excessive penetration occurs. Penalty values are normally recalculated every time there is a change in stiffness. However, if SLINEKSFACT is negative, penalty values are not recalculated. This setting is generally not recommended. Note that for heat transfer solutions with the SLINEKSFACT2TC model parameter set to ON, *sfact* will be interpreted as contact capacitance (see Section 5, *Parameters*, for more information on SLINEKSFACT2TC).
3. The value of frictional stiffness should be chosen carefully. A method of choosing a value is to divide the expected frictional strength (*mu* ∗ expected normal force) by reasonable value of the relative displacement before slip occurs. A large stiffness value may cause poor convergence, while too small a value may result in reduced accuracy.
4. *maxad* is the contact surface normal and radial tolerance for generating a contact element. A recommended value is a distance approximately 10% larger than the largest gap to be recognized as contact (or welded). If *maxad* is not specified it will be internally calculated by multiplying the model reference dimension by 1.0E-04. Note that when *maxad* is specified, the SLINEOFFSETTOL model parameter will be set to this value. (See Section 5, *Parameters*, for more information on SLINEOFFSETTOL.)
5. The contact plane is defaulted to the xy-plane of the master nodes. A positive value of *w0* offsets the contact plane in the element z-direction and results in a contact condition occurring when a slave node penetrates the offset plane.
6. There are two methods for adaptive stiffness updates normal to the contact plane: proximity stiffness based and displacement based. If *tmax* ≠ 0.0, the displacement based update method is selected. When *tmax* = 0.0 (default), the proximity stiffness based update method is selected. The recommended allowable penetration *tmax* is between 1% and 10% of the element thickness for plates or the equivalent thickness for other elements that are connected to the contact element.
7. The CONTACTGEN and CONTACTTOL model parameters provide the same functionality as this command. See Section 5, *Parameters*, for more information on CONTACTGEN and CONTACTTOL.
8. The default element identification number is one plus the maximum element identification number in the model.

CONTACTSET Active Contact Set Definition

Description: Defines the active contact set.

Format:

$$\text{CONTACTSET} = \begin{Bmatrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{Bmatrix}$$

Example:

CONTACTSET = 12

Option	Definition	Type	Default
ALL	All slide line and contact surface elements will be active.	Character	✓
n	Set identification of previously appearing SET command. Only slide line and contact surface elements whose identification numbers appear on this SET command will be active.	Integer > 0	
NONE	All slide line and contact surface elements will be inactive.	Character	

Remarks:

1. This command is only applicable to nonlinear static and dynamic solutions with slide line and contact surface element types. For other element types see the ELEMSET command in Section 3, *Case Control*.

CORRELATE Modal Assurance Criterion and Cross-Orthogonality Request

Description: Requests that Modal Assurance Criterion (MAC) and Modal Cross-Orthogonality (MXO) checks be performed.

Format:

$$\text{CORRELATE}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\end{bmatrix}, \begin{bmatrix}\text{MAC}\\\text{MXO}\\\text{MALL}\end{bmatrix}, [\text{CTOL}\ \textit{value}]\right)\right] = \begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

CORRELATE(PLOT, MAC, CTOL 0.1) = ALL

Option	Definition	Type	Default
PRINT	Modal assurance data will be output to both the Model Results Output File and displayed graphically in the Autodesk Nastran Editor.	Character	✓
PLOT	Modal assurance data will be output only to the Autodesk Nastran Editor.	Character	
MAC	Modal assurance criterion data output request.	Character	
MXO	Modal cross-orthogonality data output request.	Character	
MALL	Both MAC and MXO will be output.	Character	
CTOL	Off-diagonal output tolerance. See Remark 2.	Real $\geq$ 0.0	0.0
ALL	Modal assurance data for all modes will be output.	Character	
n	Set identification of previously appearing SET command. Only modal assurance data for modes whose identification numbers appear on this SET command will be output.	Integer $>$ 0	
NONE	Modal assurance data will not be output.	Character	✓

Remarks:

1. This command is used to compare modal data from two different sources defined using the DATINFILE1 and DATINFILE2 Model Initialization directives. See Section 2, *Initialization*, for more information on DATINFILE1 and DATINFILE2.
2. Output of off-diagonal Modal Assurance Criterion (MAC) and Modal Cross-Orthogonality (MXO) matrix terms will be suppressed if less than the output tolerance, CTOL.

CYSYMGENERATE — Cyclic Symmetry Boundary Condition Generation

Description: Defines parameters for automatic cyclic symmetry boundary condition generation on an axisymmetric model.

Format:

CYSYMGEN, *cid*, *ptol, type*

Example:

CYSYMGEN, 12, 1.-6

Option	Definition	Type	Default
cid	Reference cylindrical coordinate system id that matches a CORD1C or CORD2C Bulk Data entry.	Integer > 0	Required
ptol	Near tolerance used to identify boundary grid points for the application of cyclic symmetric boundary conditions.	Real or blank	1.0E-10
type	If mesh on cyclic boundaries is mapped, use SYMMETRIC. If mesh on cyclic boundaries is unmapped use UNSYMMETRIC	SYMMETRIC/ UNSYMMETRIC	SYMMETRIC

Remarks:

1. When set to a valid cylindrical coordinate system id, boundary conditions are automatically generated which force cyclic symmetric behavior. Grid points are automatically identified at each r-z boundary plane based on the specified near tolerance, *ptol*. The two symmetry planes must have identical mesh patterns.
2. The near tolerance is used to identify boundary grid points for the application of cyclic symmetric boundary conditions. The actual tolerance is derived using *ptol* and a model reference dimension. Each r-z boundary is identified as all grid points within this tolerance at the minimum and maximum θ values of the model.
3. The CYSYMGEN and CYSYMTOL model parameters provide the same functionality as this command. See Section 5, *Parameters*, for more information on CYSYMGEN and CYSYMTOL.

DDAM — Dynamic Design Analysis Method Data Set Selection

Description: Selects the DDAMDAT Bulk Data entry to be used in the DDAM analysis. DDAM is a form of response spectrum analysis.

Format:

DDAM = *n*

Example:

DDAM = 12

Option	Definition	Type
n	Set identification of a DDAMDAT Bulk Data entry to be used in DDAM analysis.	Integer > 0

Remarks:

1. DDAM must reference a DDAMDAT Bulk Data entry to perform DDAM analysis.

DEFORM Element Deformation Static Load

Description: Selects the Element Deformation Set to be applied to the model.

Format:

DEFORM = *n*

Example:

DEFORM = 27

Option	Definition	Type
n	Set identification of DEFORM Bulk Data entries.	Integer > 0

Remarks:

1. DEFORM Bulk Data entries will not be used unless selected in the Case Control Section.
2. The total load applied will be the sum of external (LOAD command), element deformation (DEFORM command), constrained displacement (SPC command), and thermal (TEMPERATURE command) loads.
3. Static, thermal, and element deformation loads should have unique set identification numbers.

DISPINTERPOLATE — Enforced Displacement Interpolation

Description: Interpolates grid point enforced displacement data from a known set of input grid points and displacements to a set of output grid points and displacements based on geometric position in 2d or 3d space.

Format:

DISPINTERPOLATE, *ossid*, *ogsid*, *issid*, *igsid, nnri, ndlsf, cgsize, maxnus*

Example:

DISPINTERPOLATE, 100, 10, 1, 1

Option	Definition	Type	Default
olsid	Output single-point constraint set identification number (see Remark 1).	Integer > 0	Required
ogsid	Output grid set identification number. Set identification of previously appearing SET command. Only grid points whose identification numbers appear on this SET command will be used.	Integer > 0 or blank	All grid points in model
issid	Input single-point constraint set identification number (see Remark 2).	Integer > 0	Required
igsid	Input grid set identification number. Set identification of previously appearing SET command. Only grid points whose identification numbers appear on this SET command will be used.	Integer > 0 or blank	All grid points in constraint set
nnri	Number of interpolation nodes within radius of influence.	Integer > 0 or blank	See Remark 3
ndlsf	Number of data nodes in least squares fit.	Integer > 0 or blank	See Remark 4
cgsize	Number of rows, columns, and planes in the cell grid. A box containing the nodes is partitioned into cells in order to increase search efficiency.	Integer > 0 or blank	See Remark 5
maxnus	Maximum number of unique solution occurrences.	Integer > 0 or blank	See Remark 6

Remarks:

1. Output is SPC Bulk Data entries at grid points defined by *ogsid*.
2. Input is GRID and SPC Bulk Data entries which need not be associated with the analysis model. (See Section 4, *Bulk Data*, for more information on GRID and SPC Bulk Data entries.)
3. The valid range for *nnri* is $1 \leqslant nnri \leqslant \min(100, n-1)$, where *n* is the number of input data points. The default is 100. A lower value may increase performance at the cost of accuracy. A value greater than or equal to 32 is recommended.

4. The valid range for *ndlsf* is $9 \leqslant ndlsf \leqslant \min(100, n-1)$, where *n* is the number of input data points. The default is 100. A lower value may increase performance at the cost of accuracy. A value greater than or equal to 17 is recommended.
5. The recommended value for *cgsize* is:

$$cgsize = \left(\frac{n}{3}\right)^{\frac{1}{3}}$$

   where *n* is the number of input data points. The default is determined using the above formula.
6. A 3d interpolation algorithm is used initially, but will automatically revert to a 2d algorithm if the number of no unique solution errors exceeds *maxnus* while processing the input data points. Models that are dominantly flat but still have 3d features that default to the 2d interpolation algorithm may not be interpolated accurately. A larger *maxnus* value can be used to force a 3d interpolation. It is advisable to always check the interpolated loads.
7. Generated SPC Bulk Data entries can be exported using the TRSLBULKDATA Model Initialization directive. (See Section 2, *Initialization*, for more information on TRSLBULKDATA.)

DISPLACEMENT

Displacement Output Request

Description: Requests displacement vector output.

Format:

$$\text{DISPLACEMENT}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\\text{PHASE}\end{bmatrix},\begin{bmatrix}\text{ABS}\\\text{REL}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\\text{ATOC}\\\text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\\text{NONE}\end{Bmatrix}$$

Example:

DISPLACEMENT = ALL

Option	Definition	Type	Default
PRINT	Grid point displacements will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point displacements will be output only to the results neutral file system.	Character	
PUNCH	Grid point displacements will be output additionally to the Model Results Punch File.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ABS	Requests output as absolute displacement (see Remark 3).	Character	✓
REL	Requests output as relative displacement (see Remark 3).	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
ALL	Displacements for all grid points will be output.	Character	✓
n	Set identification of previously appearing SET command. Only displacements of grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point displacements will not be output.	Character	

Remarks:

1. DISPLACEMENT results are output in the global coordinate system. (See CD field on the GRID Bulk Data entry in Section 4, *Bulk Data.*)
2. The translation components are in the same units of measure as the model. The rotation components are in radians.

3. Relative displacement output is only applicable to modal transient and linear direct transient response solutions. The reference point for relative motion is defaulted to the direct enforced motion input point. When direct enforced motion is not specified the point with the largest mass in the model is used. The reference point may be specified explicitly using the DYNSOLRELGRID model parameter. See Section 5, *Parameters*, for more information on DYNSOLRELGRID.

DLOAD — Dynamic Load Set Selection

Description: Selects a dynamic load to be applied in a transient or frequency response problem.

Format:

DLOAD = *n*

Example:

DLOAD = 10

Option	Definition	Type
n	Set identification of a DLOAD, RLOAD1, RLOAD2, TLOAD1, or TLOAD2 Bulk Data entry.	Integer > 0

Remarks:

1. TLOAD1 and TLOAD2 may only be selected in a transient response problem.
2. RLOAD1 and RLOAD2 may only be selected in a frequency response problem.

DMIGADD — DMIG Combination

Description: Combines multiple DMIG matrixes referenced in the Bulk Data for selection in the Case Control using K2GG, K2PP, M2GG, etc. commands.

Format:

$$\text{DMIGADD } \mathit{name} = \left\{ \mathit{name}1 \left[, \ \mathit{name}2, \ \mathit{name}3 \right] \right\}$$

Example:

DMIGADD KALL = K1, K2, K3, K4

Option	Definition	Type	Default
name	The name of the combined DMIG matrix. See Remark 1.	Character	Required
name1, name2, etc.	The names of existing DMIG matrixes. See Remark 2.	Character	Required

Remarks:

1. The combined name should be unique with respect to all other DMIG names.
2. This command may not refer to a DMIG name generated from another DMIGADD command.

ECHO — Bulk Data Echo Request

Description: Requests echo of the Bulk Data.

Format:

$$\text{ECHO} = \left\{ \begin{matrix} \text{SORT} \\ \text{UNSORT} \\ \text{NONE} \end{matrix} \right\}$$

Example:

ECHO = NONE

Option	Definition	Type	Default
SORT	Sorted echo will be output.	Character	
UNSORT	Unsorted echo will be output.	Character	
NONE	No echo will be output.	Character	✓

Remarks:

1. Default is to not echo the Bulk Data.
2. A translated Case Control and Bulk Data output file can be requested with the Initialization Directive, TRSLMODLDATA = ON. See Section 2, *Initialization*.
3. This command is equivalent to the Initialization Directive, BULKDATAOUT. See Section 2, *Initialization*.

ELEMDELETE — Model Database Element Deletion

Description: Deletes elements in the specified set from the model database.

Format:

ELEMDELETE = *n*

Example:

ELEMDELETE = 21

Option	Definition	Type
n	Set identification of previously appearing SET command. Only elements whose identification numbers appear on this SET command will be deleted.	Integer > 0

Remarks:

1. This command can be used along with the SETGENERATE Case Control command and the RCN option to delete elements that have a result quantity obtained in a previous run which is above a threshold value. (See the SETGENERATE command in Section 3, *Case Control.*)

ELEMSET — Active Element Set Definition

Description: Defines the active element set.

Format:

$$\text{ELEMSET} = \begin{Bmatrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{Bmatrix}$$

Example:

ELEMSET = 15

Option	Definition	Type	Default
ALL	All structural elements will be active.	Character	✓
n	Set identification of previously appearing SET command. Only elements whose identification numbers appear on this SET command will be active.	Integer > 0	
NONE	All structural elements will be inactive.	Character	

Remarks:

1. This command is only applicable to nonlinear static and dynamic solutions excluding slide line and contact surface element types. For these element types see the CONTACTSET command in Section 3, *Case Control*.

ELFORCE — Element Force Output Request

Description: Requests element force output.

Format:

$$\text{ELFORCE}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{CENTER}\\ \text{CORNER}\\ \text{GAUSS}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

ELFORCE = ALL

Option	Definition	Type	Default
PRINT	Element forces will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Element forces will be output only to the results neutral file system.	Character	
PUNCH	Element forces will be output additionally to the Model Results Punch File.	Character	
CENTER	Output shell and solid element forces at the center only.	Character	✓
CORNER	Output shell and solid element forces at the center and corner nodes.	Character	
GAUSS	Output shell and solid element forces at the center and gauss/integration points.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
ALL	Element forces for all elements will be output.	Character	
n	Set identification of previously appearing SET command. Only forces for elements whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Element forces will not be output.	Character	✓

Remarks:

1. FORCE is an alternate form and is entirely equivalent to ELFORCE.

2. Not available for solid elements.
3. Shell elements must be referenced on a SURFACE. (See the SURFACE command in Section 3, *Case Control.*)

ELSTRAIN — Element Strain Output Request

Description: Requests element strain output.

Format:

$$\text{ELSTRAIN}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{CENTER}\\ \text{CORNER}\\ \text{GAUSS}\end{bmatrix},\begin{bmatrix}\text{SHEAR}\\ \text{VONMISES}\\ \text{TRESCA}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{STRCUR}\\ \text{FIBER}\end{bmatrix},\begin{bmatrix}\text{THERMAL}\\ \text{MECH}\\ \text{TOTAL}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{RALL}\end{bmatrix},\begin{bmatrix}\text{VRMS}\\ \text{BIAX}\\ \text{VALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

ELSTRAIN(VONMISES, CORNER) = 45

Option	Definition	Type	Default
PRINT	Element strains will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Element strains will be output only to the results neutral file system.	Character	
PUNCH	Element strains will be output additionally to the Model Results Punch File.	Character	
CENTER	Output shell and solid element strains at the center only.	Character	✓
CORNER	Output shell and solid element strains at the center and corner nodes.	Character	
GAUSS	Output shell and solid element strains at the center and gauss/integration points.	Character	
SHEAR	Maximum shear strain request for shell elements and octahedral shear strain request for solid elements.	Character	
VONMISES	Von Mises strain request for shell and solid elements.	Character	✓
TRESCA	Tresca strain request for shell and solid elements.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
STRCUR	Strain at reference plane and curvatures are output for shell elements.	Character	
FIBER	Strain at locations Z1 and Z2 are output for shell elements.	Character	✓
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
VRMS	RMS von Mises output request.	Character	

Option	Definition	Type	Default
BIAX	Biaxiality ratio output request.	Character	
VALL	RMS von Mises, RMS principal, RMS maximum shear, and biaxiality ratio will be output.	Character	
THERMAL	Thermal strain request for shell and solid elements.	Character	
MECH	Mechanical strain request for shell and solid elements.	Character	
TOTAL	Total strain (thermal plus mechanical) request for shell and solid elements.	Character	✓
ALL	Element strains for all elements will be output.	Character	
n	Set identification of previously appearing SET command. Only strains for elements whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Element strains will not be output.	Character	✓

Remarks:

1. STRAIN is an alternate form and is entirely equivalent to ELSTRAIN.
2. Both STRESS and STRAIN cannot be requested in the same subcase.
3. Shell elements must be referenced on a SURFACE and solid elements must be referenced in a VOLUME. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)
4. See the STRAIN command in Section 3, *Case Control,* for the equations used to calculate strain invariants.
5. If the MECHSTRAIN model parameter is set to ON (default is OFF), mechanical strain will be output regardless of settings on this command. (See Section 5, *Parameters*, for more information on MECHSTRAIN.)

ELSTRESS — Element Stress Output Request

Description: Requests element stress output.

Format:

$$\text{ELSTRESS}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{CENTER}\\\text{CORNER}\\\text{GAUSS}\end{bmatrix},\begin{bmatrix}\text{SHEAR}\\\text{VONMISES}\\\text{TRESCA}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\\text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\\text{ATOC}\\\text{RALL}\end{bmatrix},\begin{bmatrix}\text{VRMS}\\\text{BIAX}\\\text{VALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\\text{NONE}\end{Bmatrix}$$

Example:

ELSTRESS(CORNER, SHEAR) = ALL

Option	Definition	Type	Default
PRINT	Element stresses will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Element stresses will be output only to the results neutral file system.	Character	
PUNCH	Element stresses will be output additionally to the Model Results Punch File.	Character	
CENTER	Output shell and solid element stresses at the center only.	Character	✓
CORNER	Output shell and solid element stresses at the center and corner nodes.	Character	
GAUSS	Output shell and solid element stresses at the center and gauss/integration points.	Character	
SHEAR	Maximum shear stress request for shell elements and octahedral shear stress request for solid elements.	Character	
VONMISES	Von Mises stress request for shell and solid elements.	Character	✓
TRESCA	Tresca stress request for shell and solid elements.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
VRMS	RMS von Mises output request.	Character	

Option	Definition	Type	Default
BIAX	Biaxiality ratio output request.	Character	
VALL	RMS von Mises, RMS principal, RMS maximum shear, and biaxiality ratio will be output.	Character	
ALL	Element stresses for all elements will be output.	Character	
n	Set identification of previously appearing SET command. Only stresses for elements whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Element stresses will not be output.	Character	✓

Remarks:

1. STRESS is an alternate form and is entirely equivalent to ELSTRESS.
2. Both STRESS and STRAIN cannot be output in the same subcase.
3. Shell elements must be referenced on a SURFACE and solid elements must be referenced in a VOLUME. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)
4. See the STRESS command in Section 3, *Case Control,* for the equations used to calculate stress invariants.
5. VRMS, von Mises RMS stress, is calculated by evaluating the PSD response of the peak RMS stresses calculated at each frequency step in a frequency or random response analysis. It is used as a measure of the total component stress.
6. BIAX, Biaxiality Ratio, is the ratio of the minimum and maximum principal stress and is used in conjunction with the von Mises RMS stress to assess the nature of stress components in a frequency or random response analysis. Values that tend towards -1 indicates a pure shear state, 0 indicates uniaxial state, and 1 indicates equal biaxial loading.

ENTHALPY — Heat Transfer Enthalpy Output Request

Description: Requests enthalpy vector output in transient heat transfer analysis.

Format:

$$\text{ENTHALPY}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\n\\\text{NONE}\end{Bmatrix}$$

Example:

ENTHLAPY = 10

Option	Definition	Type	Default
PRINT	Grid point enthalpy will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point enthalpy will be output only to the results neutral file system.	Character	
PUNCH	Grid point enthalpy will be output additionally to the Model Results Punch File.	Character	
ALL	Enthalpy for all grid points will be output.	Character	
n	Set identification of previously appearing SET command. Only enthalpy for grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point enthalpy will not be output.	Character	✓

Remarks:

1. ENTHALPY = NONE is used to override a previous ENTHALPY = n or ENTHALPY = ALL command.

ESE — Element Strain Energy Output Request

Description: Requests element strain energy output.

Format:

$$ESE\left[\left(\begin{bmatrix}PRINT\\PLOT\\PUNCH\end{bmatrix}\right)\right]=\begin{Bmatrix}ALL\\n\\NONE\end{Bmatrix}$$

Example:

ESE = ALL

Option	Definition	Type	Default
PRINT	Element strain energy will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Element strain energy will be output only to the results neutral file system.	Character	
PUNCH	Element strain energy will be output additionally to the Model Results Punch File.	Character	
ALL	Strain energy for all elements will be output.	Character	
n	Set identification number of a previously appearing SET command. Only elements whose identification numbers appear on this SET command will be included in the element strain energy output.	Integer > 0	
NONE	Element strain energy will not be output.	Character	✓

Remarks:

1. The strain energy calculations do not include the contribution of thermal strain.
2. Strain energy density (element strain energy divided by element volume) is also computed.
3. Shell elements must be referenced on a SURFACE and solid elements must be referenced in a VOLUME. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)

EXTSEOUT — Superelement Matrix Export

Description: Specifies the type, format, and media for superelement data storage.

Format:

EXTSEOUT [([STIF],[MASS],[DAMP],[LOAD],[MODEL],[DMIGOUT],[DMIGBDF],[DMIGOP2],[DMIGSFIX *name*])]

Examples:

EXTSEOUT(STIF, MASS, DMIGBDF)

EXTSEOUT(MASS, DMIGOP2)

Option	Definition	Type	Default
STIF	Include global stiffness matrix output.	Character	✓
MASS	Include global mass matrix output.	Character	✓
DAMP	Include global damping matrix output.	Character	✓
LOAD	Include global load vector output.	Character	✓
MODEL	Requests model data translation to the Bulk Data Output File.	Character	
DMIGOUT	Requests global matrix output to the Model Results Output File.	Character	
DMIGBDF	Requests global matrix export in DMIG format to the Bulk Data Output File.	Character	
DMIGOP2	Requests global matrix export to a NASTRAN Output 2 formatted results file.	Character	
DMIGSFIX	Matrix name. Specifies the name field in the exported DMIG Bulk Data entry. See Remark 4.	Character	

Remarks:

1. If no matrix type is specified all matrixes will be exported.
2. If multipoint constraints or RBEi elements are included in the model the exported matrixes will be modified. If ASET or QSET reduction is performed the exported matrixes will be reduced.
3. The GLBMATRIX command provides additional options for matrix output to the Model Results Output File. (See the GLBMATRIX command in Section 3, *Case Control.*)
4. The exported DMIG matrix name is generated by concatenating the matrix type with the DMIGSFIX name where the boundary stiffness matrix name becomes K*cccccc*, the mass M*cccccc*, the damping B*cccccc*, the load P*cccccc*, and *cccccc* the name specified after DMIGSFIX. DMIGSFIX is only applicable when DMIGBDF is also specified.

FATIGUE — Multiaxial Fatigue Analysis Data Set Selection

Description: Selects the FATIGUE Bulk Data entry to be used in multiaxial fatigue analysis.

Format:

FATIGUE = *n*

Example:

FATIGUE = 10

Option	Definition	Type
n	Set identification of a FATIGUE Bulk Data entry to be used in multiaxial fatigue analysis.	Integer > 0

Remarks:

1. FATIGUE must reference a FATIGUE Bulk Data entry to perform multiaxial fatigue analysis.

FLUX Element Thermal Gradient and Heat Flux Output Request

Description: Requests element thermal gradient and heat flux output in heat transfer analysis.

Format:

$$\text{FLUX}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{CENTER}\\\text{CORNER}\\\text{GAUSS}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

FLUX = ALL

Option	Definition	Type	Default
PRINT	Element thermal gradients and heat fluxes will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Element thermal gradients and heat fluxes will be output only to the results neutral file system.	Character	
PUNCH	Element thermal gradients and heat fluxes will be output additionally to the Model Results Punch File.	Character	
CENTER	Output thermal gradients and heat fluxes at the center only.	Character	✓
CORNER	Output thermal gradients and heat fluxes at the center and corner nodes.	Character	
GAUSS	Output thermal gradients and heat fluxes at the center and gauss/integration points.	Character	
ALL	Thermal gradients and heat fluxes for all elements will be output.	Character	
n	Set identification number of a previously appearing SET command. Only gradients and fluxes for elements whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Element thermal gradient and heat flux will not be output.	Character	✓

Remarks:

1. Shell elements must be referenced on a SURFACE and solid elements must be referenced in a VOLUME. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)

FORCE — Element Force Output Request

Description: Requests element force output.

Format:

$$\text{FORCE}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{CENTER}\\ \text{CORNER}\\ \text{GAUSS}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

FORCE = ALL

Option	Definition	Type	Default
PRINT	Element forces will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Element forces will be output only to the results neutral file system.	Character	
PUNCH	Element forces will be output additionally to the Model Results Punch File.	Character	
CENTER	Output shell and solid element forces at the center only.	Character	✓
CORNER	Output shell and solid element forces at the center and corner nodes.	Character	
GAUSS	Output shell and solid element forces at the center and gauss/integration points.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
ALL	Element forces for all elements will be output.	Character	
n	Set identification of previously appearing SET command. Only forces for elements whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Element forces will not be output.	Character	✓

Remarks:

1. ELFORCE is an alternate form and is identical to FORCE.
2. Not available for solid elements.
3. Shell elements must be referenced on a SURFACE. (See the SURFACE command in Section 3, *Case Control.*)

FREQUENCY

Frequency Set Selection

Description: Selects the set of solution frequencies to be solved in frequency response problems.

Format:

FREQUENCY = *n*

Example:

FREQUENCY = 20

Option	Definition	Type
n	Set identification number of FREQ, FREQ1, FREQ2 FREQ3, FREQ4 Bulk Data entries.	Integer > 0

Remarks:

1. One or more FREQi entries must be selected to perform frequency response analysis.
2. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored. Two frequencies are considered duplicated if

$$|f_i - f_{i-1}| < \text{DFREQ} * |f_{MAX} - f_{MIN}|$$

where DFREQ is defaulted to 1.0E-5 and f_{MAX} and f_{MIN} are the maximum and minimum solution frequencies of the combined FREQi entries. (See Section 5, *Parameters*, for more information on DFREQ.)

GEOMCHECK — Geometry Check Options

Description: Specifies tolerance values and options for element geometry checks.

Format:

$$\text{GEOMCHECK}\left[\textit{testtype} = \textit{tol}\right], \left[\text{MSGLIMIT} = n\right], \left[\text{MSGTYPE} = \begin{matrix}\text{WARNING}\\ \text{FATAL}\end{matrix}\right], \left[\text{SUMMARY}\right]$$

Examples:

Set the tolerance for the CQUAD4 element skew angle test to 30.0 degrees and limit element geometry warning/fatal error messages to 100:

GEOMCHECK Q4_SKEW = 30.0, MSGLIMIT = 100

Set the message type to fatal for CQUAD4 element taper tests:

GEOMCHECK Q4_TAPER, MSGLIMIT = FATAL

Request summary table output using default tolerance values:

GEOMCHECK SUMMARY

Option	Definition	Type	Default
testtype	Element geometry test type: One of the character variables shown in Remark 3.	Character	
tol	Tolerance value for the specified *testtype*.	Real $>$ 0.0	See Remark 3
n	Maximum number of element geometry warning/fatal error messages. See Remark 4.	Integer	See Remark 4
FATAL	Geometry tests that exceed tolerance values produce fatal error messages.	Character	
WARN	Geometry tests that exceed tolerance values produce warning messages.	Character	✓
SUMMARY	Option to output individual element geometry statistics. See Remark 5.	Character	

Remarks:

1. The GEOMCHECK command combines several Geometry Processor Parameters which control element geometry check tolerances, the number and severity of associated warning and fatal error messages, and output of additional tabular summary information. Multiple GEOMCHECK statements may be present.

2. Autodesk Nastran performs a number of element checks internally for every analysis. These are done to identify elements that can potentially cause numerical issues, such as singularities. The GEOMCHECK element checks are included to help a user identify elements that will potentially give bad results. Virtually all the distortions that are checked can cause elements to be too stiff (as compared with ideal elements). Additionally, the extrapolation of calculated results from Gauss points to corner nodes is adversely affected in distorted elements. If only centroid output is requested, many of the checks can be relaxed, but highly distorted elements may still be too stiff. The default values for the checks represent limits beyond which the element results may be compromised significantly.

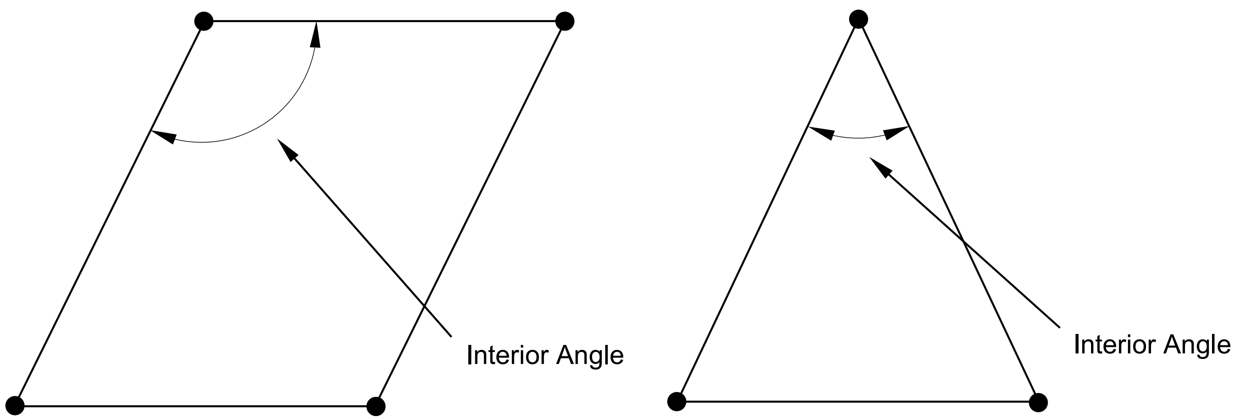
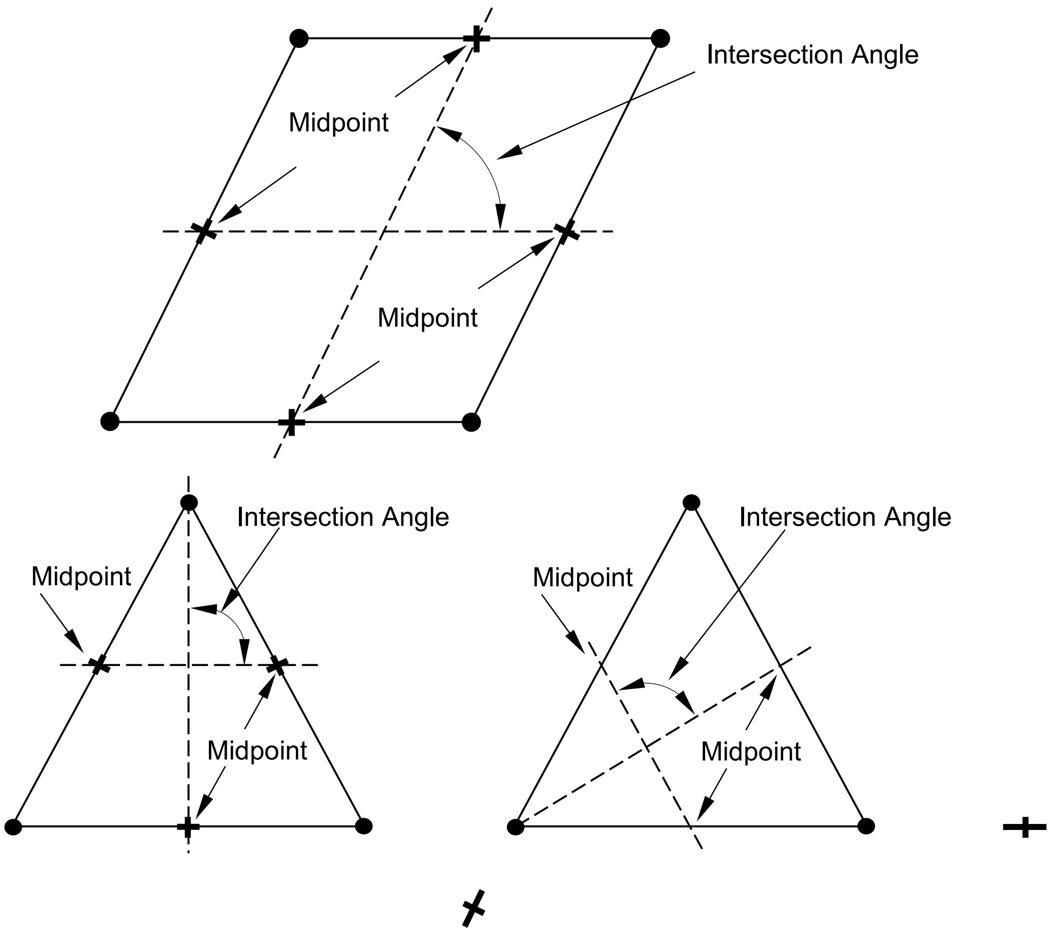
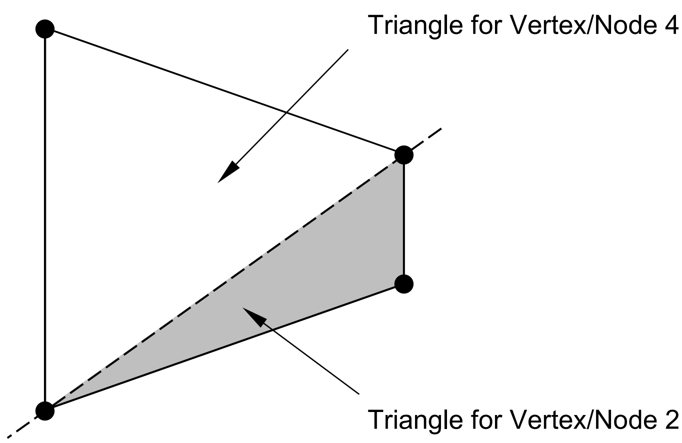


Figure 1. Interior Angle Check.



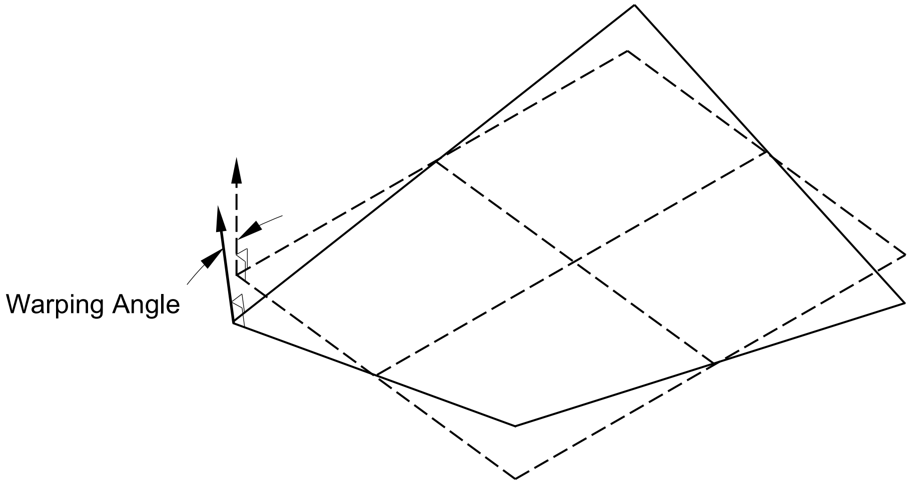
$$\text{Skew Angle} = \left|\text{Max Intersection Angle} - 90°\right|$$

Figure 2. Skew Angle Check.



$$\text{TaperRatio} = 2 * Area_{\text{tri}} / Area_{\text{quad}} - 1$$

Figure 3. Taper Ratio Check.



Warping Angle = Max Element Corner Normal Angular Deviation from Normal of Mean Plane

Figure 4. Warping Angle Check.

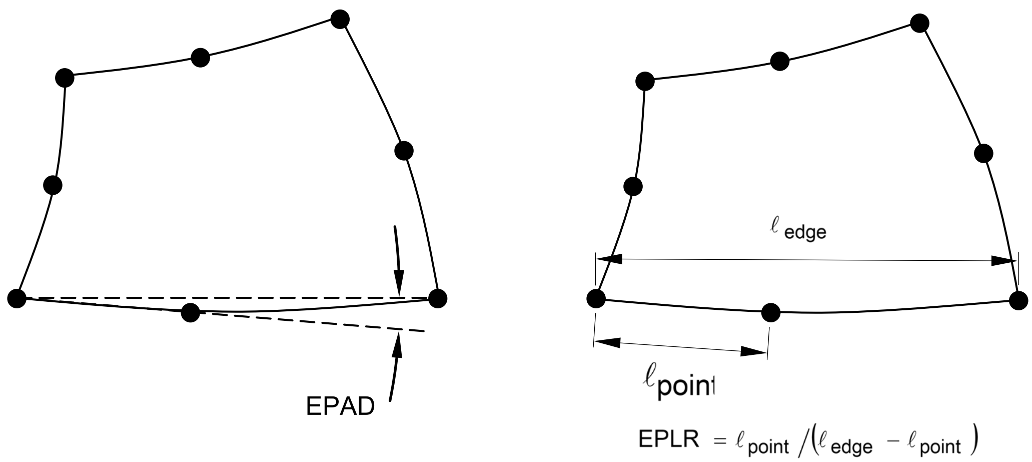


Figure 5. Edge-Point Angular Deviation and Length Ratio Checks.

(Continued)

3. The following table lists the *testtype* character variable options and the associated model parameter which may also be used to change the default setting.

***Testtype* Character Variable**	**Equivalent Model Parameter**	**Description**	**Default Value**
HEX_AR	HEXARTOL	Hex element aspect ratio.	100.0
HEX_IAMAX	HEXFACEMAXIATOL	Hex element face maximum interior angle (degrees).	165.0
HEX_IAMIN	HEXFACEMINIATOL	Hex element face minimum interior angle (degrees).	25.0
HEX_SKEW	HEXFACESKEWTOL	Hex element face skew angle (degrees).	65.0
HEX_TAPER	HEXFACETAPERTOL	Hex element face taper ratio.	0.75
HEX_WARP	HEXFACEWARPTOL	Hex element face warping angle (degrees).	45.0
HEX_EPAD	HEXMAXEPADTOL	Hex element maximum edge-point angular deviation (degrees).	30.0
HEX_EPLR	HEXMINEPLRTOL	Hex element minimum edge-point length ratio.	0.5
PENT_IAMAX	PENTFACEMAXIATOL	Pent element face maximum interior angle (degrees).	165.0
PENT_IAMIN	PENTFACEMINIATOL	Pent element face minimum interior angle (degrees).	25.0
PENT_SKEW	PENTFACESKEWTOL	Pent element face skew angle (degrees).	65.0
PENT_TAPER	PENTFACETAPERTOL	Pent element face taper ratio.	0.75
PENT_WARP	PENTFACEWARPTOL	Pent element face warping angle (degrees).	45.0
PENT_EPAD	PENTMAXEPADTOL	Pent element maximum edge-point angular deviation (degrees).	30.0
PENT_EPLR	PENTMINEPLRTOL	Pent element minimum edge-point length ratio.	0.5
PYR_IAMAX	PYRFACEMAXIATOL	Pyr element face maximum interior angle (degrees).	165.0
PYR_IAMIN	PYRFACEMINIATOL	Pyr element face minimum interior angle (degrees).	25.0
PYR_SKEW	PYRFACESKEWTOL	Pyr element face skew angle (degrees).	65.0
PYR_TAPER	PYRFACETAPERTOL	Pyr element face taper ratio.	0.75
PYR_WARP	PYRFACEWARPTOL	Pyr element face warping angle (degrees).	45.0
PYR_IAMAX	PYRFACEMAXIATOL	Pyr element face maximum interior angle (degrees).	165.0
PYR_EPAD	PYRMAXEPADTOL	Pyr element maximum edge-point angular deviation (degrees).	30.0
PYR_EPLR	PYRMINEPLRTOL	Pyr element minimum edge-point length ratio.	0.5
Q4_AR	QUADARTOL	Quad element aspect ratio.	100.0
Q4_EPAD	QUADMAXEPADTOL	Quad element maximum edge-point angular deviation (degrees).	30.0
Q4_IAMAX	QUADMAXIATOL	Quad element maximum interior angle (degrees).	165.0
Q4_EPLR	QUADMINEPLRTOL	Quad element minimum edge-point length ratio.	0.5
Q4_IAMIN	QUADMINIATOL	Quad element minimum interior angle (degrees).	25.0
Q4_SKEW	QUADSKEWTOL	Quad element skew angle (degrees).	65.0
Q4_TAPER	QUADTAPERTOL	Quad element taper ratio.	0.75
Q4_WARP	QUADWARPTOL	Quad element warping angle (degrees).	45.0

Testtype Character Variable	Equivalent Model Parameter	Description	Default Value
TET_AR	TETARTOL	Tet element aspect ratio.	100.0
TET_IAMAX	TETFACEMAXIATOL	Tet element face maximum interior angle (degrees).	170.0
TET_IAMIN	TETFACEMINIATOL	Tet element face minimum interior angle (degrees).	5.0
TET_SKEW	TETFACESKEWTOL	Tet element face skew angle (degrees).	80.0
TET_EPAD	TETMAXEPADTOL	Tet element maximum edge-point angular deviation (degrees).	30.0
TET_EPLR	TETMINEPLRTOL	Tet element minimum edge-point length ratio.	0.5
T3_AR	TRIARTOL	Tri element aspect ratio.	100.0
T3_EPAD	TRIMAXEPADTOL	Tri element maximum edge-point angular deviation (degrees).	30.0
T3_IAMAX	TRIMAXIATOL	Tri element maximum interior angle (degrees).	170.0
T3_EPLR	TRIMINEPLRTOL	Tri element minimum edge-point length ratio.	0.5
T3_IAMIN	TRIMINIATOL	Tri element minimum interior angle (degrees).	10.0
T3_SKEW	TRISKEWTOL	Tri element skew angle (degrees).	65.0

Notes:

- *Testtype* character variables starting with the characters Q4 are applicable to CQUAD4 and CQUADR elements. *Testtype* character variables starting with the characters Q8 are applicable to CQUAD8 elements. *Testtype* character variables starting with the characters T3 are applicable to CTRIA3 and CTRIAR elements. *Testtype* character variables starting with the characters T6 are applicable to CTRIA6 and CTRIAX6 elements. *Testtype* character variables names starting with the characters TET are applicable to CTETRA elements. *Testtype* character variables starting with the characters HEX are applicable to CHEXA elements. *Testtype* character variables starting with the characters PENT are applicable to CPENTA elements. *Testtype* character variables starting with the characters PYR are applicable to CPYRA elements.
- Aspect ratio is defined as the ratio of the length of the longest element side to the length of the shortest side. This check looks at all element edges to find the maximum and minimum lengths. For solid elements, edges along all faces are considered. Only element corners are used. Edge nodes of parabolic elements are ignored. Quad and hex elements are often very tolerant of large aspect ratios especially for in-plane loads, hence the large default value. For shear and twisting loads, however, a significantly lower tolerance should be considered. For tri and tet elements, high aspect ratios can result in poor extrapolation of results to corner nodes. It is recommended that a lower tolerance be used for these elements if corner stresses are required.
- 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. A perfect rectangle would have four 90 degree interior angles. An equilateral triangle would have three 60 degree interior angles. Internal angles are evaluated against both a minimum and a maximum tolerance. Like skew, large and small internal angles result in poor element performance, especially as they approach the upper and lower default limits of this check. And internal angle of 180 degrees or more will result in a singular element. An internal angle of zero degrees is also singular.

  Skew angle for a quadrilateral element or solid element face is a measure of how much of a parallelogram it is relative to a rectangle. It is defined to be the angle between the lines that join midpoints of the opposite sides of the quadrilateral minus 90 degrees. A rectangle would have a zero skew angle. Skew angle for a triangular element or solid element face is a measure of how close it is in shape relative to an equilateral triangle. Skew angle for the triangular element is defined to be the angle between the lines that join midpoints of two opposite sides relative to the line through their vertex and the midpoint of the remaining side minus 90 degrees. An equilateral triangle would have a zero skew angle. Each vertex of the triangle is examined and the largest skew angle reported. Element accuracy can be sensitive to element skew angle.

- 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 minus one). The largest of the four ratios is compared against the tolerance value. Note that as the ratio approaches zero, the shape approaches a rectangle. A large taper ratio implies an element that is trapezoid shaped, with a short edge opposite a long edge. High tapers affect the ability of an element to extrapolate Gauss point values to corner nodes accurately.
- The warping angle is the angle formed between the normal vectors located at diagonally opposite corner points. The warping angle is zero when all four corner points are in the same plane. Quad elements are very sensitive to even small amounts of warping and users should keep elements as flat as possible, breaking them up if necessary to prevent warpage.
- The edge point length ratio and edge point interior angle tests are only performed for solid and shell elements when edge node points exist. The length ratio 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 edge point angular deviation is the angle between the lines joining the edge node and the end points. For curved elements, some angular deviation is necessary and expected, but high values will compromise the stiffness of the element.

4. The default limit on the number of warning/fatal error messages output for element geometry checks is either 10,000 or the number of lines in the Model Input File, whichever is larger.
5. The specification of the SUMMARY character variable is equivalent to PARAM, ELEMGEOMOUT, ON. When ELEMGEOMOUT is set to ON, the following statistics are output to the Model Results Output File for each element:
   - Aspect ratio
   - Taper ratio
   - Skew angle
   - Warping angle
   - Normalized Jacobian

   The data is sorted based on normalized Jacobian determinant, skew angle, and aspect ratio in ascending order for each element type. See Section 5, *Parameters*, for more information on ELEMGEOMOUT.

GLBMATRIX Global Matrix Output Request

Description: Requests output of the global stiffness, differential stiffness, damping, and mass matrices at selected phases of analysis at specified grid points.

Format:

$$\text{GLBMATRIX}\left[([\text{KG}],[\text{KN}],[\text{KF}],[\text{KA}],[\text{MG}],[\text{MN}],[\text{MF}],[\text{MA}],[\text{BG}],[\text{BN}],[\text{BF}],[\text{BA}]) = \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix} \right]$$

Example:

GLBMATRIX(KA, MA) = 25

Option	Definition	Type	Default
KG	Include output of the global stiffness matrix before modification for multipoint constraints.	Character	
KN	Include output of the global stiffness matrix after modification for multipoint constraints.	Character	
KF	Include output of the global stiffness matrix after modification for single point constraints.	Character	
KA	Include output of the global stiffness matrix after reduction to the analysis set.	Character	
MG	Include output of the global mass matrix before modification for multipoint.	Character	
MN	Include output of the global mass matrix after modification for multipoint.	Character	
MF	Include output of the global stiffness matrix after modification for single point constraints.	Character	
MA	Include output of the global mass matrix after reduction to the analysis set.	Character	
BG	Include output of the global damping or differential stiffness matrix before modification for multipoint constraints.	Character	
BN	Include output of the global damping or differential stiffness matrix after modification for multipoint constraints.	Character	
BF	Include output of the global damping or differential stiffness matrix after modification for single point constraints.	Character	
BA	Include output of the global damping or differential stiffness matrix after reduction to the analysis set.	Character	

Option	Definition	Type	Default
ALL	The specified matrices at all grid points will be output.	Character	✓
n	Set identifications number of a previously appearing SET command. Only grid points whose identification numbers appear on this SET will be included in the output.	Integer $>$ 0	
NONE	Matrix output will be suppressed.	Character	✓

Remarks:

1. Selecting ALL for even small models may result in a very large Model Results File.
2. If no matrix type (KG, KN, KF, KA, etc.) is specified, all types will be output.
3. Output is not supported for all matrix types at all phases of analysis. A request for a phase that is not supported will result in no output for that phase with no warning message.
4. The B matrix types can be used to output global differential stiffness in solutions where this matrix is generated (i.e., buckling and prestress).

GPDISCONT — Grid Point Discontinuity

Description: Requests mesh discontinuity output based on grid point stress or strain.

Format:

$$\text{GPDISCONT}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

GPDISCONT = 3

Option	Definition	Type	Default
PRINT	Grid point mesh discontinuities will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point mesh discontinuities will be output only to the results neutral file system.	Character	
PUNCH	Grid point mesh discontinuities will be output additionally to the Model Results Punch File.	Character	
ALL	Grid point mesh discontinuities for all grid points will be output.	Character	
n	Set identification number of a previously appearing SET command. Only mesh discontinuities for grid points whose identification numbers appear on this SET will be output.	Integer > 0	
NONE	Grid point mesh discontinuities will not be output.	Character	✓

Remarks:

1. Only mesh discontinuities for grid points connected to elements used to define the surface or volume are output. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)
2. If grid point stresses are requested, then stress discontinuities will be output. If grid point strains are requested, then strain discontinuities will be output.
3. If the STRESSERROR model parameter is set to ON, normalized grid point stress error (mesh convergence error) will be output regardless of settings on this command. STRESSERROR provides both a grid point error and an overall mesh convergence error estimate. (See Section 5, *Parameters*, for more information on STRESSERROR.)

GPFLUX Grid Point Thermal Gradient and Heat Flux Output Request

Description: Requests grid point thermal gradient and heat flux output in heat transfer analysis.

Format:

$$\text{GPFLUX}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\\text{NONE}\end{Bmatrix}$$

Example:

GPFLUX = ALL

Option	Definition	Type	Default
PRINT	Grid point thermal gradients and heat fluxes will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point thermal gradients and heat fluxes will be output only to the results neutral file system.	Character	
PUNCH	Grid point thermal gradients and heat fluxes will be output additionally to the Model Results Punch File.	Character	
ALL	Thermal gradients and heat fluxes for all grid points will be output.	Character	
n	Set identification number of a previously appearing SET command. Only gradients and fluxes for grid points whose identification numbers appear on this SET command will be output.	Integer >0	
NONE	Grid point thermal gradient and heat flux will not be output.	Character	✓

Remarks:

1. Shell elements must be referenced on a SURFACE and solid elements must be referenced in a VOLUME. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)

GPFORCE

Grid Point Force Output Request

Description: Requests a static equilibrium summary.

Format:

$$\text{GPFORCE}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

GPFORCE = ALL

Option	Definition	Type	Default
PRINT	Grid point thermal gradient and heat flux will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point force balance will be output only to the results neutral file system.	Character	
ALL	Grid point force balance for all grid points will be output.	Character	
n	Set identification number of a previously appearing SET command. Only grid points whose identification numbers appear on this SET will be included in the grid point force balance.	Integer > 0	
NONE	Grid point force balance will not be output.	Character	✓

Remarks:

1. If the GPFORCEMETHOD model parameter is set to NORAN (default is NASTRAN), only grid points connected to elements specified by FORCE or ELFORCE are output. This feature allows users to break out loads in critical areas of a large model. These loads can then be used in loading a detailed model of the critical area. (See the FORCE and ELFORCE commands in Section 3, *Case Control,* and the GPFORCEMETHOD model parameter in Section 5, *Parameters.*)

GPSTRAIN — Grid Point Strain Output Request

Description: Requests grid point strain output.

Format:

$$\text{GPSTRAIN}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{SHEAR}\\\text{VONMISES}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\\text{PHASE}\end{bmatrix},\begin{bmatrix}\text{THERMAL}\\\text{MECH}\\\text{TOTAL}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\\text{ATOC}\\\text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\\text{NONE}\end{Bmatrix}$$

Example:

GPSTRAIN = ALL

Option	Definition	Type	Default
PRINT	Grid point strains will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point strains will be output only to the results neutral file system.	Character	
PUNCH	Grid point strains will be output additionally to the Model Results Punch File.	Character	
SHEAR	Maximum shear stress request for shell elements and octahedral shear stress request for solid elements.	Character	
VONMISES	Von Mises stress request for shell and solid elements.	Character	✓
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
THERMAL	Thermal strain request for shell and solid elements.	Character	
MECH	Mechanical strain request for shell and solid elements.	Character	
TOTAL	Total strain (thermal plus mechanical) request for shell and solid elements.	Character	✓
ALL	Grid point strains for all elements will be output.	Character	
n	Set identification of previously appearing SET command. Only strains for grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point strains will not be output.	Character	✓

Remarks:

1. Only grid points connected to elements used to define the surface or volume are output. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)
2. See the STRAIN command in Section 3, *Case Control,* for the equations used to calculate strain invariants.
3. If the MECHSTRAIN model parameter is set to ON (default is OFF), mechanical strain will be output regardless of settings on this command. (See Section 5, *Parameters*, for more information on MECHSTRAIN.)

GPSTRESS — Grid Point Stress Output Request

Description: Requests grid point stress output.

Format:

$$\text{GPSTRESS}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{SHEAR}\\\text{VONMISES}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\\text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\\text{ATOC}\\\text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\n\\\text{NONE}\end{Bmatrix}$$

Example:

GPSTRESS(VONMISES) = ALL

Option	Definition	Type	Default
PRINT	Grid point stresses will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point stresses will be output only to the results neutral file system.	Character	
PUNCH	Grid point stresses will be output additionally to the Model Results Punch File.	Character	
SHEAR	Maximum shear stress request for shell elements and octahedral shear stress request for solid elements.	Character	
VONMISES	Von Mises stress request for shell and solid elements.	Character	✓
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
ALL	Grid point stresses for all elements will be output.	Character	
n	Set identification of previously appearing SET command. Only stresses for grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point stresses will not be output.	Character	✓

Remarks:

1. Only grid points connected to elements used to define the surface or volume are output. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)

2. See the STRESS command in Section 3, *Case Control,* for the equations used to calculate stress invariants.

GRIDOFFSET

Grid Point Offset

Description: Specifies the offset vector used to translate all model grid point data.

Format:

GRIDOFFSET*, o1, o2, o3*

Example:

GRIDOFFSET, 3.12, 4.4, 22.76

Option	Definition	Type	Default
o1, o2, o3	Components of the model offset vector in the location coordinate system of the grid point. See Remark 1.	Real	0.0

Remarks:

1. Offsets are applied to all grid points translated in the Bulk Data and are relative to the coordinate system specified for the grid coordinates in field three of the GRID entry.
2. This command can only be used to offset the Model Input File grid data.

GRIDSCALEFACTOR Grid Point Scale Factor

Description: Specifies the scale factors used to scale all model grid point data.

Format:

GRIDSCALEFACTOR, *s1, s2, s3*

Example:

GRIDSCALEFACTOR, 0.5, 1.0, 1.0

Option	Definition	Type	Default
s1, s2, s3	Scale factors for each component coordinate. See Remark 1.	Real ≠ 0.0	1.0

Remarks:

1. Scale factors are applied to all grid points translated in the Bulk Data and are relative to the coordinate system specified for the grid point coordinates in field three of the GRID entry.
2. This command can only be used to offset the Model Input File grid data.

GROUNDCHECK

Rigid Body Motion Grounding Check

Description: Perform grounding check analysis on the stiffness matrix to expose unintentional constraints by moving the model rigidly.

Format:

$$\text{GROUNDCHECK}\left[(\text{SET}=(\begin{bmatrix}\text{G,N,F,A}\\ \text{ALL}\end{bmatrix}),\text{GRID}=gid,\text{THRESH}=etol,\text{DATAREC}=\text{YES/NO},\text{RTHRES}=rtol)\right]=\begin{Bmatrix}\text{YES}\\ \text{NO}\end{Bmatrix}$$

Examples:

GROUNDCHECK=YES

GROUNDCHECK(SET=(N), GRID=12, THRESH=1.-5, DATAREC=YES) = YES

Option	Definition	Type	Default
SET	Specifies at what point in the solution sequence to perform the rigid body motion grounding check. One of the characters variables: G Perform checks after stiffness matrix assembly before multipoint constraints are applied. N Perform checks after multipoint constraints are applied before single point constraints are applied. F Perform checks after single point constraints are applied before static condensation. A Perform checks after static condensation before decomposition. ALL Perform all checks.	Character	G
gid	Reference grid point for the calculation of the rigid body motion.	Integer > 0	
etol	Maximum strain energy which passes the check.	Real ≥ 0.0	See Remark 1
DATAREC	Option for outputting grounding forces. One of the following character variables: YES or NO.	Character	NO
rtol	Percent tolerance for grounding force output when DATAREC=YES.	Real ≥ 0.0	10.0

Remarks:

1. The default THRESH value is computed by dividing the largest term in the stiffness matrix by 1.0E+10.
2. If DATAREC=YES, GROUNDCHECK forces will be output in the grid displacement coordinate system.

HDOT — Heat Transfer Rate of Change of Enthalpy Output Request

Description: Requests rate of change of enthalpy vector output in transient heat transfer analysis.

Format:

$$\text{HDOT}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

HDOT = 10

Option	Definition	Type	Default
PRINT	Grid point rate of change of enthalpy will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point rate of change of enthalpy will be output only to the results neutral file system.	Character	
PUNCH	Grid point rate of change of enthalpy will be output additionally to the Model Results Punch File.	Character	
ALL	Rate of change of enthalpy for all grid points will be output.	Character	
n	Set identification of previously appearing SET command. Only rates of change of enthalpy for grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point rate of change of enthalpy will not be output.	Character	✓

Remarks:

1. HDOT = NONE is used to override a previous HDOT = n or HDOT = ALL command.

IC — Transient Initial Condition Set Selection

Description: Selects the initial conditions for transient response analysis.

Format:

IC = *n*

Example:

IC = 15

Option	Definition	Type
n	Set identification of a TIC Bulk Data entry.	Integer > 0

Remarks:

1. TIC entries will not be used (no initial conditions) unless selected in the Case Control Section.

IMPACTGENERATE — Automated Impact Analysis

Description: Automated Impact Analysis (AIA). Automatically sets up a nonlinear transient impact analysis including contact definition, initial conditions, damping, time increment, and duration.

Format:

IMPACTGENERATE, *gid, cid, v0, a, t1, t2, t3, tdelta, ttotal, nta, nto, desid*

Example:

IMPACTGENERATE, 134, , 124.5, 386.4, 0.004, 1.05, 0.654

Option	Definition	Type	Default
gid	Grid point identification number on projectile. See Remark 1.	Integer > 0	Required
cid	Projectile translation vector, *ti*, coordinate system identification number. See Remark 2.	Integer ≥ 0 or -1	0
v0	Initial projectile velocity magnitude in the direction of the projectile translation vector. See Remark 3.	Real ≥ 0.0	0.0
a	Projectile and part acceleration magnitude in the direction of the projectile translation vector. See Remark 4.	Real ≥ 0.0	0.0
t1, t2, t3	Projectile translation vector.	Real ≥ 0.0	0.0, 0.0, 0.0
tdelta	Time increment. See Remark 5.	Real ≥ 0.0	Model dependent
ttotal	Total duration. See Remark 6.	Real ≥ 0.0	Model dependent
nta	Number of analysis time steps. See Remark 6.	Integer > 0 or blank	See Remark 6
nto	Number of output time steps. See Remark 7.	Integer > 0 or blank	See Remark 7
desid	Damped element set identification number. Set identification of previously appearing SET command. Only elements whose identification numbers appear on this SET command will be used. See Remarks 8 and 9.	Integer > 0 or blank	Projectile element set

Remarks:

1. *gid* is a point on the projectile through the projectile translation vector defined by *ti*. The grid point should be selected at the approximate impact point and defines the base of the projectile axis.
2. A *cid* value equal to -1 specifies that the *ti* are in the basic coordinate system and the vector magnitude is the exact translation distance required to place the projectile on the surface of the body. A *cid* value greater than or equal to zero ignores the projectile vector magnitude and estimates the distance to impact by translating the projectile to the part so that it contacts near the vector base.

3. The initial projectile velocity magnitude, *v0*, is required if the acceleration magnitude, *a*, is not specified.
4. The acceleration magnitude, *a*, is required if the initial projectile velocity, *v0*, is not specified.
5. The transient time increment may be omitted and a value based on the estimate of the contact frequency at impact will be used. The contact frequency is determined using

$$f_c = \mathrm{MAX}(f_{fp}, f_{fb})$$

   where f_{fp} is the natural frequency of the projectile fixed at the point of contact and f_{fb} is the natural frequency of the part fixed at the user defined boundary conditions. The time increment is then determined using

$$\Delta t = \frac{1}{2 * f_c}$$

6. Duration is the total time duration of the analysis. If both duration (*ttotal*) and the number of analysis time steps (*nta*) are omitted, a duration value will be calculated using

$$\mathrm{MAX}\left(\frac{d_t}{V_a}, 100 * \Delta t\right)$$

   where d_t is the projectile translation distance to impact, V_a is the average velocity before impact equal to $\sqrt{v0^2 + 2 * a * d_t}$, and Δt is *tdelta* if specified or the calculated time increment. When *nta* is specified, a duration value will be calculated using

$$nta * \Delta t$$

7. If the MAXIMPACTSTEP model parameter is set to a value other than zero and *nta* is omitted, the transient time increment and duration will be adjusted to limit the number of output steps to MAXIMPACTSTEP (see Section 5, *Parameters*, for more information on MAXIMPACTSTEP). When *nta* and *nto* are both specified, the number of output steps is set to *nto* if *nto* is less than *nta* or *nta* if it is greater.
8. The damped element set, *desid*, is generally not required but may be needed for more complicated models where the object of interest is the projectile. It is used for the following
   a) To define all elements and grid points contained in the projectile set when there is a discontinuity in a complex projectile such as a surface weld element between two discontinuous parts. By default the projectile set is automatically identified using the projectile grid point, *gid*. The body set is defined as all elements not in the projectile set. If there is a discontinuity in a complex projectile it will be necessary to explicitly define the projectile using the damped element set, *desid*.
   b) To specify the object that the damping frequency of interest should be based on. This is typically the object of interest. When the damped element set is specified the damping frequency is calculated using a normal modes analysis where the body is fixed and the frequency of the mode with the largest scaled displacement at the impact point in the direction of the projectile translation vector is used.
9. When the damped element set, *desid*, is not specified, the damping frequency of interest is based on element structural damping. If element structural damping is specified on any element in the projectile set the damping frequency will be based on a normal modes analysis of the projectile. Otherwise the body is used as the basis where the projectile is fixed and a normal modes analysis is performed on the body.

INITSTRAIN — Initial Strain Set Selection

Description: Selects the initial strain state in nonlinear analysis.

Format:

INITSTRAIN = *n*

Example:

INITSTRAIN = 10

Option	Definition	Type
n	Set identification of a STRAIN Bulk Data entry.	Integer > 0

Remarks:

1. STRAIN entries will not be used (initial strain state set to zero) unless selected in the Case Control Section.

INCLUDE — Insert External File

Description: Inserts an external file into the Model Input File.

Format:

INCLUDE [*d*:] [*path*] *filename*[.*ext*]

Example:

The following INCLUDE statement shows how to fetch the Bulk Data from another file called Bolt.NAS:

```
TITLE = STATIC ANALYSIS
SPC = 1
LOAD = 2
BEGIN BULK
INCLUDE ‘Bolt.NAS’
ENDDATA
```

Remarks:

1. The INCLUDE statement may appear anywhere in the Model Input File.
2. Maximum file specification length is 72 characters.
3. INCLUDE statements cannot be nested (i.e., no INCLUDE statement can appear inside the external file).
4. Quotation marks on the file specification are optional.

K2GG — Direct Input Stiffness Matrix Selection

Description: Selects a direct input stiffness matrix.

Format:

K2GG = *name*

Example:

K2GG = KDMIG

Option	Definition	Type
name	Name of the $[K_{gg}^2]$ matrix that is defined on the DMIG Bulk Data entry.	Character

Remarks:

1. Direct input matrices will not be used unless selected.
2. Terms are added to the global stiffness matrix before any constraints are applied.
3. The matrix must be symmetric in form (field 4 on DMIG Bulk Data entry must contain the integer 6).
4. A scale factor may be applied to this input using PARAM, CK2.

K2PP — Direct Input Stiffness Matrix Selection

Description: Selects a direct input stiffness matrix.

Format:

K2PP = *name*

Example:

K2PP = KDMIG

Option	Definition	Type
name	Name of the $\left[K_{pp}^{2}\right]$ matrix that is defined on the DMIG Bulk Data entry.	Character

Remarks:

1. Direct input matrices will not be used unless selected.
2. Terms are added to the global stiffness matrix after constraints are applied.
3. The matrix must be square or symmetric in form (field 4 on DMIG Bulk Data entry must contain the integer 1 or 6).
4. This command is only supported in complex eigenvalue solutions.

LABEL — Output Label

Description: Defines a character label that will appear on the third heading line of each page of output for each subcase.

Format:

LABEL = *Any character string*

Example:

LABEL = 100.0 LB Parabolic Edge Load In Y-Direction

Remarks:

1. Maximum label length is 74 characters.
2. LABEL appearing at the subcase level will label output for that subcase only.
3. LABEL appearing outside a subcase level will label all output unless another LABEL command is encountered at the subcase level.
4. If no LABEL command is supplied, the label line will be blank.
5. LABEL information is also placed on the third line of each results neutral file. Only the first 67 characters appear.

LINE Data Lines Per Page

Description: Defines the number of data lines per output page.

Format:

LINE = *n*

Example:

LINE = 51

Option	Definition	Type	Default
n	Number of data lines per page.	Integer > 0	66

Remarks:

1. This value should correspond to the number of printed lines per page of the system printer.

LOAD — External Static Load Set Selection

Description: Selects the external static load set to be applied to the model.

Format:

LOAD = *n*

Example:

LOAD = 15

Option	Definition	Type
n	Set identification of at least one external load Bulk Data entry. The set identification must appear on at least one FORCE, FORCE1, GRAV, MOMENT, MOMENT1, LOAD, PLOAD1, PLOAD2, PLOAD4, or SPCD entry.	Integer > 0

Remarks:

1. The above static load entries will not be used unless selected in the Case Control Section.
2. The total load applied will be the sum of external (LOAD command), element deformation (DEFORM command), constrained displacement (SPC command), and thermal (TEMPERATURE command) loads.
3. Static, thermal, and element deformation loads should have unique set identification numbers.

LOADINTERPOLATE — Load Interpolation

Description: Interpolates grid point force, moment, pressure, and heat flux data from a known set of input grid points and loads to a set of output grid points and loads based on geometric position in 2d or 3d space.

Format:

LOADINTERPOLATE, *olsid*, *ogsid*, *ilsid*, *igsid, ltype, nnri, ndlsf, cgsize, maxnus*

Example:

LOADINTERPOLATE, 100, 10, 1, 1

Option	Definition	Type	Default
olsid	Output load set identification number. See Remark 1.	Integer > 0	Required
ogsid	Output grid set identification number. Set identification of previously appearing SET command. Only grid points whose identification numbers appear on this SET command will be used.	Integer > 0 or blank	All grid points in model
ilsid	Input load set identification number. See Remark 2.	Integer > 0	Required
igsid	Input grid set identification number. Set identification of previously appearing SET command. Only grid points whose identification numbers appear on this SET command will be used.	Integer > 0 or blank	All grid points in load set
ltype	Load type. One of the following character variables: FORCE, SFORCE, MOMENT, PRESSURE, PNORMAL, or HEATFLUX. See Remark 3.	Character	Required
nnri	Number of interpolation nodes within radius of influence.	Integer > 0 or blank	See Remark 4
ndlsf	Number of data nodes in least squares fit.	Integer > 0 or blank	See Remark 5
cgsize	Number of rows, columns, and planes in the cell grid. A box containing the nodes is partitioned into cells in order to increase search efficiency.	Integer > 0 or blank	See Remark 6
maxnus	Maximum number of unique solution occurrences.	Integer > 0 or blank	See Remark 7

Remarks:

1. For *ltype* equal to FORCE or MOMENT output is FORCE and MOMENT Bulk Data entries at grid points defined by *ogsid*. For *ltype* equal to PRESSURE output is PLOAD4 Bulk Data entries on element faces that have grid points defined by *ogsid*.
2. Input is GRID, FORCE, MOMENT, PLOADG, and QBDYG Bulk Data entries which need not be associated with the analysis model. See Section 4, *Bulk Data*, for more information on GRID, FORCE, MOMENT, PLOADG, and QBDYG Bulk Data entries.

3. FORCE, SFORCE, or MOMENT interpolation is not recommended for non-planar structures. FORCE and MOMENT interpolation may result in an output load total different than the input one. SFORCE provides a scaled output load total that is equal to the input total. It is recommended that the input forces are in a consistent direction. Multiple LOADINTERPOLATE commands may be required to affect this. PNORMAL is similar to PRESSURE, except the pressure vector is forced to be normal to the element surface (pressure magnitude interpolation only). This option is recommended when the input pressure is normal to applied surface.
4. The valid range for *nnri* is $1 \leqslant nnri \leqslant \min(100, n-1)$), where *n* is the number of input data points. The default is 100. A lower value may increase performance at the cost of accuracy. A value greater than or equal to 32 is recommended.
5. The valid range for *ndlsf* is $9 \leqslant ndlsf \leqslant \min(100, n-1)$, where *n* is the number of input data points. The default is 100. A lower value may increase performance at the cost of accuracy. A value greater than or equal to 17 is recommended.
6. The recommended value for *cgsize* is:

$$cgsize = \left(\frac{n}{3}\right)^{\frac{1}{3}}$$

   where *n* is the number of input data points. The default is determined using the above formula.
7. A 3d interpolation algorithm is used initially, but will automatically revert to a 2d algorithm if the number of no unique solution errors exceeds *maxnus* while processing the input data points. Models that are dominantly flat but still have 3d features that default to the 2d interpolation algorithm may not be interpolated accurately. A larger *maxnus* value can be used to force a 3d interpolation. It is advisable to always check the interpolated loads.
8. Generated FORCE and MOMENT Bulk Data entries can be exported using the TRSLBULKDATA Model Initialization directive. See Section 2, *Initialization*, for more information on TRSLBULKDATA.

LOADSET

Static Load Set Selection for Use in Dynamics

Description: Selects a sequence of static load sets which can be referenced by dynamic load commands.

Format:

LOADSET = *n*

Example:

LOADSET = 100

Option	Definition	Type
n	Set identification number of at least one LSEQ Bulk Data Entry.	Integer $>$ 0

Remarks:

1. The number of static load vectors created is the number of unique DAREA fields defined on all LSEQ Bulk Data entries.
2. This command is only applicable in transient and frequency response analysis.

M2GG Direct Input Mass Matrix Selection

Description: Selects a direct input mass matrix.

Format:

M2GG = *name*

Example:

M2GG = MDMIG

Option	Definition	Type
name	Name of the $\left[M_{gg}^{2}\right]$ matrix that is defined on the DMIG Bulk Data entry.	Character

Remarks:

1. Direct input matrices will not be used unless selected.
2. Terms are added to the global mass matrix before any constraints are applied.
3. The matrix must be symmetric in form (field 4 on DMIG Bulk Data entry must contain the integer 6).
4. M2GG input is not affected by PARAM, WTMASS. M2GG input must either be in consistent mass units or scaled using PARAM, CM2.

M2PP Direct Input Mass Matrix Selection

Description: Selects a direct input mass matrix.

Format:

M2PP = *name*

Example:

M2PP = MDMIG

Option	Definition	Type
name	Name of the $\left[M^2_{pp}\right]$ matrix that is defined on the DMIG Bulk Data entry.	Character

Remarks:

1. Direct input matrices will not be used unless selected.
2. Terms are added to the global mass matrix after constraints are applied.
3. The matrix must be square or symmetric in form (field 4 on DMIG Bulk Data entry must contain the integer 1 or 6).
4. M2PP input is not affected by PARAM, WTMASS. M2PP input must be in consistent mass units.
5. This command is only supported in complex eigenvalue solutions.

MASTER — Redefine the MASTER Subcase

Description: Allows the redefinition of a MASTER subcase.

Format:

SUBCASE *n*

Example:

SUBCASE 20
MASTER

Remarks:

1. All commands in a MASTER subcase apply to the following subcases until a new MASTER subcase is defined.
2. In the following example subcase 101 and on reference SPC set 10 and MPC set 10 until subcase 201 where a new MASTER subcase is defined referencing SPC set 20 and MPC set 20:

```
DISP = ALL
STRESS(CORNER) = ALL
SUBCASE 101
      MASTER
      LABEL = FIXED BOUNDARY
      SPC = 10
      MPC = 10
      LOAD = 101
SUBCASE 102
      LOAD = 102
SUBCASE 103
      LOAD = 103
SUBCASE 201
      MASTER
      LABEL = PINNED BOUNDARY
      SPC = 20
      MPC = 20
      LOAD = 201
SUBCASE 202
      LOAD = 202
SUBCASE 203
      LOAD = 203
```

3. The MASTER command must appear immediately after a SUBCASE command.

METHOD — Real Eigenvalue Extraction Method Selection

Description: Selects the real eigenvalue extraction parameters.

Format:

METHOD = *n*

Example:

METHOD = 33

Option	Definition	Type
n	Set identification of an EIGRL Bulk Data entry.	Integer > 0

Remarks:

1. This command should only be specified once in transient and frequency response solutions.

MFLUID — Fluid Boundary Element Selection

Description: Selects the MFLUID Bulk Data entries to be used to specify the fluid-structure.

Format:

MFLUID = *n*

Example:

MFLUID = 105

Option	Definition	Type
n	Set identification of one or more MFLUID Bulk Data entries.	Integer > 0

Remarks:

1. MFLUID entries will not be used unless selected in Case Control.

MODES

Subcase Repeater

Description: Allows alternate eigenvalue results output selection.

Format:

MODES = *n*

Example:

MODES = 15

Option	Definition	Type
n	Number of modes to be output for the specified subcase.	Integer > 0

Remarks:

1. This command is best described with an example. It is desired to output element forces for the first four modes only, then element strain energy for the next two, and element stress for all remaining modes. The following example demonstrates this:

```
SUBCASE 1 $ FOR MODES 1 THRU 4
      MODES = 4
      FORCE = ALL
SUBCASE 5 $ FOR MODES 5 AND 6
      MODES = 2
      ESE = ALL
SUBCASE 7 $ FOR MODES 7 AND REMAINING MODES
      STRESS = ALL
```

2. If this command is excluded, all eigenvalue results are considered to be part of a single subcase.
3. This command can also be used to suppress output after a certain number of modes have been output. For example, to suppress all eigenvalue output for modes beyond the first five, the following Case Control could be used:

```
SUBCASE 1
      MODES = 5
      STRESS = ALL
SUBCASE 6
      DISPLACEMENTS = NONE
```

MODESET — Mode Set Generation

Description: Modal set generation.

Format:

MODESET, *method*, *value*

Examples:

MODESET, SET, 20

MODESET, TOP, 5

MODESET, PERCENT, 2.5

MODESET, CUTOFF, 80.0

MODESET, INCLUDE, SET 5

MODESET, EXCLUDE, SET 4

Option	Definition		Type	Default
method	The search method used, one of the following character variables: SET, INCLUDE, EXCLUDE, TOP, PERCENT, or CUTOFF.		Character	Required
value	Value is based on method as follows:		Integer > 0 or real	Required
	SET	Previously appearing SET command which defines which modes are to be included in the solution set. Equivalent to INCLUDE.		
	INCLUDE	Previously appearing SET command which defines which modes are to be included in the solution set. Equivalent to SET.		
	EXCLUDE	Previously appearing SET command which defines which modes are to be excluded in the solution set.		
	TOP	The number of modes to be retained in the solution set starting with the highest modal effective mass.		
	PERCENT	Modes with a percent modal effective mass greater than this value are included in the solution set.		
	CUTOFF	Modes starting with the highest modal effective mass and stopping when the sum of percent modal effective mass is equal to this value.		

Remarks:

1. This command may be repeated with different options to generate the modal set.

MPC — Multipoint Constraint Set Selection

Description: Selects a multipoint constraint set.

Format:

MPC = *n*

Example:

MPC = 24

Option	Definition	Type
n	The set identification of a multipoint constraint set and hence must appear on a MPC or MPCADD Bulk Data entry.	Integer > 0

Remarks:

1. MPC or MPCADD entries will not be used unless selected in Case Control.

MPCFORCES

Multipoint Forces of Constraint Set Selection

Description: Requests multipoint constraint force vector output.

Format:

$$\text{MPCFORCES}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\\text{NONE}\end{Bmatrix}$$

Example:

MPCFORCES = 8

Option	Definition	Type	Default
PRINT	Multipoint constraint forces will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Multipoint constraint forces will be output only to the results neutral file system.	Character	
PUNCH	Multipoint constraint forces will be output additionally to the Model Results Punch File.	Character	
ALL	Multipoint constraint forces for all grid points will be output.	Character	
n	Set identification of previously appearing SET command. Only multipoint constraint forces for grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Multipoint constraint forces will not be output.	Character	✓

Remarks:

1. MPCFORCE output is in the global coordinate system. (See CD field on the GRID Bulk Data entry in Section 4, *Bulk Data.*)

MPRES — Fluid Pressure Output Request

Description: Requests fluid pressure for selected grid points in fluid-structure interaction problems.

Format:

$$\text{MPRES}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

MPRES = 5

Option	Definition	Type	Default
PRINT	Fluid pressure will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Fluid pressure will be output only to the results neutral file system.	Character	
PUNCH	Fluid pressure will be output additionally to the Model Results Punch File.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
ALL	Fluid pressure for all grid points will be output.	Character	
n	Set identification of previously appearing SET command. Only pressures for grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Fluid pressures will not be output.	Character	✓

Remarks:

1. Fluid pressure output is only supported in dynamic response solutions and limited to the virtual fluid mass wet surface.

NASTRAN — Model Initialization Directive Specification

Description: Specifies Model Initialization directives in the Case Control Section.

Format:

NASTRAN *directive1=option1*, …, *directiven = optionn*

Example:

NASTRAN DECOMPMETHOD=PCGLSS, SPARSEITERTOL=1.-5

Remarks:

1. Maximum length is 80 characters.
2. More than one NASTRAN command may be specified.
3. Directives specified on this command will override ones specified in the Model Initialization File.

NLPARM Nonlinear Static Analysis Parameter Selection

Description: Selects the parameters used for nonlinear static analysis.

Format:

NLPARM = *n*

Example:

NLPARM = 5

Option	Definition	Type
n	Set identification of an NLPARM Bulk Data entry.	Integer $>$ 0

Remarks:

1. An NLPARM entry in the Bulk Data will not be used unless selected.

NLSTRESS — Nonlinear Element Stress Output

Description: Request nonlinear element stress output in nonlinear solutions.

Format:

$$\text{NLSTRESS}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

NLSTRESS = 10

Option	Definition	Type	Default
PRINT	Nonlinear element stresses will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Nonlinear element stresses will be output only to the results neutral file system.	Character	
PUNCH	Nonlinear element stresses will be output additionally to the Model Results Punch File.	Character	
ALL	Element stresses for all nonlinear elements will be output.	Character	✓
n	Set identification of previously appearing SET command. Only stresses for elements whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Nonlinear element stresses will not be output.	Character	

Remarks:

1. If the NLSTRESS command is not specified the default is NLSTRESS = ALL.

NONLINEAR Nonlinear Dynamic Load Set Selection

Description: Selects nonlinear dynamic load set for transient problems.

Format:

NONLINEAR = *n*

Example:

NONLINEAR = 10

Option	Definition	Type
n	Set identification of NOLINi Bulk Data entry.	Integer > 0

Remarks:

1. NOLINi Bulk Data entries will not be used unless selected in the Case Control Section.

OFREQUENCY Output Frequency Set

Description: Selects a set of frequencies for output requests.

Format:

$$\text{OFREQUENCY} = \begin{Bmatrix} \text{ALL} \\ n \end{Bmatrix}$$

Example:

OFREQUENCY = ALL

Option	Definition	Type	Default
ALL	Output for all frequencies will be computed.	Character	✓
n	Set identification number of a previously appearing SET command. Output for frequencies closest to those given on this SET command will be output.	Integer > 0	

Remarks:

1. If the OFREQUENCY command is not supplied in the Case Control Section, then OFREQUENCY is defaulted to ALL.
2. This command is particularly useful for requesting a subset of the output (e.g., stresses at only peak frequencies, etc.).

OLOAD — Applied Load Output Request

Description: Requests applied load vector output.

Format:

$$\text{OLOAD}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

OLOAD = ALL

Option	Definition	Type	Default
PRINT	Grid point applied loads will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point applied loads will be output only to the results neutral file system.	Character	
PUNCH	Grid point applied loads will be output additionally to the Model Results Punch File.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
ALL	Grid point applied loads for all grid points will be output.	Character	
n	Set identification of previously appearing SET command. Only applied loads for grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point applied loads will not be output.	Character	✓

Remarks:

1. Indirect loads generated via the SPCD Bulk Data entry are not included in OLOAD output.
2. OLOAD results are output in the global coordinate system. (See CD field on the GRID Bulk Data entry in Section 4, *Bulk Data.*)

OTIME — Output Time Set

Description: Selects a set of times for output requests.

Format:

$$\text{OTIME} = \begin{Bmatrix} \text{ALL} \\ n \end{Bmatrix}$$

Example:

OTIME = ALL

Option	Definition	Type	Default
ALL	Output for all times will be computed.	Character	✓
n	Set identification number of a previously appearing SET command. Output for times closest to those given on this SET command will be output.	Integer > 0	

Remarks:

1. If the OTIME command is not supplied in the Case Control Section, then OTIME is defaulted to ALL.
2. This command is particularly useful for requesting a subset of the output (e.g., stresses at only peak times, etc.).

P2G Direct Input Load Matrix Selection

Description: Selects a direct input load matrix.

Format:

P2G = *name*

Example:

P2G = PDMIG

Option	Definition	Type
name	Name of the matrix that is defined on the DMIG Bulk Data entry.	Character

Remarks:

1. Direct input matrices will not be used unless selected.
2. Terms are added to the global load vector before any constraints are applied.
3. The matrix must be columnar in form (field 4 on DMIG Bulk Data entry must contain the integer 9).
4. A scale factor may be applied to this input using PARAM, CP2.

PARAM — Parameter Specification

Description: Specifies values for parameters to be used at certain places in the control sequence.

Format:

PARAM, *n, v*

Example:

PARAM, K6ROT, 1.+3

Option	Definition	Type	Default
n	Parameters name (one to 16 alphanumeric characters, the first of which is alphabetic).	Character	Required
v	Parameter value based on parameter type.	Character, real, or integer	Required

Remarks:

1. Parameters with names that are less than or equal to 8 characters can appear anywhere in the model input file prior to ENDDATA. Parameters with names greater than 8 characters must be specified in the Case Control Section.
2. For a list and detailed description of each parameter, see Section 5, *Parameters*.

RANDOM — Random Analysis Set Selection

Description: Selects the RANDPS and RANDT1 Bulk Data entries to be used in random analysis.

Format:

RANDOM = *n*

Example:

RANDOM = 120

Option	Definition	Type
n	Set identification of RANDPS and RANDT1 Bulk Data entries to be used in random analysis.	Integer > 0

Remarks:

1. RANDOM must reference one or more RANDPS Bulk Data entries to perform random analysis.
2. RANDOM must appear in the first subcase. RANDPS Bulk Data entries may not reference subcases in a different loop. Loops are defined by a change in the FREQUENCY command.

RESULTLIMITS

Result Limits Output Request

Description: Requests a subcase and global results search for result limits (max/min).

Format:

RESULTLIMITS, *sid*, *ssid*, *msid*, *etype*, *osid*, *stype*, *column*

Example:

RESULTLIMITS, 3, 4, 6, QUAD, 22, ELEM, 9

Option	Definition	Type	Default
sid	Results limits search set Identification number.	Integer > 0	Required
ssid	Subcase set identification number. Set identification of previously appearing SET command. Only subcases whose identification numbers appear on this SET command will be output. The character variable ALL may be used to specify all subcases. See Remark 2.	Integer > 0 or blank	ALL
msid	Step set identification number. Set identification of previously appearing SET command. Only time, frequency, or load steps whose identification numbers appear on this SET command will be output. The character variable ALL may be used to specify all steps as applicable.	Integer > 0 or blank	ALL
etype	Element type to be searched for within element identification number range, one of the following character variables: Element Results (*stype* = ELEM, see below) ELAS, WELD, PIPE, CABLE, GAP, BEAM, BAR, ROD, QUAD, TRI, SHEAR, TET, PENT, PYR, HEX, and ALL Grid Point Results (*stype* = GRID, see below) SHELL, SOLID, or ALL	Character or blank	ALL
osid	Output set identification number. Set identification of previously appearing SET command. Only elements or grid points whose identification numbers appear on this SET command will be output. The character variable ALL may be used to specify all elements or grid points as applicable. See Remark 3.	Integer > 0 or blank	ALL
stype	Output set identification type, one of the following character variables: GRID or ELEM.	Character	ELEM
column	Results column number. See Remark 4.	Integer > 0	Required

Remarks:

1. This command is used for determining results limits (i.e., max/min: stress, force, strain energy, etc.).

2. The subcase set identification number, *ssid*, will forced to ALL unless the RSLTFILETYPE directive is set to either PATRANBINARY or PATRANASCII.
3. The output set identification type must be consistent with the specified element type.
4. See Appendix A, *Results Neutral File Formats* for result column number definition.

RESVEC

Residual Vector Selection

Description: Specifies options for the calculation of residual vectors.

Format:

$$\text{RESVEC}\left[\left(\begin{bmatrix}\text{INRLOAD}\\ \text{NOINRL}\end{bmatrix},\begin{bmatrix}\text{APPLOAD}\\ \text{NOAPPL}\end{bmatrix},\begin{bmatrix}\text{RVDOF}\\ \text{NORVDOF}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ON}\\ \text{OFF}\end{Bmatrix}$$

Examples:

RESVEC = ON

RESVEC(APPLOAD, RVDOF) = ON

RESVEC = OFF

Option	Definition	Type	Default
INRLOAD	Enables the calculation of residual vectors based on inertia relief.	Character	
NOINRL	Disables the calculation of residual vectors based on inertia relief.	Character	
APPLOAD	Enables the calculation of residual vectors based on applied loads.	Character	✓
NOAPPL	Disables the calculation of residual vectors based on applied loads.	Character	
RVDOF	Enables the calculation of residual vectors based on RVDOFi entries.	Character	✓
NORVDOF	Disables the calculation of residual vectors based on RVDOFi entries.	Character	
ON	Requests the calculation of residual vectors based on inertia relief, applied loads, and RVDOFi entries.	Character	
OFF	Disables the calculation of residual vectors.	Character	✓

Remarks:

1. PARAM, RESVEC, ON is equivalent to the command RESVEC = ON. See Section 5, *Parameters*, for more information on RESVEC.

SDAMPING Structural Damping Selection

Description: Requests damping as a function of frequency in modal transient and frequency response solutions.

Format:

SDAMPING = *n*

Example:

SDAMPING = 25

Option	Definition	Type
n	Set identification number of a TABDMP1 Bulk Data entry.	Integer > 0

Remarks:

1. SDAMPING must reference a TABDMP1 entry.

SELEMGENERATE — Superelement Generation

Description: Superelement generation.

Format:

SELEMGENERATE, *seid, stype, esid, btype*

Examples:

SELEMGENERATE, 10, ELEM, 15

SELEMGENERATE, 10, GRID, 32, SELEM

Option	Definition	Type	Default
seid	Superelement id.	Integer ≥ 0	See Remark 1
stype	Output set type, one of the following character variables: GRID or ELEM.	Character	ELEM
esid	Element or grid point set identification number. Set identification of previously appearing SET command. Only elements or grid points whose identification numbers appear on this SET command will be used.	Integer > 0	Required.
btype	Boundary type: RSET or SELEM. See Remark 2. RSET Boundary grid points will be put in residual set. SELEM Boundary grid points will be left in superelement.	Character	RSET

Remarks:

1. A blank or zero value will automatically generate the next available superelement identification number.
2. The default RSET boundary type moves unassigned grid points on the superelement boundary into the residual set. Grid points assigned a superelement id via the GRID Bulk Data entry field 9 or the SESET Bulk Data entry will not be moved.

SET Set Definition

Description: Defines the following lists:

1. Identification numbers (grid point, element, or mode) for processing and output requests.
2. Output frequencies for frequency response problems or output times for transient response problems using OFREQ and OTIME commands, respectively.

Formats:

$\text{SET } n = \{ \text{i1}[,\ \text{i2},\ \text{i3 THRU i4}] \}$

$\text{SET } n = \{ \text{r1}[,\ \text{r2},\ \text{r3}, \text{r4},] \}$

$\text{SET } n = \text{ALL}$

Examples:

SET 15 = 7

SET 55 = 1 THRU 200000

SET 22 = 1, 5, 7, 8, 9, 15 THRU 66, 77, 79, 106 THRU 400,
544, 625, 1005 THRU 2067, 3005, 4020

SET 12 = 1.0, 2.0, 3.0, 4.0

SET 35 = 1.07-2, 8.05, 16.145, 2.456+2

Option	Definition	Type	Default
n	Set identification number.	Integer > 0	Required
i1, i2, etc.	Identification numbers. Identification numbers that do not exist are ignored.	Integer > 0	
i3 THRU i4	Identification number range (i3 $<$ i4). Identification numbers that do not exist are ignored.	Integer > 0	
ALL	All identification numbers are included.	Character	
r1, r2, etc.	Output frequencies or times. The nearest solution frequency or time will be output.	Real	
ALL	All frequencies or times are included.	Character	

Remarks:

1. Multiple SET commands with the same set identification number are allowed and will be treated as one set.
2. A comma at the end of the command signifies a continuation.
3. A THRU symbol may not be used for a continuation without the ending identification number.

SETGENERATE — Set Generation

Description: Element and grid point set generation.

Format:

SETGENERATE, *sid*, *stype*, *etype*, *method*, *value*, *id*, *threshold*

Example:

SETGENERATE, 3, ELEM, QUAD, MID, 105

Option	Definition	Type	Default
sid	Generated set identification number.	Integer $>$ 0	Required
stype	Target output set type, one of the following character variables: GRID or ELEM.	Character	Required
etype	Element type to be searched for within element identification number range, one of the following character variables: ELAS, WELD, PIPE, CABLE, GAP, BEAM, BAR, ROD, QUAD, TRI, SHEAR, TET, PENT, PYR, HEX, SHELL, SOLID, or ALL.	Character or blank	ALL
method	The search method used, one of the following character variables: RCN, PID, MID, R, T, P, X, Y, Z, or ALL	Character	ALL
value	Depending on the character variable supplied for method, this is either an integer property or material identification number (PID or MID), a real coordinate component (R, T, P, X, Y, or Z), an integer element results column number (RCN), or blank (*method* = ALL).	Integer $>$ 0 or real	See Remark 1
id	Depending on the character variable supplied for method, this is either a coordinate system identification number (R, T, P, X, Y, or Z) or a subcase identification number.	Integer $>$ 0 or blank	0; See Remark 2
threshold	The element result threshold value corresponding to the specified element results column number. Elements that have a result value greater than this value will be included in the generated set.	Real or blank	0.0; See Remark 2

Remarks:

1. Required when the *method* character variable is not set to ALL.
2. Required when the *method* character variable is RCN.
3. See Appendix A, *Results Neutral File Formats*, for result column number (RCN) definition.

SKIP — Case Control Processing Delimiter

Description: Activates or deactivates the execution of subsequent commands in the Case Control.

Format:

$$\text{SKIP}\begin{Bmatrix}\text{ON}\\\text{OFF}\end{Bmatrix}$$

Example:

SKIPOFF

Remarks:

1. SKIPON and SKIPOFF commands may appear as many times as needed in the Case Control.
2. SKIPON ignores subsequent commands until either a SKIPOFF or BEGIN BULK command is encountered. This allows requests to be omitted without deleting them or commenting them out. In the following example the second subcase will be skipped:

```
SUBCASE 101
      SPC = 101
      LOAD = 101
      NLPARM = 101
SKIPON $ SKIP SUBCASE 102
SUBCASE 102
      SPC = 102
      LOAD = 102
      NLPARM = 102
SKIPOFF $ RESUME PROCESSING CASE CONTROL
SUBCASE 103
      SPC = 103
      LOAD = 103
      NLPARM = 103
SKIPON $ SKIP SET AND VOLUME COMMANDS
SET 5 = 1, 5, 67, 37
VOLUME 1, SET 5, SYSTEM BASIC
BEGIN BULK
```

SOLUTION — Solution Sequence

Description: Select the type of solution.

Format:

SOLUTION = *type*

Example:

SOLUTION = LINEAR STATIC

Alternate Format and Example:

SOLUTION = 101

Option	Definition	Type
type	Type of solution sequence. Available solution types depend on the license purchased. This directive may also be specified in the Model Initialization File (see Section 2, *Initialization*, for more information on SOLUTION).	Character

Remarks:

1. The following table gives the solution number corresponding to each solution type. Either one may be used.

Solution Character Variable	Solution Number
LINEAR STATIC or STEADY STATE HEAT TRANSFER	101
MODAL	103
LINEAR BUCKLING	105
NONLINEAR STATIC	106
DIRECT FREQUENCY RESPONSE	108
DIRECT TRANSIENT RESPONSE	109
MODAL COMPLEX EIGENVALUE	110
MODAL FREQUENCY RESPONSE	111
MODAL TRANSIENT RESPONSE	112
NONLINEAR TRANSIENT RESPONSE	129
NONLINEAR STEADY STATE HEAT TRANSFER	153
NONLINEAR TRANSIENT HEAT TRANSFER	159
NONLINEAR BUCKLING	180
PRESTRESS STATIC	181
LINEAR PRESTRESS MODAL	182
LINEAR PRESTRESS FREQUENCY RESPONSE	183
LINEAR PRESTRESS TRANSIENT RESPONSE	184
NONLINEAR PRESTRESS MODAL	185
NONLINEAR PRESTRESS FREQUENCY RESPONSE	186
NONLINEAR PRESTRESS TRANSIENT RESPONSE	187
LINEAR PRESTRESS COMPLEX EIGENVALUE	188
NONLINEAR PRESTRESS COMPLEX EIGENVALUE	189

SPC — Single-Point Constraint Set Selection

Description: Selects the single-point constraint set to be applied to the model.

Format:

SPC = *n*

Example:

SPC = 10

Option	Definition	Type
n	The set identification of a single-point constraint set and hence must appear on a SPC, SPC1, or SPCADD Bulk Data entry.	Integer > 0

Remarks:

1. SPC, SPC1 or SPCADD Bulk Data entries will not be used unless selected in Case Control.
2. SPCD entries cannot be referenced with this command. The LOAD command must be used.

SPCFORCES — Single-Point Forces of Constraint Set Selection

Description: Requests single-point constraint force vector output.

Format:

$$\text{SPCFORCES}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

SPCFORCES = 5

Option	Definition	Type	Default
PRINT	Single-point constraint forces will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Single-point constraint forces will be output only to the results neutral file system.	Character	
PUNCH	Single-point constraint forces will be output additionally to the Model Results Punch File.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
ALL	Single-point constraint forces for all grid points will be output.	Character	
n	Set identification of previously appearing SET command. Only single-point constraint forces for grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Single point constraint forces will not be output.	Character	✓

Remarks:

1. SPCFORCE output is in the global coordinate system. (See CD field on the GRID Bulk Data entry in Section 4, *Bulk Data.*)

STATSUB Static Solution Selection for Differential Stiffness

Description: Selects the static solution to use in forming the differential stiffness matrix for linear buckling, normal modes, and modal response analysis.

Format:

$$\text{STATSUB}\left[\left(\begin{bmatrix}\text{BUCKLING}\\ \text{PRELOAD}\end{bmatrix}\right)\right] = n$$

Example:

STATSUB(PRELOAD) = 3

STATSUB(BUCKLING) = 4

Option	Definition	Type	Default
BUCKLING	Subcase identification number corresponds to a static buckling or varying load.	Character	See Remark 2
PRELOAD	Subcase identification number corresponds to a static preload or constant load.	Character	See Remark 2
n	Subcase identification number of an existing SUBCASE specified for static analysis.	Integer > 0	

Remarks:

1. STATSUB may be used in linear static and modal response solutions (SOL 101, 103, 105, 110, 111, and 112).
2. BUCKLING is the default option for linear buckling and PRELOAD is the default for linear static and modal response solutions.
3. The STATSUB command is not required for linear buckling analysis when a preload is not required. In this case the default for STATSUB is the first static subcase identification.
4. In linear static and modal response solutions only one STATSUB command may be specified. In linear buckling analysis with a preload, both STATSUB(BUCKLING) and STATSUB(PRELOAD) must be specified.

STRAIN — Element Strain Output Request

Description: Requests element strain output.

Format:

$$\text{STRAIN}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{CENTER}\\ \text{CORNER}\\ \text{GAUSS}\end{bmatrix},\begin{bmatrix}\text{SHEAR}\\ \text{VONMISES}\\ \text{TRESCA}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{STRCUR}\\ \text{FIBER}\end{bmatrix},\begin{bmatrix}\text{THERMAL}\\ \text{MECH}\\ \text{TOTAL}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{RALL}\end{bmatrix},\begin{bmatrix}\text{VRMS}\\ \text{BIAX}\\ \text{VALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

STRAIN(VONMISES, CORNER) = 45

Option	Definition	Type	Default
PRINT	Element strains will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Element strains will be output only to the results neutral file system.	Character	
PUNCH	Element strains will be output additionally to the Model Results Punch File.	Character	
CENTER	Output shell and solid element strains at the center only.	Character	✓
CORNER	Output shell and solid element strains at the center and corner nodes.	Character	
GAUSS	Output shell and solid element strains at the center and gauss/integration points.	Character	
SHEAR	Maximum shear strain request for shell elements and octahedral shear strain request for solid elements.	Character	
VONMISES	Von Mises strain request for shell and solid elements.	Character	✓
TRESCA	Tresca strain request for shell and solid elements.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
STRCUR	Strain at reference plane and curvatures are output for shell elements.	Character	
FIBER	Strain at locations Z1 and Z2 are output for shell elements.	Character	✓
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
VRMS	RMS von Mises output request.	Character	

Option	Definition	Type	Default
BIAX	Biaxiality ratio output request.	Character	
VALL	RMS von Mises, RMS principal, RMS maximum shear, and biaxiality ratio will be output.	Character	
THERMAL	Thermal strain request for shell and solid elements.	Character	
MECH	Mechanical strain request for shell and solid elements.	Character	
TOTAL	Total strain (thermal plus mechanical) request for shell and solid elements.	Character	✓
ALL	Element strains for all elements will be output.	Character	
n	Set identification of previously appearing SET command. Only strains for elements whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Element strains will not be output.	Character	✓

Remarks:

1. ELSTRAIN is an alternate form and is identical to STRAIN.
2. Both STRESS and STRAIN cannot be requested in the same subcase.
3. Shell elements must be referenced on a SURFACE and solid elements must be referenced in a VOLUME. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)
4. Solid element invariants are defined as follows:

Octahedral shear strain:

$$\varepsilon_o = \left[\frac{1}{9}\left\{(\varepsilon_x - \varepsilon_y)^2 + (\varepsilon_y - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_x)^2\right\} + \frac{1}{6}\left(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{zx}^2\right)\right]^{\frac{1}{2}}$$

von Mises equivalent strain:

$$\varepsilon_v = \left[\frac{2}{9}\left\{(\varepsilon_x - \varepsilon_y)^2 + (\varepsilon_y - \varepsilon_z)^2 + (\varepsilon_z - \varepsilon_x)^2\right\} + \frac{1}{3}\left(\gamma_{xy}^2 + \gamma_{yz}^2 + \gamma_{zx}^2\right)\right]^{\frac{1}{2}}$$

Tresca strain:

$$\varepsilon_t = |\varepsilon_{max} - \varepsilon_{min}|$$

5. Shell element invariants for plane stress analysis are defined as follows:

   Maximum shear strain:

$$\gamma_{max} = \left[\left(\varepsilon_x - \varepsilon_y \right)^2 + \gamma_{xy}^2 \right]^{\frac{1}{2}}$$

   von Mises equivalent strain:

$$\varepsilon_v = \left[\frac{4}{9} \left(\varepsilon_x^2 + \varepsilon_y^2 - \varepsilon_x \varepsilon_y \right) + \frac{1}{3} \gamma_{xy}^2 \right]^{\frac{1}{2}}$$

   Tresca strain:

$$\varepsilon_t = \left| \varepsilon_{max} - \varepsilon_{min} \right|$$

6. Shell element Tresca stress is defined using the maximum and minimum of three stress measures:
   a) Inplane major principal stress
   b) Inplane minor principal stress
   c) Through thickness stress defined as the negative of the applied pressure at the element surface.
7. VRMS, von Mises RMS strain, is calculated by evaluating the PSD response of the peak RMS strains calculated at each frequency step in a frequency or random response analysis. It is used as a measure of the total component stress.

STRESS — Element Stress Output Request

Description: Request element stress output.

Format:

$$\text{STRESS}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PUNCH}\\\text{PLOT}\end{bmatrix},\begin{bmatrix}\text{CENTER}\\\text{CORNER}\\\text{GAUSS}\end{bmatrix},\begin{bmatrix}\text{SHEAR}\\\text{VONMISES}\\\text{TRESCA}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\\text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\\text{ATOC}\\\text{RALL}\end{bmatrix},\begin{bmatrix}\text{VRMS}\\\text{BIAX}\\\text{VALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\n\\\text{NONE}\end{Bmatrix}$$

Example:

STRESS(SHEAR) = ALL

Option	Definition	Type	Default
PRINT	Element stresses will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Element stresses will be output only to the results neutral file system.	Character	
PUNCH	Element stresses will be output additionally to the Model Results Punch File.	Character	
CENTER	Output shell and solid element stresses at the center only.	Character	✓
CORNER	Output shell and solid element stresses at the center and corner nodes.	Character	
GAUSS	Output shell and solid element stresses at the center and gauss/integration points.	Character	
SHEAR	Maximum shear stress request for shell elements and octahedral shear stress request for solid elements.	Character	
VONMISES	Von Mises stress request for shell and solid elements.	Character	✓
TRESCA	Tresca stress request for shell and solid elements.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
VRMS	RMS von Mises output request.	Character	

Option	Definition	Type	Default
BIAX	Biaxiality ratio output request.	Character	
VALL	RMS von Mises, RMS principal, RMS maximum shear, and biaxiality ratio will be output.	Character	
ALL	Element stresses for all elements will be output.	Character	
n	Set identification of previously appearing SET command. Only stresses for elements whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Element stresses will not be output.	Character	✓

Remarks:

1. ELSTRESS is an alternate form and is identical to STRESS.
2. Both STRESS and STRAIN cannot be requested in the same subcase.
3. Shell elements must be referenced on a SURFACE and solid elements must be referenced in a VOLUME. (See the SURFACE and VOLUME commands in Section 3, *Case Control.*)
4. Solid element invariants are defined as follows:

Mean pressure:

$$p_o = -\frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$$

Octahedral shear stress:

$$\tau_o = \frac{1}{3}\left[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6\tau_{yz}^2 + 6\tau_{zx}^2 + 6\tau_{xy}^2\right]^{\frac{1}{2}}$$

von Mises equivalent stress:

$$\tau_v = \left(\frac{3}{\sqrt{2}}\right)\tau_o$$

Tresca stress:

$$\sigma_t = |\sigma_{max} - \sigma_{min}|$$

5. Shell element invariants for plane stress analysis are defined as follows:

   Maximum shear stress:

$$\tau_{max} = \left[\left(\frac{\sigma_x - \sigma_y}{2} \right)^2 + \tau_{xy}^2 \right]^{\frac{1}{2}}$$

   von Mises equivalent stress:

$$\tau_v = \left[\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2 \right]^{\frac{1}{2}}$$

   Tresca stress:

$$\sigma_t = \left| \sigma_{max} - \sigma_{min} \right|$$

6. Shell element Tresca stress is defined using the maximum and minimum of three stress measures:
   a) Inplane major principal stress
   b) Inplane minor principal stress
   c) Through thickness stress defined as the negative of the applied pressure at the element surface.
7. VRMS, von Mises RMS stress, is calculated by evaluating the PSD response of the peak RMS stresses calculated at each frequency step in a frequency or random response analysis. It is used as a measure of the total component stress.
8. BIAX, Biaxiality Ratio, is the ratio of the minimum and maximum principal stress and is used in conjunction with the von Mises RMS stress to assess the nature of stress components in a frequency or random response analysis. Values that tend towards -1 indicate a pure shear state, 0 indicates uniaxial state, and 1 indicates equal biaxial loading.

SUBCASE — Subcase Delimiter

Description: Delimits and identifies a subcase.

Format:

SUBCASE *n*

Example:

SUBCASE 101

Option	Definition	Type
n	Subcase identification number.	Integer > 0

Remarks:

1. RANDPS requests refer to n. (See Section 4, *Bulk Data*, for more information on the RANDPS Bulk Data entry.)

SUBCOM — Combination Subcase Delimiter

Description: Delimits and identifies a combination subcase.

Format:

SUBCOM *n*

Example:

SUBCOM 205

Option	Definition	Type
n	Subcase identification number.	Integer > 2

Remarks:

1. A SUBSEQ command must follow this command.
2. SUBCOM may only be used in linear problems.
3. Output requests above the subcase level will be used.
4. The following is an example of a simple combination:

```
SUBCASE 101
      LOAD = 101
SUBCASE 102
      LOAD = 102
SUBCOM 110
      LABEL = COMBINE SUBCASES 101 AND 102
      SUBSEQ = 1.0, 1.0
SUBCASE 201
      LOAD = 201
SUBCASE 202
      LOAD = 202
SUBCOM 210
      LABEL = COMBINE SUBCASES 201 AND 202
      SUBSEQ = 1.0, 1.0
```

SUBSEQ

Subcase Sequence Coefficients

Description: Specifies the coefficients for forming a linear combination of previous subcases.

Format:

$\text{SUBSEQ} = \text{R1}\ [,\ \text{R2},\ \text{R3}, \ldots,\ \text{Rn}\ ]$

Example:

SUBSEQ = -1.0, 1.5, 0.0, 3.0

Option	Definition	Type	Default
Ri	Coefficients of the previously occurring subcases. See Remark 4.	Real	0.0

Remarks:

1. The SUBSEQ command can only appear after a SUBCOM command.
2. 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:

```
DISP = 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
```

SUBTITLE — Output Subtitle

Description: Defines a character subtitle which will appear on the second heading line of each page of output.

Format:

SUBTITLE = *Any character string*

Example(s):

SUBTITLE = 2IN. X 10IN. CANTILEVER BEAM

Remarks:

1. Maximum SUBTITLE length is 71 characters.
2. SUBTITLE may appear anywhere in the Case Control Section. If no SUBTITLE command is present, the subtitle line will be blank.
3. SUBTITLE information is also placed on the second line of each results neutral file.

SURFACE — Surface Definition

Description: Shell element results coordinate system definition.

Format:

SURFACE *id*, SET *esid*, [SYSTEM *system*], [AXIS *x-axis*], [NORMAL *normal*]

Example:

SURFACE 12, SET 3, SYSTEM CORD 2, AXIS X, NORMAL Z

Option	Definition	Type	Default
id	Surface identification number.	Integer > 0	Required
SET *esid*	Element set identification number. Set identification of previously appearing SET command. Only shell elements whose identification numbers appear on this SET command will be included as part of the defined SURFACE. The character variable ALL may be used to specify all elements.	Integer > 0 or blank	Required
SYSTEM *system*	Coordinate system for results output, one of the following character variables: ELEMENT, BASIC, MATERIAL, GRID, or CORD followed by a coordinate system identification number.	Character or blank, or integer > 0	See Remark 3
AXIS *x-axis*	Surface x-axis definition, one of the following character variables: R, T, P, X, Y, or Z. See Remark 4.	Character	See Remark 3
NORMAL *normal*	Surface normal definition, one of the following character variables: R, X, Y, or Z. See Remark 4.	Character	See Remark 3

Remarks:

1. The SURFACE command is used to align element normals and define the output coordinate system for shell element and grid point results. A shell element must be defined on a SURFACE in order to have results calculated for it.
2. When the system option is equal to ELEMENT (or MATERIAL with no material coordinate system defined), element normals are not aligned and element results output is in the element coordinate system. Grid point results will default to the global coordinate system.
3. The default SURFACE is defined as ALL shell elements in the coordinate system specified by the ELEMRSLTCORD model parameter (default MATERIAL) and ALL shell element grid points in the global coordinate system.
4. AXIS and NORMAL are ignored when SYSTEM is set to ELEMENT or MATERIAL.

TEMPERATURE

Temperature Set Selection

Description: Selects the temperature set to be used in either the calculation of temperature-dependent material properties or the generation of thermal loads.

Format:

$$\text{TEMPERATURE}\left[\left(\begin{bmatrix}\text{INITIAL}\\ \text{MATERIAL}\\ \text{LOAD}\\ \text{BOTH}\end{bmatrix}\right)\right] = n$$

Examples:

TEMPERATURE(LOAD) = 12

TEMPERATURE(MATERIAL) = 34

TEMPERATURE = 5

Option	Definition	Type	Default
INITIAL	The selected temperature set will be used to determine an initial temperature distribution.	Character	See Remark 6
LOAD	The selected temperature set will be used to determine thermal loads.	Character	See Remark 5
MATERIAL	The selected temperature set will be used to determine temperature-dependent material properties indicated on the MATTi Bulk Data entries.	Character	See Remark 5
BOTH	Both MATERIAL and LOAD will use the same temperature set.	Character	✓
n	Set identification number of TEMP, TEMPD, TEMPP1, or TEMPRB Bulk Data entries.	Integer > 0	

Remarks:

1. For LINEAR STATIC solutions, temperature-dependent material properties are updated for each subcase.
2. Equivalent material properties generated from PCOMP Bulk Data entries are evaluated at the reference temperature specified in the PCOMP entry TREF field.
3. The total load applied will be the sum of external (LOAD command), thermal (TEMPERATURE command), element deformation (DEFORM command), and constrained displacement (SPC command) loads.
4. Static, thermal, and element deformation loads should have unique set identification numbers.
5. If TEMPERATURE(LOAD) is specified without TEMPERATURE(MATERIAL), the thermal load set will be used for the calculation of temperature-dependent material properties.
6. The specification of TEMPERATURE(INITIAL) above the subcase level is recommended in all nonlinear solutions. When TEMPERATURE(INITIAL) is not specified, the initial temperature distribution is obtained from the TREF field on the MATi Bulk Data entry.

TEMPGENERATE Temperature Generation

Description: Grid point temperature generation.

Format:

TEMPGENERATE, *sid*, *esid*, *gradient*, *temperature*, *component*, *cid*

Example:

TEMPGENERATE, 23, 4, 25.34, 100.0, Z, 2

Option	Definition	Type	Default
sid	Generated temperature set identification number.	Integer > 0	Required
esid	Element set identification number. Set identification of previously appearing SET command. Only elements whose identification numbers appear on this SET command will be used.	Integer > 0	All
gradient	Thermal gradient. See Remark 1.	Real	0.0
temperature	Reference temperature. See Remark 1.	Real	0.0
component	Gradient component direction, one of the following character variables: R, T, P, X, Y, or Z.	Character	Required if gradient ≠ 0.0
cid	Gradient component coordinate system.	Integer > 0	0

Remarks:

1. Grid point temperatures (via the TEMP Bulk Data entry) are generated using the following relation:

$$T(\chi) = T'\chi + T_0$$

where,

T' is the specified gradient

χ is the component coordinate in the specified coordinate system

T_0 is the reference temperature

TEMPINTERPOLATE Temperature Interpolation

Description: Interpolates grid point temperature data from a known set of input grid points and temperatures to a set of output grid points and temperatures based on geometric position in 2d or 3d space.

Format:

TEMPINTERPOLATE, *otsid*, *ogsid*, *itsid*, *igsid, nnri, ndlsf, cgsize, maxnus*

Example:

TEMPINTERPOLATE, 100, 10, 1, 1

Option	Definition	Type	Default
otsid	Output temperature set identification number. See Remark 1.	Integer > 0	Required
ogsid	Output grid set identification number. Set identification of previously appearing SET command. Only grid points whose identification numbers appear on this SET command will be used.	Integer > 0 or blank	All grid points in model
itsid	Input temperature set identification number. See Remark 2.	Integer > 0	Required
igsid	Input grid set identification number. Set identification of previously appearing SET command. Only grid points whose identification numbers appear on this SET command will be used.	Integer > 0 or blank	All grid points in temperature set
nnri	Number of interpolation nodes within radius of influence.	Integer > 0 or blank	See Remark 3
ndlsf	Number of data nodes in least squares fit.	Integer > 0 or blank	See Remark 4
cgsize	Number of rows, columns, and planes in the cell grid. A box containing the nodes is partitioned into cells in order to increase search efficiency.	Integer > 0 or blank	See Remark 5
maxnus	Maximum number of unique solution occurrences.	Integer > 0 or blank	See Remark 6

Remarks:

1. Output is TEMP Bulk Data entries at grid points defined by *ogsid*.
2. Input is GRID and TEMP Bulk Data entries which need not be associated with the analysis model. See Section 4, *Bulk Data*, for more information on GRID and TEMP Bulk Data entries.
3. The valid range for *nnri* is $1 \leq nnri \leq \min(100, n-1)$), where *n* is the number of input data points. The default is 100. A lower value may increase performance at the cost of accuracy. A value greater than or equal to 32 is recommended.
4. The valid range for *ndlsf* is $9 \leq ndlsf \leq \min(100, n-1)$, where *n* is the number of input data points. The default is 100. A lower value may increase performance at the cost of accuracy. A value greater than or equal to 17 is recommended.

5. The recommended value for *cgsize* is:

$$cgsize = \left(\frac{n}{3}\right)^{\frac{1}{3}}$$

   where *n* is the number of input data points. The default is determined using the above formula.

6. A 3d interpolation algorithm is used initially, but will automatically revert to a 2d algorithm if the number of no unique solution errors exceeds *maxnus* while processing the input data points. Models that are dominantly flat but still have 3d features that default to the 2d interpolation algorithm may not be interpolated accurately. A larger *maxnus* value can be used to force a 3d interpolation. It is advisable to always check the interpolated loads.

7. Generated TEMP Bulk Data entries can be exported using the TRSLBULKDATA Model Initialization directive. (See Section 2, *Initialization*, for more information on TRSLBULKDATA.)

TEMPSCALEFACTOR

Temperature Scale Factor

Description: Specifies scale factors for the generation of grid point temperatures from existing temperature set definitions.

Format:

TEMPSCALEFACTOR, *sid, scale, xsid*

Example:

TEMPSCALEFACTOR, 2, 2.5, 1

Option	Definition	Type	Default
sid	Generated set identification number.	Integer > 0	Required
scale	Scale factor applied to temperatures specified on temperature entries that reference the specified existing temperature set.	Real	Required
xsid	Set identification number of an existing temperature set.	Integer > 0	Required

Remarks:

1. Grid point temperatures (via the TEMP Bulk Data entry) are generated using this command.

THERMAL

Temperature Output Request

Description: Requests grid point temperature output.

Format:

$$\text{THERMAL}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

THERMAL = 10

Option	Definition	Type	Default
PRINT	Grid point temperatures will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point temperatures will be output only to the results neutral file system.	Character	
PUNCH	Grid point temperatures will be output additionally to the Model Results Punch File.	Character	
ALL	Temperatures for all grid points will be output.	Character	
n	Set identification number of a previously appearing SET command. Only temperatures of points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point temperatures will not be output.	Character	✓

Remarks:

1. Temperature output is only available for in heat transfer solutions.

TITLE

Output Title

Description: Defines a character title that will appear on the first heading line of each page of output.

Format:

TITLE = *Any character string*

Example:

TITLE = F22 Wing Box

Remarks:

1. Maximum TITLE length is 71 characters.
2. TITLE may appear anywhere in the Case Control Section. If no TITLE command is present, the title line will be blank.
3. TITLE information is also placed on the second line of each results neutral file.

TSTEP — Transient Time Step Set Selection for Linear Analysis

Description: Select integration and output time steps for linear transient response problems.

Format:

TSTEP = *n*

Example:

TSTEP = 35

Option	Definition	Type
n	Set identification number of a TSTEP Bulk Data entry.	Integer > 0

Remarks:

1. A TSTEP entry must be selected to perform transient response analysis.

TSTEPNL Transient Time Step Set Selection for Nonlinear Analysis

Description: Select integration and output time steps for nonlinear transient response problems.

Format:

TSTEPNL = *n*

Example:

TSTEPNL = 45

Option	Definition	Type
n	Set identification number of a TSTEPNL Bulk Data entry.	Integer > 0

Remarks:

1. A TSTEPNL entry must be selected to perform nonlinear transient response analysis.

VECTOR — Displacement Output Requests

Description: Requests displacement vector output.

Format:

$$\text{VECTOR}\left[\left(\begin{bmatrix}\text{PRINT}\\\text{PLOT}\\\text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\\text{PHASE}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\\text{ATOC}\\\text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\n\\\text{NONE}\end{Bmatrix}$$

Example:

VECTOR = ALL

Option	Definition	Type	Default
PRINT	Grid point displacements will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point displacements will be output only to the results neutral file system.	Character	
PUNCH	Grid point displacements will be output additionally to the Model Results Punch File.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
ALL	Displacements for all grid points will be output.	Character	✓
n	Set identification of previously appearing SET command. Only displacements of grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point displacements will not be output.	Character	

Remarks:

1. VECTOR displacement results are output in the global coordinate system. (See CD field on the GRID Bulk Data entry in Section 4, *Bulk Data.*)
2. The translation components are in the same units of measure as the model. The rotation components are in radians.

VELOCITY

Velocity Output Request

Description: Requests velocity vector output.

Format:

$$\text{VELOCITY}\left[\left(\begin{bmatrix}\text{PRINT}\\ \text{PLOT}\\ \text{PUNCH}\end{bmatrix},\begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix},\begin{bmatrix}\text{ABS}\\ \text{REL}\end{bmatrix},\begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{RALL}\end{bmatrix}\right)\right]=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Example:

VELOCITY = 25

Option	Definition	Type	Default
PRINT	Grid point velocities will be output to both the Model Results Output File and the results neutral file system.	Character	✓
PLOT	Grid point velocities will be output only to the results neutral file system.	Character	
PUNCH	Grid point velocities will be output additionally to the Model Results Punch File.	Character	
REAL or IMAG	Requests complex output in rectangular format (real and imaginary).	Character	✓
PHASE	Requests complex output in polar format (magnitude and phase). Phase output is in degrees.	Character	
ABS	Requests output as absolute displacement (see Remark 2).	Character	✓
REL	Requests output as relative displacement (see Remark 2).	Character	
PSDF	Power spectral density function, RMS, and number of positive crossings output request.	Character	
ATOC	Autocorrelation function output request.	Character	
RALL	Both PSDF and ATOC will be output.	Character	
ALL	Velocities for all grid points will be output.	Character	✓
n	Set identification of previously appearing SET command. Only velocities of grid points whose identification numbers appear on this SET command will be output.	Integer > 0	
NONE	Grid point velocities will not be output.	Character	

Remarks:

1. Velocity results are output in the global coordinate system. (See CD field on the GRID Bulk Data entry in Section 4, *Bulk Data.*)

2. Relative velocity output is only applicable to modal transient and linear direct transient response solutions. The reference point for relative motion is defaulted to the direct enforced motion input point. When direct enforced motion is not specified the point with the largest mass in the model is used. The reference point may be specified explicitly using the DYNSOLRELGRID model parameter. See Section 5, *Parameters*, for more information on DYNSOLRELGRID.

VIBFATIGUE

Vibration Fatigue Analysis Data Set Selection

Description: Selects the VFATIGUE Bulk Data entry to be used in vibration fatigue analysis.

Format:

VIBFATIGUE = *n*

Example:

VIBFATIGUE = 15

Option	Definition	Type
n	Set identification of a VIBFATIGUE Bulk Data entry to be used in vibration fatigue analysis.	Integer > 0

Remarks:

1. VIBFATIGUE must reference a VIBFATIGUE Bulk Data entry to perform vibration fatigue analysis.

VOLUME — Volume Definition

Description: Solid element results coordinate system definition.

Format:

VOLUME *id*, SET *esid*, [SYSTEM *system*]

Example:

VOLUME 12, SET 3, SYSTEM BASIC

Option	Definition	Type	Default
id	Volume identification number.	Integer > 0	Required
SET *esid*	Element set identification number. Set identification of previously appearing SET command. Only solid elements whose identification numbers appear on this SET command will be included as part of the defined SURFACE. The character variable ALL may be used to specify all elements.	Integer > 0 or blank	ALL
SYSTEM *system*	Coordinate system for results output, one of the following character variables: ELEMENT, BASIC, MATERIAL, GRID, or CORD followed by a coordinate system identification number.	Character or blank, or integer > 0	See Remark 3

Remarks:

1. The VOLUME command is used to define the output coordinate system for solid element and grid point results. A solid element must be defined on a VOLUME in order to have results calculated for it.
2. When the system option is equal to ELEMENT (or MATERIAL with no material coordinate system defined), element results output is in the element coordinate system. Grid point results will default to the global coordinate system.
3. The default VOLUME is defined as ALL solid elements in the coordinate system specified by the ELEMRSLTCORD model parameter (default MATERIAL) and ALL solid element grid points in the global coordinate system.

WELDGENERATE

Spot Weld Element Generation

Description: CWELD element generation. Converts a specified set of CBAR elements into CWELD elements.

Format:

WELDGENERATE, *ftype, ctype, esid, diameter*

Examples:

WELDGENERATE, ELEMID, SPOT, 1, 0.3

WELDGENERATE, ALIGN, GENERAL, 2, 0.1

Option	Definition	Type	Default
ftype	Connection format type, one of the following character variables: ELEMID or ALIGN. See Remark 1. ELEMID Connection to the shell element nearest to the reference bar element end point. ALIGN Connection to one or more shell element vertex grid points.	Character	Required
ctype	Weld connection type, one of the following character variables: SPOT or GENERAL. See Remark 2. SPOT Weld type connection. GENERAL General connection.	Character	GENERAL
esid	Element set identification number. Set identification of previously appearing SET command. Only bar elements whose identification numbers appear on this SET command will be used.	Integer > 0	See Remark 3
diameter	Diameter of the connector. See Remark 4.	Real > 0.0 or blank	See Remark 4

Remarks:

1. Both ELEMID and ALIGN function similarly to the corresponding options in the CWELD Bulk Data entry. For *ftype* = ELEMID, connection will be to the shell element with its origin nearest the reference bar element end point. For *ftype* = ALIGN, the reference bar element is already connected to a shell element vertex.
2. For *ctype* = SPOT and *ftype* = ELEMID, the effective length for the stiffness of the weld element is set to $\ell_e = (t_A + t_B)/2$ regardless of the reference bar element distance GA to GB. t_A and t_B are the shell thicknesses of SHIDA and SHIDB which are located automatically based on proximity. For all other cases, the effective length of the weld element is equal to the true length, the distance of the reference bar GA to GB, provided the ratio of length to diameter is in the range $0.2 \le L/D \le 5.0$. If L is below this range, the effective length is set to $\ell_e = 0.2D$ and if L is above this range, the effective length is set to $\ell_e = 5.0D$.

3. If *esid* is blank, all CBAR elements will be converted to CWELD elements.
4. The reference bar element material property will be used for the corresponding CWELD element generated. If *diameter* is not specified, the reference bar area will be used to generate an equivalent diameter.
5. See the CWELD and PWELD Bulk Data entries for more information.

XSETGENERATE — Degree of Freedom Set Generation

Description: ASET and ESET degree of freedom set generation.

Format:

XSETGENERATE, *stype*, *method*, *gsid*, *ptol, component*

Examples:

XSETGENERATE, ASET, SURFACE, , , 123

XSETGENERATE, ESET, INTER, , 0.1, 123456

Option	Definition	Type	Default
stype	Target output set type, one of the following character variables: ASET or ESET.	Character	Required
method	The search method used, one of the following character variables: SURFACE or SET for ASET or NEAR or INTER for ESET. See Remark 1.	Character	Required
gsid	Grid set identification number. Set identification of previously appearing SET command. Only grid points whose identification numbers appear on this SET command will be used.	Integer > 0	See Remark 2
ptol	Position tolerance used for ESET generation. Grid points defined in the XSET within a radius equal to *ptol* are moved into the ESET.	Real or blank	See Remark 3
component	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6 or blank	123456

Remarks:

1. The *method* field defines how the set will be generated. SURFACE and SET are only applicable to ASET generation. When *method* is set to SURFACE, only grid points on the exterior of the model will be included in the ASET. When *method* is set to SET, only grid points listed in the output set defined by *setid* are included. NEAR and INTER are applicable to ESET generation. Both methods look for grid points in the model near points defined in the XSET within a radius defined by *ptol*. The INTER method interpolates data in each *component* direction specified at the near point using the XSET data.
2. Required if *method* is equal to SET.
3. If *ptol* is blank or zero and *method* is set to INTER, all grid points not in the XSET will be moved into the ESET. If an ESET is already defined, the ESET will not be changed.

XYDATA — Generate X-Y Plots at a Specified Grid Point or Element

Description: Requests the generation of results x-y plots at a specified grid point or element.

Format:

XYDATA, *gid/eid, component/column, group, stype*

Example:

XYDATA, 10, 3, 1, GRID

XYDATA, 15, 22, 3, ELEM

Option	Definition	Type	Default
gid	Grid point identification number for *stype* equals GRID.	Integer > 0	Required
eid	Element identification number for *stype* equals ELEM.	Integer > 0	Required
component	Component number of global coordinate for *stype* equals GRID. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	123456
column	Results column number for *stype* equals ELEM. See Remark 4.	Integer > 0	Required
group	Group identification number.	Integer > 0	0
stype	Output set identification type, one of the following character variables: GRID or ELEM.	Character	GRID

Remarks:

1. A separate plot is generated for each vector result requested in the Case Control.
2. XYDATA commands with the same group identification number will be plotted on the same x-y axes.
3. The XYPLOTCSVOUT directive can be used to generate an MS Excel Comma Separated Variable file containing the plot data in tabular form. See Section 2, *Initialization*, for more information on XYPLOTCSVOUT.
4. See Appendix A, *Results Neutral File Formats,* for result column number definition.

XYDATAGENERATE — Generate X-Y Plots at Specified Grid Points or Elements

Description: Requests the generation of results x-y plots at specified grid points or elements.

Format:

XYDATAGENERATE, *gsid/esid, component/column, group, stype*

Examples:

XYDATAGENERATE, 5, 1, 2, GRID

XYDATAGENERATE, 15, 22, 3, ELEM

Option	Definition	Type	Default
gsid	Grid set identification number for *stype* equals GRID. Set identification of previously appearing SET command. Only grid points whose identification numbers appear on this SET command will be used.	Integer > 0	Required
esid	Element set identification number for *stype* equals ELEM. Set identification of previously appearing SET command. Only elements whose identification numbers appear on this SET command will be used.	Integer > 0	Required
component	Component number of global coordinate for *stype* equals GRID. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	123456
column	Results column number for *stype* equals ELEM. See Remark 4.	Integer > 0	Required
group	Group identification number.	Integer > 0	0
stype	Output set identification type, one of the following character variables: GRID or ELEM.	Character	GRID

Remarks:

1. A separate plot is generated for each vector result requested in the Case Control.
2. XYDATA commands with the same group identification number will be plotted on the same x-y axes.
3. The XYPLOTCSVOUT directive can be used to generate an MS Excel Comma Separated Variable file containing the plot data in tabular form. See Section 2, *Initialization*, for more information on XYPLOTCSVOUT.
4. See Appendix A, *Results Neutral File Formats,* for result column number definition.

Section 4

BULK DATA

The Bulk Data Section

The Bulk Data Section contains entries that define the model. This consists of model geometry, element connectivity, element and material properties, constraints, and loads. Certain entries, such as loads and constraints, are not active unless selected by an appropriate Case Control command.

Bulk Data Entry Descriptions

Each Bulk Data entry is described using the following format:

Description
A single sentence **Description** is given which states the function of the Bulk Data entry.

Format
The entry syntax is defined under **Format**. The first field gives the entry name. The following fields are referenced under **Field** and **Definition**. Light shaded fields are optional. Dark shaded fields must be left blank. If field 10 is dark shaded, then no continuation entries are permitted.

Example
A typical example is given under **Example**.

Field, Definition, Type, and Default
Each of the fields 2 through 9 that are named under **Format** is briefly described under **Definition**. The field's type (e.g., Integer, Real, or Character) and allowable range are specified under **Type**. If the field has a default, then it will be given under **Default**. If user input is required, then "Required" will be specified.

Remarks
Additional information about the entry is given under **Remarks**.

$ Comment

Description: Used to add comments to the Model Input File.

Format:

$ followed by any characters out to column 80.

Example:

```
$ NITROGEN TANK PROPERTIES
```

Remarks:

1. Comments are ignored by the program and may appear anywhere within the Model Input File.
2. Comments will not appear in either the sorted or unsorted echo of the Bulk Data or in the Bulk Data File.

ASET Analysis Set Definition

Description: Defines degrees of freedom in the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
ASET	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

ASET	15	3	17	456	7	4			

Field	Definition	Type	Default
Gi	Grid point identification number(s).	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required

Remarks:

1. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees of freedom not otherwise constrained (i.e., SPCi or MPC entries) will be placed in the omitted set (o-set).
2. ASET generation can be automated using the XSETGENERATE Case Control command.

ASET1 — Analysis Set Definition, Alternate Form

Description: Defines degrees of freedom in the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
ASET1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	- etc.-					

Example:

ASET1	123	6	3	7	10	18	14	11	
	19	23							

Alternate Format and Example:

ASET1	C	G1	THRU	G2					

ASET1	456	15	THRU	512					

Field	Definition	Type	Default
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required
Gi	Grid point identification number(s).	Integer > 0; G1 < G2	Required

Remarks:

1. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees of freedom not otherwise constrained (i.e., SPCi or MPC entries) will be placed in the omitted set (o-set).
2. If the alternate form is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will be skipped.
3. ASET generation can be automated using the XSETGENERATE Case Control command.

BAROR CBAR Entry Default Values

Description: 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		

Example:

BAROR		56			0.5	2.7	-3.2		

Alternate Format and Example:

BAROR		PID			G0				

BAROR		46			14				

Field	Definition	Type	Default
PID	Property identification number of a PBAR entry.	Integer > 0	Required
X1, X2, X3	Components of vector $\vec{\mathbf{v}}$, from GA, in the displacement coordinate system at GA (see Figure 1).	Real or blank	
G0	Grid point identification number to optionally supply X1, X2, and X3. Direction of orientation vector is GA to G0.	Integer 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. If field 6 is an integer, then G0 is used. If field 6 is blank or real, then X1, X2, X3 is used.

BCONP — Slide Line Contact Parameters

Description: Defines the parameters for a slide line contact region.

Format:

1	2	3	4	5	6	7	8	9	10
BCONP	ID	SECONDARY	PRIMARY		SFACT	FRICID	PTYPE	CID	
	V0	TMAX	MAR	TRMIN	SMAX	CTC			

Example:

BCONP	15	10	20		10.0		2		

Field	Definition	Type	Default
ID	Contact region identification number.	Integer > 0	Required
SECONDARY	Secondary region identification number.	Integer > 0	Required
PRIMARY	Primary region identification number.	Integer > 0	Required
SFACT	Stiffness scaling factor used to scale the penalty values determined automatically. See Remark 4.	Real > 0.0	1.0
FRICID	Contact friction identification number. See Remark 5.	Integer ≥ 0 or blank	
PTYPE	Penetration type. See Remarks 6 and 7. 1 = Unsymmetric general contact (slave penetration only) 2 = Symmetric general contact 3 = Unsymmetric welded contact 4 = Symmetric welded contact 5 = Unsymmetric bi-directional sliding contact 6 = Symmetric bi-directional sliding contact 7 = Unsymmetric rough contact 8 = Symmetric rough contact	1 ≤ Integer ≤ 8	1
CID	Coordinate system identification number to define plane of contact. See Remark 9.	Integer ≥ 0 or blank	0
V0	Penetration edge offset. See Remark 10.	Real	0.0
TMAX	Maximum allowable penetration used in the adjustment of penalty values normal to the slide line. A positive value activates the penalty value adjustment. See Remark 11.	Real ≥ 0.0	See Remark 11
MAR	Maximum allowable adjustment ratio for adaptive penalty values K and FSTIF. See Remark 12.	Real > 1.0	100.0
TRMIN	Fraction of TMAX defining the lower bound for the allowable penetration. See Remark 13.	0.0 ≤ Real ≤ 1.0	0.001

Field	Definition	Type	Default
SMAX	Maximum allowable slip used in the adjustment of penalty values parallel to the contact plane (FSTIF). A positive value activates the penalty value adjustment. See Remark 14.	Real ≥ 0.0	0.0
CTC	Contact thermal conductance. See Remark 15.	Real ≥ 0.0	∞

Remarks:

1. Contact region identification number must be unique with respect to all other BCONP identification numbers.
2. The SLAVE field defines the slave line by referencing a BLSEG Bulk Data entry. The width of each slave segment is defined via the BWIDTH Bulk Data entry. The width must be defined to get the proper contact stress if symmetrical penetration is specified.
3. The MASTER field defines the master line by referencing a BLSEG Bulk Data entry. The width of each master segment is defined via the BWIDTH Bulk Data entry. The width must be defined to get the proper contact stress.
4. SFACT may be used to scale the penalty values that are determined automatically based on adjacent diagonal stiffness matrix coefficients. Additionally, penalty values calculated may be further scaled by the SLINEKSFACT model parameter (see Section 5, *Parameters*, for more information on SLINEKSFACT). The penalty value is then equal to $k * SFACT * |SLINEKSFACT|$, where k is a value selected for each slave node based on the diagonal stiffness matrix coefficient and SFACT is specified in the SFACT field above. Note that the SLINEKSFACT value applies to all contact regions in the model. Penalty values are normally recalculated every time there is a change in stiffness. However, if SLINEKSFACT is negative, penalty values are not recalculated. This setting is recommended if problems with convergence are encountered.
5. The referenced FRICIC is the identification number of the BFRIC Bulk Data entry. The BFRIC defines friction properties for the contact region.
6. For unsymmetric contact, only the penetration of the slave node into the master segments is checked. This may lead to the master nodes penetrating the slave segments. This error is reduced as the mesh density is increased. For symmetric penetration, both the slave and master nodes are checked for penetration. This is accomplished by generating a slave node, master segment element using the MASTER line for the slave nodes and the SLAVE line for the master segments.
7. Welded contact behavior is accomplished by selecting the unsymmetric or symmetric welded contact setting (3 or 4). With either setting the element will behave the same in tension as in compression and will not slide. Note that for linear solutions general contact will default to welded behavior. Bi-directional sliding contact behavior is accomplished by selecting the unsymmetric or symmetric bi-directional contact setting (5 or 6). With either setting the element will act similar to a welded contact element in tension and compression, but will slide in-plane. Bi-directional sliding contact is intended for use on planar surfaces and is available in all solutions. Rough contact behavior is accomplished by selecting the unsymmetric or symmetric rough contact setting (7 or 8). With either setting the element will act similar to a general contact element in tension and compression, but will not permit sliding in-plane.
8. This element will default to welded contact in linear solutions including linear static analysis with linear contact enabled. A nonlinear solution must be selected for general contact behavior.

9. Figure 1 shows a typical slide line contact definition. The slide line coordinate system z-axis defines the slide line contact plane. An alternate coordinate axis other than the z-axis may be specified using PARAM, SLINEPLANEZDIR (see Section 5, *Parameters*, for more information on SLINEPLANEZDIR). Relative motions outside the slide line plane are ignored and should be small compared to a typical master segment. The normal direction for a slide line segment is formed from the cross product of the slide line plane vector and the vector from master node 1 to master node 2. The definition of the coordinate system should be such that the normal direction points toward the slave region. For symmetric penetration the normals of the master and slave segments must face each other. This is generally accomplished by ordering the nodes on the master and slave lines either clockwise or counterclockwise depending on the direction of the slide line plane.
10. A positive value of V0 offsets the contact line in the element y-direction and results in a contact condition occurring when a slave node penetrates the offset line.
11. There are two methods for adaptive stiffness updates normal to the slide line: proximity stiffness based and displacement based.
    a) When TMAX ≠ 0.0, the displacement based stiffness update method is selected. The value specified defines the allowable penetration of the slave node into the master line. The recommended TMAX value is between 1% and 10% of the element thickness for plates or the equivalent thickness for other elements that are connected to the contact element.
    b) When TMAX = 0.0 (default), the update method selected is dependent on the SLINESLIDETYPE and SLINEMAXDISPTOL model parameter settings. When SLINESLIDETYPE is set to DYNAMIC, the proximity stiffness based update method is selected. When SLINESLIDETYPE is set to STATIC, the displacement based stiffness update method is selected where SLINEMAXDISPTOL defines the default TMAX value using

    $$\mathrm{TMAX} = \mathrm{SLINEMAXDISPTOL} * \ell$$

    where ℓ is the total length of the master slide line. See Section 5, *Parameters*, for more information on SLINESLIDETYPE and SLINEMAXDISPTOL.
12. The maximum adjustment ratio MAR defines the upper and lower bounds of the adjusted value by

    $$\frac{K_{initial}}{\mathrm{MAR}} \le K \le K_{initial} * \mathrm{MAR}$$
13. TRMIN is used for the penalty value adjustment and defines the lower bound for the allowable penetration computed by TRMIN $*$ TMAX. The penalty values are decreased if the penetration is below the lower bound.
14. There are two methods for adaptive stiffness updates parallel to the contact plane: proximity stiffness based and displacement based. If SMAX ≠ 0.0, the displacement based update method is selected. When SMAX = 0.0 (default), the proximity stiffness based update method is selected. If FSTIF is specified it will be used as the penalty stiffness for stick when the proximity stiffness method is used. If SMAX ≠ 0.0, the FSTIF value will be adjusted internally to achieve the SMAX displacement specified.
15. The thermal contact conductance CTC is defined as

    $$C_{tc} = q / \Delta T$$

    where ΔT is the change in temperature between the slave node and average of the master nodes and q is the heat flux through the slide line. Thermal contact conductance is only applicable in heat transfer solutions.

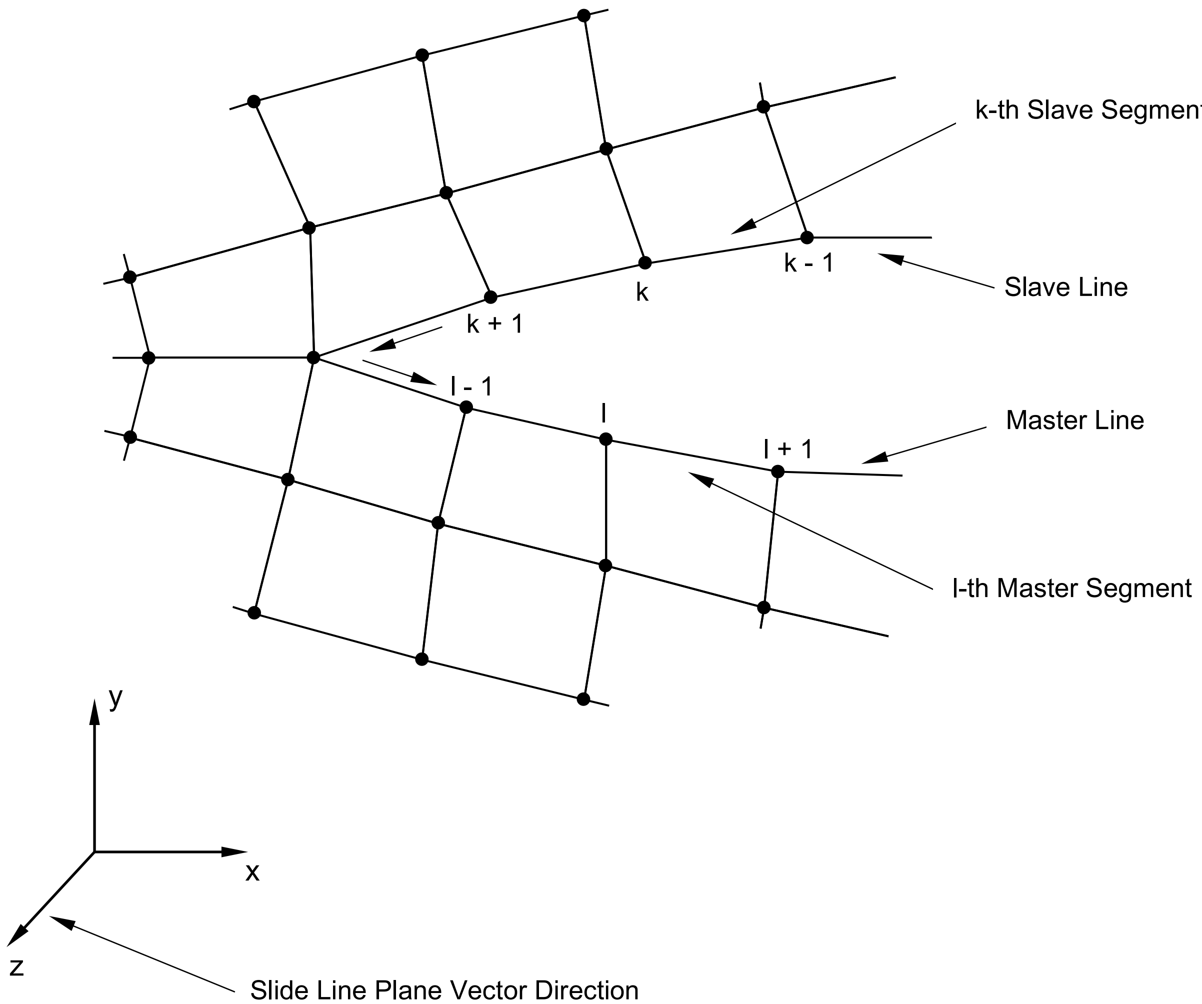


Figure 1. Slide Line Contact Definition.

BEAMOR — CBEAM Entry Default Values

Description: 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		

Example:

BEAMOR		56			0.5	2.7	-3.2		

Alternate Format and Example:

BEAMOR		PID			G0				

BEAMOR		46			14				

Field	Definition	Type	Default
PID	Property identification number of a PBEAM entry.	Integer > 0	Required
X1, X2, X3	Components of vector $\vec{\mathbf{v}}$, from GA, in the displacement coordinate system at GA (see Figure 1).	Real or blank	
G0	Grid point identification number to optionally supply X1, X2, and X3. Direction of orientation vector is GA to G0.	Integer or blank	

Remarks:

1. The contents of fields on this entry will be assumed for any CBEAM entry whose corresponding fields are blank.
2. Only one BEAMOR entry is allowed.
3. If field 6 is an integer, then G0 is used. If field 6 is blank or real, then X1, X2, X3 is used.

BFRIC — Contact Friction

Description: Defines frictional properties between two bodies in slide line contact.

Format:

1	2	3	4	5	6	7	8	9	10
BFRIC	FID		FSTIF	MU					

Example:

BFRIC	15			0.1					

Field	Definition	Type	Default
FID	Friction identification number.	Integer > 0	Required
FSTIF	Frictional stiffness for stick. See Remark 3.	Real ≥ 0.0	Model dependent
MU	Coefficient of static friction.	Real ≥ 0.0	0.0

Remarks:

1. Friction identification number must be unique with respect to all other BFRIC identification numbers.
2. This entry is used in the FRICID field of the BCONP Bulk Data entry.
3. The value of frictional stiffness should be chosen carefully. A method of choosing a value is to divide the expected frictional strength (MU ∗ expected normal force) by a reasonable value of the relative displacement which may be permitted before slip occurs. A large stiffness value may cause poor convergence, while too small a value may cause reduced accuracy.

BLSEG Boundary Line Segments

Description: Defines a curve which is comprised of a number of line segments via grid points that may come in contact with another curve.

Format:

1	2	3	4	5	6	7	8	9	10
BLSEG	ID	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	- etc.-					

Example:

BLSEG	2	3	5	7	9	11	13	15	
	17	21	27						

Alternate Format and Example:

BLSEG	ID	G1	THRU	G2	BY	INC			

BLSEG	10	23	THRU	55	BY	2			

Field	Definition	Type	Default
ID	Boundary line identification number.	Integer > 0	Required
Gi	Grid point identification number(s). Grid points form line segments of a curve and must be ordered so that the normal to the segment points toward the other curve. See Remark 2.	Integer > 0	Required
INC	Grid point identification number increment.	Integer or blank	

Remarks:

1. Boundary line identification numbers must be unique with respect to all other BLSEG and BSSEG entries.
2. A line segment is defined between every two consecutive grid points. The number of segments defined equals the number of grid points specified minus one.
3. The width of each segment is defined via the BWIDTH Bulk Data entry. The BWIDTH entry requires the same ID as the BLSEG entry. For each segment defined on the BLSEG entry a corresponding width is defined on the BWIDTH entry.
4. The normal to the segment is determined by the cross product of the slide line plane vector (i.e., the z-direction of the coordinate system defined in the CID field of the BCONP Bulk Data entry) and the vector formed from node 1 to node 2 of the segment.

BOLT Bolt Definition

Description: Selects CBEAM or CBAR elements for bolt preload analysis.

Format:

1	2	3	4	5	6	7	8	9	10
BOLT	BID		EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	EID8	- etc.-						

Example:

BOLT	10		15	18	22	25	32	45	
	47	51							

Alternate Format and Example:

BOLT	BID		EID1	THRU	EID2	BY	INC		

BOLT	10		11	THRU	15				

Field	Definition	Type	Default
BID	Bolt identification number.	Integer > 0	Required
EIDi	Element identification number of CBEAM or CBAR element(s) to be included in bolt preload analysis.	Integer > 0	Required

Remarks:

1. Bolt preloads are supported in the following solutions:

Solution Character Variable	Solution Number
LINEAR STATIC	101
LINEAR BUCKLING	105
NONLINEAR STATIC	106
NONLINEAR TRANSIENT RESPONSE	129
NONLINEAR BUCKLING	180
PRESTRESS STATIC	181
LINEAR PRESTRESS MODAL	182
LINEAR PRESTRESS FREQUENCY RESPONSE	183
LINEAR PRESTRESS TRANSIENT RESPONSE	184
NONLINEAR PRESTRESS MODAL	185
NONLINEAR PRESTRESS FREQUENCY RESPONSE	186
NONLINEAR PRESTRESS TRANSIENT RESPONSE	187
LINEAR PRESTRESS COMPLEX EIGENVALUE	188
NONLINEAR PRESTRESS COMPLEX EIGENVALUE	189

2. In buckling solutions (105 and 180) both the bolt preload and externally applied loads will be scaled to determine the critical load.

BOLTFOR — Preload Force on Bolt Elements

Description: Defines a preload force applied to bolt elements.

Format:

1	2	3	4	5	6	7	8	9	10
BOLTFOR	SID	LOAD	B1	B2	B3	B4	B5	B6	
	B7	B8	- etc.-						

Example:

BOLTFOR	10	1500.0	15	18	22	25	32	45	
	47	51	57						

Alternate Format and Example:

BOLTFOR	SID	LOAD	B1	THRU	B2	BY	INC		

BOLTFOR	10	1500.0	11	THRU	15				

Field	Definition	Type	Default
SID	BOLTLD set identification number.	Integer > 0	Required
LOAD	Preload force.	Real	Required
Bi	Bolt identification number(s).	Integer > 0; EID1 < EID2	Required
INC	Bolt identification number increment.	Integer or blank	

Remarks:

1. Bolt preload analysis sets must be selected in the Case Control Section (BOLTLD = SID).
2. If the alternate form is used, all bolts B1 through B2 that do not exist will be skipped.
3. The same bolt id must not be specified more than once.

BOUTPUT Output Slide Line Contact

Description: Specifies slave nodes for slide line contact output.

Format:

1	2	3	4	5	6	7	8	9	10
BOUTPUT	ID	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	- etc.-					

Example:

BOUTPUT	2	3	5	7	9	11	13	15	
	17	21	27						

Alternate Format and Example:

BOUTPUT	ID	G1	THRU	G2	BY	INC			

BOUTPUT	10	23	THRU	55	BY	2			

Field	Definition	Type	Default
ID	Corresponding BLSEG entry identification number. See Remark 1.	Integer > 0	Required
Gi	Grid point identification number of the slave node for which output is requested. See Remark 2.	Integer > 0	Required
INC	Grid point identification number increment.	Integer or blank	

Remarks:

1. The BOUTPUT entry requires the same ID as the BLSEG entry.
2. For each segment defined on the BLSEG entry a corresponding output request is defined on the BOUTPUT entry. The ALL character variable may be used to request output for all segments.

BSCONP Surface Contact Parameters

Description: Defines the parameters for a surface contact region.

Format:

1	2	3	4	5	6	7	8	9	10
BSCONP	ID	SECONDARY	PRIMARY	SFACT	FSTIF	MU	PTYPE	MAXAD	
	W0	TMAX	MAR	TRMIN	MAXRAD	MAXNAD	SMAX	CTC	
	FT	SDMAXT	SDMAXS	UDINITT	UDINITS	UDMAXT	UDMAXS	PRSFIT	

Example:

BSCONP	11	2	5			0.2	2		
								1.0+5	

Field	Definition	Type	Default
ID	Contact region identification number.	Integer > 0	Required
SECONDARY	Secondary region identification number.	Integer > 0	Required
PRIMARY	Primary region identification number.	Integer > 0	Required
SFACT	Stiffness scaling factor used to scale the penalty values determined automatically. See Remark 4.	Real > 0.0	1.0
FSTIF	Frictional stiffness for stick. See Remarks 5 and 12.	Real ≥ 0.0	Model dependent
MU	Coefficient of static friction.	Real ≥ 0.0	0.0
PTYPE	Penetration type. See Remarks 6 and 7. 1 = Unsymmetric general contact (slave penetration only) 2 = Symmetric general contact 3 = Unsymmetric welded contact 4 = Symmetric welded contact 5 = Unsymmetric bi-directional sliding contact 6 = Symmetric bi-directional sliding contact 7 = Unsymmetric rough contact 8 = Symmetric rough contact 9 = Unsymmetric offset welded contact 10 = Symmetric offset welded contact 11 = RBE3 element	1 ≤ Integer ≤ 11	1
MAXAD	Maximum activation distance. See Remark 8.	Real ≥ 0.0 or AUTO	See Remark 8
W0	Penetration surface offset. See Remark 9.	Real	0.0

Field	Definition	Type	Default
TMAX	Maximum allowable penetration used in the adjustment of penalty values normal to the contact plane. A positive value activates the penalty value adjustment. See Remark 10.	Real ≥ 0.0	See Remark 10
MAR	Maximum allowable adjustment ratio for adaptive penalty values K and FSTIF. See Remark 11.	Real > 1.0	100.0
TRMIN	Fraction of TMAX defining the lower bound for the allowable penetration. See Remark 12.	0.0 ≤ Real ≤ 1.0	0.001
MAXRAD	Maximum radial activation distance. See Remark 13.	Real ≥ 0.0	0.0
MAXNAD	Maximum normal activation distance. See Remark 13.	Real ≥ 0.0	0.0
SMAX	Maximum allowable slip used in the adjustment of penalty values parallel to the contact plane (FSTIF). A positive value activates the penalty value adjustment. See Remark 14.	Real ≥ 0.0	0.0
CTC	Thermal contact conductance. See Remark 15.	Real ≥ 0.0	∞
FT	Failure theory. The following weld bond failure theories are allowed. WFM for the Weld Failure Model CZM for the Cohesive Zone Model	Character or blank	WFM
SDMAXT	Tensile stress of the weld bonding material when damage initiates. See Remark 16.	Real ≥ 0.0 or blank	0.0
SDMAXS	Shear stress of the weld bonding material when damage initiates. See Remark 16.	Real ≥ 0.0 or blank	0.0
UDINITT	Separation normal to the master weld surface when bond damage initiates. See Remark 16.	Real ≥ 0.0 or blank	0.0
UDINITS	Slip tangential to the master weld surface when bond damage initiates. See Remark 16.	Real ≥ 0.0 or blank	0.0
UDMAXT	Separation normal to the master weld surface when bond damage results in complete failure. See Remark 16.	Real ≥ 0.0 or blank	0.0
UDMAXS	Slip tangential to the master weld surface when bond damage results in complete failure. See Remark 16.	Real ≥ 0.0 or blank	0.0
PRSFIT	Press fit option, one of the following character variables: YES or NO. See Remark 17.		

Remarks:

1. Contact region identification number must be unique with respect to all other BCONP and BSCONP identification numbers.
2. The SECONDARY field defines the secondary surface by referencing a BSSEG Bulk Data entry.
3. The PRIMARY field defines the primary surface by referencing a BSSEG Bulk Data entry.

4. SFACT may be used to scale the penalty values that are determined automatically based on adjacent diagonal stiffness matrix coefficients. Additionally, penalty values calculated may be further scaled by the SLINEKSFACT model parameter (see Section 5, *Parameters*, for more information on SLINEKSFACT). The penalty value is then equal to $k * SFACT * |SLINEKSFACT|$, where k is a value selected for each secondary node based on the diagonal stiffness matrix coefficient and SFACT is specified in the SFACT field above. Note that the SLINEKSFACT value applies to all contact regions in the model. The use of a scale factor (SFACT or SLINEKSFACT) less than one is recommended when convergence problems arise and a value greater than one when excessive penetration occurs. Penalty values are normally recalculated every time there is a change in stiffness. However, if SLINEKSFACT is negative, penalty values are not recalculated. This setting is generally not recommended.
5. The value of frictional stiffness should be chosen carefully. A method of choosing a value is to divide the expected frictional strength (MU $*$ expected normal force) by reasonable value of the relative displacement before slip occurs. A large stiffness value may cause poor convergence, while too small a value may result in reduced accuracy. An alternative method is to specify the value of relative displacement using SMAX.
6. For unsymmetric contact, only the penetration of the secondary node into the primary segments is checked. This may lead to the primary nodes penetrating the secondary segments. This error is reduced as the mesh density is increased. For symmetric penetration, both the secondary and primary nodes are check for penetration. This is accomplished by generating a secondary node, primary segment element using the PRIMARY surface for the secondary nodes and the SECONDARY surface for the primary segments.
7. Welded contact behavior is accomplished by selecting the unsymmetric or symmetric welded contact setting (3, 4, 9, 10, or 11). With either setting the element will behave the same in tension as in compression and will not slide. Note that for linear solutions with the LINEARCONTACT model parameter set to OFF, general contact will default to welded behavior (see Section 5, *Parameters*, for more information on LINEARCONTACT). Bi-directional sliding contact behavior is accomplished by selecting the unsymmetric or symmetric bi-directional contact setting (5 or 6). With either setting the element will act similar to a welded contact element in tension and compression, but will slide in-plane. Bi-directional sliding contact is intended for use on planar surfaces and is available in all solutions. Rough contact behavior is accomplished by selecting the unsymmetric or symmetric rough contact setting (7 or 8). With either setting the element will act similar to a general contact element in tension and compression, but will not permit sliding in-plane. The offset weld setting (9 or 10) is intended for welded connections with significant separation between contact surfaces. Welded contact with a separation less than the value defined by the SLINEOFFSETTOL model parameter is automatically converted to an offset weld (see Section 5, *Parameters*, for more information on SLINEOFFSETTOL).
8. MAXAD may be used to prevent unnecessary generation of contact segments when little or no sliding is expected. Elements are only generated if the distance from any contact surface master node to the potential slave node is less than $(1.0E-5) * \ell_{13} + MAXAD$, where ℓ_{13} is the distance from node 1 to node 3 of the contact surface. The default MAXAD value is set by the model parameter SLINEMAXACTDIST and permits general sliding in any direction. The AUTO setting is recommended for optimal performance when little or no sliding is expected.
9. The contact plane is defaulted to the xy-plane of the master nodes. A positive value of W0 offsets the contact plane in the element z-direction and results in a contact condition occurring when a slave node penetrates the offset plane.
10. There are two methods for adaptive stiffness updates normal to the contact plane: proximity stiffness based and displacement based.
    a) When TMAX ≠ 0.0, the displacement based stiffness update method is selected. The value specified defines the allowable penetration of the slave node into the master surface. The recommended TMAX value is between 1% and 10% of the element thickness for plates or the equivalent thickness for other elements that are connected to the contact element.

b) When TMAX = 0.0 (default), the update method selected is dependent on the SLINESLIDETYPE and SLINEMAXDISPTOL model parameter settings. When SLINESLIDETYPE is set to DYNAMIC, the proximity stiffness based update method is selected. When SLINESLIDETYPE is set to STATIC, the displacement based stiffness update method is selected where SLINEMAXDISPTOL defines the default TMAX value using

$$\text{TMAX} = \text{SLINEMAXDISPTOL} * \sqrt{Area}$$

where $Area$ is the total area of the contact element master surface. See Section 5, *Parameters*, for more information on SLINESLIDETYPE and SLINEMAXDISPTOL.

11. TRMIN is used for the penalty value adjustment and defines the lower bound for the allowable penetration computed by TRMIN $*$ TMAX. The penalty values are decreased if the penetration is below the lower bound.
12. The maximum adjustment ratio MAR defines the upper and lower bounds of the adjusted value by

$$\frac{K_{initial}}{\text{MAR}} \le K \le K_{initial} * \text{MAR}$$

13. MAXRAD and MAXNAD are an alternative to MAXAD. If either one is set to a non-zero value MAXAD will be ignored and MAXRAD and/or MAXNAD will be used instead. When MAXRAD is specified elements are only generated if the element in-plane distance from any contact surface master node to the potential slave node is less than $(1.0\text{E}-5)\cdot \ell_{13} + \text{MAXRAD}$, where ℓ_{13} is the distance from node 1 to node 3 of the contact surface. When MAXNAD is specified elements are only generated if the element normal distance from any contact surface primary node to the potential secondary node is less than MAXNAD.
14. There are two methods for adaptive stiffness updates parallel to the contact plane: proximity stiffness based and displacement based. If SMAX ≠ 0.0, the displacement based update method is selected. When SMAX = 0.0 (default), the proximity stiffness based update method is selected. If FSTIF is specified it will be used as the penalty stiffness for stick when the proximity stiffness method is used. If SMAX ≠ 0.0, the FSTIF value will be adjusted internally to achieve the SMAX displacement specified.
15. The thermal contact conductance TCC is defined as

$$C_{tc} = q/\Delta T$$

where ΔT is the change in temperature between the secondary node and average of the primary nodes and q is the heat flux through the contact surface. Thermal contact conductance is only applicable in heat transfer solutions.

16. There are two failure theories available for weld bond failure: WFM (Weld Failure Model) and CZM (Cohesive Zone Model). The WFM failure theory has two damage models used for modeling weld failure: stress-based and deformation-based. The usage of SDMAXi, UDINITi, and UDMAXi and default values are given below. One or both components of SDMAXi, UDINITi, or UDMAXi may be specified. SDMAXi values are ignored if UDINITi values are specified. Stress-based and deformation-based weld failure is only supported when PTYPE equals 3 or 4. Deformation-based weld failure is also supported for PTYPE is set to 9 or 10 (offset welded contact) or when PTYPE is set to 3 or 4 and reverts to 9 or 10 due to a separation greater than PARAM, SLINEOFFSETTOL. (See Section 5, *Parameters*, for more information on SLINEOFFSETTOL.) Stress-based weld failure is not supported for offset welded contact.

SDMAXi	UDINITi	UDMAXi	WFM Damage Model and Default Values
✓			Stress-based damage model where UDINITi is calculated using SDMAXi and the equivalent weld stress and displacement from the first load increment. UDMAXi is the incremental deformation to failure after damage initiation and is set to 0.1% of the calculated UDINITi value.
✓		✓	Stress-based damage model where UDINITi is calculated using SDMAXi and the equivalent weld stress and displacement from the first load increment. UDMAXi is the incremental deformation to failure after damage initiation.
	✓	✓	Deformation-based damage model.
	✓		Deformation-based damage model where UDMAXi is defaulted to 2 ∗UDINITi.
		✓	Deformation-based damage model where UDINITi is defaulted to 0.5 ∗UDMAXi.
			No damage model is used.

The CZM failure theory requires either SDMAXT and UDMAXT or SDMAXS and UDMAXS to be specified. UDINITi are ignored. CZM is only supported when PTYPE equals 3 or 4 and is not supported for offset welded contact.

17. Setting the PRSFIT field to ON enables press fit contact which does the following:
    - PARAM, SLINESLIDETYPE will be forced to STATIC. (See Section 5, *Parameters*, for more information on SLINESLIDETYPE).
    - The grid points associated with the contact surfaces will not be adjusted regardless of the PARAM, NCONTACTGEOMITER setting. (See Section 5, *Parameters*, for more information on NCONTACTGEOMITER).
    - The SFACT value will be lowered to 0.01 and the MAR value will be updated to bring the initial penetration, set either through actual penetrating geometry or using W0, to zero at the end of the first subcase.

BSET Fixed Analysis Set Definition

Description: 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	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

BSET	15	3	17	456	7	4			

Field	Definition	Type	Default
Gi	Grid point identification number(s).	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required

Remarks:

1. If there are no CSETi or BSETi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi entries present, any a-set degrees of freedom not listed are placed in the free boundary set (c-set). If there are both BSETi and CSETi entries present, the c-set degrees of freedom are defined by the CSETi entries, and any remaining a-set points are placed in the b-set.

BSET1 — Fixed Analysis Set Definition, Alternate Form

Description: 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	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	- etc.-					

Example:

BSET1	123	6	3	7	10	18	14	11	
	19	23							

Alternate Format and Example:

BSET1	C	G1	THRU	G2					

BSET1	456	15	THRU	512					

Field	Definition	Type	Default
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required
Gi	Grid point identification number(s).	Integer > 0; G1 < G2	Required

Remarks:

1. If there are no CSETi or BSETi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi entries present, any a-set degrees of freedom not listed are placed in the free boundary set (c-set). If there are both BSETi and CSETi entries present, the c-set degrees of freedom are defined by the CSETi entries, and any remaining a-set points are placed in the b-set.

BSSEG Boundary Surface Segments

Description: Defines a surface which is comprised of a quadrilateral or triangular segments via grid points that may come in contact with another surface.

Format:

1	2	3	4	5	6	7	8	9	10
BSSEG	ID	G1A	G2A	G3A	G4A	G1B	G2B	G3B	
	G4B	G1C	G2C	G3C	G4C	- etc.-			

Example:

BSSEG	2	3	5	7	9	11	13	15	
		21	27	33	38				

Alternate Format and Example:

BSSEG	ID	G1	THRU	G2	BY	INC			

BSSEG	10	23	THRU	55	BY	2			

Field	Definition	Type	Default
ID	Boundary surface identification number.	Integer > 0	Required
Gi	Grid point identification number(s). Grid points form quadrilateral or triangular segments of a surface and must be ordered so that the normal to the segment points toward the other surface using the right hand rule. See Remark 2.	Integer > 0	Required

Remarks:

1. Boundary surface identification numbers must be unique with respect to all other BLSEG and BSSEG entries.
2. A triangular segment is defined by specifying a zero or blank for the fourth node.
3. The normal to the segment is determined by the ordering of the segment nodes using the right hand rule. Each segment normal of a contact surface must point toward the opposite surface.
4. The alternate format should only be used when referenced as a slave surface on a BSCONP entry with unsymmetric penetration specified.

BWIDTH Boundary Line Segment Width

Description: Specifies widths for line segments defined on BLSEG Bulk Data entries.

Format:

1	2	3	4	5	6	7	8	9	10
BWIDTH	ID	W1	W2	W3	W4	W5	W6	W7	
	W8	W9	W10	- etc.-					

Example:

BWIDTH	2	2.0	2.1	2.2	2.5	2.8	2.4	2.2	
	1.9	1.5							

Alternate Format and Example:

BWIDTH	ID	W1	THRU	W2	BY	INC			

BWIDTH	10	2.1	THRU	3.2	BY	0.1			

Field	Definition	Type	Default
ID	Corresponding BLSEG entry identification number. See Remark 1.	Integer > 0	Required
Wi	Width values for the corresponding line segments defined in the BLSEG entry. See Remark 2.	Real > 0.0	Required
INC	Width value increment.	Real or blank	1.0

Remarks:

1. The BWIDTH entry requires the same ID as the BLSEG entry.
2. For each segment defined on the BLSEG entry a corresponding width is defined on the BWIDTH entry. If only one width is specified, the remaining segments will be set to that value.
3. If the BWIDTH entry is omitted, a default width of 1.0 will be used.

CBAR Simple Beam Element Connection

Description: Defines a simple beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3		
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
			F0						

Example:

CBAR	10	100	101	102	0.0	0.0	1.0		
	456		0.5	0.0	0.0	0.5	0.0	0.0	
			1.+4						

Alternate Format and Example:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	G0/X1	X2	X3		
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
			F0						

CBAR	2	39	7	6	105				
		45							
			1.+4						

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PBAR entry.	Integer > 0	Required
GA, GB	Grid point identification numbers of connection points.	Integer > 0; GA ≠ GB	Required
X1, X2, X3	Components of vector $\vec{\mathbf{v}}$, from GA, in the displacement coordinate system at GA (see Figure 1).	Real or blank	
G0	Grid point identification number to optionally supply X1, X2, and X3. Direction of orientation vector is GA to G0.	Integer or blank	

Field	Definition	Type	Default
PA, PB	Pin flags for bar ends A and B, respectively (up to 5 of the unique digits 1-6 anywhere in the field with no embedded blanks). 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 1). 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.	Integer $>$ 0 or blank	None
WiA, WiB	Components of offset vectors $\vec{\mathbf{W}}$ iA and $\vec{\mathbf{W}}$ iB, respectively, in displacement coordinate systems at points GA and GB, respectively (see Figure 1).	Real or blank	0.0
F0	Preload.	Real or blank	0.0

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. If field 6 is an integer, then G0 is used. If field 6 is blank or real, then X1, X2, X3 is used.
3. G0 cannot be located at GA or GB.
4. If there are no pin flags or offsets, the continuation may be omitted.
5. Offset vectors are treated like rigid elements and are therefore subject to the same limitations.
   a) Offset vectors do not affect thermal loads.
   b) The specification of offset vectors is not recommended in solutions that compute differential stiffness because the offset vector remains parallel to its original orientation (differential stiffness is computed in buckling, prestress, and nonlinear analysis with PARAM, LGDISP, ON).

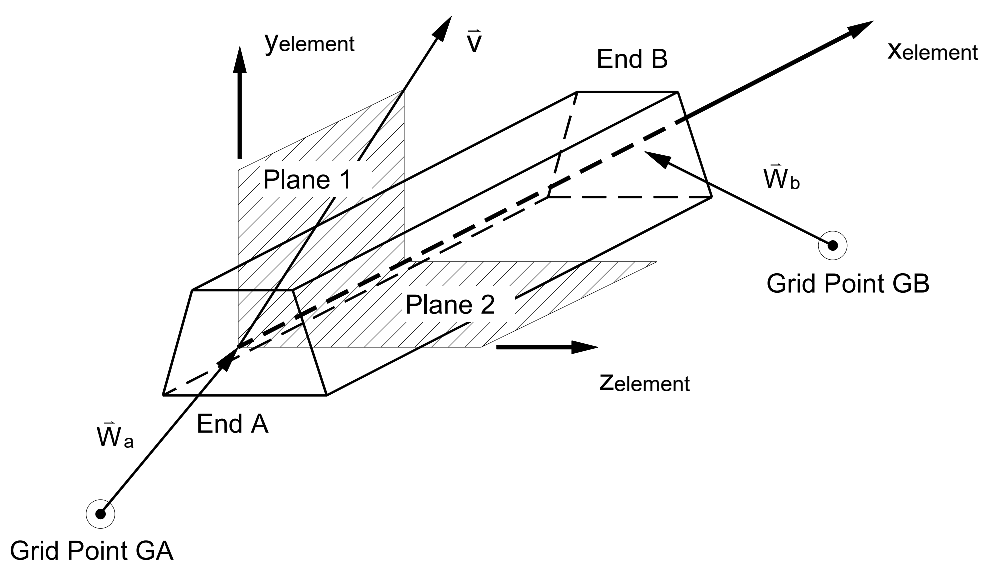


Figure 1. CBAR Element Geometry.

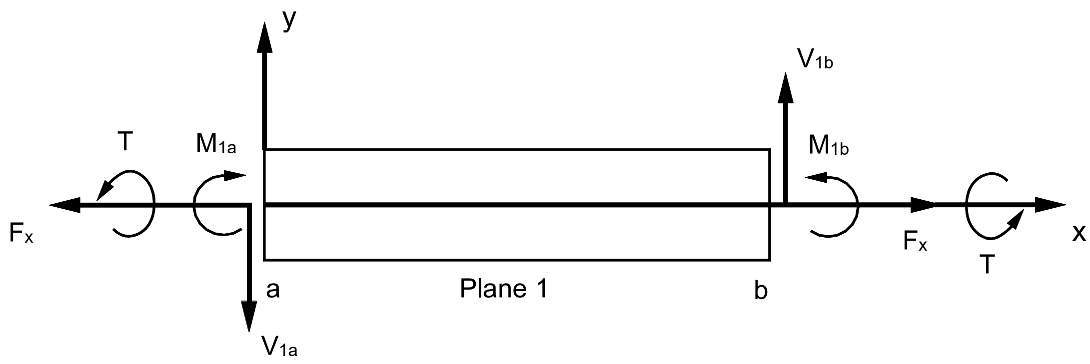


Figure 2. CBAR Element Internal Forces and Moments (xy-Plane).

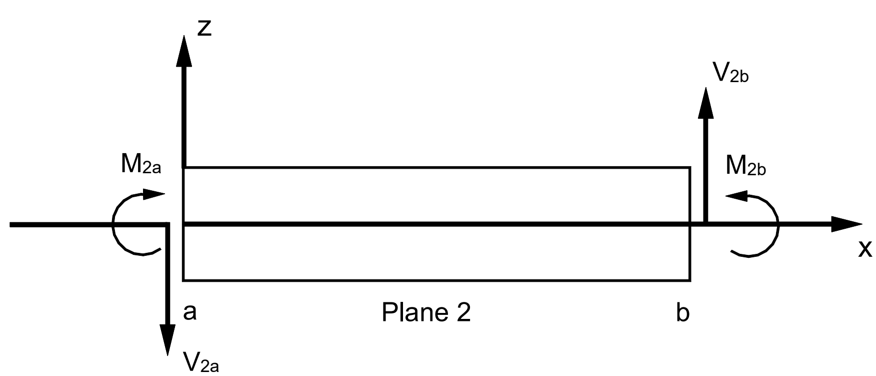


Figure 3. CBAR Element Internal Forces and Moments (xz-Plane).

CBARAO — Auxiliary Output Points Along Bar Element Axis

Description: A series of points along a bar element x-axis may be defined with this 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	1270	FR	0.3	0.4	0.5	0.7			

Alternate Format and Example:

CBARAO	EID	SCALE	NPTS	X1	DELTAX				

CBARAO	1270	FR	4	0.2	0.2				

Field	Definition	Type	Default
EID	CBAR element identification number.	Integer > 0	Required
SCALE	Defines scale of Xi values. Must be one of following character variables: LE or FR.	Character	Required
Xi	Series of locations along element x-axis for stress and force data recovery.	Real > 0.0	0.0
DELTAX	Incremental distance along element x-axis.	Real	0.0
NPTS	Number of stress recovery points, not including the endpoints.	Integer > 0	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 distance along the length if SCALE = LE. If SCALE = FR, the values of Xi are ratios of actual distance to the bar length.
2. When the alternate format is used, a series of locations Xi = X[i-1] + DELTAX, i = 1, 2, 3…, NPTS are generated.
3. If a CBARAO entry is specified for a bar element 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 endpoints of the beam will always be output.

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 the definition of additional points is not necessary.
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 and nine with the alternate form. The endpoints should 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

Description: Defines a beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	G0/X1	X2	X3		
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
			F0						

Example:

CBEAM	10	45	5	21	0.5	7.0	-1.3		
		123		1.0			1.0		
			1.+4						

Alternate Format and Example:

CBEAM	EID	PID	GA	GB	G0				
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
			F0						

CBEAM	12	29	7	6	105				
		45							
			1.+4						

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PBEAM entry.	Integer > 0	Required
GA, GB	Grid point identification numbers of connection points.	Integer > 0; GA ≠ GB	Required
X1, X2, X3	Components of vector $\vec{v}$, from GA, in the displacement coordinate system at GA.	Real or blank	
G0	Grid point identification number to optionally supply X1, X2, and X3. Direction of orientation vector is GA to G0.	Integer or blank	

Field	Definition	Type	Default
PA, PB	Pin flags for bar ends A and B, respectively (up to 5 of the unique digits 1-6 anywhere in the field with no embedded blanks). 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 1). 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.	Integer > 0 or blank	None
WiA, WiB	Components of offset vectors $\vec{w}$ iA and $\vec{w}$ iB, respectively, in displacement coordinate systems at points GA and GB, respectively (see Figure 1).	Real or blank	0.0
F0	Preload.	Real or blank	0.0

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The following figure defines beam element geometry:

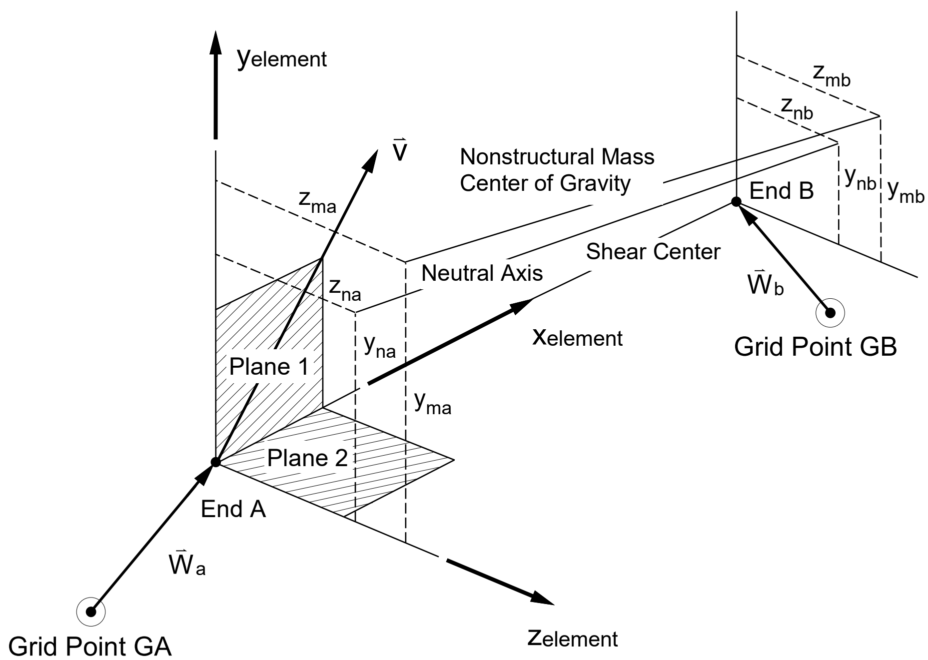


Figure 1. CBEAM Element Geometry System.

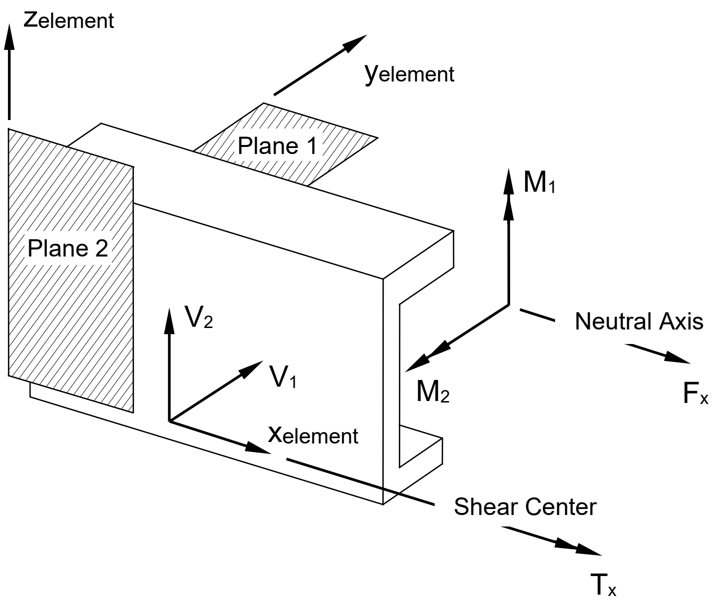


Figure 2. CBEAM Internal Element Forces and Moments.

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 continuation may be omitted if there are no pin flags or offsets.
6. Offset vectors are treated like rigid elements and are therefore subject to the same limitations.
    a) Thermal loads are not affected by offset vectors.
    b) The specification of offset vectors is not recommended in solutions that compute differential stiffness because the offset vector remains parallel to its original orientation (differential stiffness is computed in buckling, prestress, and nonlinear analysis with PARAM, LGDISP, ON).

CBUSH — Generalized Spring and Damper Connection

Description: 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	G0/X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

Example:

CBUSH	45	5	11	67	78				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PBUSH entry.	Integer > 0	Required
GA, GB	Grid point identification number of connection points.	Integer > 0	See Remark 6
X1, X2, X3	Components of vector $\vec{v}$, from GA, in the displacement coordinate system at GA (see Figure 1).	Real or blank	
G0	Grid point identification number to optionally supply X1, X2, and X3. Direction of orientation vector is GA to G0.	Integer or blank	
CID	Element coordinate system identification. A 0 means the basic coordinate system. If CID is blank, then the element coordinate system is determined from G0 or Xi. See Figure 1 and Remark 3.	Integer ≥ 0 or blank	
S	Location of spring damper. See Figure 2.	0.0 ≤ Real ≤ 1.0	0.5
OCID	Coordinate system identification of spring-damper offset. See Remark 8.	Integer ≥ -1	-1
S1, S2, S3	Components of spring-damper offset in the OCID coordinate system if OCID ≥ 0. See Figure 2 and Remark 8.	Real	

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The bush element geometry is shown in Figure 1.

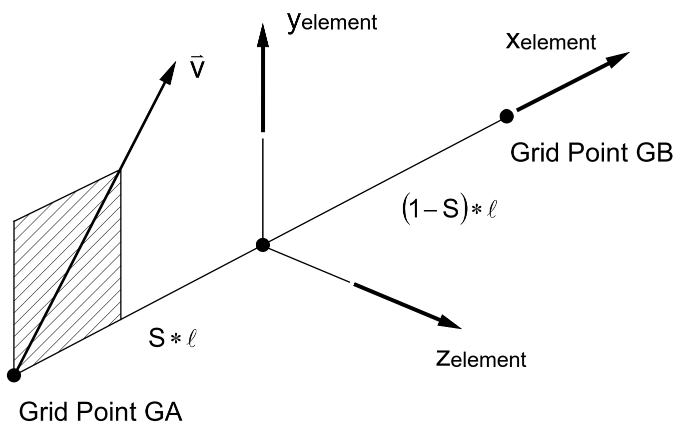


Figure 1. CBUSH Element Coordinate System.

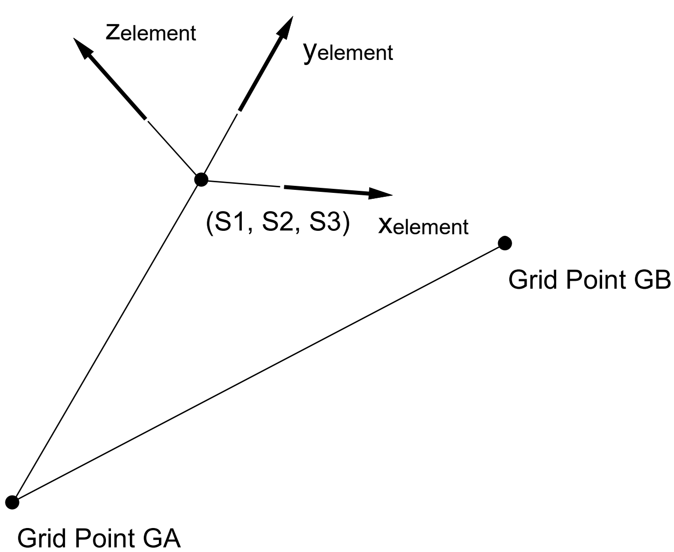


Figure 2. Definition of Offset S1, S2, S3.

3. CID ≥ 0 overrides G0 and (X1, X2, X3). 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 selected to define the element x-axis.
4. For noncoincident grids (GA ≠ GB), when G0 or (X1, X2, X3) is given and no CID is specified, then the line GA – GB is the element x-axis and the orientation vector $\vec{\mathbf{v}}$ lies in the x-y plane (similar to the CBEAM element).
5. For noncoincident grids (GA ≠ GB), if neither G nor (X1, X2, X3) is specified and no CID is specified, then the line GA – GB 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. A blank in field 5 may be used to indicate a grounded terminal GB. A grounded terminal is a point whose displacement is constrained to zero.
7. If GA and GB are coincident, or if GB is blank, then CID must be specified.
8. If OCID = -1 or blank (default) then S is used and S1, S2, S3 are ignored. If OCID ≥ 0, then S is ignored and S1, S2, S3 are used.

CBUSH1D — Rod Type Spring and Damper Connection

Description: 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	30	105	109	114					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PBUSH1D entry.	Integer > 0	Required
GA, GB	Grid point identification number of connection points.	Integer > 0	See Remark 4
CID	Element coordinate system identification.	Integer ≥ 0 or blank	

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. For noncoincident grids GA ≠ GB and if CID is blank, the line GA to GB is the element axis. In nonlinear analysis with large displacement effects turned on, the element axis follows the deformation of grids GA and GB (see Figure 1).
3. If CID ≥ 0 is specified, the x-axis of the CID coordinate system is the element axis. In nonlinear analysis with large displacement effects turned on, the element axis remains fixed.
4. A blank in field 5 may be used to indicate a grounded terminal GB. A grounded terminal is a point whose displacement is constrained to zero.
5. If GA and GB are coincident or if GB is blank, then CID ≥ 0 must be specified and the element axis is the x-axis of CID.

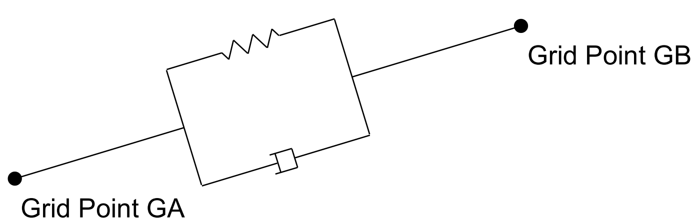


Figure 1. Spring and Damper Element.

CCABLE — Cable Element Connection

Description: Defines a tension-only element with optional bending stiffness.

Format:

1	2	3	4	5	6	7	8	9	10
CCABLE	EID	PID	G1	G2					

Example:

CCABLE	62	12	105	110					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PCABLE property entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0, G1 ≠ G2	Required

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. This element will default to a circular bar in linear solutions. A nonlinear solution must be selected for tension-only behavior.

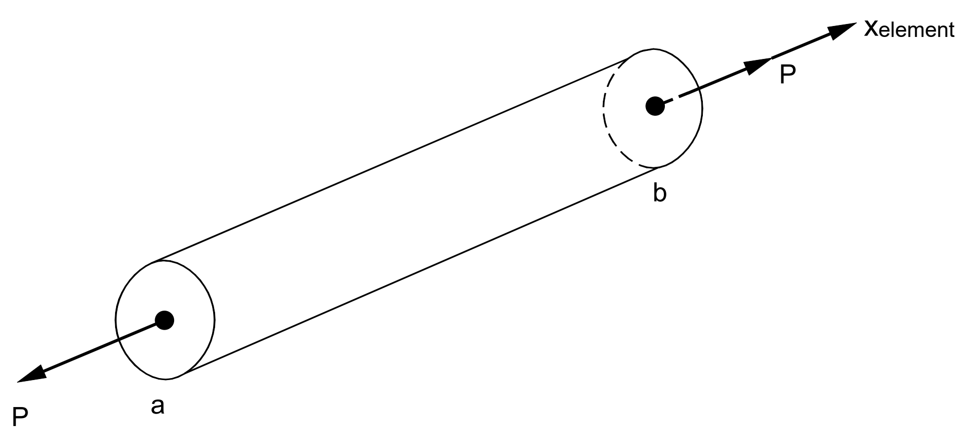


Figure 1. CCABLE Element Internal Forces.

CDAMP1 Scalar Damper Connection

Description: 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	20	2	30	2			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PDAMP property entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer ≥ 0	See Remark 2
C1, C2	Component numbers.	0 ≤ Integer ≤ 6	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. When this entry is used in heat transfer analysis, it generates a lumped heat capacity.
5. If Gi refers to a grid point then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.

CDAMP2 Scalar Damper Property and Connection

Description: 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	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
B	Value of scalar damper.	Real	Required
G1, G2	Grid point identification numbers of connection points.	Integer ≥ 0	See Remark 2
C1, C2	Component numbers.	0 ≤ Integer ≤ 6	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. When this entry is used in heat transfer analysis, it generates a lumped heat capacity.
5. If Gi refers to a grid point then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.

CDAMP3 — Scalar Damper Connection to Scalar Points Only

Description: 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	19	6	20	30					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PDAMP property entry.	Integer > 0	Required
S1, S2	Scalar point identification numbers of connection points.	Integer ≥ 0	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. S1 or S2 may be blank indicating a constrained coordinate.
3. When this entry is used in heat transfer analysis, it generates a lumped heat capacity.

CDAMP4 Scalar Damper Property and Connection to Scalar Points

Description: Defines a scalar damper element that is 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	2.98	32	55					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
B	Value of scalar damper.	Real	Required
S1, S2	Scalar point identification numbers of connection points.	Integer ≥ 0	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. S1 or S2 may be blank indicating a constrained coordinate.
3. When this entry is used in heat transfer analysis, it generates a lumped heat capacity.

CELAS1 — Scalar Spring Connection

Description: 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	12	101			22	4			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PELAS entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0	See Remark 2
C1, C2	Component numbers.	0 ≤ Integer ≤ 6	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. If Gi refers to a grid point then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.

CELAS2 — Scalar Spring Property and Connection

Description: 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	124	1.+4	44	5	45	5			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
K	Stiffness value.	Real	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0	See Remark 2
C1, C2	Component numbers.	0 ≤ Integer ≤ 6	See Remark 2
GE	Structural element damping coefficient. See Remark 5.	Real or blank	0.0
S	Stress coefficient.	Real or blank	0.0

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. This single entry completely defines the element since no material or geometric properties are required.
4. The two connection points (G1, C1) and (G2, C2) must be distinct.
5. If Gi refers to a grid point then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.
6. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.

CELAS3 — Scalar Spring Connection to Scalar Points Only

Description: Defines a scalar spring element that is connected only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS3	EID	PID	S1	S1					

Example:

CELAS3	12	101	25	35					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PELAS entry.	Integer > 0	Required
S1, S2	Scalar point identification numbers of connection points.	Integer > 0	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. If Gi refers to a grid point then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.

CELAS4 Scalar Spring Property and Connection to Scalar Points Only

Description: Defines a scalar spring element that is connected only to scalar points and without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS4	EID	K	S1	S1			GE	S	

Example:

CELAS4	124	1.+4	44	5					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
K	Stiffness value.	Real	Required
S1, S2	Scalar point identification numbers of connection points.	Integer > 0	See Remark 2
GE	Structural element damping coefficient. See Remark 4.	Real or blank	0.0
S	Stress coefficient.	Real or blank	0.0

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. This single entry completely defines the element since no material or geometric properties are required.
4. If Gi refers to a grid point then Ci refers to component numbers in the displacement coordinate system specified by CD on the GRID entry.
5. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
6. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.

CGAP Gap Element Connection

Description: Defines a gap or friction element.

Format:

1	2	3	4	5	6	7	8	9	10
CGAP	EID	PID	GA	GB	G0/X1	X2	X3	CID	

Example:

CGAP	20	1	100	101	4.7	1.2	0.		

Alternate Format and Example:

CGAP	EID	PID	GA	GB	GO			CID	

CGAP	17	2	110	112	13				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PGAP entry.	Integer > 0	Required
GA, GB	Grid point identification numbers of connection points.	Integer > 0; GA ≠ GB	Required
X1, X2, X3	Components of vector $\vec{\mathbf{v}}$, from GA, in the displacement coordinate system at GA (see Figure 1).	Real or blank	
G0	Grid point identification number to optionally supply X1, X2, X3. Direction of orientation vector is GA to G0.	Integer or blank	
CID	Element coordinate system identification number. CID must be specified if GA and GB are coincident. See Remark 7.	Integer ≥ 0 or blank	

Remarks:

1. The CGAP element is intended for use in nonlinear static analysis. It will produce a linear stiffness matrix for all other solutions. The stiffness used depends on the gap state.

2. The gap element coordinate system is defined by one of two following methods:
   a) 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 $\vec{\mathbf{v}}$ will be ignored in this case.
   b) If the CID field is blank and the grid points GA and GB are not coincident, then the line AB is the element x-axis and the orientation vector $\vec{\mathbf{v}}$ lies in the x-y plane.
3. The element coordinate system does not rotate as a result of deflections.
4. Initial gap openings are defined on the PGAP entry and not by 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. This element will default to a linear spring in linear solutions including linear static analysis with linear contact enabled. A nonlinear solution must be selected for general contact behavior.
7. 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 to define the element orientation.

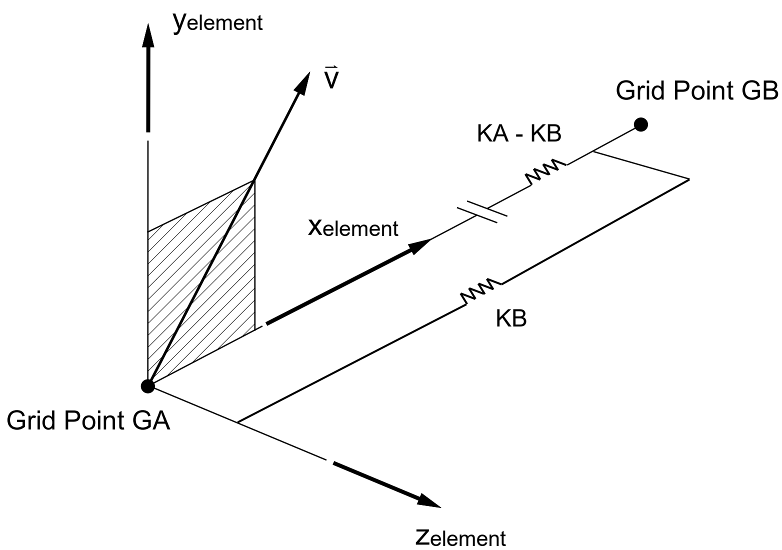


Figure 1. CGAP Element Coordinate System.

CHBDYG Geometric Surface Element Definition (Grid Form)

Description: Defines a boundary condition surface element for heat transfer analysis without reference to a property form.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE	IVIEW		RADMID			
	G1	G2	G3	G4	G5	G6	G7	G8	

Example:

CHBDYG	5		AREA3						
	22	35	33	12					

Field	Definition	Type	Default
EID	Surface element identification number.	Integer > 0	Required
TYPE	Surface type, see Remark 2.	Character	Required
IVIEW	A VIEW identification number.	Integer > 0	
RADMID	RADM identification number.	Integer > 0	
Gi	Grid point identification numbers of grids bounding the surface.	Integer > 0	Required

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. TYPE specifies the kind of element surface. Supported types are REV, AREA3, AREA4, AREA6, and AREA8.
   - TYPE = REV

     The REV type has two primary grid points that must lie in the x-z plane of the basic coordinate system. 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 (see Figure 1).
   - 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 (see Figures 2 and 3).
3. These types have three and four primary grid points, respectively, which define a triangular or quadrilateral surface and must be ordered to go around the boundary.
4. For defining the front face, the right-hand rule is used on the sequence G1 to G2 to … Gn of the grid points.

5. All conduction elements to which any boundary condition is to be applied must be individually identified with one of the surface element entries: CHBDYG or CHBDYP.

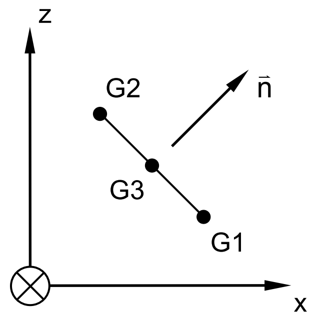


Figure 1. Normal Vector for CHBDYG Element TYPE = REV.

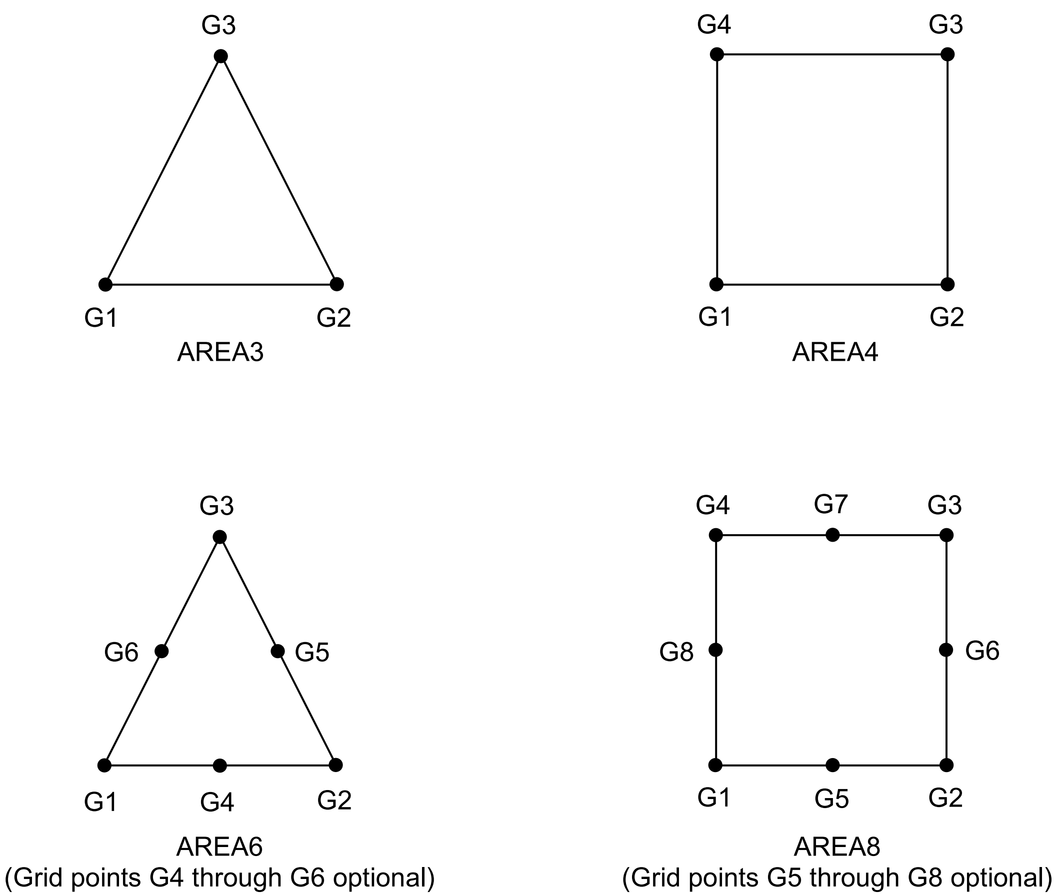


Figure 2. Surface TYPE Definition.

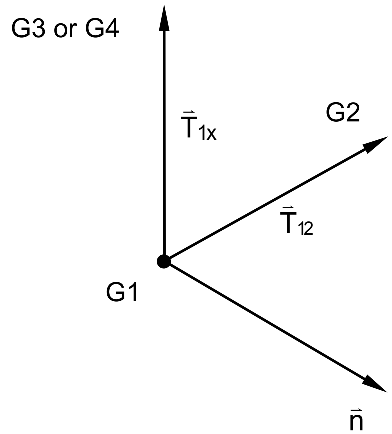


Figure 3. Normal Vector for CHBDYG Element TYPE = AREAi.

The unit normal is given by:

$$\vec{n} = \frac{(\bar{T}_{12} \times \bar{T}_{1x})}{|\bar{T}_{12} \times \bar{T}_{1x}|}$$

(G3 is used for triangles and G4 is used for quadrilaterals).

CHBDYP Geometric Surface Element Definition (Property Form)

Description: 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	IVIEW		G1	G2	G0	
	RADMID			CID	X1	X2	X3		

Example:

CHBDYP	4	10	POINT			15			
					0.0	0.0	1.0		

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PHBDY entry.	Integer > 0	Required
TYPE	Surface type, see Remark 3.	Character	Required
IVIEW	A VIEW identification number.	Integer > 0	
Gi	Grid point identification numbers of connection points of the surface.	Integer > 0	
G0	Grid point identification number to optionally supply X1, X2, and X3. Direction of orientation vector is G1 to G0.	Integer > 0 or blank	
RADMID	RADM identification number.	Integer > 0	
CID	Coordinate system for defining orientation vector.	Integer > 0	0
Xi	Components of the orientation vector in the coordinate system defined in field 5. The origin of the orientation vector is a grid point G1.	Real or blank	

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. For types POINT and LINE geometric orientation is required. The required information is sought in the following order:
   - If G0 > 0 is found on the CHBDYP entry, it is used.
   - Otherwise, if a non-blank CE is found on the CHBDYP continuation entry, this CE and the corresponding vectors E1, E2, and E3 are used.
   - If none of the above apply, a warning message is issued.
3. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of either surface element entries: CHBDYG or CHBDYP.

4. TYPE specifies the kind of element surface. Supported types are POINTS and LINE.

a) TYPE = POINT

The POINT type has one primary grid point, requires a property entry, and the normal vector Vi must be specified.

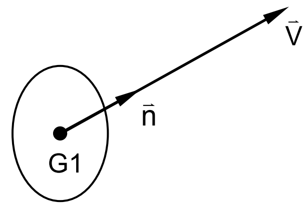


Figure 1. Normal Vector for CHBDYP Element with Type Equal to POINT.

The unit normal is given by:

$$\vec{n} = \frac{\vec{V}}{|\vec{V}|}$$

b) TYPE = LINE

The LINE type has two primary grid points, requires a property entry, and the vector is required.

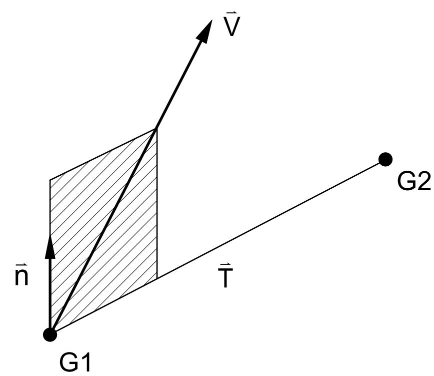


Figure 2. Normal Vector for CHBDYP Element with Type Equal to LINE.

The unit normal lies in the plane $\vec{V}$ and $\vec{T}$, is perpendicular to $\vec{T}$, and is given by:

$$\vec{n} = \frac{\vec{T} \times (\vec{V} \times \vec{T})}{|\vec{T} \times (\vec{V} \times \vec{T})|}$$

5. The geometric orientation can be defined by either GO or the vector E1, E2, E3.

- If GO $>$ zero:

  For a POINT-type surface, the normal to the front face is the vector from G1 to GO. For the LINE-type 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.

- If GO is zero:

  For a POINT-type surface, the normal to the front face is the orientation vector. For the LINE-type 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.

CHEXA Six-Sided Solid Element Connection

Description: Defines the connections of a six-sided isoparametric 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	G116	G17	G18	G19	G20			

Example:

CHEXA	71	4	3	4	5	6	7	8	
	9	10	30	31			53	54	
	55	56	57	58	59	60			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank, all unique	Required

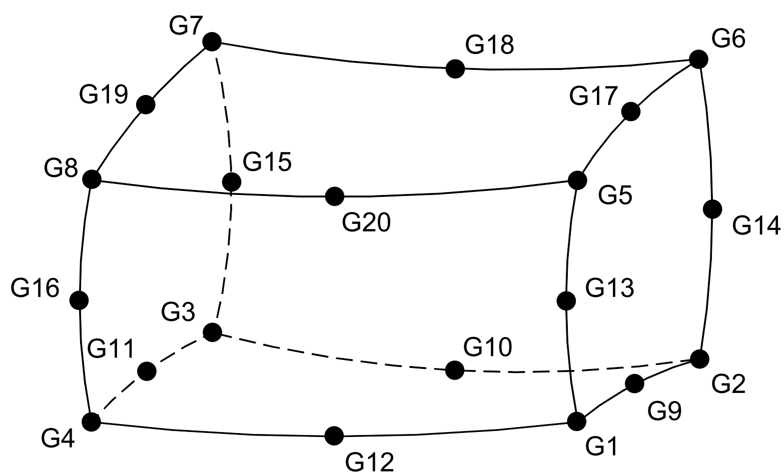


Figure 1. CHEXA Element Connection.

Remarks:

1. Element identification numbers must 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. Grid points G5 through G8 must be in order in the same direction around the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. Any or all of the edge points, G9 through G20, may be deleted. If the ID of any edge connection points is left blank or set to zero (as for G11 and G12 in the example), the element equations are modified to give the correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. Components of stress are output in the volume coordinate system. (See the VOLUME command in Section 3, *Case Control.*)
5. The material coordinate system is defined on the PSOLID entry.
6. The second continuation is optional.
7. 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.
8. It is recommended that the edge points be located within the middle third of the edge.
9. By default, all of the twelve edges of the element are considered straight unless an edge node is specified using G9 through G20.

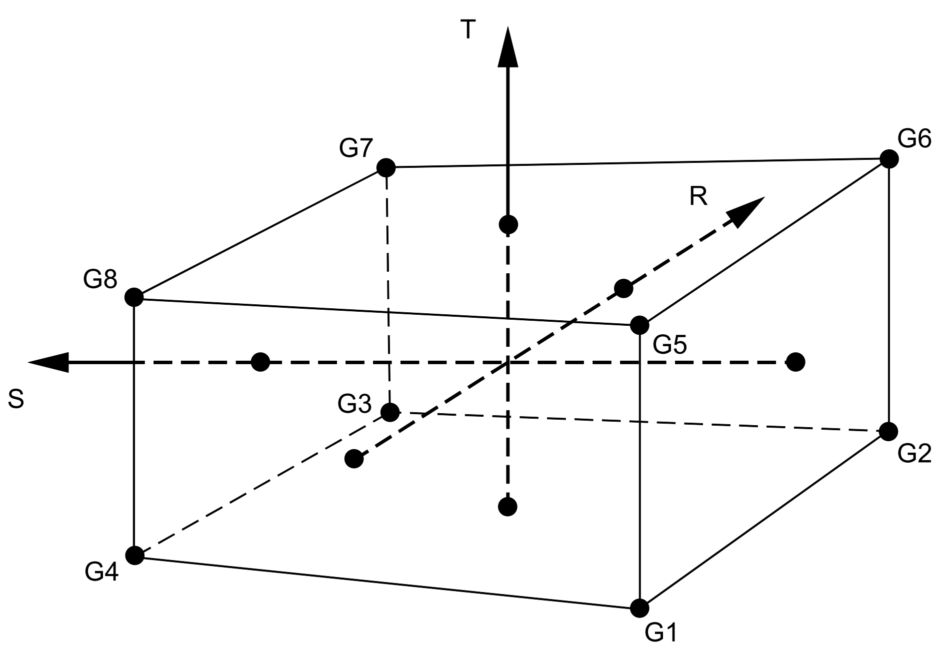


Figure 2. CHEXA Element R, S, and T Vectors.

CMASS1 Scalar Mass Connection

Description: 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	55	2	2	3	5	3			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PMASS entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0	See Remark 2
C1, C2	Component numbers.	0 ≤ Integer ≤ 6	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. The two connection points (G1, C1) and (G2, C2) must not be coincident.
4. A scalar point specified on this entry need not be defined on an SPOINT entry.

CMASS2 — Scalar Mass Property and Connection

Description: 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	128	145.0	5	2	9	2			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
M	Mass value.	Real	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0	See Remark 2
C1, C2	Component numbers.	0 ≤ Integer ≤ 6	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. A 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 whose displacement is constrained to zero.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. A scalar point specified on this entry need not be defined on an SPOINT entry.

CMASS3 Scalar Mass Connection to Scalar Points Only

Description: 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	S1					

Example:

CMASS3	55	2	2	5					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PMASS entry.	Integer > 0	Required
S1, S2	Scalar point identification numbers of connection points.	Integer > 0	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. S1 or S2 may be blank indicating a constrained coordinate.
3. 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

Description: Defines a scalar mass element that is connected only to scalar points and without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS4	EID	M	S1	S2					

Example:

CMASS4	128	145.0	5	9					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
M	Mass value.	Real	Required
S1, S2	Scalar identification numbers of connection points.	Integer > 0	See Remark 2

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. S1 or S2 may be blank indicating a constrained coordinate.
3. A scalar point specified on this entry need not be defined on an SPOINT entry.

CONCRETE — Concrete Material Property Definition

Description: Defines material properties for use in fully nonlinear analysis of quasi-brittle materials (concrete).

Format:

1	2	3	4	5	6	7	8	9	10
CONCRETE	MID	SINITT	SINITC	SMAXT	SMAXC	GT	GC	SBYC	
	KDT	KDC	ALPHAP	LT	LC				

Example:

CONCRETE	101	3.3+6	3.+7		3.5+7	2.5+2	2.5+4		
	0.5	0.4	0.2						

Field	Definition	Type	Default
MID	Identification number of a MAT1 entry.	Integer > 0	Required
SINITT	Initial tensile strength.	Real > 0.0	See Remark 1
SINITC	Initial compressive strength.	Real > 0.0 and SMAXC > SINITC	See Remark 1
SMAXT	Maximum tensile strength.	Real > 0.0	See Remark 1
SMAXC	Maximum compressive strength.	Real > 0.0 and SMAXC > SINITC	See Remark 1
GT	Tensile crushing fracture energy.	Real > 0.0	See Remark 1
GC	Compressive crushing fracture energy.	Real > 0.0	See Remark 1
SBYC	Initial biaxial yield compressive stress.	Real ≥ 0.0 or blank	0.0
KDT	Uniaxial tensile elastic stiffness degradation factor.	Real > 0.0 or blank	0.5
KDC	Uniaxial compressive elastic stiffness degradation factor.	Real > 0.0 or blank	0.4

Field	Definition	Type	Default
ALPHAP	Coefficient of plastic potential.	Real > 0.0 or blank	0.2
LT	Tensile characteristic length parameter.	Real > 0.0 or blank	See Remark 2
LC	Compressive characteristic length parameter.	Real > 0.0 or blank	See Remark 2

Remarks:

1. The following are values for fields 3 through 8 for standard concrete in metric units:

Variable	Value
SINITT	3.3E+6 Pa
SINITC	3.0E+7 Pa
SMAXT	3.3E+6 Pa
SMAXC	3.5E+7 Pa
GT	2.5E+2 N/m
GC	2.5E+4 N/m

2. The default tensile and compressive characteristic length parameter values are based on the maximum element reference length in the model.

CONM1 Concentrated Mass Element Connection, General Form

Description: Defines a 6-by-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	5	25	6	6.5	8.4	7.9			
	7.8	45.7							
		56.3						43.7	

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
G	Grid point identification number	Integer > 0	Required
CID	Coordinate system identification number for the mass matrix.	Integer ≥ 0	0
Mij	Mass matrix values.	Real	Required

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. See the CONM2 entry description for a less general means of defining concentrated mass at grid points.

CONM2 — Concentrated Mass Element Connection

Description: 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	1	2	12	20.0	22	4			
	23.5		32.6		12.8				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
G	Grid point identification number	Integer > 0	Required
CID	Coordinate system identification number. For CID of -1, see X1, X2, X3 below.	Integer ≥ -1	0
M	Mass value.	Real	Required
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 of the center of gravity of the mass in the basic coordinate system.	Real or blank	0.0
Iij	Mass moments of inertia measured at the center of gravity in the coordinate system defined by field 4. If CID = -1, mass moments of inertia measured at the center of gravity in the basic coordinate system.	I11, I22, and I33; Real > 0.0; I21, I31, and I32, Real	0.0

Remarks:

1. Element identification numbers must 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 entry may be omitted.
4. If CID = -1, offsets are calculated internally as the difference between the grid point location and X1, X2, X3. If the grid point locations are defined in a non-basic coordinate system, the values of Iij must be in a coordinate system that parallels the basic coordinate system.
5. If CID ≥ 0, then X1, X2, X3 are defined by a local Cartesian system similar to the method in which displacement coordinate systems are defined.

6. The form of the inertia matrix about its center of gravity is taken as:

$$\begin{bmatrix} M & & & & & \\ & M & & & & \\ & & M & & & \\ & & & I_{11} & & \\ & & & -I_{21} & I_{22} & \\ & & & -I_{31} & -I_{32} & I_{33} \end{bmatrix}$$

where,

$$M = \int \rho dV$$

$$I_{11} = \int \rho \left(x_2^2 + x_3^2\right) dV$$

$$I_{22} = \int \rho \left(x_1^2 + x_3^2\right) dV$$

$$I_{33} = \int \rho \left(x_1^2 + x_2^2\right) dV$$

$$I_{21} = \int \rho x_1 x_2 dV$$

$$I_{31} = \int \rho x_1 x_3 dV$$

$$I_{32} = \int \rho x_2 x_3 dV$$

and x_1, x_2, x_3 are components of distance from the center of gravity in the coordinate system defined in field 4. Only the magnitude of I_{ij} should be supplied, the negative signs for the off-diagonal terms are supplied automatically. A warning message is issued of the inertia matrix is non-positive definite. A non-positive definite inertia matrix may cause fatal errors in the eigenvalue extraction module.

CONROD

Rod Element Property and Connection

Description: Defines a tension-compression-torsion 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	61	12	17	45	0.05				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0, G1 ≠ G2	Required
MID	Material identification number.	Integer > 0	Required
A	Area of rod cross-section.	Real	Required
J	Torsional constant.	Real or blank	0.0
C	Coefficient to determine torsional stress.	Real or blank	0.0
NSM	Nonstructural mass per unit length.	Real or blank	0.0

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. For structural problems, PROD entries may only reference MAT1 material entries.
3. The formula used to compute torsional stress is

$$\tau = \frac{Tc}{J}$$

where T is the torsional moment.

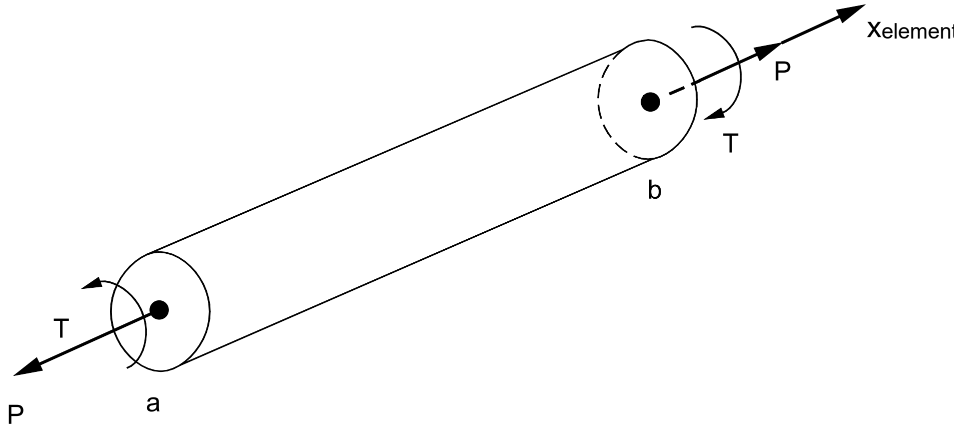


Figure 1. CONROD Element Internal Forces and Moments.

CONV — Heat Boundary Element Free Convection Entry

Description: Specifies a free convection boundary condition for heat transfer analysis through connections to a surface element (CHBDYi entry).

Format:

1	2	3	4	5	6	7	8	9	10
CONV	EID	PID	FLMND	CNTRLND	TA1	TA2	TA3	TA4	
	TA5	TA6	TA7	TA8	CTID1	CTID2	CTID3	ATID1	
	ATID2	ATID3							

Example:

CONV	1	50	5		62				

Field	Definition	Type	Default
EID	CHBDYG or CHBDYP surface identification number.	Integer > 0	Required
PID	Convection property identification number of a PCONV entry.	Integer > 0	Required
FLMND	Point for film convection fluid property temperature.	Integer ≥ 0 or blank	0
CNTRLND	Control point for free convection boundary condition.	Integer ≥ 0 or blank	0
TAi	Ambient points used for convection.	Integer > 0 for TA1 Integer ≥ 0 for TA2 through TA8	TA1
CTID1, CTID2, CTID3	TABLEDi set identification numbers that define control point position dependent scale factors in the x, y, and z directions of the basic coordinate system. See Remark 1.	Integer > 0 or blank	
ATID1, ATID2, ATID3	TABLEDi set identification numbers that define ambient point position dependent scale factors in the x, y, and z directions of the basic coordinate system. See Remark 1.	Integer > 0 or blank	

Remarks:

1. The basic exchange relationship can be expressed in one of the following forms:

   a) $q = \mathrm{H} * \mathrm{u}_{\mathrm{CNTRLND}} * c\,(x, y, z) * [\mathrm{T - TAMB} * a\,(x, y, z)]$, CNTRLND ≠ 0

   b) $q = \mathrm{H} * [\mathrm{T - TAMB} * a\,(x, y, z)]$, CNTRLND = 0

   where $c(x, y, z)$ is defined as the product of scale factors returned by tables defined in fields 6, 7, and 8 on the first continuation entry and $a(x, y, z)$ is defined as the product of scale factors returned by tables defined in field 9 on the first continuation entry and fields 2 and 3 on the second continuation entry.

2. CONV is used with a CHBDYi (CHBDYG or CHBDYP) entry having the same EID.
3. The temperature of the film convection point must be specified to determine the convection film coefficient. If FLMND = 0, the default temperature is the average of the ambient points (average) and element grid point temperatures (average).
4. If only one ambient point is specified then all the ambient points are assumed to have the same temperature. If mid-side ambient points are missing, the temperature of these points is assumed to be the average of the connecting corner points.
5. See the PCONV Bulk Data entry for an explanation of the mathematical relationships involved in free convection and the reference temperature for convection film coefficient.

CORD1C Cylindrical Coordinate System Definition, Form 1

Description: Defines a cylindrical coordinate system by reference to the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1C	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1C	4	2	44	67					

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
GiA, GiB	Grid point identification numbers.	Integer > 0, G1A ≠ G2A ≠ G3A, G1B ≠ G2B ≠ G3B	Required

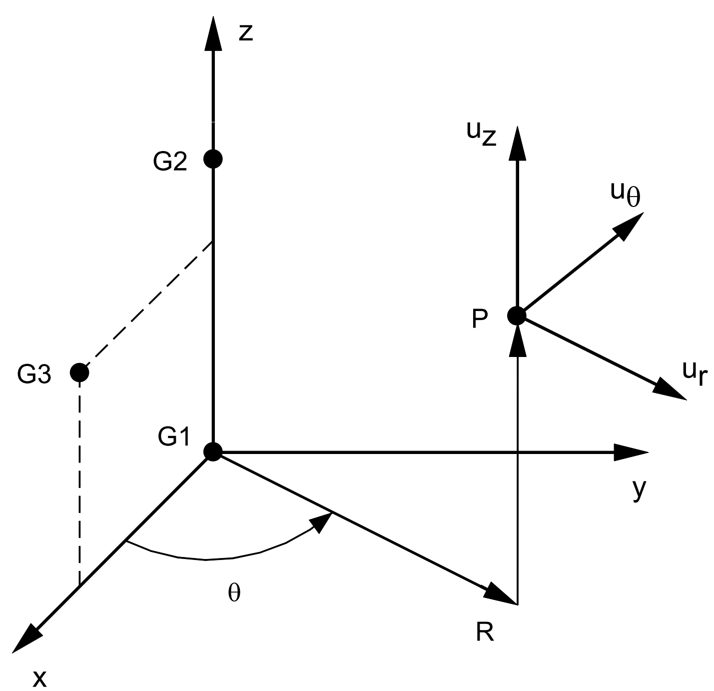


Figure 1. CORD1C Definition.

Remarks:

1. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique. One or two coordinate systems may be defined on a single entry.
2. GiA and GiB must be defined in coordinate whose definition 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 azimuth origin. The three grid points GiA (or GiB) must be noncollinear and not coincident.
3. Coordinate systems defined using CORD1C, CORD1R, and CORD1S entries cannot be used as reference coordinate systems on CORD2C, CORD2R, and CORD2S entries.
4. The location of a grid point (P in the sketch) 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. Points on the z-axis may not have their displacement directions defined in this coordinate system since ambiguity results. In this case the basic rectangular system will be used.

CORD1R — Rectangular Coordinate System Definition, Form 1

Description: Defines a rectangular coordinate system by reference to the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1R	CID	G1A	G2A	G3A	CID	G1B	G2B	G3B	

Example:

CORD1R	3	16	32	19					

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
GiA, GiB	Grid point identification numbers.	Integer > 0, G1A ≠ G2A ≠ G3A, G1B ≠ G2B ≠ G3B	0

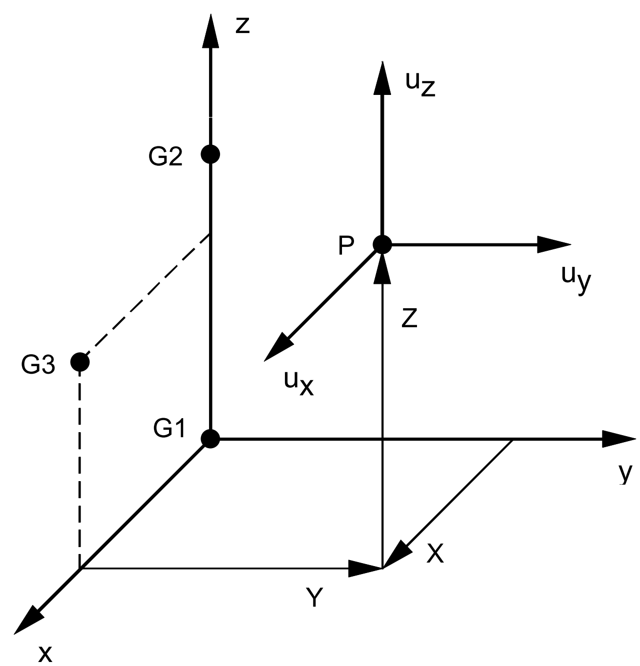


Figure 1. CORD1R Definition.

Remarks:

1. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate whose definition 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 azimuth origin. The three grid points GiA (or GiB) must be noncollinear and not coincident.
4. Coordinate systems defined using CORD1C, CORD1R, and CORD1S entries cannot be used as reference coordinate systems on CORD2C, CORD2R, and CORD2S entries.
5. The location of a grid point (P in the sketch) in this coordinate system is given by (X, Y, Z).
6. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_x, u_y, u_z).

CORD1S — Spherical Coordinate System Definition, Form 1

Description: Defines a spherical coordinate system by reference to the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1S	CID	G1A	G2A	G3A	CID	G1B	G2B	G3B	

Example:

CORD1S	4	5	43	55					

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
GiA, GiB	Grid point identification numbers.	Integer > 0, G1A ≠ G2A ≠ G3A, G1B ≠ G2B ≠ G3B	Required

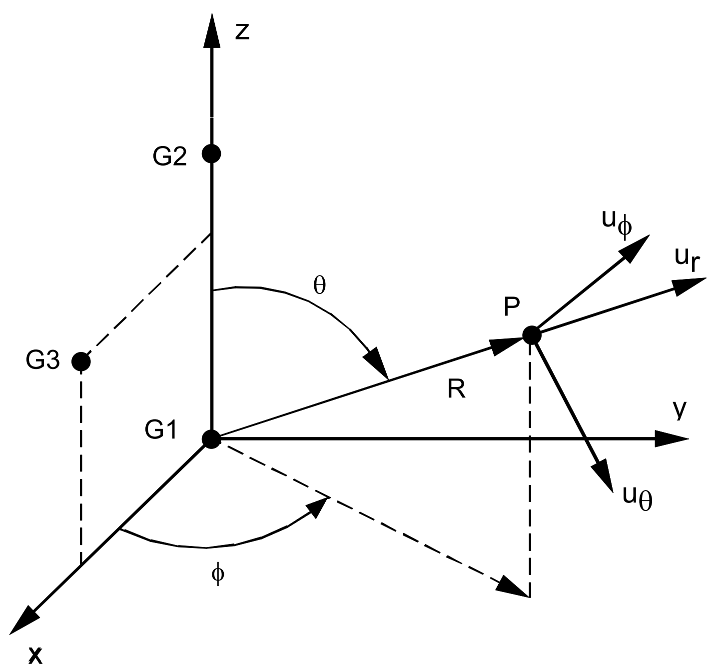


Figure 1. CORD1S Definition.

Remarks:

1. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate whose definition 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 azimuth origin. The three grid points GiA (or GiB) must be noncollinear and not coincident.
4. Coordinate systems defined using CORD1C, CORD1R, and CORD1S entries cannot be used as reference coordinate systems on CORD2C, CORD2R, and CORD2S entries.
5. The location of a grid point (P in the sketch) 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 dependent on the location of P as shown above by (u_r, u_θ, u_ϕ).
7. Points on the z-axis may not have their displacement directions defined in this coordinate system since ambiguity results. In this case the basic rectangular system will be used.

CORD2C — Cylindrical Coordinate System Definition, Form 2

Description: Defines a cylindrical coordinate system by reference to 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	5		0.0	0.0	0.0	0.0	0.0	1.0	
	1.0	1.0	0.0						

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
RID	Identification number of a coordinate system that is defined independently from this coordinate system.	Integer ≥ 0	0
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3.	Real	Required

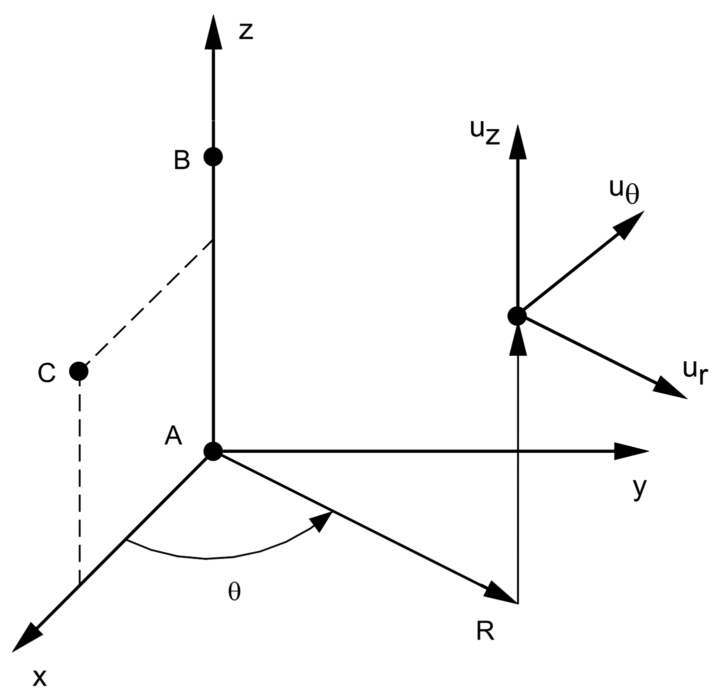


Figure 1. CORD2C Definition.

Remarks:

1. Continuation entry must be present.
2. The three points (A1, A2, A3), (B1, B2, B3), (C1, C2, C3) must be unique and noncollinear. The model translator checks for noncollinearity.
3. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
4. The reference coordinate system must be independently defined.
5. A RID of zero (or blank) references the basic coordinate system.
6. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.
7. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z).
8. Points on the z-axis may not have their displacement directions defined in this coordinate system since ambiguity results. In this case the basic rectangular system will be used.

CORD2R Rectangular Coordinate System Definition, Form 2

Description: Defines a rectangular coordinate system by reference to 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	5		0.0	0.0	0.0	0.0	0.0	1.0	
	1.0	1.0	0.0						

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
RID	Identification number of a coordinate system that is defined independently from this coordinate system.	Integer ≥ 0	0
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3.	Real	Required

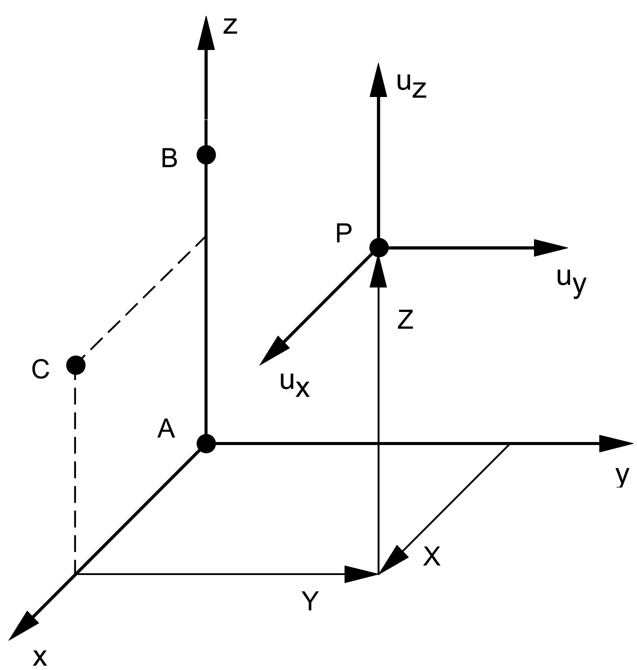


Figure 1. CORD2R Definition.

Remarks:

1. Continuation entry must be present.
2. The three points (A1, A2, A3), (B1, B2, B3), (C1, C2, C3) must be unique and noncollinear. The model translator checks for noncollinearity.
3. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
4. The reference coordinate system must be independently defined.
5. A RID of zero (or blank) references the basic coordinate system.
6. The location of a grid point (P in the sketch) in this coordinate system is given by (X, Y, Z).
7. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_x, u_y, u_z).

CORD2S Spherical Coordinate System Definition, Form 2

Description: Defines a spherical coordinate system by reference to 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	5		0.0	0.0	0.0	0.0	0.0	1.0	
	1.0	1.0	0.0						

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer > 0	Required
RID	Identification number of a coordinate system that is defined independently from this coordinate system.	Integer ≥ 0	0
Ai, Bi, Ci	Coordinates of three points in coordinate system defined in field 3.	Real	Required

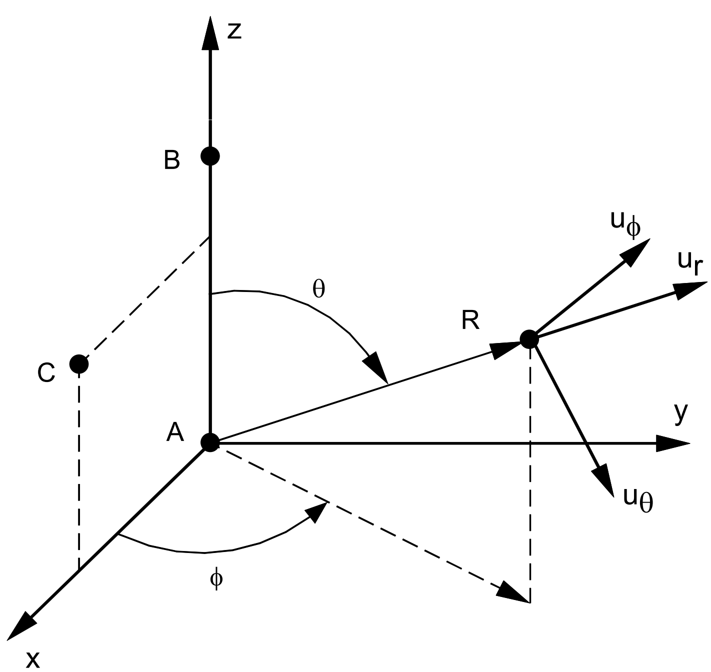


Figure 1. CORD2S Definition.

Remarks:

1. Continuation entry must be present.
2. The three points (A1, A2, A3), (B1, B2, B3), (C1, C2, C3) must be unique and noncollinear. The model translator checks for noncollinearity.
3. Coordinate system identification numbers on all CORD1C, CORD1R, CORD1S, CORD2C, CORD2R, and CORD2S entries must all be unique.
4. The reference coordinate system must be independently defined.
5. A RID of zero (or blank) references the basic coordinate system.
6. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, ϕ) where θ and ϕ are measured in degrees.
7. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_ϕ).
8. Points on the z-axis may not have their displacement directions defined in this coordinate system since ambiguity results. In this case the basic rectangular system will be used.

CPENTA Five-Sided Solid Element Connection

Description: Defines the connections of a five-sided isoparametric 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		34		125	
	130								

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank	Required

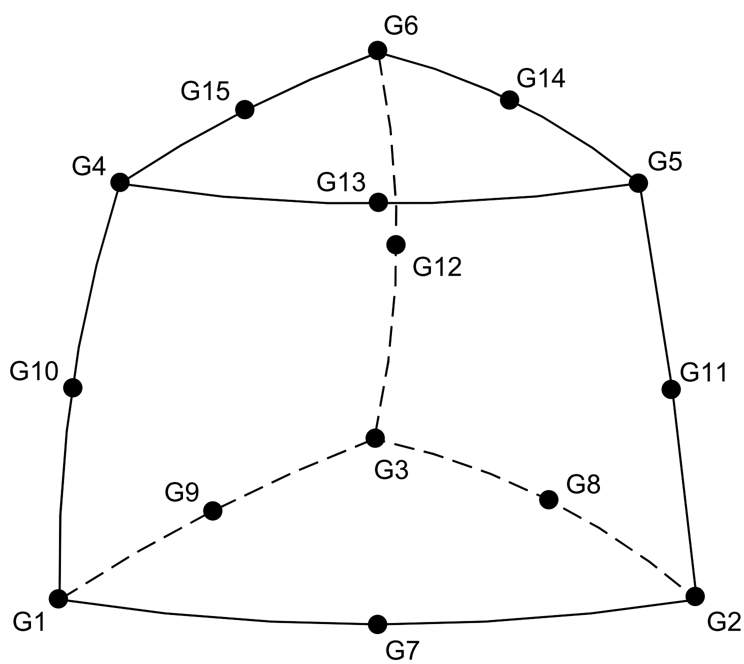


Figure 1. CPENTA Element Connection.

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The topology of the diagram must be preserved; i.e., G1, G2, G3 define a triangular face G1, G10, and G4 are on the same edge, etc.
3. Any or all of the edge points, G7 through G15, may be deleted. If the ID of any edge connection points is left blank or set to zero (as for G11 and G13 in the example), the element equations are modified to give the correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. Components of stress are output in the volume coordinate system. (See the VOLUME command in Section 3, *Case Control*.)
5. The material coordinate system is defined on the PSOLID entry.
6. It is recommended that the edge grid points be located within the middle third of the edge.
7. The element coordinate system is defined as follows:
8. The origin is located at the midpoint of a straight line joining points G1-G4. The x-axis passes through the midpoint of a straight line joining G2-G5. The z-axis is normal to a plane passing through the midpoints of straight lines joining G1-G4, G2-G5, and G3-G6.

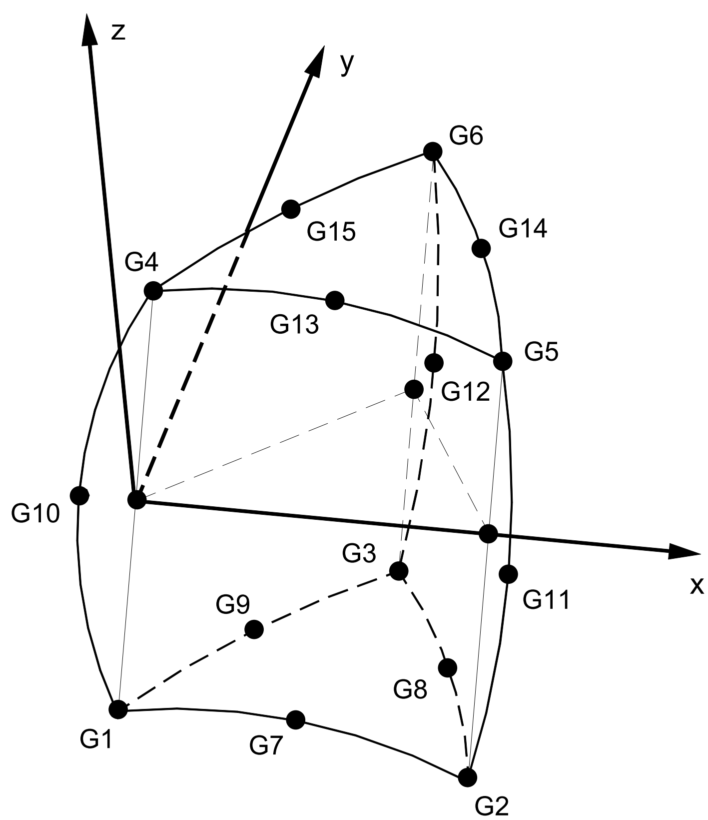


Figure 2. CPENTA Element Coordinate System.

CPIPE — Pipe Element Connection

Description: Defines a pipe element.

Format:

1	2	3	4	5	6	7	8	9	10
CPIPE	EID	PID	G1	G2					

Example:

CPIPE	50	20	301	302					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PPIPE property entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0, G1 ≠ G2	Required

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.

CPYRA — Five-Sided Solid Element Connection

Description: Defines the connections of a five-sided isoparametric solid element with five to thirteen grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CPYRA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13		

Example:

CPYRA	111	3	12	15	14	5	101	115	
	25	13	22	28		45			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank, all unique	Required

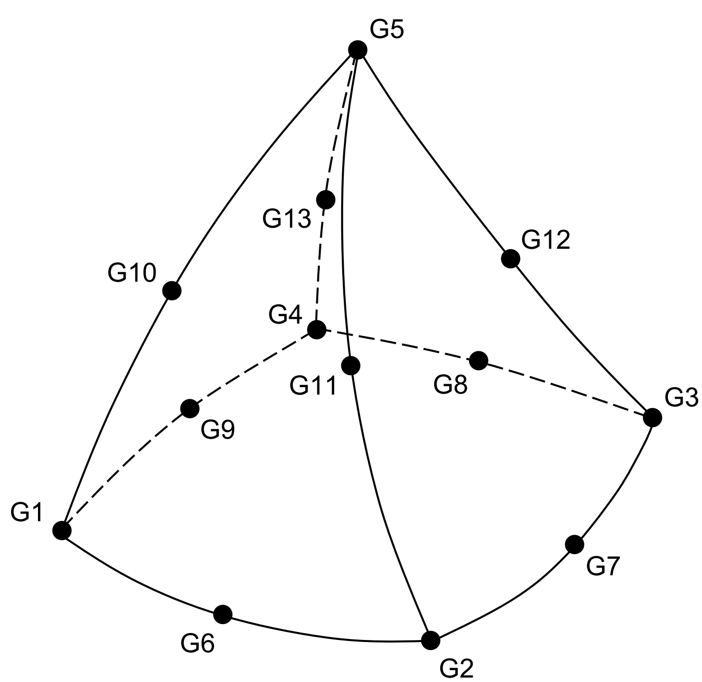


Figure 1. CPYRA Element Connection.

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The topology of the diagram must be preserved; i.e., G1, G2, G3, G4 define a quadrilateral face G1, G10, and G5 are on the same edge, etc.
3. Any or all of the edge points, G6 through G13, may be deleted. If the ID of any edge connection points is left blank or set to zero (as for G6 and G13 in the example), the element equations are modified to give the correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. Components of stress are output in the volume coordinate system. (See the VOLUME command in Section 3, *Case Control.)*
5. It is recommended that the edge grid points be located within the middle third of the edge.
6. The element coordinate system is defined as follows:

   The origin is located at G1 and the x-axis lies on the G1-G2 edge. The y-axis lies in the G1-G2-G4 plane and is perpendicular to the x-axis. The positive y-axis lies on the same side of the G1-G2 edge as node G4. The z-axis is orthogonal to the x and y axes.

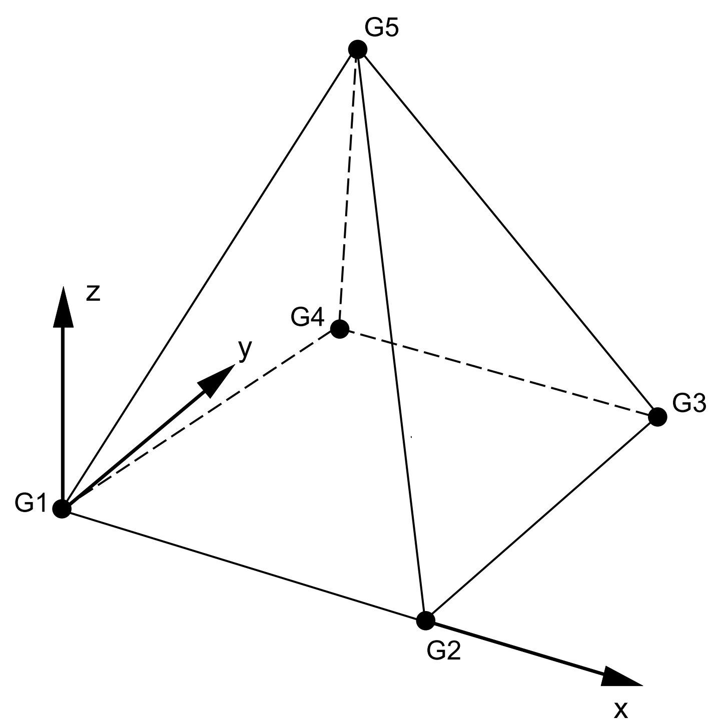


Figure 2. CPYRA Element Coordinate System.

CQUAD4 Quadrilateral Plate Element Connection

Description: Defines a quadrilateral, isoparametric membrane-bending or plane strain plate element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA/MCID	ZOFFS	
			T1	T2	T3	T4			

Example:

CQUAD4	61	11	101	111	201	202	0.0	1.0	

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHELL or PCOMP property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required
THETA	Material property orientation angle in degrees.	Real or blank	See Remark 6
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 6
ZOFFS	Offset from the surface of grid points to the element reference plane (see Remark 5).	Real or blank	0.0
Ti	Membrane thickness of element at G1, G2, G3, and G4.	Real ≥ 0.0 or blank	See Remark 7

Remarks:

1. Element identification numbers must 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. Stresses are output in the surface coordinate system. (See the SURFACE command in Section 3, *Case Control.*)
5. Elements may be offset from the grid point surface by means of ZOFFS. Other data, such as stress fiber locations are given relative to the reference plane. Positive offset implies that the element reference plane lies above the grid points. Use of a non-zero value for ZOFFS will produce membrane-bending coupling. Users must specify values for MID1, MID2, and MID3 in the PSHELL entry for the element if a non-zero value of ZOFFS is used. ZOFFS values must only be used when membrane and bending action is specified for the element. Absence of either of the actions does not allow development of membrane-bending coupling.

6. If THETA/MCID is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed when a non-isotropic material is referenced.
7. If Ti in fields 4 through 7 of the continuation entry are blank, field 4 of the PSHELL entry will be used. This is the preferred way of specifying element thickness if the thickness does not vary over the element.

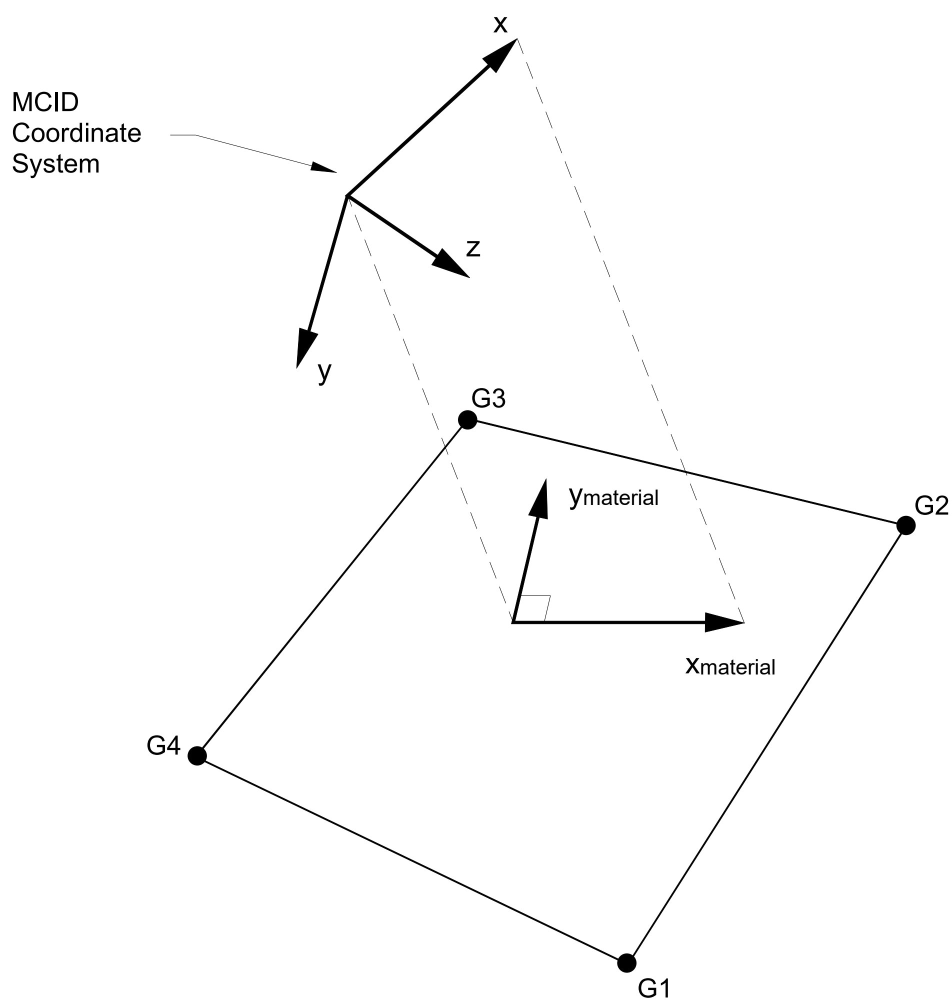


Figure 1. MCID Coordinate System Definition.

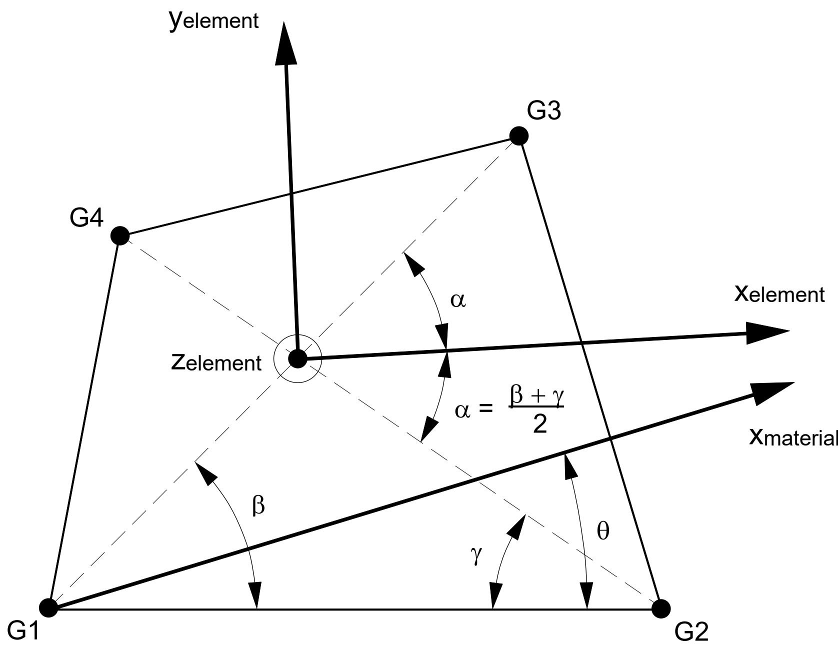


Figure 2. CQUAD4 Element Geometry and Coordinate System.

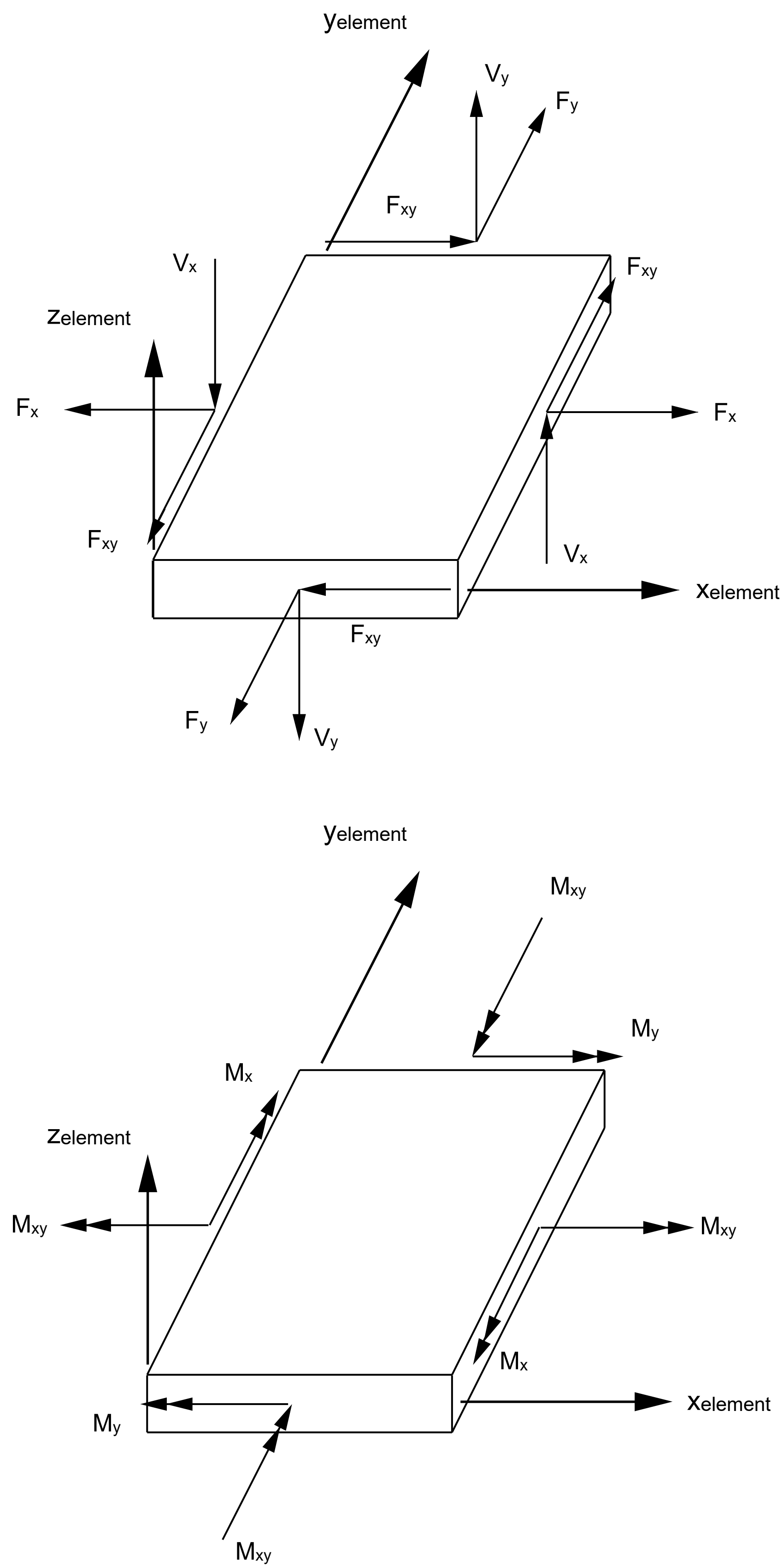


Figure 3. Forces and Moments in CQUAD4 Elements.

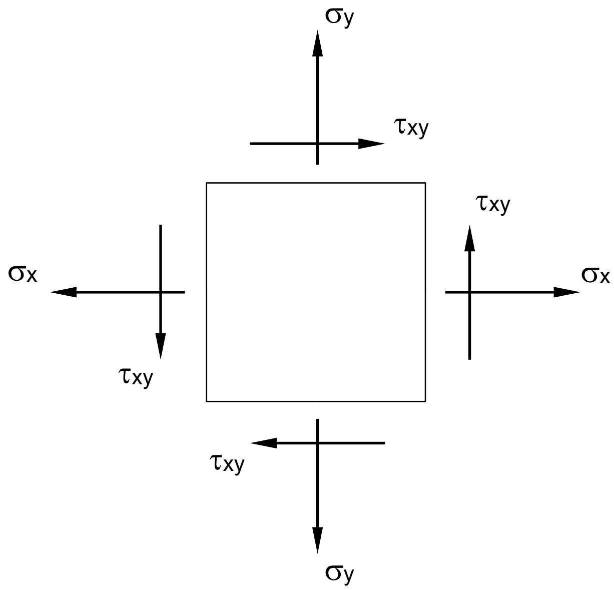


Figure 4. Stresses in CQUAD4 Elements.

CQUAD8 — Quadrilateral Plate Element Connection

Description: Defines a curved quadrilateral isoparametric shell or plane strain element with four to 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/MCID	ZOFFS	

Example:

CQUAD8	65	15	31	35	37	39	45	48	
	58	65					30.0		

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHELL or PCOMP property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required
Ti	Membrane thickness of element at G1, G2, G3, and G4.	Real ≥ 0.0 or blank	See Remark 9
THETA	Material property orientation angle in degrees.	Real or blank	See Remark 8
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 8
ZOFFS	Offset from the surface of grid points to the element reference plane (see Remark 7).	Real or blank	0.0

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G1 through G8 must be ordered as shown.
3. Any or all of the edge points, G5 through G8, may be deleted. If the ID of any edge connection points is left blank or set to zero, the element equations are modified to give the correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. 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 element may become singular.
5. All the interior angles must be less than 180°.
6. Stresses are output in the surface coordinate system. (See the SURFACE command in Section 3, *Case Control.*)

7. Elements may be offset from the grid point surface by means of ZOFFS. Other data, such as stress fiber locations are given relative to the reference plane. Positive offset implies that the element reference plane lies above the grid points. Use of a non-zero value for ZOFFS will produce membrane-bending coupling. Users must specify values for MID1, MID2, and MID3 in the PSHELL entry for the element if a non-zero value of ZOFFS is used. ZOFFS values must only be used when membrane and bending action is specified for the element. Absence of either of the actions does not allow development of membrane-bending coupling.
8. If THETA/MCID is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed when a non-isotropic material is referenced.
9. If Ti in fields 4 through 7 of the continuation entry are blank, field 4 of the PSHELL entry will be used. This is the preferred way of specifying element thickness if the thickness does not vary over the element.

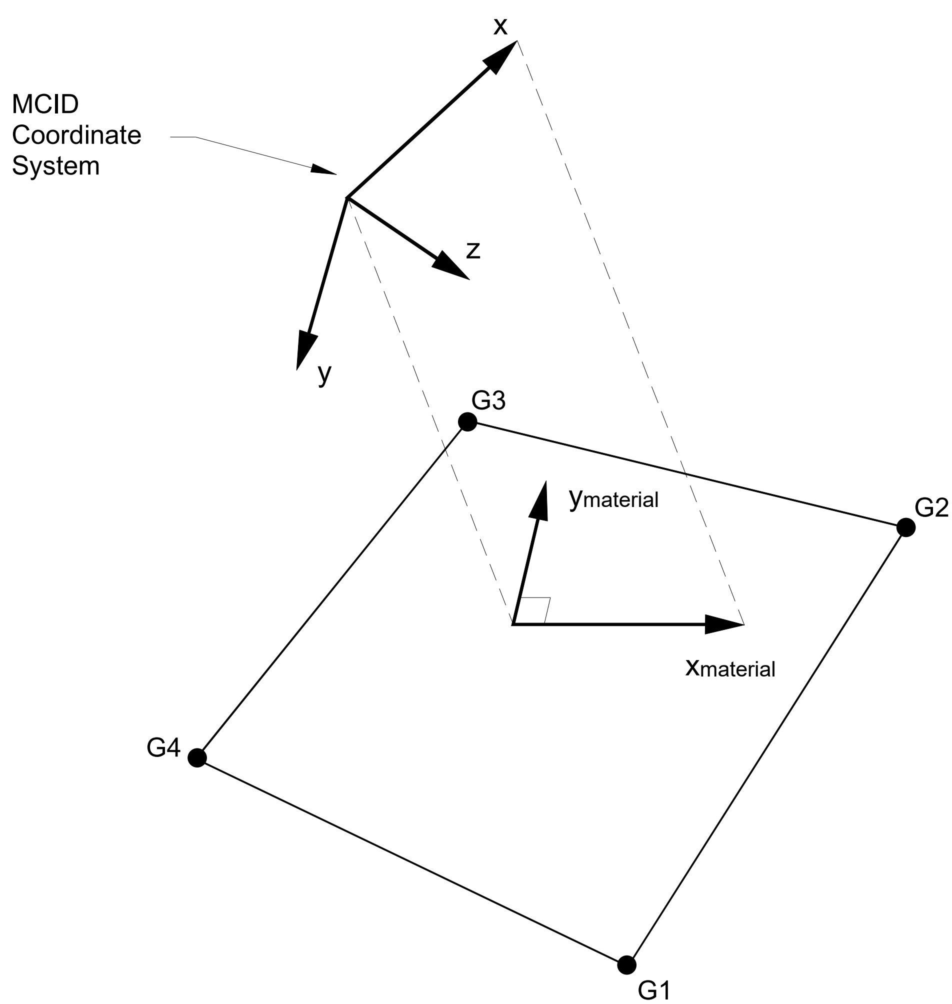


Figure 1. MCID Coordinate System Definition.

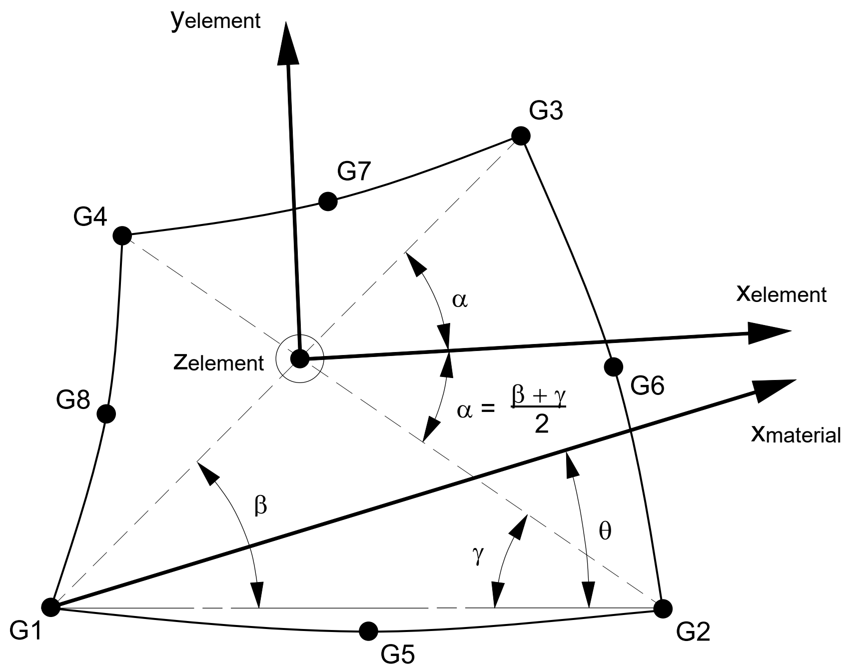


Figure 2. CQUAD8 Element Geometry and Coordinate System.

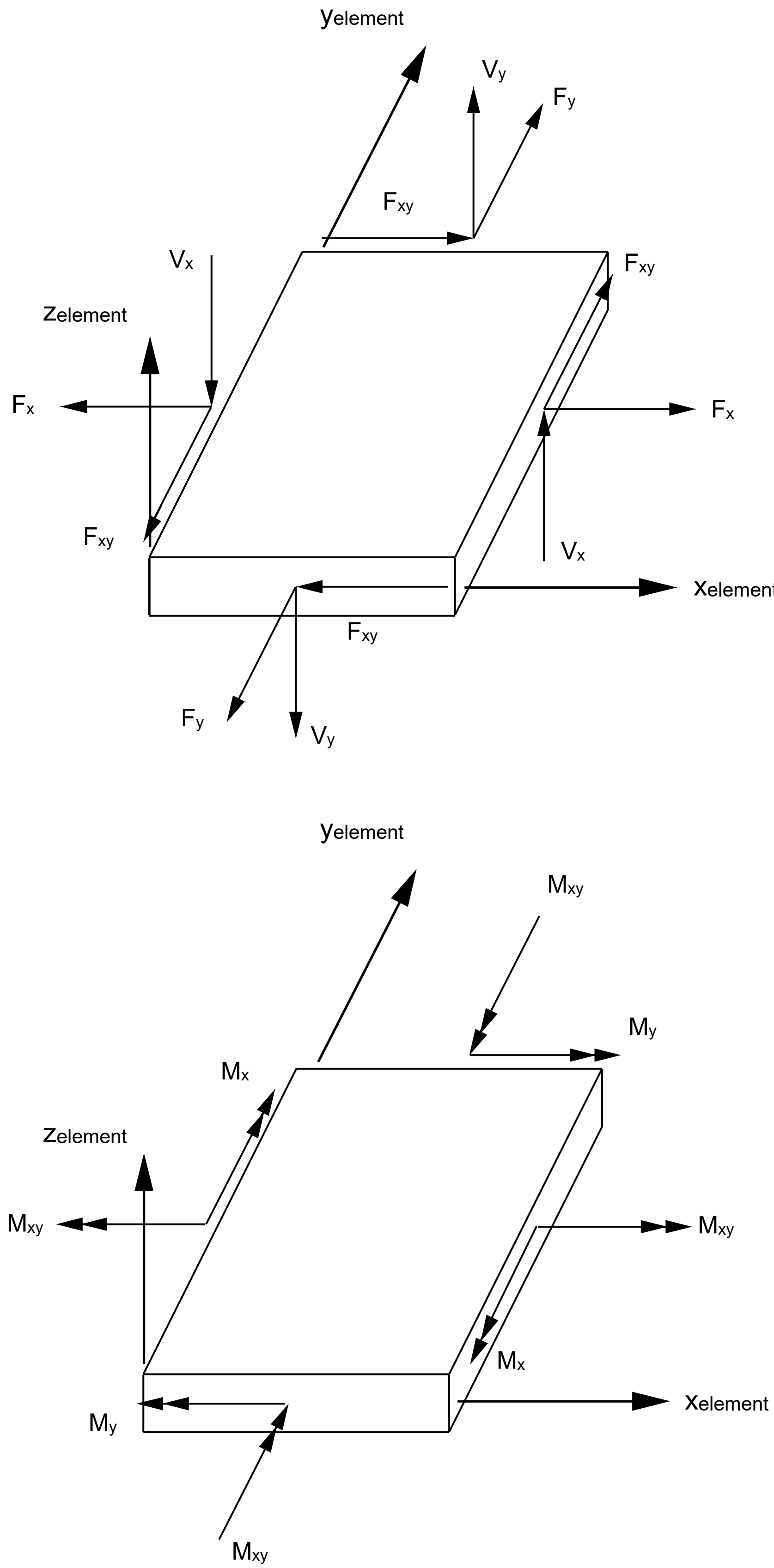


Figure 3. Forces and Moments in CQUAD8 Elements.

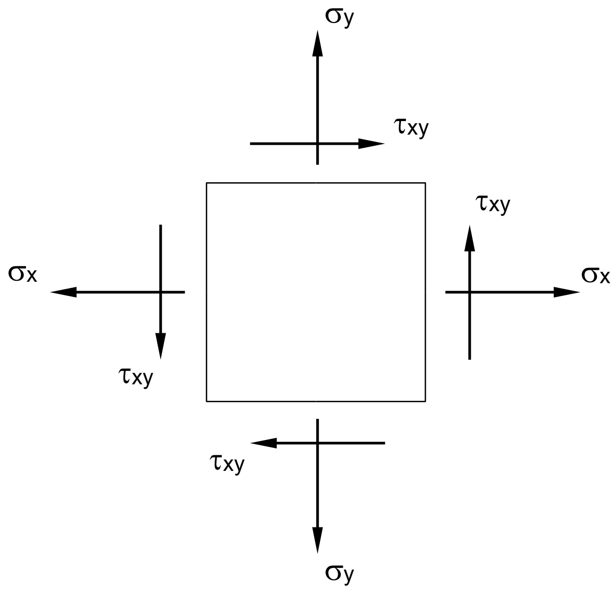


Figure 4. Stresses in CQUAD8 Elements.

CQUADR — Quadrilateral Plate Element Connection

Description: Defines a quadrilateral, isoparametric membrane-bending or plane strain plate element with vertex rotations.

Format:

1	2	3	4	5	6	7	8	9	10
CQUADR	EID	PID	G1	G2	G3	G4	THETA/MCID		
			T1	T2	T3	T4			

Example:

CQUADR	61	11	101	111	201	202			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHELL or PCOMP property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0 all unique	Required
THETA	Material property orientation angle in degrees.	Real	
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 7
Ti	Membrane thickness of element at G1, G2, G3, and G4.	Real	See Remark 8

Remarks:

1. Element identification numbers must 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. Components of stress are output in the surface coordinate system. (See the SURFACE command in Section 3, *Case Control.*)
5. The rotational degrees of freedom at the connection points and 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 generated.
6. This element is less sensitive to initial distortion and Poisson's ratio than the CQUAD4 element and is more compatible with the CBAR and CTRIAR elements that also have 6 degrees of freedom per node.
7. If THETA/MCID is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed when a non-isotropic material is referenced.

8. If Ti in fields 4 through 7 of the continuation entry is blank, field 4 of the PSHELL entry will be used. This is the preferred way of specifying element thickness if the thickness does not vary over the element.

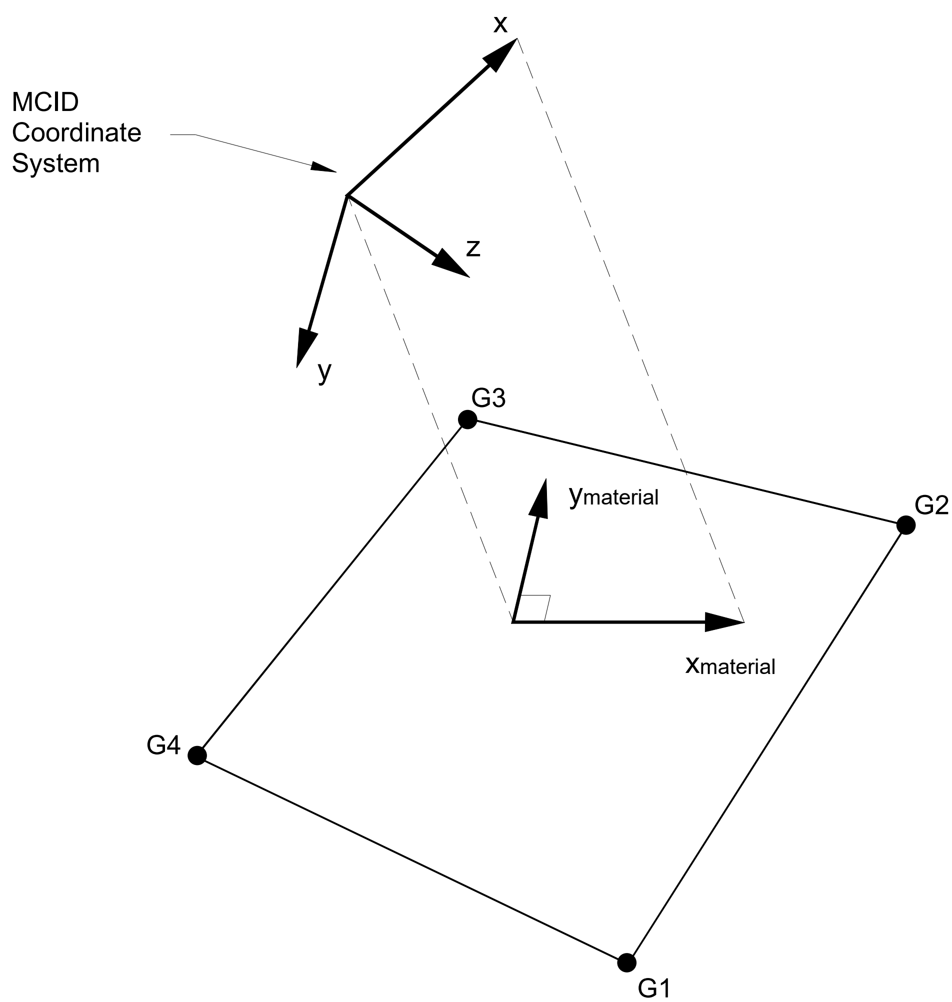


Figure 1. MCID Coordinate System Definition.

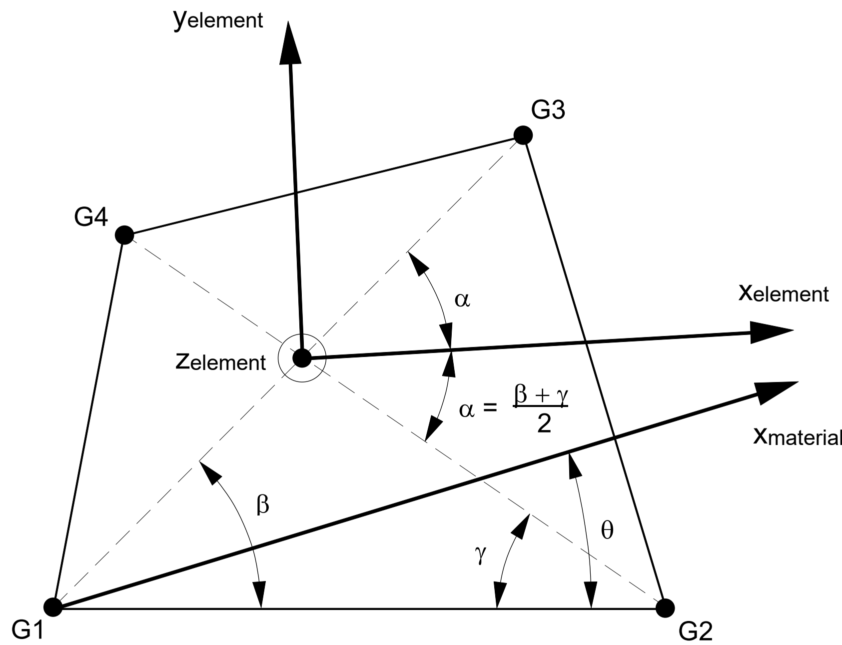


Figure 2. CQUADR Element Geometry and Coordinate System.

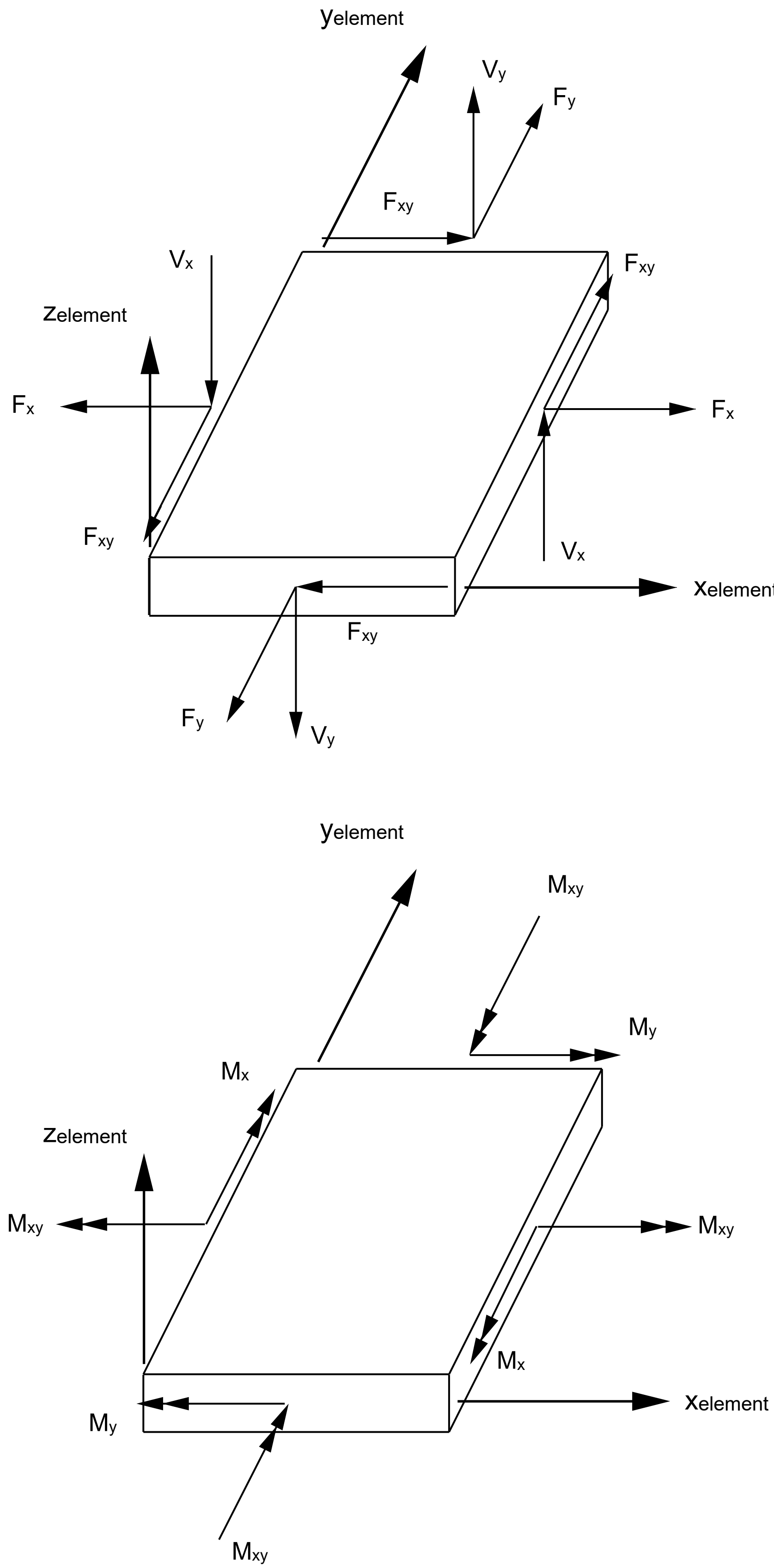


Figure 3. Forces and Moments in CQUADR Elements.

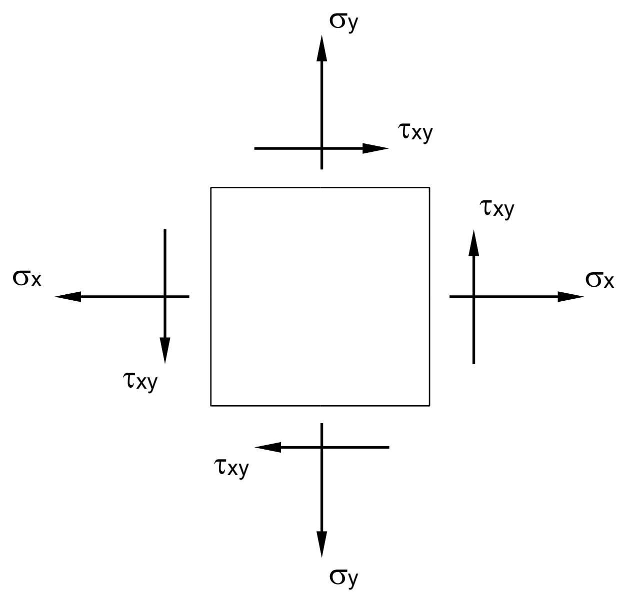


Figure 4. Stresses in CQUADR Elements.

CREEP Creep Characteristics

Description: Defines creep characteristics based on experimental data or known empirical creep law.

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	10	1000.		CRLAW					
	122	7.984-5	2.612	6.151-4	0.2271	7.63-9	0.1760	3.0	

Field	Definition	Type	Default
MID	Identification number of a MAT1, MAT2, MAT9, or MAT12 entry.	Integer > 0	Required
T0	Reference temperature at which creep characteristics are defined. See Remark 2.	Real or blank	0.0
EXP	Temperature-dependent term, $e^{-\Delta H / (R * T0)}$, in the creep rate expression. See Remark 2.	Real or blank	1.0E-9
FORM	Form of the input data defining creep characteristics, one of the following character variables: CRLAW for empirical creep law or TABLE for tabular input data of creep model parameters.	Character	Required
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	Required
THRESH	Threshold limit for creep process. Threshold stress under which creep does not occur is computed as THRESH multiplied by Young's modulus.	Real or blank	1.0E-5
TYPE	Identification number of the empirical creep law type, one of the following integers: 111, 112, 121, 122, 211, 212, 221, 222, or 300. Not required if FORM = TABLE. See Remarks 1 and 3.	Integer > 0	
a – g	Coefficients of the empirical creep law specified in TYPE. Not required if FORM = TABLE. See Remark 1.	Real or blank	

Remarks:

1. This entry will be activated if a MAT1, MAT2, MAT9, or MAT12 entry with the same MID is used and the NLPARM entry is prepared for creep analysis.
2. The creep formulation is principally suited for isotropic materials and when used with anisotropic materials may produce incorrect results. However, slightly anisotropic materials may produce acceptable results.
3. Two classes of empirical creep law are available.

   Creep Law Class 1:

   The first creep law class is expressed as:

$$\varepsilon^{c}(\sigma,t) = A(\sigma)\left[1-e^{-R(\sigma)t}\right] + K(\sigma)t$$

   Parameters $A(\sigma)$, $R(\sigma)$, and $K(\sigma)$ are specified in the following form, as recommended by Oak Ridge National Laboratory:

Parameters	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
$K(\sigma)$	$e*[\sinh(f\sigma)]^{g}$	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 is expressed as:

$$\varepsilon^{c}(\sigma,t) = a\sigma^{b}t^{d}$$

   where the values of b and d must be defined as follows:

$$1.0 < b < 8.0$$

and

$$0.2 < d < 1.0$$

   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.
4. 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, TEMP(MATERIAL) = SID, or TEMP(BOTH) = SID within the subcase.

From the thermodynamic consideration, the creep rate is expressed as:

$$\dot{\varepsilon}^c = \dot{\varepsilon}_A \left(e^{-\Delta H / (R * T0)} \right)$$

where ΔH = energy of activation

R = gas constant (1.98 cal/mole °K)

T = absolute temperature (°K)

$\dot{\varepsilon}^c$ = strain/second per activation

If the creep characteristics are defined at temperature T0, the creep rate at temperature T is corrected by a factor

$$\frac{\dot{\varepsilon}^c}{\dot{\varepsilon}^c_o} = \text{EXP}^{\left(\frac{T0}{T} - 1 \right)}$$

where $\dot{\varepsilon}^c$ = corrected creep rate

$\dot{\varepsilon}^c_o$ = creep rate at T0

$\text{EXP}^{\left(\frac{T0}{T} - 1 \right)}$ = correction factor

5. Creep model parameters K_p, C_p, and C_s represent parameters of the uniaxial rheological model as shown in the following figure.

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}.

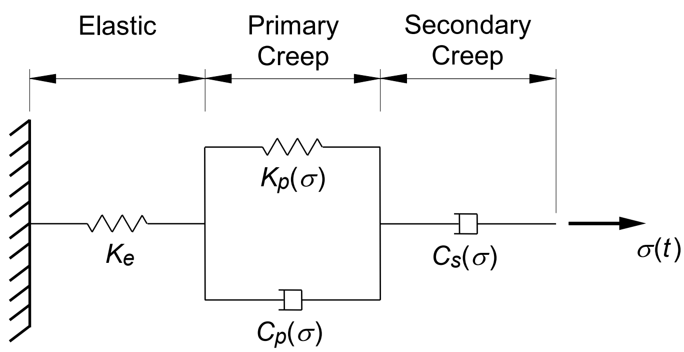


Figure 1. CREEP Parameter Idealization.

Creep model parameters, as shown in the figures 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 will be issued.

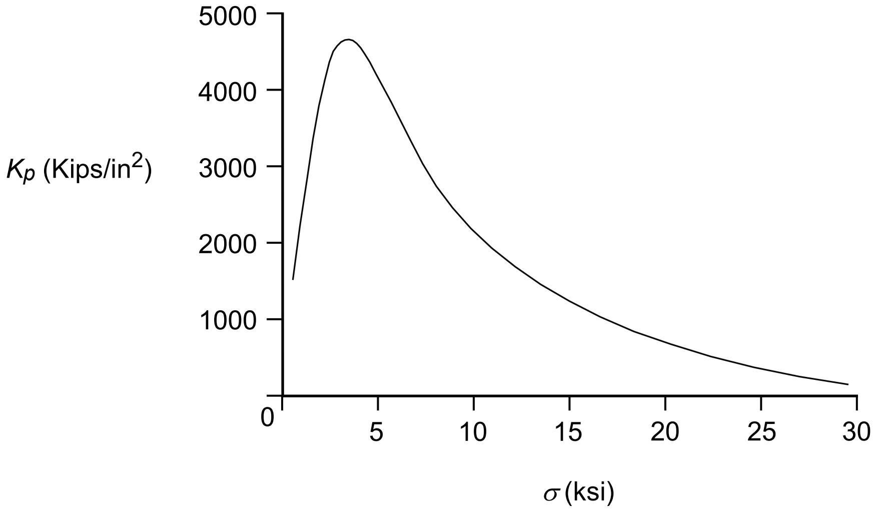


Figure 2. K_p Versus σ Example for CREEP.

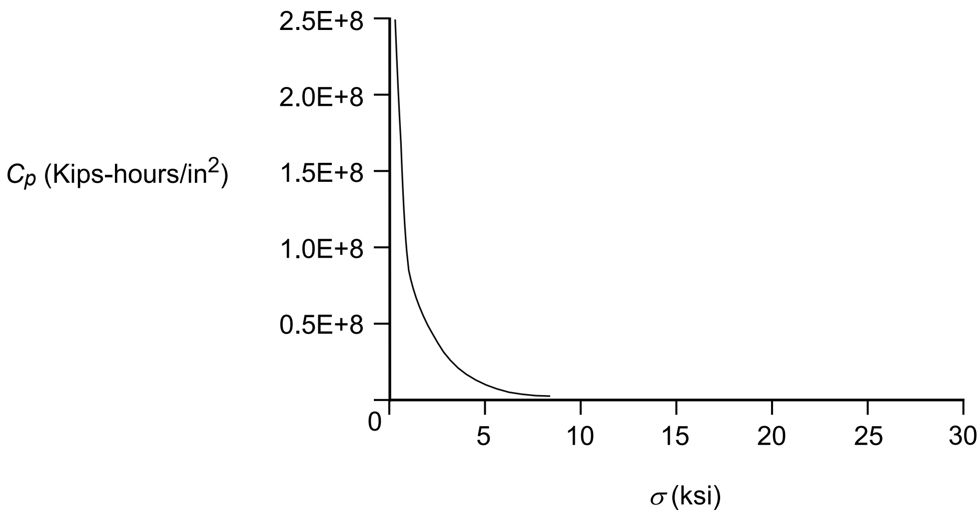


Figure 3. C_p Versus σ Example for CREEP.

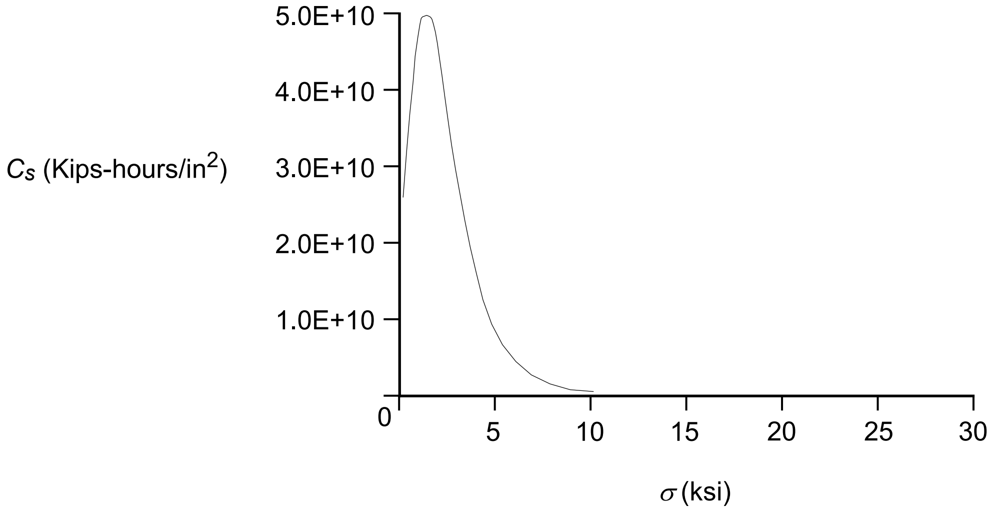


Figure 4. C_s Versus σ Example for CREEP

CROD Rod Element Connection

Description: Defines a tension-compression-torsion element.

Format:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

Example:

CROD	61	11	101	111					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PROD property entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0, G1 ≠ G2	Required

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.

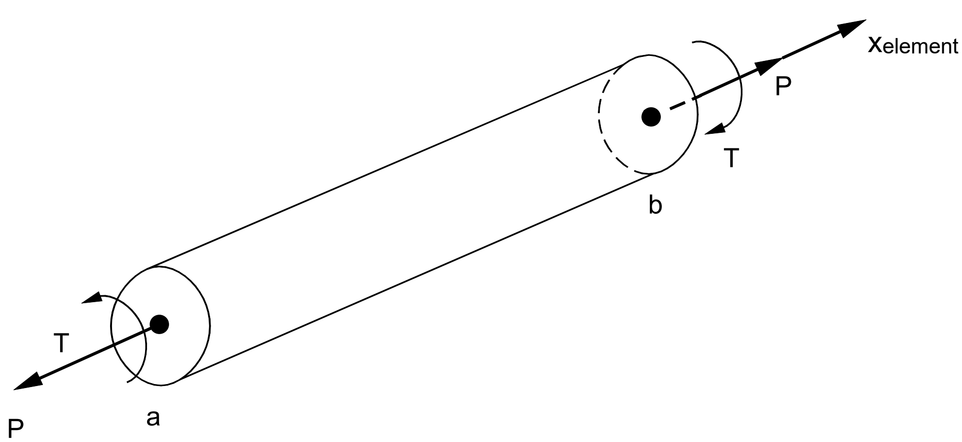


Figure 1. CROD Element Internal Forces and Moments.

CSET — Free Boundary Analysis Set Definition

Description: 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
CSET	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

CSET	15	3	17	456	7	4			

Field	Definition	Type	Default
Gi	Grid point identification number(s).	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required

Remarks:

1. If there are no CSETi or BSETi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi entries present, any a-set degrees of freedom not listed are placed in the free boundary set (c-set). If there are both BSETi and CSETi entries present, the c-set degrees of freedom are defined by the CSETi entries, and any remaining a-set points are placed in the b-set.

CSET1 Free Boundary Analysis Set Definition, Alternate Form

Description: 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
CSET1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	- etc.-					

Example:

CSET1	123	6	3	7	10	18	14	11	
	19	23							

Alternate Format and Example:

CSET1	C	G1	THRU	G2					

CSET1	456	15	THRU	512					

Field	Definition	Type	Default
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required
Gi	Grid point identification number(s).	Integer $>$ 0; G1 < G2	Required

Remarks:

1. If there are no CSETi or BSETi entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi entries present, any a-set degrees of freedom not listed are placed in the free boundary set (c-set). If there are both BSETi and CSETi entries present, the c-set degrees of freedom are defined by the CSETi entries, and any remaining a-set points are placed in the b-set.

CSHEAR — Shear Panel Element Connection

Description: 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	61	11	101	111	201	202			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHEAR property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required

Remarks:

1. Element identification numbers must 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°.

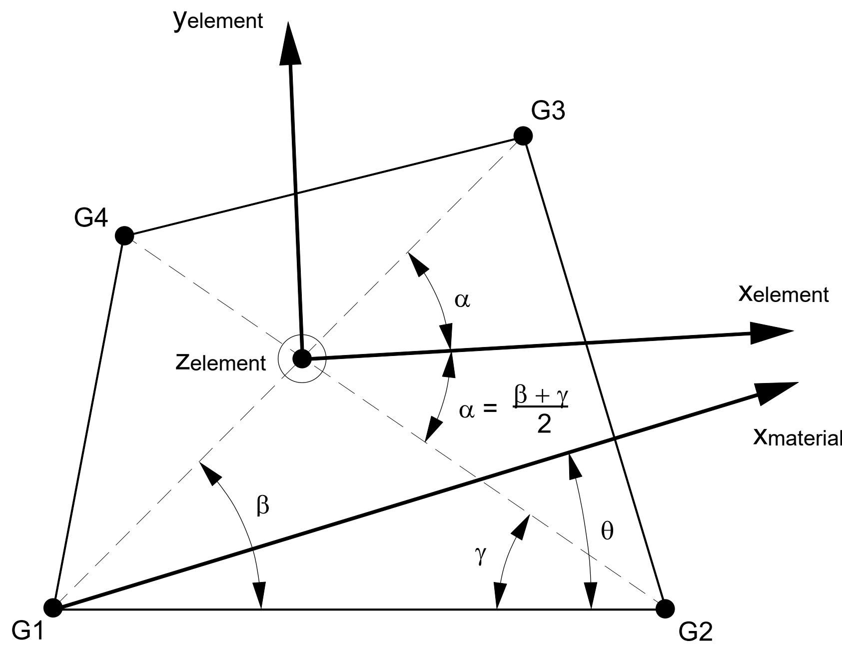


Figure 1. CSHEAR Element Connection and Coordinate System.

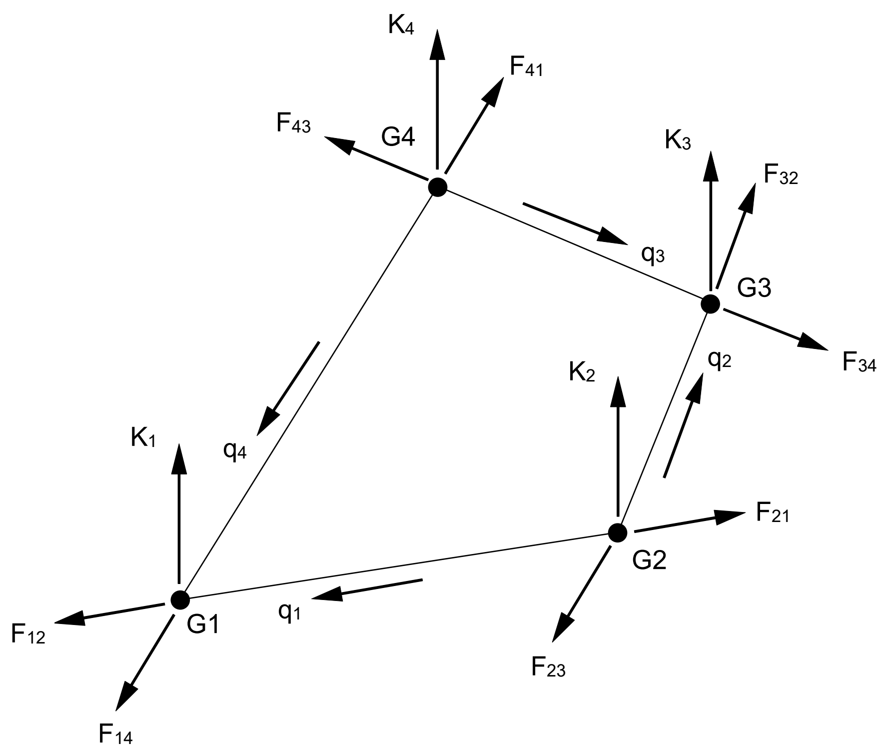


Figure 2. CSHEAR Element Corner Forces and Shear Flows.

CTETRA — Four-Sided Solid Element Connection

Description: Defines the connections of a four-sided isoparametric 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			27					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PSOLID entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank, all unique	Required

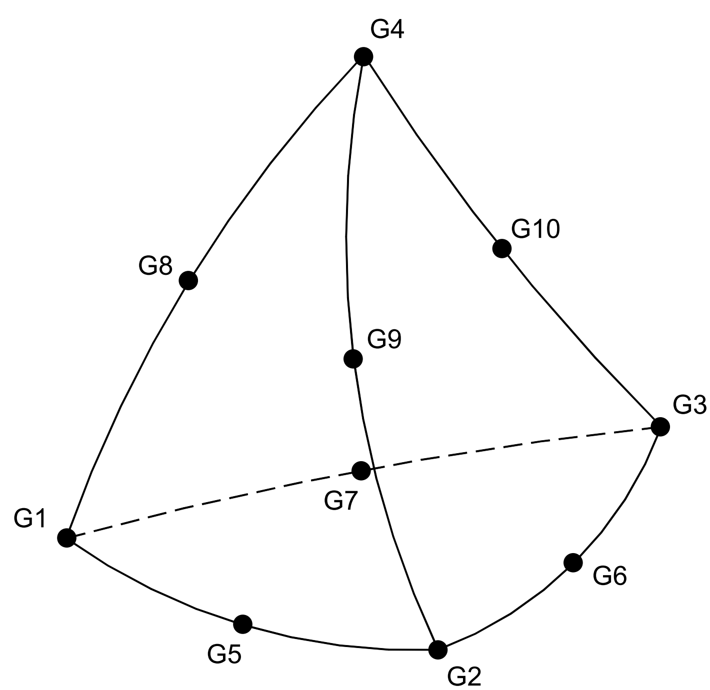


Figure 1. CTETRA Element Connection.

Remarks:

7. Element identification numbers must be unique with respect to all other element identification numbers.
8. 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.
9. Any or all of the edge points, G5 through G10, may be deleted. If the ID of any edge connection points is left blank or set to zero (as for G8 and G9 in the example), the element equations are modified to give the correct results for the reduced number of connections. Corner grid points cannot be deleted.
10. Components of stress are output in the volume coordinate system. (See the VOLUME command in Section 3, *Case Control.)*
11. It is recommended that the edge grid points be located within the middle third of the edge.
12. The element coordinate system is defined as follows:

    The origin is located at G1 and the x-axis lies on the G1-G2 edge. The y-axis lies in the G1-G2-G3 plane and is perpendicular to the x-axis. The positive y-axis lies on the same side of the G1-G2 edge as node G3. The z-axis is orthogonal to the x and y axes.

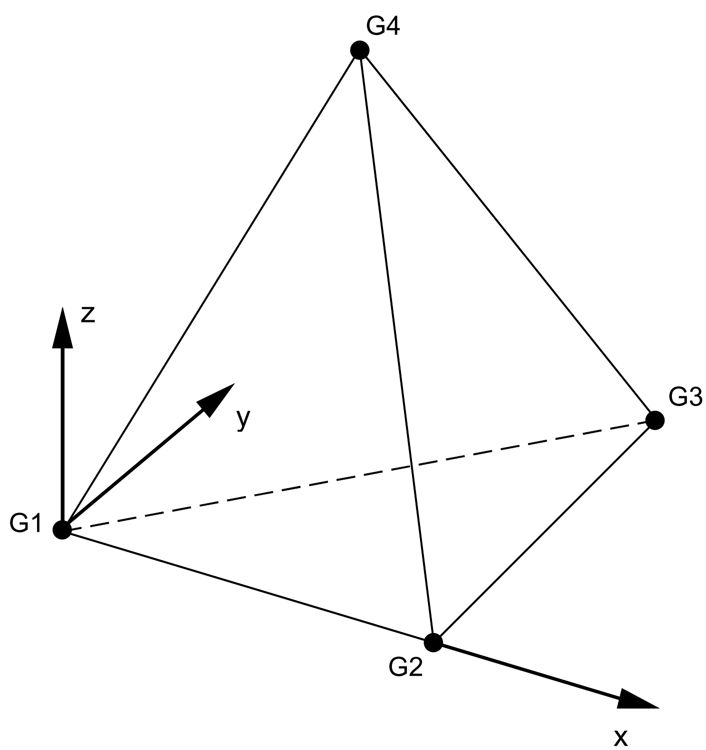


Figure 2. CTETRA Element Coordinate System.

CTRAX6 Axisymmetric Triangular Element Connection

Description: Defines an isoparametric axisymmetric triangular cross-section solid element with midside grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRAX	EID	MID	G1	G2	G3	G4	G5	G6	
	THETA								

Example:

CTRAX	21	100	20	21	22	31	32	33	
	15.0								

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
MID	Material identification number.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required
THETA	Material property orientation angle in degrees.	Real	0.0

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. 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. If the ID of any edge connection points is left blank or set to zero, the element equations are modified to give the correct results for the reduced number of connections. Corner grid points cannot be deleted.
3. For structural problems, the MID must reference a MAT1 or MAT3 material entry.
4. The continuation is optional.
5. Material properties (if defined on a MAT3 entry) and stresses are given in the $(r_{material} - z_{material})$ coordinate system shown in Figure 2.
6. 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:

   $$(100\,N/m)\,2\pi\,(0.5\,m) = 314.159\,N$$

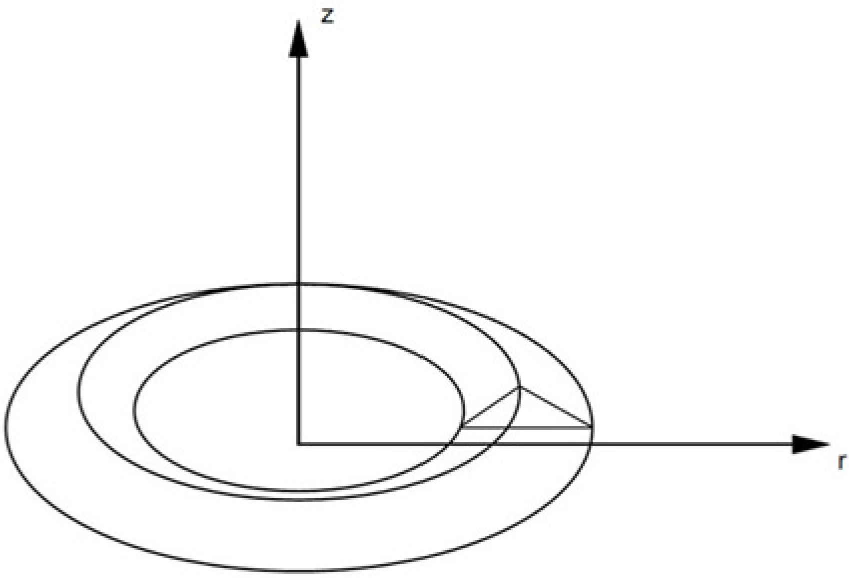


Figure 1. CTRAX6 Element Idealization

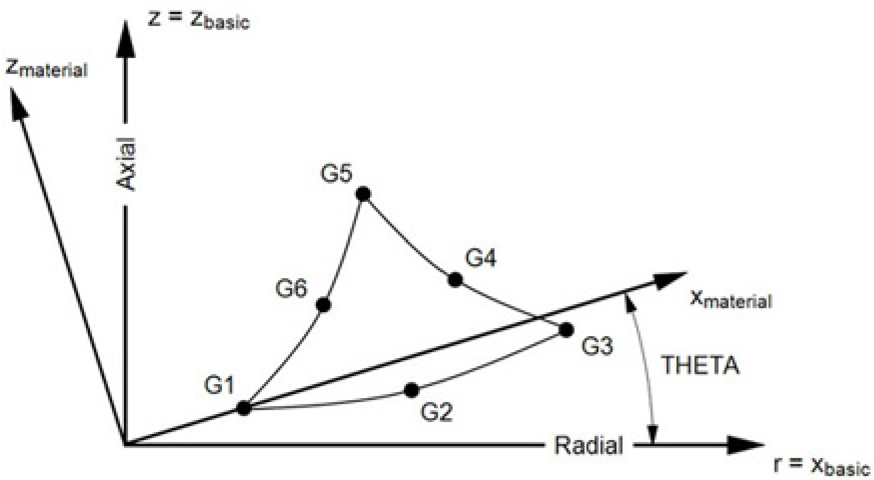


Figure 2. CTRAX6 Element Geometry and Coordinate Systems

(Continued)

CTRIA3 Triangular Element Connection

Description: Defines a triangular, isoparametric membrane-bending or plane strain plate element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA/MCID	ZOFFS		
			T1	T2	T3				

Example:

CTRIA3	61	11	101	111	202				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHELL or PCOMP property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required
THETA	Material property orientation angle in degrees.	Real	See Remark 4
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 4
ZOFFS	Offset from the surface of grid points to the element reference plane (see Remark 3).	Real	0.0
Ti	Membrane thickness of element at G1, G2, and G3.	Real ≥ 0.0	See Remark 5

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Stresses are output in the surface coordinate system. (See the SURFACE command in Section 3, *Case Control.*)
3. Elements may be offset from the grid point surface by means of ZOFFS. Other data such as stress fiber locations are given relative to the reference plane. Positive offset implies that the element reference plane lies above the grid points in the sketch. Use of a non-zero value for ZOFFS will produce membrane-bending coupling. Users must specify values for MID1, MID2, and MID3 in the PSHELL entry for the element if a non-zero value of ZOFFS is used. ZOFFS values must only be used when membrane and bending action is specified for the element. Absence of either of the actions does not allow development of membrane-bending coupling.
4. If THETA/MCID is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed when a non-isotropic material is referenced.

5. If Ti in fields 4 through 6 of the continuation entry are blank, field 4 of the PSHELL entry will be used. This is the preferred way of specifying element thickness if the thickness does not vary over the element.

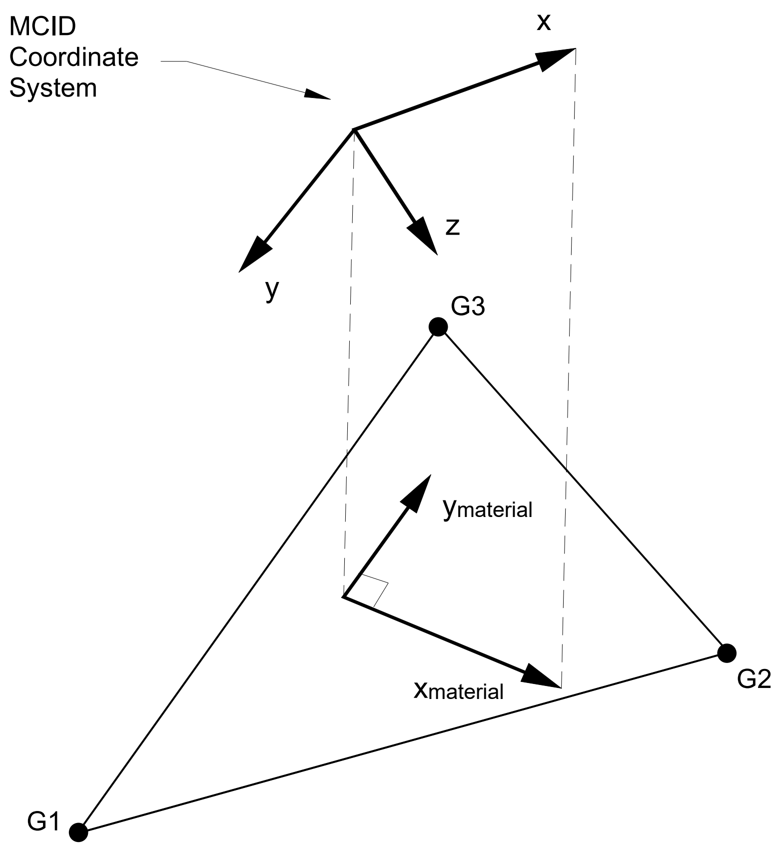


Figure 1. MCID Coordinate System Definition.

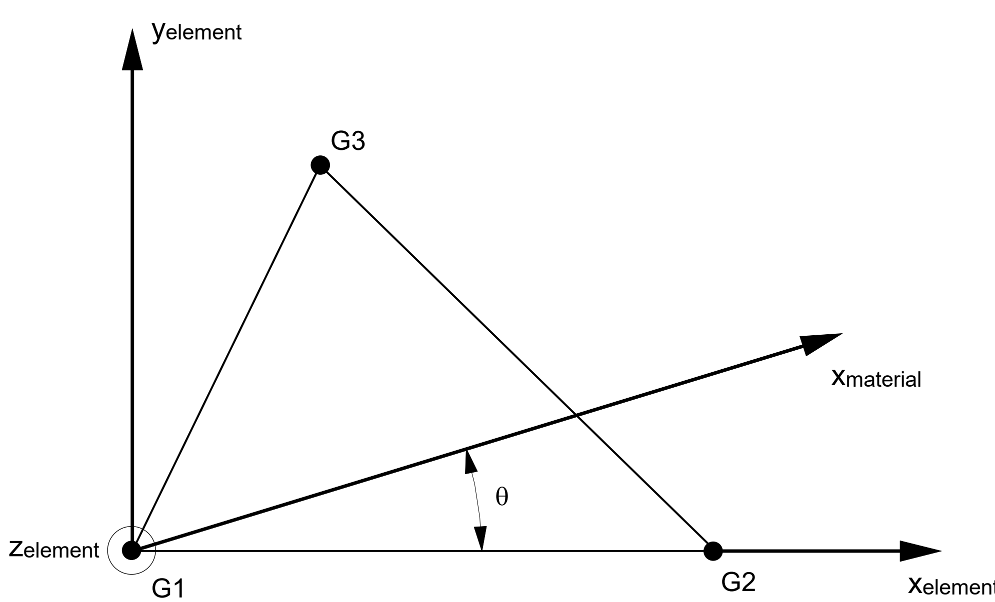
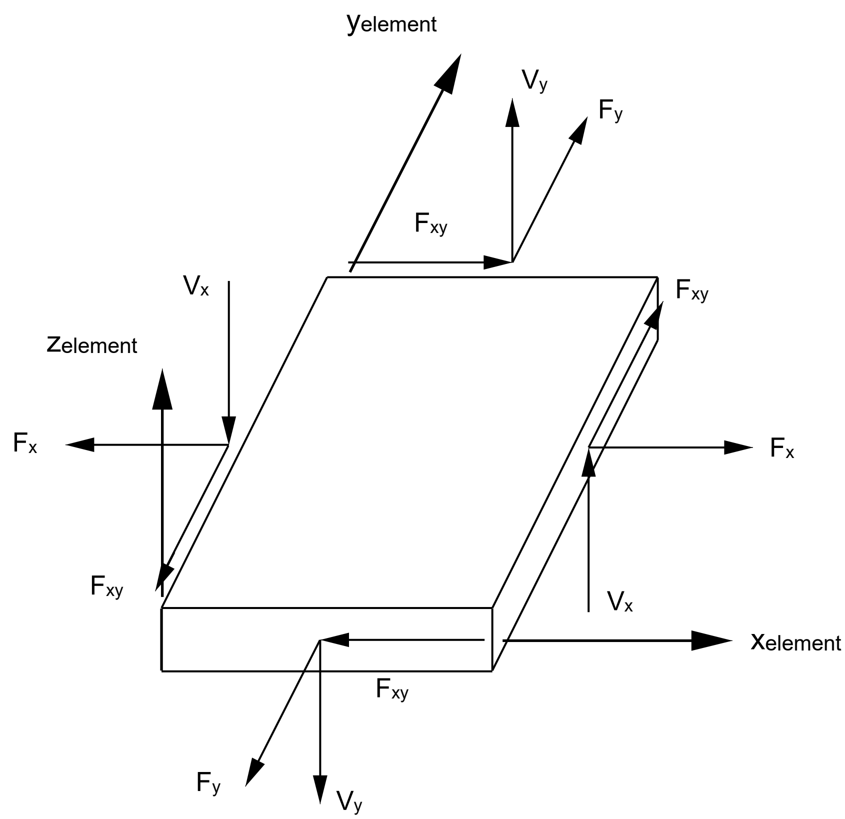


Figure 2. CTRIA3 Element Geometry and Coordinate Systems.



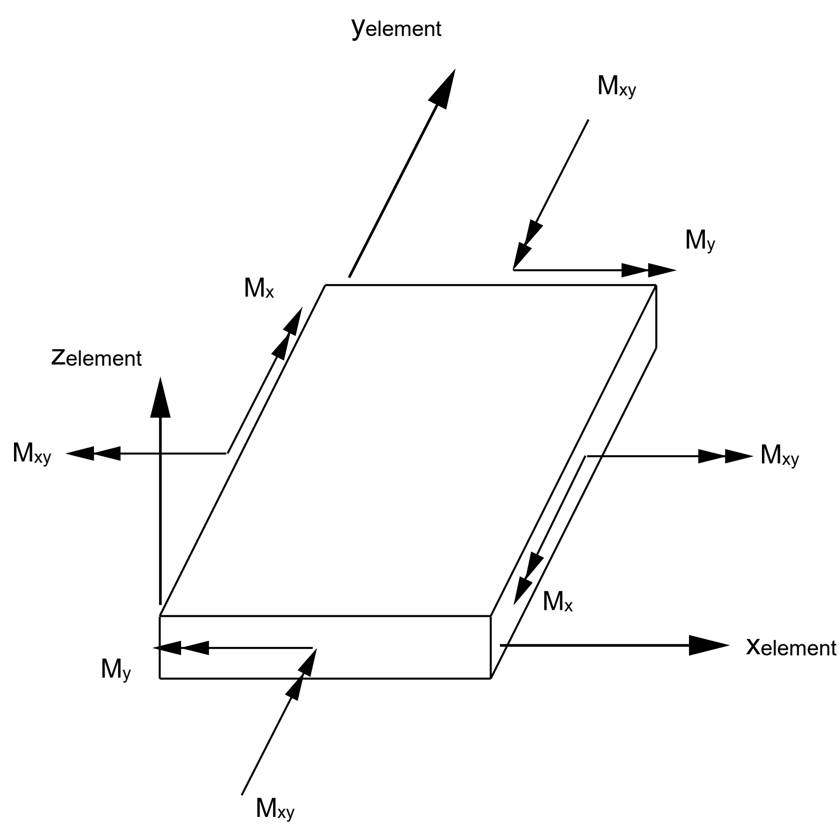


Figure 3. Forces and Moments in CTRIA3 Elements.

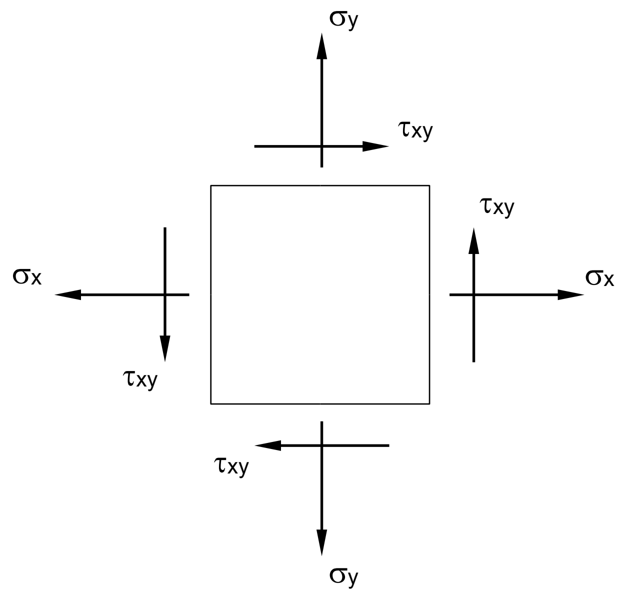


Figure 4. Stresses in CTRIA3 Elements.

CTRIA6 Triangular Element Connection

Description: Defines a curved triangular isoparametric shell or plane strain element with three to six grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA6	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA/MCID	ZOFFS	T1	T2	T3				

Example:

CTRIA6	65	15	45	48	50	67	89	95	
	45.0								

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHELL or PCOMP property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required
THETA	Material property orientation angle in degrees.	Real	See Remark 7
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 7
ZOFFS	Offset from the surface of grid points to the element reference plane (see Remark 6).	Real	0.0
Ti	Membrane thickness of element at G1, G2, and G3.	Real ≥ 0.0	See Remark 8

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G1 through G6 must be ordered as shown.
3. Any or all of the edge points, G4 through G6, may be deleted. If the ID of any edge connection points is left blank or set to zero, the element equations are modified to give the correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. 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 element may become singular.
5. Stresses are output in the surface coordinate system. (See the SURFACE command in Section 3, *Case Control.*)

6. Elements may be offset from the grid point surface by means of ZOFFS. Other data such as stress fiber locations are given relative to the reference plane. Positive offset implies that the element reference plane lies above the grid points in the sketch. Use of a non-zero value for ZOFFS will produce membrane-bending coupling. Users must specify values for MID1, MID2, and MID3 in the PSHELL entry for the element if a non-zero value of ZOFFS is used. ZOFFS values must only be used when membrane and bending action is specified for the element. Absence of either of the actions does not allow development of membrane-bending coupling.
7. If THETA/MCID is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed when a non-isotropic material is referenced.
8. If Ti in fields 4 through 6 of the continuation entry are blank, field 4 of the PSHELL entry will be used. This is the preferred way of specifying element thickness if the thickness does not vary over the element.

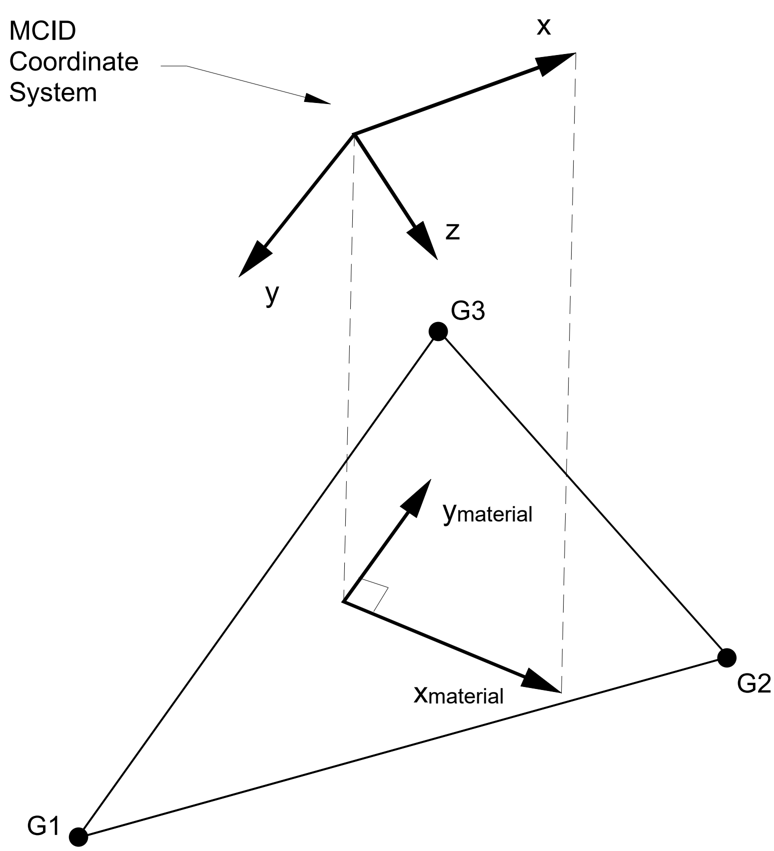


Figure 1. MCID Coordinate System Definition.

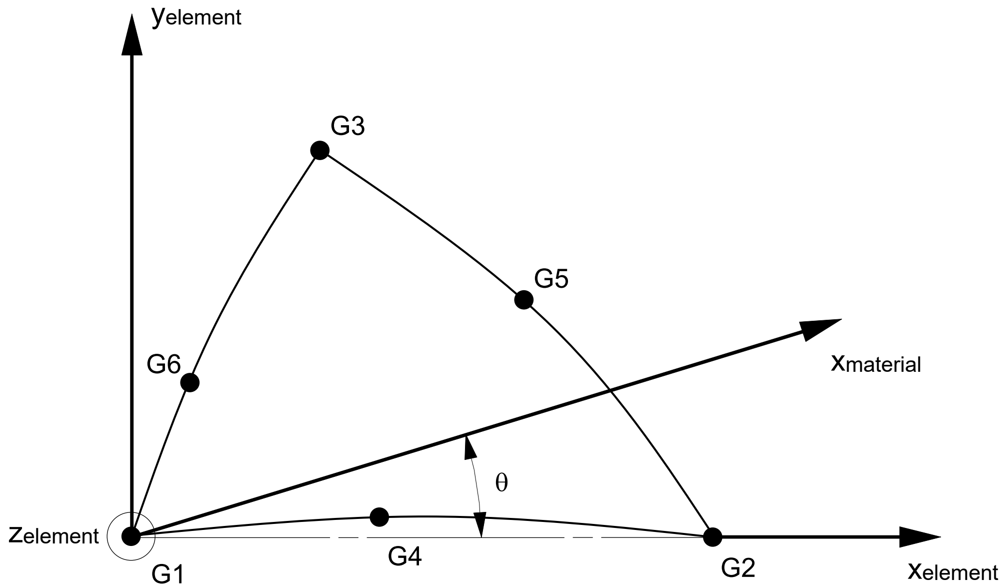


Figure 2. CTRIA6 Element Geometry and Coordinate Systems.

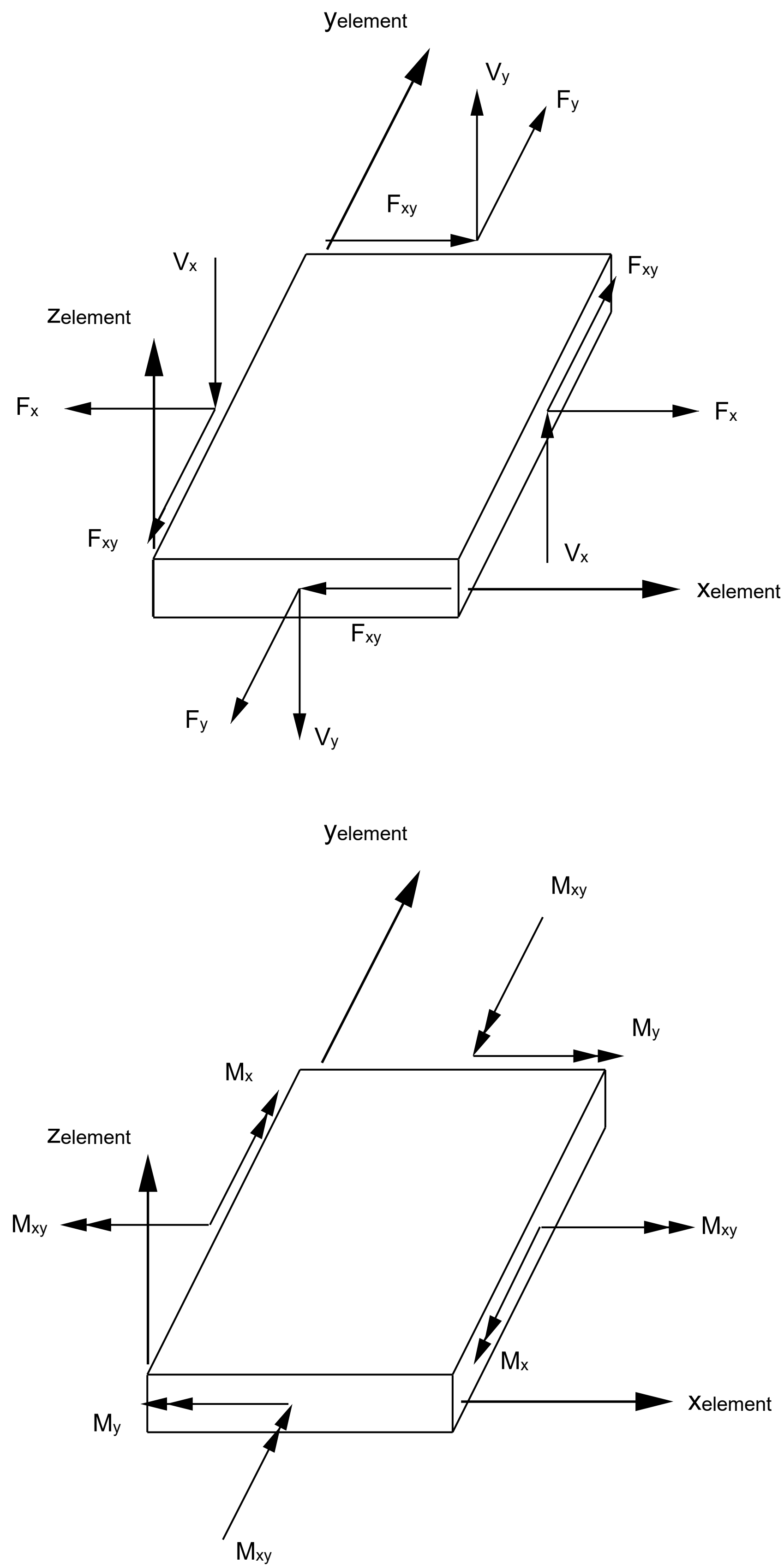


Figure 3. Forces and Moments in CTRIA6 Elements.

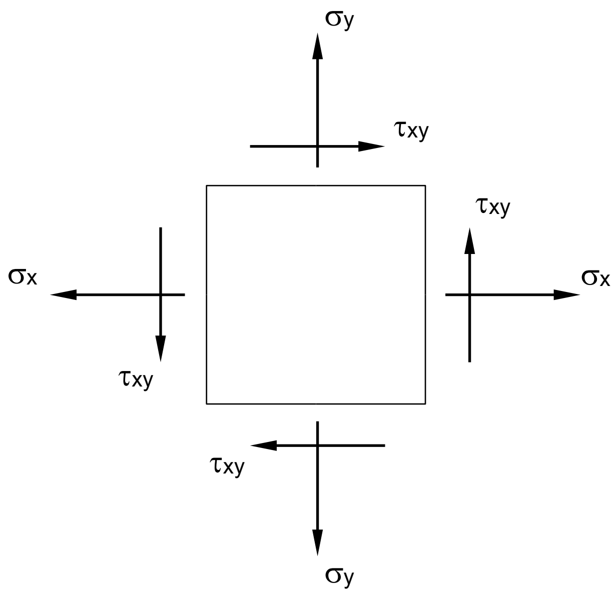


Figure 4. Stresses in CTRIA6 Elements.

CTRIAR Triangular Element Connection

Description: Defines a triangular, isoparametric membrane-bending or plane strain plate element with vertex rotations.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAR	EID	PID	G1	G2	G3	THETA/MCID			
			T_1	T_2	T_3				

Example:

CTRIAR	61	11	101	111	202				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PSHELL or PCOMP property entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required
THETA	Material property orientation angle in degrees.	Real	See Remark 5
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 5
Ti	Membrane thickness of element at G1, G2, and G3.	Real ≥ 0.0	See Remark 6

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Stresses are output in the surface coordinate system. (See the SURFACE command in Section 3, *Case Control.*)
3. The rotational degrees of freedom at the connection points and 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.
4. This element is less sensitive to initial distortion and Poisson's ratio than the CTRIA3 element and is more compatible with the CBAR and CQUADR elements which also have 6 degrees of freedom per node.
5. If THETA/MCID is blank, field 5 of the PSHELL continuation entry will be used. If this field is also blank, then THETA = 0.0 is assumed when a non-isotropic material is referenced.
6. If Ti in fields 4 through 7 of the continuation entry are blank, field 4 of the PSHELL entry will be used. This is the preferred way of specifying element thickness if the thickness does not vary over the element.

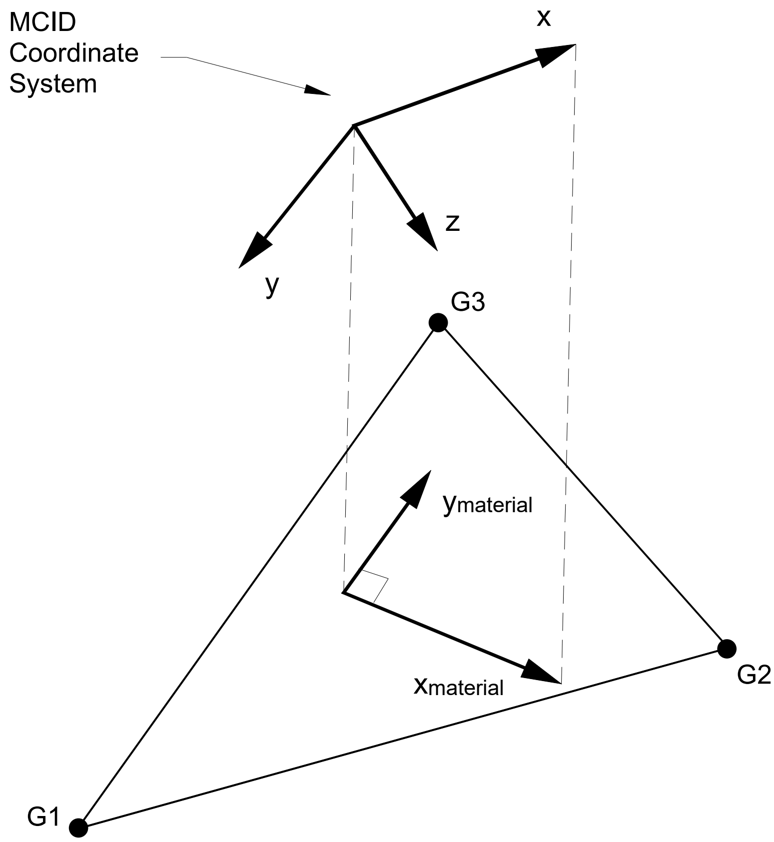


Figure 1. MCID Coordinate System Definition.

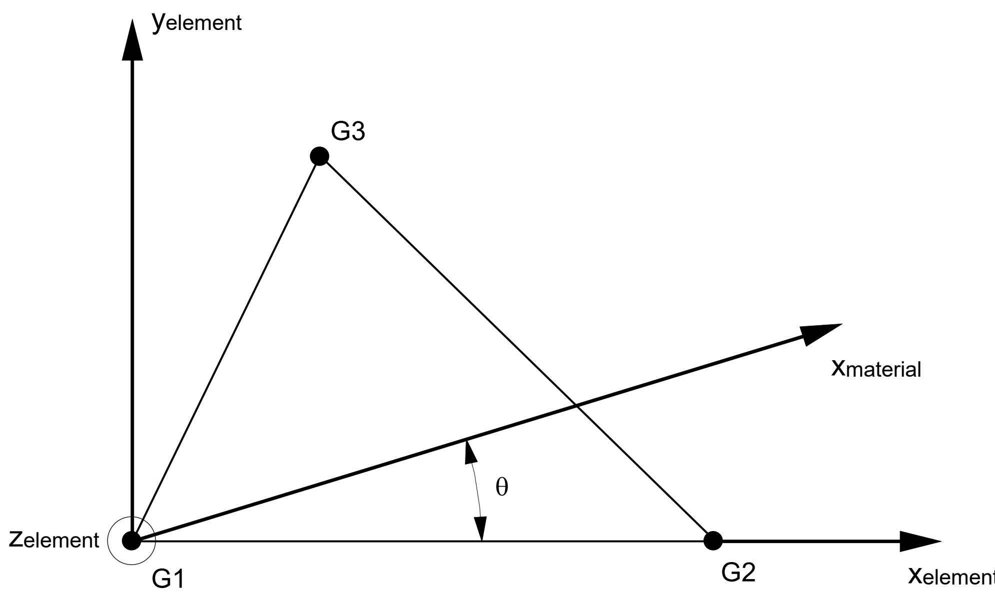


Figure 2. CTRIAR Element Geometry and Coordinate Systems.

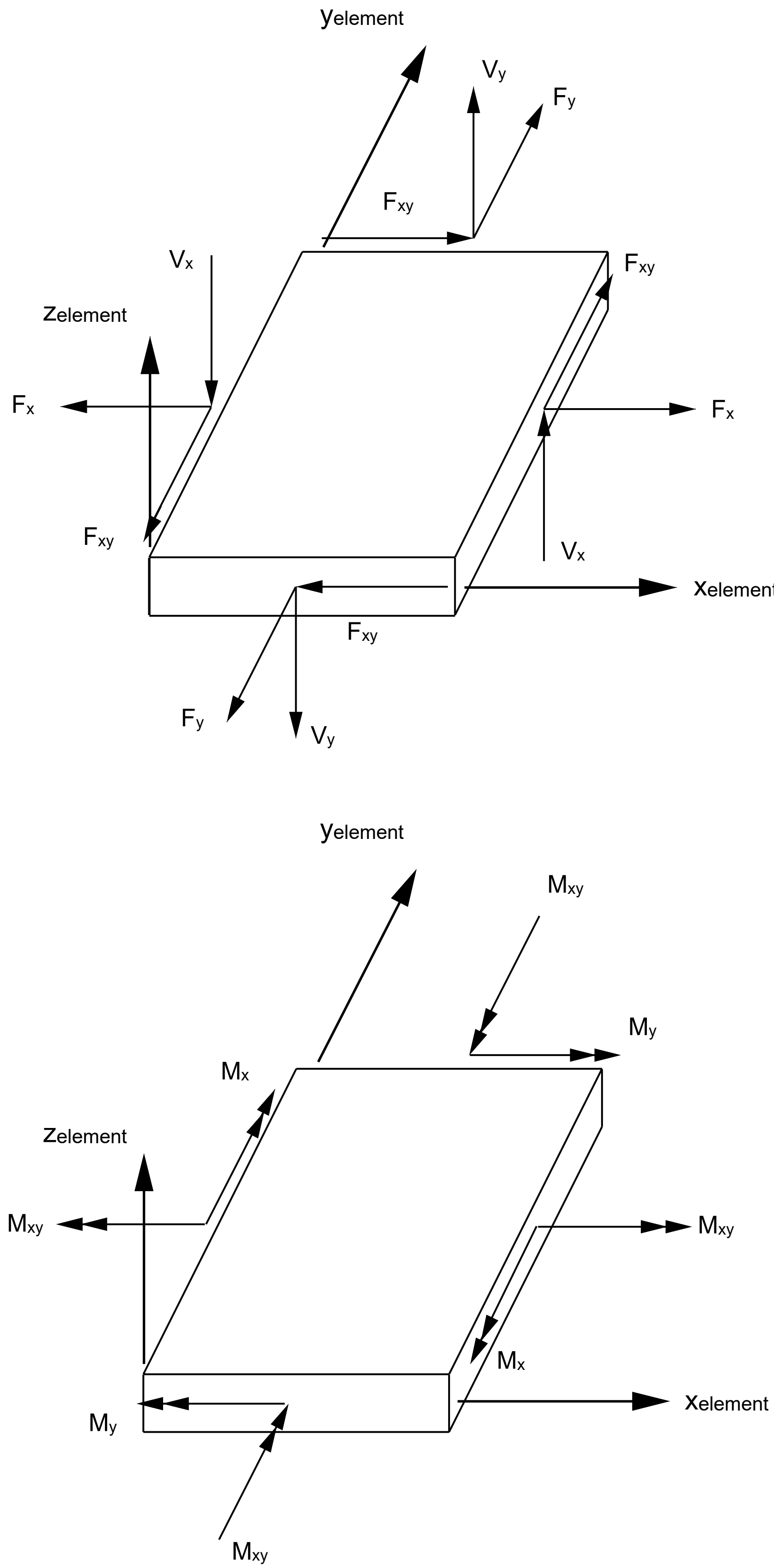


Figure 3. Forces and Moments in CTRIAR Elements.

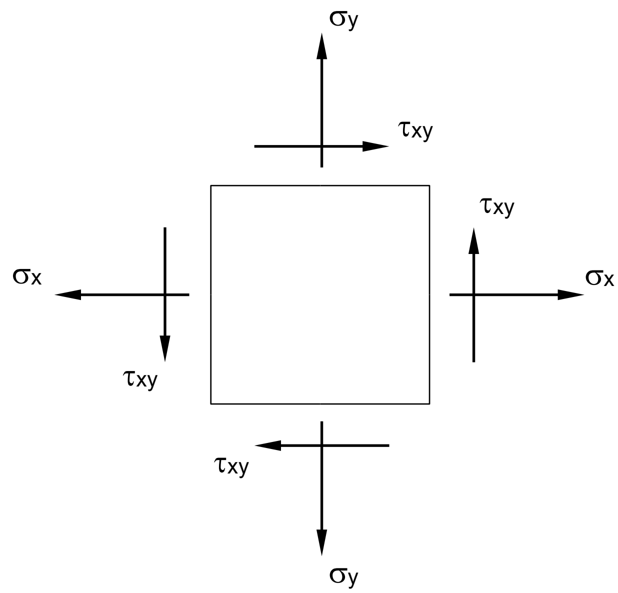


Figure 4. Stresses in CTRIAR Elements.

CTRIAX6 — Axisymmetric Triangular Element Connection

Description: Defines an isoparametric axisymmetric triangular cross-section solid 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	
	THETA								

Example:

CTRIAX6	21	100	20	21	22	31	32	33	
	15.0								

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
MID	Material identification number.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer > 0, all unique	Required
THETA	Material property orientation angle in degrees.	Real	0.0

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The grid points must lie in the x-z plane of the basic coordinate system, with x = r ≥ 0. The grid points must be listed consecutively beginning at a vertex and proceeding around the perimeter in either direction. If the ID of any edge connection points is left blank or set to zero, the element equations are modified to give the correct results for the reduced number of connections. Corner grid points cannot be deleted.
3. For structural problems, the MID must reference a MAT1 or MAT3 material entry
4. The continuation is optional.
5. Material properties (if defined on a MAT3 entry) and stresses are given in the $(r_{material} - z_{material})$ coordinate system shown in Figure 2.
6. 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:

$$(100\, N/m)\, 2\pi\, (0.5\, m) = 314.159\, N$$

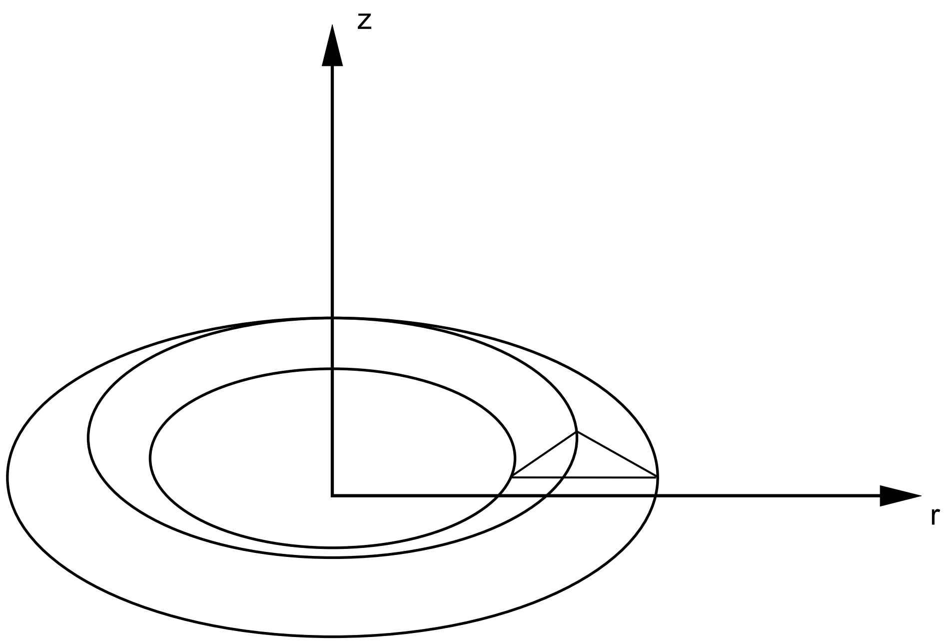


Figure 1. CTRIAX6 Element Idealization.

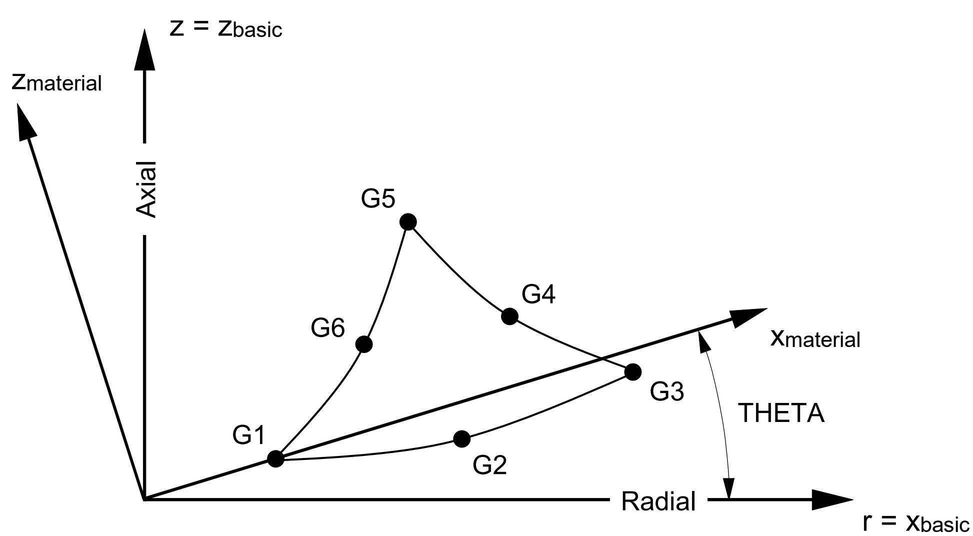


Figure 2. CTRIAX6 Element Geometry and Coordinate Systems.

CTUBE — Tube Element Connection

Description: 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	51	21	201	202					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PTUBE property entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0, G1 ≠ G2	Required

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.

CVISC Viscous Damper Connection

Description: Defines a viscous damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CVISC	EID	PID	G1	G2					

Example:

CVISC	13	327	15	23					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Identification number of a PVISC property entry.	Integer > 0	Required
G1, G2	Grid point identification numbers of connection points.	Integer > 0, $G1 \neq G2$	Required

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

Description: Defines a weld or fastener connecting two surface patches or points.

Format:

1	2	3	4	5	6	7	8	9	10
CWELD	EID	PID	GS	FTYPE	GA	GB			
	GA1	GA2	GA3	GA4	GA5	GA6	GA7	GA8	
	GB1	GB2	GB3	GB4	GB5	GB6	GB7	GB8	

Example:

CWELD	8	24	156	GRIDID					
	12	18	21	25					
	6	4	9	16					

Alternate Formats and Examples:

CWELD	EID	PID	GS	ELEMID					
	SHIDA	SHIDB							

CWELD	EID	PID		ALIGN	GA	GB			

CWELD	5	15	56	ELEMID					
	25	26							

CWELD	12	28		ALIGN	108	199			

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
PID	Property identification number of a PWELD entry.	Integer > 0	Required
GS	Identification number of a grid point which defines the location of the connector. Required for GRIDID and ELEMID. See Remark 2	Integer > 0	

Field	Definition		Type	Default
FTYPE	Connection format type, one of the following character variables: GRIDID, ELEMID, or ALIGN.		Character	Required
	GRIDID	Connection defined using grid identification numbers GAi and GBi. See Remark 4.		
	ELEMID	Connection defined using shell element identification numbers SHIDA and SHIDB. See Remark 5.		
	ALIGN	Connection defined between two shell vertex grid points GA and GB. See Remark 6.		
GA, GB	For FTYPE = GRIDID or ELEMID the grid identification numbers of piercing points on surface A and surface B, respectively. For FTYPE = ALIGN the vertex grid identification numbers of the first and second shell elements respectively.		Integer > 0	See Remark 7
GAi	For FTYPE = GRIDID the grid identification numbers of the first surface patch. GA1 to GA3 are required. See Remark 6.		Integer > 0	See Remark 8
GBi	For FTYPE = GRIDID the grid identification numbers of the second surface patch. See Remark 6.		Integer > 0	See Remark 8
SHIDA	For FTYPE = ELEMID the element identification number of the first shell element.		Integer > 0	See Remark 5
SHIDB	For FTYPE = ELEMID the element identification number of the second shell element.		Integer > 0	See Remark 5

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.

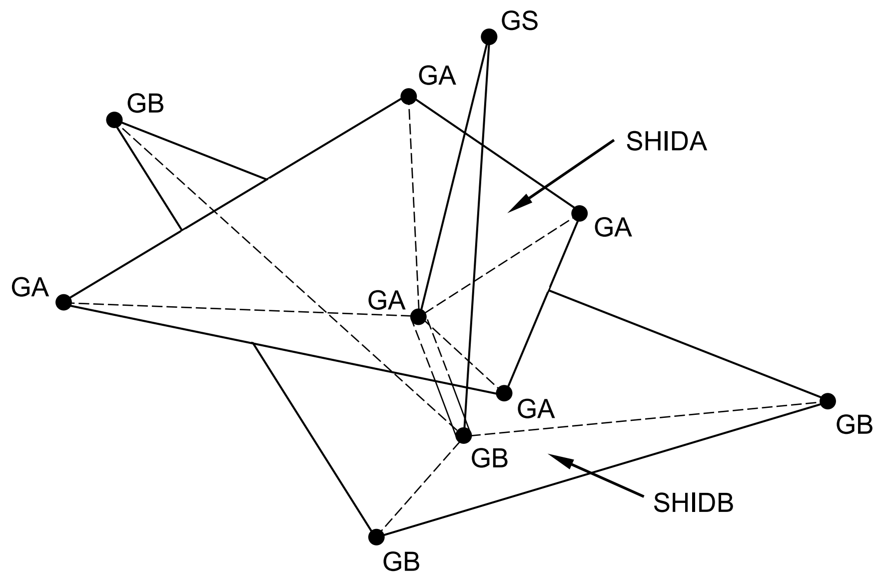


Figure 1. Patch-to-Patch Connection Defined with FTYPE Equal to GRIDID or ELEMID.

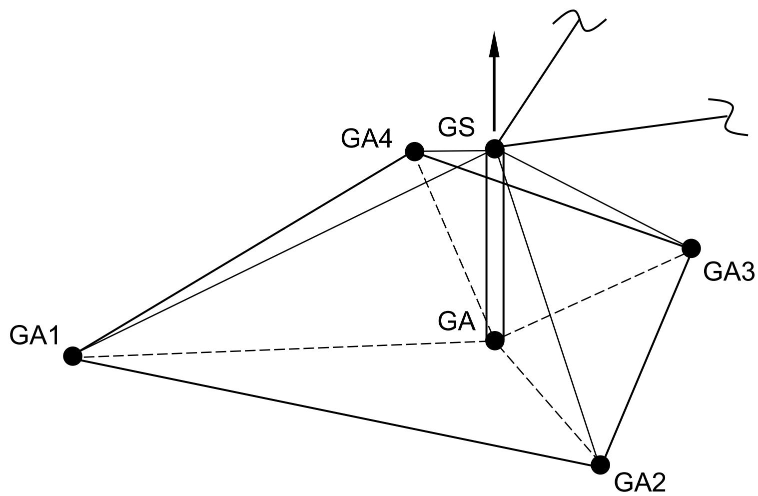


Figure 2. Patch-to-Point Connection Defined with FTYPE Equal to GRIDID or ELEMID.

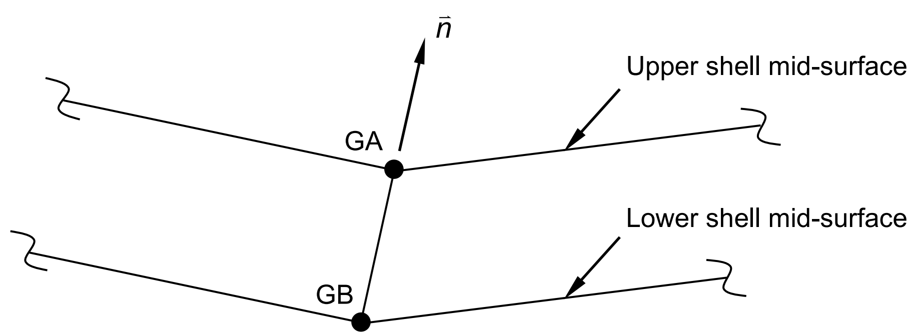


Figure 3. Point-to-Point Connection Defined with FTYPE Equal to ALIGN.

2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The location of the connector element is defined by a projection of grid point GS normal to surface patches A and B. A normal projection must exist in order to define a valid element. GS need not lie on either surface patch, and is ignored if FTYPE = ALIGN.
4. FTYPE = GRIDID defines either a point to patch or a patch to patch connection. For the point to patch connection, the user must define GS and GAi. Then it is assumed that GS is a shell vertex grid and GAi are grids describing a surface patch. For the patch to patch connection, the user must define GS, GAi and GBi. Then GAi describes the first surface patch and GBi the second surface patch.
5. FTYPE = ELEMID defines a point to patch connection, GS to SHIDA or a patch to patch connection, SHIDA to SHIDB. SHIDA and SHIDB must be valid shell element identification numbers.
6. FTYPE = ALIGN defines a point to point connection. GA and GB are required, and they must be vertex nodes of shell elements. GA and GB are not required for the other formats.
7. The input of the piercing points GA and GB is optional for FTYPE = GRIDID and ELEMID. If GA and GB are not specified, they are generated from the normal projection of GS on surface patch A and B. If GA and GB are specified, their locations may be corrected so that they lie on surface patch A and B, respectively. The length of the connector is the distance from GA to GB.

8. GAi are required for FTYPE = GRIDID. At least 3 and at most 8 grid point identification numbers may be specified for GAi and GBi, respectively. Triangular and quadrilateral element definition sequences apply for the order of GAi and GBi.

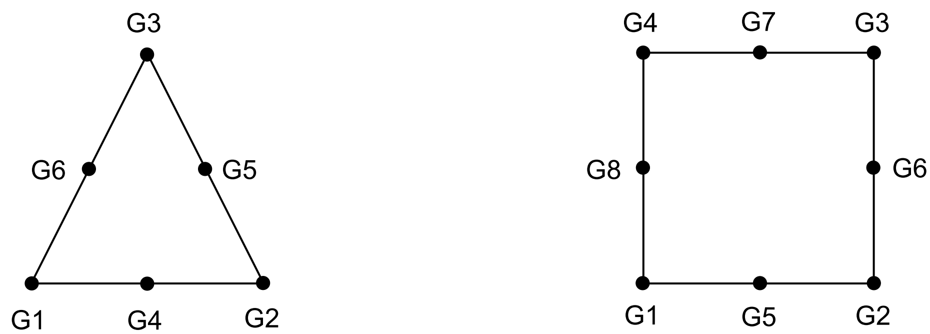


Figure 4. Triangular and Quadrilateral Surface Patches Defined with Format GRIDID.

9. Forces and moments are output in the element coordinate system. The element x-axis is in the direction of GA to GB. The element y-axis is perpendicular to the element x-axis and is lined up with the closest axis of the basic coordinate system. The element z-axis is the cross product of the element x-axis and y-axis. The output of the forces and moments including the sign convention is the same as in the CBAR element.

DAREA — Dynamic Load Scale Factor

Description: The entry is used in conjunction with the TLOAD1 and TLOAD2 entries and defines the point where the dynamic load is to be applied with the scale (area) factor A.

Format:

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

Example:

DAREA	10	3	2	4.4	12	3	16.9		

Field	Definition	Type	Default
SID	Identification number of DAREA set.	Integer > 0	Required
Pi	Grid point identification number.	Integer > 0	Required
Ai	Scale (area) factor.	Real	Required
Ci	Component number of global coordinate (up to six unique digits may be placed in the field with no embedded blanks).	0 ≤ Integer ≤ 6	Required

Remarks:

1. One or two scale factors may be defined on a single entry.
2. Refer to TLOAD1 or TLOAD2 entries for the formulas that define the scale factor Ai.
3. Component numbers refer to the displacement coordinate system.
4. DAREA entries may be used with LSEQ Bulk Data entries. The LSEQ and static load entries will be used to internally generate DAREA entries.

DCONADD Design Constraint Set Combination

Description: Defines a design optimization constraint set as a union of constraints defined via 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:

1	2	3	4	5	6	7	8	9	10
DCONADD	10	4	6	8					

Field	Definition	Type	Default
DCID	Design constraint set identification number.	Integer > 0	Required
DCi	DCONSTR entry identification number.	Integer > 0; SID ≠ Si	Required

Remarks:

1. The DCi values must be unique.
2. Design constraint sets must be selected in the Case Control Section (DESSUB = DCID or DESGLB = DCID) to be used.
3. No DCi may be the identification number of a design constraint set defined by another DCONADD entry.

4.

DCONSTR Design Constraint

Description: Defines a design constraint used in design optimization analysis.

Format:

1	2	3	4	5	6	7	8	9	10
DCONSTR	DCID	RID	LALLOW	UALLOW		UADJUST			

Example:

1	2	3	4	5	6	7	8	9	10
DCONSTR	20	8		1.5					

Field	Definition	Type	Default
DCID	Design constraint identification number.	Integer > 0	Required
RID	DRESP1 entry identification number.	Integer > 0	Required
LALLOW	Lower bound of the response quantity.	Real > 0.0	See Remark 3.
UALLOW	Upper bound of the response quantity.	Real > 0.0	See Remark 3.
UADJUST	Adjustable bound option, one of the following character variables: YES or NO. See Remark 5.	Character	NO

Remarks:

5. The DCONSTR entry may be selected in the Case Control Section by the DESSUB or DESGLB command.
6. DCID may reference a DCONADD Bulk Data entry.
7. Either LALLOW or UALLOW or both must be specified.
8. The units of LALLOW and UALLOW must be consistent with the referenced response defined on the DRESP1 entry.
9. UADJUST is currently only available when the referenced DRESP1 specifies a FRMASS response. When set to ON, the upper bound will increase as required to bring other design or manufacturing constraints within their respective bounds when it is determined that without an adjustment the solution will not converge. Additionally, the upper bound will decrease when a sufficient margin exists between the upper bound and PARAM, TOPTMINADJVF, provided all other constraints are passing. See Section 5, *Parameters*, for more information on TOPTMINADJVF.

DDAMDAT Dynamic Design Analysis Method Data

Description: Defines data needed to perform DDAM analysis.

Format:

1	2	3	4	5	6	7	8	9	10
DDAMDAT	SID	VF1	VF2	VF3	AF1	AF2	AF3	VA	
	VB	VC	AA	AB	AC	AD	STYPE	LTYPE	
	DIRSEQ	FADIR	VDIR	GCF	MINACC	CUTOFF	MTYPE	METHOD	
	MINMEM								

Example:

DDAMDAT	10	0.25	0.5	1.0	0.25	0.50	1.0	10.0	
	20.0	50.0	10.0	45.5	6.5	15.0	SURFACE	HULL	
		3	1			100.0			
	0.5								

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
VFi	Velocity scale factors. See Remark 1.	Real	Required
AFi	Acceleration scale factors. See Remark 1.	Real	Required
VA, VB, VC	Velocity weighting factors. See Remark 1.	Real	Required
AA, AB, AC, AD	Acceleration weighting factors. See Remark 1.	Real	Required, See Remark 2
STYPE	Ship type, one of the following character variables: SURFACE for surface ship or SUBMERG for submerged.	Character	Required
LTYPE	Mounting location, one of the following character variables: DECK, HULL, or SHELL.	Character	Required
DIRSEQ	Shock direction sequence. (Up to three unique digits may be placed in the field with no embedded blanks.) See Remark 3.	$1 \leq$ Integers ≤ 3	123
FADIR	Forward-aft component number. See Remark 3.	$1 \leq$ Integer ≤ 3	1
VDIR	Vertical component number. See Remark 3.	$1 \leq$ Integer ≤ 3	3
GCF	Mass to weight conversion factor.	Real	386.4
MINACC	Minimum acceleration. See Remark 4.	Real	See Remark 4
CUTOFF	Modal mass cutoff percentage. See Remark 5.	Real	80.0

Field	Definition	Type	Default
MTYPE	Material type, one of the following character variables: ELASTIC or PLASTIC. See Remark 6.	Character	ELASTIC
METHOD	Response spectra generation method, one of the following character variables: DDS-072 or NRL-1396. See Remark 7.	Character	DDS-072
MINMEM	Minimum percent modal effective mass which defines modes that will be included after CUTOFF is achieved.	Real	1.0

Remarks:

1. The user supplied velocity, acceleration, and weighting factors are used to compute the velocity and acceleration spectra which serves as the input for response/shock spectrum analysis. The formulas for a SURFACE ship with HULL or SHELL mounted equipment (METHOD = DDS-072) or with DECK, HULL, or SHELL mounted equipment (METHOD = DDS-1396) are given by:

$$V_0 = VF_i \frac{VA(VB+M)}{(VC+M)} \qquad A_0 = AF_i \frac{AA(AB+M)(AC+M)}{(AD+M)^2}$$

For all other ship types and mounting locations the formulas are:

$$V_0 = VF_i \frac{VA(VB+M)}{(VC+M)} \qquad A_0 = AF_i \frac{AA(AB+M)}{(AC+M)}$$

Where M is the modal weight in kips calculated internally for that mode. The VFi and AFi coefficients defined in fields 3 through 8 correspond to the shock coefficients in each model direction. For example, VF1 and AF1 correspond to the shock coefficients in the model x-direction, VF2 and AF2 the y-direction, and VF3 and AF3 the z-direction.

2. The AD weighting factor is required when STYPE is SURFACE and
   - METHOD is DDS-072 and LTYPE is either HULL or SHELL.
   - METHOD is NRL-1396 and LTYPE is DECK, HULL or SHELL.
3. The DIRSEQ field defines which directions will be analyzed and the order they will be analyzed in. The FADIR and VDIR fields define which direction components in DIRSEQ correspond to the forward-aft and vertical directions, respectively. The athwartship direction is determined using the remaining direction. Each direction (1-3) corresponds to a velocity and acceleration factor defined in fields 3 through 8 on this entry. Direction 1 corresponds to the model x-direction, direction 2 the y-direction, and direction 3 the z-direction.
4. If accelerations generated in Remark 1 are less than MINACC, the MINACC value will be used. The default value for MINACC is 1.0 when METHOD is DDS-072 and 6.0 when METHOD is NRL-1396.
5. The modal mass cutoff percentage is percentage of total mass at which modal processing ceases. DDAM analysis requires that only a percentage (typically 80%) of the total modal mass needs to be included in the NRL summation.
6. The material type specified in the MTYPE field only affects the output labels and is not used in the analysis. The character variable PLASTIC does not indicate or initiate nonlinear analysis.

7. METHOD=DDS-072 specifies the equation format described in Design Data Sheet DDS-072 (Classified), 1972. This is the formal specification for the Dynamic Design Analysis Method (DDAM).
8. The default units for DDAMDAT data are IN-LBF-SEC. Other units may be used by setting the UNITS model parameter. Note that the GCF field is always in units of in/sec^2. (See Section 5, *Parameters*, for more information on UNITS.)

DDVAL Discrete Design Variable Values

Description: Defines real, discrete design variable values for design optimization.

Format:

1	2	3	4	5	6	7	8	9	10
DDVAL	ID	DVAL1	DVAL2	DVAL3	DVAL4	DVAL5	DVAL6	DVAL7	
	DVAL8	DVAL9	DVAL10	- etc.-					

Example:

DDVAL	5	0.03	0.05	0.07	0.09	0.1			

Alternate Format and Example:

DDVAL	ID	DVAL1	THRU	DVAL2	BY	INC			

DDVAL	6	0.1	THRU	0.9	BY	0.05			

Field	Definition	Type	Default
ID	Discrete value set identification number.	Integer > 0	Required
DVALi	Discrete values.	Real	Required
INC	Discrete value increment.	Real	Required when using alternate format

Remarks:

1. DDVAL entries must be referenced by a DESVAR entry in the DDVAL field.

DEFORM Element Deformation

Description: Defines enforced axial deformation for CROD, CBAR, and CBEAM elements.

Format:

1	2	3	4	5	6	7	8	9	10
DEFORM	SID	EID	D	EID	D	EID	D		

Example:

DEFORM	2	311	1.1	111	2.1				

Field	Definition	Type	Default
SID	Deformation set identification number.	Integer > 0	Required
EID	Element identification number.	Integer > 0	Required
D	Deformation ("+" = elongation).	Real	Required

Remarks:

1. The referenced element must be a CROD, CBAR, or CBEAM.
2. Deformation sets must be selected in the Case Control Section (DEFORM = SID).
3. From one to three enforced element deformations may be defined on a single entry.

DELAY — Dynamic Load Time Delay

Description: This entry is used in conjunction with the TLOAD1 and TLOAD2 entries and 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	2	31	6	3.45					

Field	Definition	Type	Default
SID	Identification number of the DELAY set.	Integer > 0	Required
Pi	Grid point identification number.	Integer > 0	Required
Ci	Component number of global coordinate (up to six unique digits may be placed in the field with no embedded blanks).	$0 \leq$ Integer ≤ 6	Required
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 referenced on a TLOAD1 or TLOAD2 entry. Refer to these entry descriptions for the formulas that define how the time delay τ is used.
3. A DAREA and/or LSEQ entry should be used to define a load at Pi and Ci.

DESVAR Design Variable

Description: Defines a design variable for design optimization.

Format:

1	2	3	4	5	6	7	8	9	10
DESVAR	ID	LABEL	XINIT	XLB	XUB	REGION	DDVAL	DVTYPE	

Example:

DESVAR	1	DR03	0.005	0.001	0.01	25	12	SHLTHK	

Field	Definition	Type	Default
ID	Design variable dentification number.	Integer > 0	Required
LABEL	Label associated with design region used for output headings.	Character	
PTYPE	Property type. Used with PID to identify the elements to be designed, one of the following character variables: PSOLID, PSHELL, or PCOMP.	Character	Required
XINIT	Initial value for design variable.	XLB ≤ XINIT ≤ XUB	See Remark 1
XLB	Lower bound for design variable.	Real > 0.0	Required
XUB	Upper bound for design variable.	Real > 0.0	Required
REGION	Design region identification number.	Integer > 0	Required
DVTYPE	Design variable type, one of the following character variables: SHLTHK	Character	See Remark 2
DDVAL	DDVAL entry identification number that provides a set of allowable discrete values.	Integer > 0 or blank	See Remark 3

Remarks:

1. If XINIT is blank and OPTION is set to SHLTHK the thickness of the PSHELL referenced in REGION will be used.
2. Currently only variable shell thickness is supported.
3. If the design variable is to be discrete (Integer > 0 in DDVAL field), and if either of the XLB and/orXUB bounds are not specified or 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 . The default of blank results in continuous design variables.

DLOAD — Dynamic Load Combination (Superposition)

Description: Defines a dynamic loading condition for transient response problems as a linear combination of load sets defined via TLOAD1 or TLOAD2 entries.

Format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	- etc. -						

Example:

DLOAD	20	1.5	2.2	6	-3.6	7	6.0	10	
	-4.5	12							

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
S	Scale factor.	Real	Required
Si	Scale factors.	Real	Required
Li	Load set identification numbers defined via entry types enumerated above.	Integer > 0; SID ≠ Li	Required

Remarks:

1. The load vector defined is given by:

$$\{P\} = S\sum_i Si\,\{P_{Li}\}$$

2. Each Li must be unique from any other Li on the same entry.
3. Dynamic load sets must be selected in the Case Control Section with DLOAD = SID.
4. A DLOAD entry may not reference a set identification number defied by another DLOAD entry.
5. TLOAD1 and TLOAD2 loads may be combined only through the use of the DLOAD entry.
6. SID must be unique for all TLOAD1 and TLOAD2 entries.

DMIG Direct Matrix Input at Points

Description: Define 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. 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
DMIG	NAME	0	IFO	TIN				NCOL	

Column Entry Format:

DMIG	NAME	GJ	CJ		G1	C1	A1		
	G2	C2	A2		- etc.-				

Example:

DMIG	STIF	0	6	1	3				
DMIG	STIF	25	1		2	3	3.54+5		
	71	5	2.36+6		81	3	5.87+6		

Field	Definition	Type	Default
NAME	Name of the matrix.	Character	Required
IFO	Form of matrix input, selected by one of the following values 1 = Square matrix 2 = Rectangular matrix 6 = Symmetric matrix 9 = Rectangular matrix	Integer	Required
TIN	Type of matrix being input, selected by one of the following values 1 = Single precision data 2 = Double precision data	Integer	Required
NCOL	Number of columns in a rectangular matrix.	Integer > 0	Required for IFO = 9
GJ	Grid, scalar or extra point identification number for column index.	Integer > 0	Required
CJ	Component number for grid point GJ.	0 < Integer ≤ 6 or blank	1

Field	Definition	Type	Default
Gi	Grid, scalar, or extra point identification number for row index.	Integer > 0	Required
Ci	Component number for Gi for a grid point.	0 < Integer ≤ 6 or blank	1
Ai	Matrix element.	Real	Required

Remarks:

1. Matrixes defined on this entry may be used in any analysis by selection in the Case Control Section with K2GG = NAME, B2GG = NAME, and M2GG = NAME for $[K]$, $[B]$, or, $[M]$ respectively. Input matrixes are added to the structural matrixes before constraints are applied. Load matrixes may be selected by P2G = NAME.
2. The header entry containing IFO and TIN is required. Each non-null column is started with a GJ, CJ pair. The entries for each row of that column follow. Only nonzero terms need be entered. The terms may be input in arbitrary order.
3. Field 3 of the header entry must contain an integer 0.
4. For symmetric matrixes (IFO = 6), a given off-diagonal element may be input either below or above the diagonal. Upper and lower triangle terms may be mixed.
5. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL.
6. The matrix names must be unique among all DMIG entries.
7. TIN should be set consistent with the number of decimal digits required to read the input data adequately. For the single precision specification (TIN=1) one eight character field is used and the input past eight characters is truncated. For the double precision specification (TIN=2) two eight character fields are combined allowing a total of 16 characters for input.
8. DMIG Bulk Data entries can be exported using the TRSLDMIDATA Model Initialization directive. (See Section 2, *Initialization*, for more information on TRSLDMIDATA.)

DPHASE — Dynamic Load Phase Lead

Description: 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	5	15	4	2.41					

Field	Definition	Type	Default
SID	Identification number of the DPHASE entry.	Integer > 0	Required
Pi	Grid point identification number.	Integer > 0	Required
Ci	Component number of global coordinate (up to six unique digits may be placed in the field with no embedded blanks).	0 ≤ Integer ≤ 6	Required
THi	Phase lead θ in degrees.	Real	

Remarks:

1. One or two dynamic load phase leads may be defined on a single entry.
2. SID must also be referenced on a RLOAD1 or RLOAD2 entry. Refer to these entry descriptions for the formulas that define how the phase lead θ is used.
3. A DAREA and/or LSEQ entry should be used to define a load at Pi and Ci

DRESP1 Design Response

Description: Defines a structural response used in design optimization analysis either as a constraint or an objective.

Format:

1	2	3	4	5	6	7	8	9	10
DRESP1	ID	LABEL	RTYPE		REGION	ATTA	ATTB	ATTC	

Example:

DRESP1	10		DISP			3		102	

Field	Definition	Type	Default
ID	Design response identification number.	Integer > 0	Required
LABEL	User-defined label.	Character	
RTYPE	Response type. See table below and Remark 5.	Character	Required
REGION	Topology design region identification number. See Remark 2.	Integer > 0 or blank	All TOPVAR regions
ATTA, ATTB, ATTC	Response attributes. See table below and Remarks 4, 6, and 7.	Integer > 0 or real or blank	

Response Type	Description		Response Attributes	
RTYPE		ATTA (Integer > 0)	ATTB (Integer > 0)	ATTC (Integer > 0)
COM	Center of mass.	Component number (1 – 6)	Blank	Grid Identification Number
COMP	Compliance.	Blank	Blank	Blank
COMPINDX	Compliance index.	Blank	Blank	Blank
DISP	Displacement	Component number (1 – 6)	Blank	Grid Identification Number
EIGN	Eigenvalue.	Mode number	Blank	Blank
FREQ	Natural frequency.	Mode number	Blank	Blank
FRMASS	Mass or volume fraction.	Blank	Blank	Blank
GLBDISP	Global displacement.	Component number (1 – 6)	Blank	Set Identification Number
GLBSTRS	Global stress.	Blank	Blank	Blank

GLBTAVE	Global average temperature.	Blank	Blank	Set Identification Number
GLBTEMP	Global temperature.	Blank	Blank	Set Identification Number
GLBTVAR	Global temperature variation.	Blank	Blank	Set Identification Number
HFLOW	Heat flow.	Blank	Blank	Grid Identification Number
LAMA	Buckling load factor.	Mode number	Blank	Blank
SPCF	Single point constraint force.	Component number (1 – 6)	Blank	Grid Identification Number
TEMP	Temperature.	Blank	Blank	Grid Identification Number

Remarks:

1. DRESP1 identification numbers must be unique.
2. REGION is used to specify which design region a constraint is applicable to. It is applicable when RTYPE is set to FREQ, FRMASS, EIGN, GLBSTRS, LAMA and SPCF. If REGION is zero or blank, then all TOPVAR regions will be used.
3. For FRMASS, the initial constraint value used is the value specified on the TOPVAR entry.
4. For DISP and SPCF the default for ATTC is the grid point with the largest absolute value in the component direction specified. If ATTA is also blank, the component defaults to the direction with the largest absolute value.
5. Compliance index is defined as the ratio of the initial compliance of the unmodified model over the compliance of the modified one. For example, a compliance index of 2.0 would be that of a structure that is 1/2 the stiffness of the original design. The lower limit of compliance index is 1.0.
6. For COM, DISP, GLBDISP, GLBTAVE, GLBTEMP, GLBTVAR, SPCF, and TEMP, ATTC is the grid point set identification number. Only grid points whose identification numbers appear on this SET command will be used. When the ATTC field is blank, all grid points in the model will be used.

DTI, SPSEL Response Spectra Generation Correlation Table

Description: Correlates output requests with frequency and damping values.

Format:

1	2	3	4	5	6	7	8	9	10
DTI	SPSEL	SID	DAMPL	FREQL	G1	G2	G3	G4	
	G5	G6	G7	- etc.-					

Example:

DTI	SPSEL	1	5	10	16	17			
DTI	SPSEL	2	12	14	1	6	10	13	
	15	19							

Field	Definition	Type	Default
SID	Spectrum identification number.	Integer > 0	Required
DAMPL	Identification number of a FREQ, FREQ1, or FREQ2 Bulk Data entry that specifies the list of damping values.	Integer > 0	Required
FREQL	Identification number of a FREQ, FREQ1, FREQ2, FREQ3, or FREQ4 Bulk Data entry that specifies the list of frequencies.	Integer > 0	Required
Gi	Grid point identification number where the response spectra will be calculated.	Integer > 0	Required

Remarks:

1. This table is used in transient response solutions for the generation of response spectra.
2. Damping values are in units of fraction of critical damping.
3. Output of response spectra requires the use of the XYPLOT…SPECTRA(SID)/Gi…Case Control command, where the Gi is restricted to the grid points listed on the (SID) entry.

EIGC Complex Eigenvalue Extraction Data

Description: 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	CTOL	ND	NIVEC	
	MAXITER	XC	ALPHA	OMEGA					

Examples:

EIGC	10						5		
		LM							

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
METHOD	Method of complex eigenvalue extraction, one of the following character variables: ARNO, HESS, or CLAN. See Remark 2.	Character	ARNO
NORM	Method for normalizing eigenvectors, one of the following character variables: MAX or POINT: MAX Normalize to unit value of the largest magnitude to a unit value for the real part and a zero value for the imaginary part. POINT Normalize the component defined in fields 5 and 6 to a unit value for the real part and a zero for the imaginary part.	Character	MAX
G	Grid point identification number.	Integer > 0	Required for NORM = POINT
C	Component number of global coordinate.	1 ≤ Integer ≤ 6	Required for NORM = POINT
CTOL	Eigenvalue convergence tolerance.	Real or blank	1.0E-6
ND	Number of roots desired.	Integer > 0	Required
NIVEC	Number of additional iteration vectors.	Integer > 0	9
MAXITER	Maximum number of iterations.	Integer > 0	100
XC	Extraction criteria, one of the following character variables: LM, SM, LR, SR, LI, SI or AUTO. See Remark 3.	Character	AUTO
ALPHA	Real component of Hessenberg shift scale.	Real or blank	
OMEGA	Imaginary component of Hessenberg shift scale.	Real or blank	

Remarks:

1. Complex eigenvalue extraction data sets must be selected with the Case Control command CMETHOD = SID.
2. METHOD = ARNO specifies that the complex Arnoldi eigensolver will be used. This is the preferred method and will handle all problems sizes. METHOD = HESS specifies that the complex general eigensolver based on the QZ method will be used. This method is only recommended when METHOD = ARNO fails and may be considerably slower for larger problem sizes. METHOD = CLAN is functionally equivalent to METHOD = ARNO.
3. The extraction criteria determines the internal sorting method and controls how the ND roots requested are extracted. The following table gives the various options.

XC Setting	Extraction Method Used
LM	Largest magnitude
SM	Smallest magnitude
LR	Largest real component
SR	Smallest real component
LI	Largest imaginary component
SI	Smallest imaginary component
AUTO	Automatic based on damping

The AUTO setting selects the best option based on the type of damping specified.

EIGR — Real Eigenvalue Extraction Data

Description: Defines data needed to perform real eigenvalue analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGR	SID		V1	V2		ND	SCHECK	NIVEC	
	NORM	G	C	MAXITER	CTOL	ADDITER	ADDIVCV		

Examples:

EIGR	10		5.0	150.0					
	MAX	45	3						

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
V1, V2	For vibration analysis: frequency range of interest. For buckling analysis: eigenvalue range of interest.	Real or blank, V1 < V2	See Remark 5
ND	Number of roots desired.	Integer > 0	See Remark 5
SCHECK	Sturm sequence check, one of the following character variables: YES or NO. See Remark 7.	Character	YES
NIVEC	Number of iteration vectors. See Remark 8.	Integer > 0	12
NORM	Method for normalizing eigenvectors, one of the following character variables: MASS, MAX, POINT:	Character	
	MASS — Normalize to unit value of the generalized mass. Not available for buckling analysis.		For vibration analysis
	MAX — Normalize to unit value of the largest eigenvector displacement.		For buckling analysis
	POINT — Normalize the component defined in fields 3 and 4 to a unit value.		
G	Grid point identification number.	Integer > 0	Required for NORM = POINT
C	Component number of global coordinate.	$1 \leq$ Integer ≤ 6	Required for NORM = POINT
MAXITER	Maximum number of iterations. See Remark 9.	Integer ≥ 0	0
CTOL	Eigenvalue convergence tolerance.	Real or blank	1.0E-5

Field	Definition	Type	Default
ADDITER	Number of additional iterations after convergence. See Remark 10.	Integer ≥ 0	1
ADDIVCV	Number of additional iteration vectors past the number of roots desired or the included range of interest that must also converge. See Remark 10.	Integer ≥ 0	5

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. In buckling, 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. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.
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. The roots are found simultaneously and sorted in increasing order for each subspace or Lanczos iteration. The number and type of roots to be found can be determined from the following table.

V1	V2	ND	Number and Type of Roots Found
V1	V2	ND	Lowest ND roots or all in range, whichever is smaller
V1	V2	blank	All in range
V1	blank	ND	Lowest ND roots in range [V1, +∞]
V1	blank	blank	Lowest root in range [V1, +∞]
blank	blank	ND	Lowest ND roots in range [-∞, +∞]
blank	blank	blank	Lowest root
blank	V2	ND	Lowest ND roots below V2
blank	V2	blank	All below V2

7. SCHECK controls whether a Sturm sequence check is performed. The Sturm sequence check determines if any roots were missed during eigenvalue extraction. Setting SCHECK equal to 0 or NO skips the Sturm sequence check and avoids an additional stiffness matrix factorization thus reducing analysis time. Setting SCHECK equal to 1 or YES performs the check and will output a warning message if any modes were missed.
8. NIVEC specifies the number of additional iteration vectors and is defaulted to 12. Increasing this value may result in a lower number of subspace iterations required but will require more memory and more solves per subspace iteration.
9. MAXITER is used to limit the number of subspace iterations to be performed. The default zero setting forces the eigensolver to iterate until convergence is reached.
10. ADDITER and ADDIVCV are used to prevent missing roots. ADDITER defines the number of additional iterations that will be forced even after all roots desired have converged. ADDIVCV defines how many roots past the desired number or range of interest must converge. A value greater than 1 is recommended when roots are closely spaced. Larger values may result in additional subspace iterations.

EIGRL — Real Eigenvalue Extraction Data

Description: Defines data needed to perform real eigenvalue (vibration or buckling) analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND	SCHECK	NIVEC	SHFSCL	NORM	
	MAXITER	CTOL	ADDITER	ADDIVCV	CTRLOPT	ORTOPT			

Example:

EIGRL	1			10			0.0		
		1.-4							

Field	Definition		Type	Default
SID	Set identification number.		Integer > 0	Required
V1, V2	For vibration analysis: frequency range of interest. For buckling analysis: eigenvalue range of interest.		Real or blank, V1 < V2	See Remark 5
ND	Number of roots desired.		Integer > 0	See Remark 5
SCHECK	Sturm sequence check, one of the following character variables: YES, NO, or AUTO. See Remark 7.		Character	AUTO
NIVEC	Number of iteration vectors. See Remark 8.		Integer > 0	See Remark 8
SHFSCL	Estimate of the first flexible mode natural frequency. See Remark 9.		Real or blank	See Remark 9
NORM	Method for normalizing eigenvectors, one of the following character variables: MASS or MAX:		Character	
	MASS	Normalize to unit value of the generalized mass. Not available for buckling analysis.		For vibration analysis
	MAX	Normalize to unit value of the largest eigenvector displacement.		For buckling analysis
MAXITER	Maximum number of iterations. See Remark 10.		Integer ≥ 0	0
CTOL	Eigenvalue convergence tolerance.		Real or blank	See Remark 11
ADDITER	Number of additional iterations after convergence. See Remark 12.		Integer ≥ 0	1
ADDIVCV	Number of additional iteration vectors past the number of roots desired or the included range of interest that must also converge. See Remark 12.		Integer ≥ 0	5
CTRLOPT	Controls solver specific operations during eigenvalue extraction. See Remark 13.		1 ≤ Integer ≤ 4	See Remark 13

Field	Definition	Type	Default
ORTOPT	Option for full or partial mass re-orthogonalization after each Lanczos iteration, one of the following character variables: FULL, PARTIAL, or AUTO. See Remark 14.	Character	AUTO

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. In buckling, 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. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.
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. The roots are found simultaneously and sorted in increasing order for each subspace or Lanczos iteration. The number and type of roots to be found can be determined from the following table.

V1	V2	ND	Number and Type of Roots Found
V1	V2	ND	Lowest ND roots or all in range, whichever is smaller
V1	V2	blank	All in range
V1	blank	ND	Lowest ND roots in range [V1, +∞]
V1	blank	blank	Lowest root in range [V1, +∞]
blank	blank	ND	Lowest ND roots in range [-∞, +∞]
blank	blank	blank	Lowest root
blank	V2	ND	Lowest ND roots below V2
blank	V2	blank	All below V2

7. SCHECK controls whether a Sturm sequence check is performed. The Sturm sequence check determines if any roots were missed during eigenvalue extraction. Setting SCHECK equal to 0 or NO skips the Sturm sequence check and avoids an additional stiffness matrix factorization thus reducing analysis time. Setting SCHECK equal to 1 or YES performs the check and will output a warning message if any modes were missed. The default setting of AUTO will always perform the check when the subspace eigensolver is selected and only for models smaller than EXTRACTAUTOSIZE when the Lanczos eigensolver is selected. (See Section 2, *Initialization*, for more information on EXTRACTAUTOSIZE.)
8. When the subspace eigensolver is selected, NIVEC specifies the number of additional iteration vectors and is defaulted to 12. Increasing this value may result in a lower number of subspace iterations required but will require more memory and more solves per subspace iteration. When the Lanczos eigensolver is selected, this option controls the Lanczos block size and the default is determined automatically. A value of 9 or 12 may increase performance for models where a large number of modes will be extracted. The maximum value for the Lanczos eigensolver is 120.
9. Specifying SHFSCL = 0.0 may improve accuracy and performance. If this field is blank, a non-zero value for SHFSCL is estimated automatically to handle unconstrained or poorly constrained structures in vibration analysis.

10. MAXITER is used to limit the number of subspace or Lanczos iterations to be performed. The default zero setting forces the eigensolver to iterate until convergence is reached.
11. The CTOL default is dependent on the OPTIMIZESETTINGS directive setting. The following table gives the various values. The default for OPTIMIZESETTINGS is NONE.

OPTIMIZESETTINGS Value	CTOL Value
SPEED	1.0E-5
ACCURACY	1.0E-7
BOTH	1.0E-6
NONE	1.0E-6

12. ADDITER and ADDIVCV are used to prevent missing roots. ADDITER defines the number of additional iterations that will be forced even after all roots desired have converged. ADDIVCV defines how many roots past the desired number or range of interest must converge. A value greater than 1 is recommended when roots are closely spaced. Larger values may result in additional subspace iterations.
13. CTRLOPT controls where the Lanczos eigensolver intermediate results are stored (in memory or on disk) and what solver mode is used (iterative or direct). Higher settings require more memory but may increase performance significantly. The default setting is the eigensolver selects the best method based on available memory. If the SPARSEITERMETHOD model parameter is set to DIRECT, the default will be a CTRLOPT setting of 4. If set to ITERATIVE and the model consists of mostly parabolic tetrahedron elements, the default will be a setting of 1. (See Section 5, *Parameters*, for more information on SPARSEITERMETHOD.) The following table gives the various options.

CTRLOPT Setting	Intermediate File Storage Location	Solver Mode
1	Disk	Iterative
2	Memory	Iterative
3	Disk	Direct
4	Memory	Direct

14. ORTOPT controls whether a full or partial mass re-orthogonalization is performed after each Lanczos iteration. Partial re-orthogonalization increases performance for models where a large number of modes (greater than 100) are requested. Partial re-orthogonalization, however, may result in a small degradation in accuracy. The AUTO setting will use partial re-orthogonalization when residual vectors are requested via the RESVEC model parameter or for models larger than EXTRACTAUTOSIZE when either an eigenvalue range is specified or the number of modes requested is greater than 100. (See Section 2, *Initialization*, for more information on EXTRACTAUTOSIZE and Section 5, *Parameters*, for more information on RESVEC.)

ELIST Element List

Description: Defines a list of structural surface 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	10	-33	9	THRU	22	28	34	41	
	49	53							

Field	Definition	Type	Default
LID	List identification number.	Integer > 0	Required
EIDi	Element identification number(s). See Remarks 1 and 2.	Integer > 0; E1 < E2	Required

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 ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element positive normal as determined by applying the right-hand rule to the sequence of its corner points.
2. If the THRU symbol is used, elements in the sequence E1 through E2 are not required to exist but E1 and E2 must have the same sign. Elements that do not exist or are not compatible will be skipped. The THRU symbol may not appear in fields 3 or 9 on the parent entry and fields 2 or 9 on the continuations.

ENDATA — Strain-Life Method Material Fatigue Data

Description: Specifies material property data needed for fatigue analysis. This entry is used if a MAT1, MAT2, MAT8, MAT9, or MAT12 entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
ENDATA	MID	SF	EF	B	C				

Example:

ENDATA	200	1.7+9	0.83	0.095	0.65				

Field	Definition	Type	Default
MID	Identification number of a MAT1, MAT2, MAT8, MAT9, or MAT12 entry.	Integer > 0	Required
SF	Coefficient of fatigue strength. See Remark 3.	Real > 0.0	See Remark 2.
EF	Coefficient of fatigue ductility. See Remark 3.	Real > 0.0	See Remark 2.
B	Exponent of fatigue strength. See Remark 3.	Real > 0.0	See Remark 2.
C	Exponent of fatigue ductility. See Remark 3.	Real > 0.0	See Remark 2.

Remarks:

1. ENDATA entries must all have unique set identification numbers.
2. VFATIGUE and FATIGUE entries provide defaults to ENDATA. Values not specified on ENDATA entries will be replaced with ones from the VFATIGUE or FATIGUE entry STRAIN continuation.
3. The ε-N curve shown in Figure 1 is characterized by the equation

$$\frac{\varepsilon}{2} = \frac{\mathrm{SF}}{E}(2N_f)^{-\mathrm{B}} + \mathrm{EF}(2N_f)^{-\mathrm{C}}$$

where,

ε is the range of strain ($\varepsilon_{max} - \varepsilon_{min}$)

$2N_f$ is the number of cycles to failure

E is the modulus of elasticity

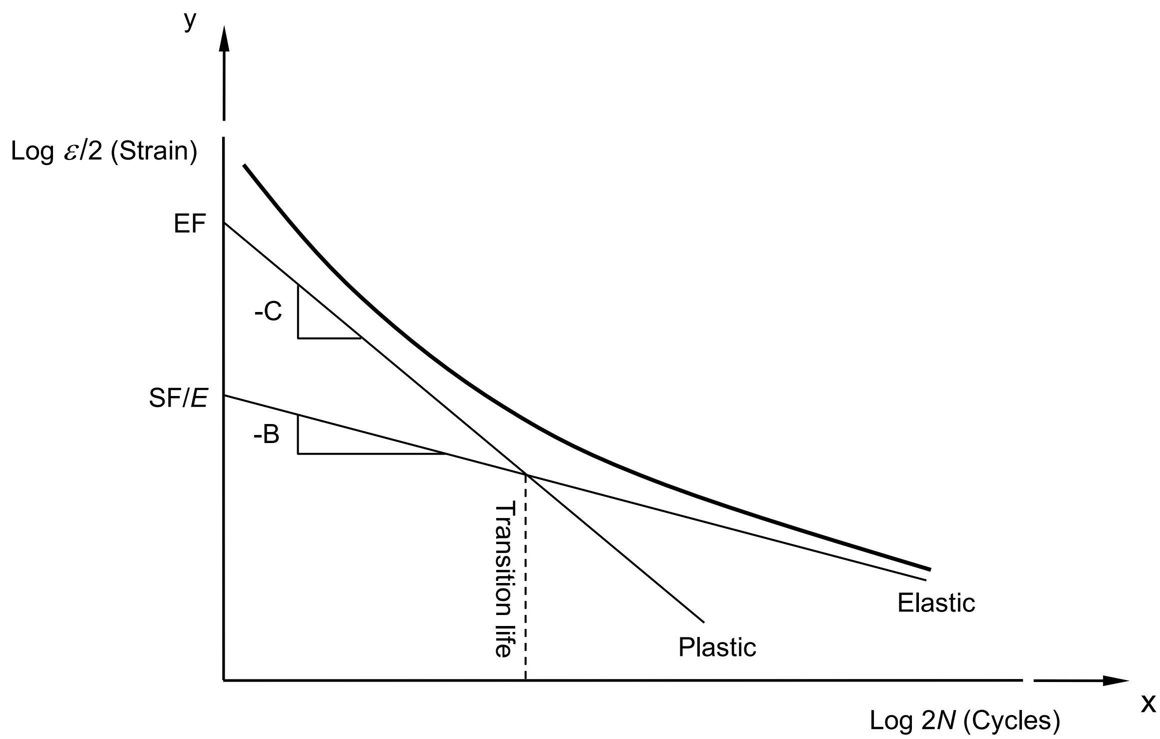


Figure 1. Strain-Life Curve Format.

ENDDATA

Bulk Data Delimiter

Description: Designates the end of the Bulk Data Section.

Format:

ENDDATA

Remarks:

1. ENDDATA is required.

EPOINT — Extra Point Definition

Description: 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	5	22	2	7	45	6			

Alternate Format and Example:

EPOINT	ID1	THRU	ID2						

EPOINT	8	THRU	245						

Field	Definition	Type	Default
IDi	Extra point identification number(s).	Integer > 0; ID2 > ID1	Required

Remarks:

1. All extra point identification numbers must be unique with respect to all other grid, scalar, and extra points.
2. At least one ID must be present on each EPOINT entry.
3. If the alternate form is used, all points ID1 through ID2 that do not exist will be skipped.
4. Extra points must not be specified more than once.
5. Continuations are not allowed.

ESET — Eigendata Set Definition

Description: Defines degrees of freedom in the reduced eigendata set (e-set) used for Modal Assurance Criterion (MAC) analysis.

Format:

1	2	3	4	5	6	7	8	9	10
ESET	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

ESET	15	3	17	456	7	4			

Field	Definition	Type	Default
Gi	Grid point identification number(s).	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks).	$1 \le$ Integers ≤ 6	Required

Remarks:

1. ESET generation can be automated using the XSETGENERATE Case Control command.

ESET1 — Eigendata Set Definition, Alternate Form

Description: Defines degrees of freedom in the reduced eigendata set (e-set) used for Modal Assurance Criterion (MAC) analysis.

Format:

1	2	3	4	5	6	7	8	9	10
ESET1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	- etc.-					

Example:

ESET1	123	6	3	7	10	18	14	11	
	19	23							

Alternate Format and Example:

ESET1	C	G1	THRU	G2					

ESET1	456	15	THRU	512					

Field	Definition	Type	Default
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks).	$1 \leq$ Integers ≤ 6	Required
Gi	Grid point identification number(s).	Integer $>$ 0; G1 < G2	Required

Remarks:

1. If the alternate form is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will be skipped.
2. ESET generation can be automated using the XSETGENERATE Case Control command.

FATIGUE — Multiaxial Fatigue Data

Description: Defines data needed for multiaxial fatigue analysis.

Format:

1	2	3	4	5	6	7	8	9	10
FATIGUE	SID	APRCH	METHOD	THRESH	SIGNINV	DT	TCF		
	STRESS	B	SU	N0	KF	BE	SE		
	STRAIN	SF	EF	B	C				

Example:

FATIGUE	200	STRAIN	1						
	STRESS	0.16	4.5+3		0.9				
	STRAIN	1.7+9	0.83	0.095	0.65				

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
APRCH	Fatigue life approach, one of the following character variables: STRESS, STRAIN, or blank.	Character	See Remark 2.
METHOD	Life calculation method, selected by one of the following values 1 = von Mises stress/strain 2 = Maximum principal stress/strain 3 = Maximum shear stress/strain	Integer	2
THRESH		Real ≥ 0.0	0.0
SIGNINV	Signed stress/strain invariant option: YES, NO, or AUTO. See Remark 7.	Character	NO
DT	Event duration used to determine life. See Remark 6.	Real > 0.0	See Remark 6.
TCF	Factor to convert DT and life output to units other than seconds. See Remark 6.	Real > 0.0	1.0
B	S-N curve slope. See Remark 3.	Real > 0.0	See Remark 2.
SU	Intercept stress level. Typically taken as the material ultimate stress. See Remark 3.	Real > 0.0	See Remark 2.
N0	Intercept cycles. See Remark 3.	Integer > 0	1000
KF	Factor applied to compensate for life reduction effects such as finish, corrosion, and notch effects. See Remark 3.	Real > 0.0	1.0
BE	Slope after endurance limit. See Remark 3.	Real > 0.0	0.1*B
SE	Endurance limit. See Remark 3.	Real ≥ 0.0	0.2*SU

Field	Definition	Type	Default
SF	Coefficient of fatigue strength. See Remark 4.	Real > 0.0	See Remark 2
EF	Coefficient of fatigue ductility. See Remark 4.	Real > 0.0	See Remark 2
B	Exponent of fatigue strength. See Remark 4.	Real > 0.0	See Remark 2
C	Exponent of fatigue ductility. See Remark 4.	Real > 0.0	See Remark 2

Remarks:

1. FATIGUE entries must all have unique set identification numbers.
2. The APRCH field is required when neither the SNDATA nor ENDATA Bulk Data entries are included. The data provided on the continuation entries serve as default values for properties normally defined on these entries. Values not specified on SNDATA entries will be replaced with ones from the STRESS continuation and values not specified on the ENDATA will be replaced with ones from the STRAIN continuation.
3. The S-N curve shown in Figure 1 is characterized by the following equations

   If $S_i \geq S_e$

   $$N_f = \mathrm{N0}\left(\frac{\mathrm{SU}}{\mathrm{KF} * S_i}\right)^{\frac{1}{\mathrm{B}}}$$

   If $S_i < S_e$

   $$N_f = N_e\left(\frac{\mathrm{SE}}{\mathrm{KF} * S_i}\right)^{\frac{1}{\mathrm{BE}}}$$

   where,

   N_f is the number of cycles to failure

   S_i is the amplitude of input stress $(S_{max} - S_{min})/2$

   N_e is the number of failure cycles at the endurance limit

   and the slope B is shown in Figure 1 is calculated by

   $$B = \frac{\log(\mathrm{SU}) - \log(\mathrm{SE})}{\log(N_e) - \log(\mathrm{N0})}$$

4. The ε-N curve shown in Figure 2 is characterized by the equation

   $$\frac{\varepsilon}{2} = \frac{\mathrm{SF}}{E}(2N_f)^{-\mathrm{B}} + \mathrm{EF}(2N_f)^{-\mathrm{C}}$$

   where,

   ε is the range of strain ($\varepsilon_{max} - \varepsilon_{min}$)

   $2N_f$ is the number of cycles to failure

   E is the modulus of elasticity

5. Amplitude filter. When the amplitude change between two sequential data is less than the threshold percent of the maximum range, the data is discarded in life calculation

6. The default value for DT is determined using the difference between the largest and smallest TABLEDi times (time range). If the specified DT is smaller that this time range, it is set equal to it. DT is useful when the event duration is different from the time range due to idling time. TCF is a time conversion factor that is typically used to convert a default DT time from seconds to another set of units such as hours. Life output will be in the same units as DT where life is defined using

$$Life = \frac{\mathrm{DT} * \mathrm{TCF}}{Damage}$$

where,

Damage is the ratio of applied cycles over cycles to failure.

7. The SIGNINV option specifies if the stress/strain used in the life calculation method should be signed. The invariant stress/strain options are by definition positive values and the sign of the load is determined using the TABLEDi load cycle data. When SIGNINV is set to YES the sign of the invariant is determined based on the principal stress/strain. When SIGNINV is set to NO or blank the invariant stress/strain is always positive.

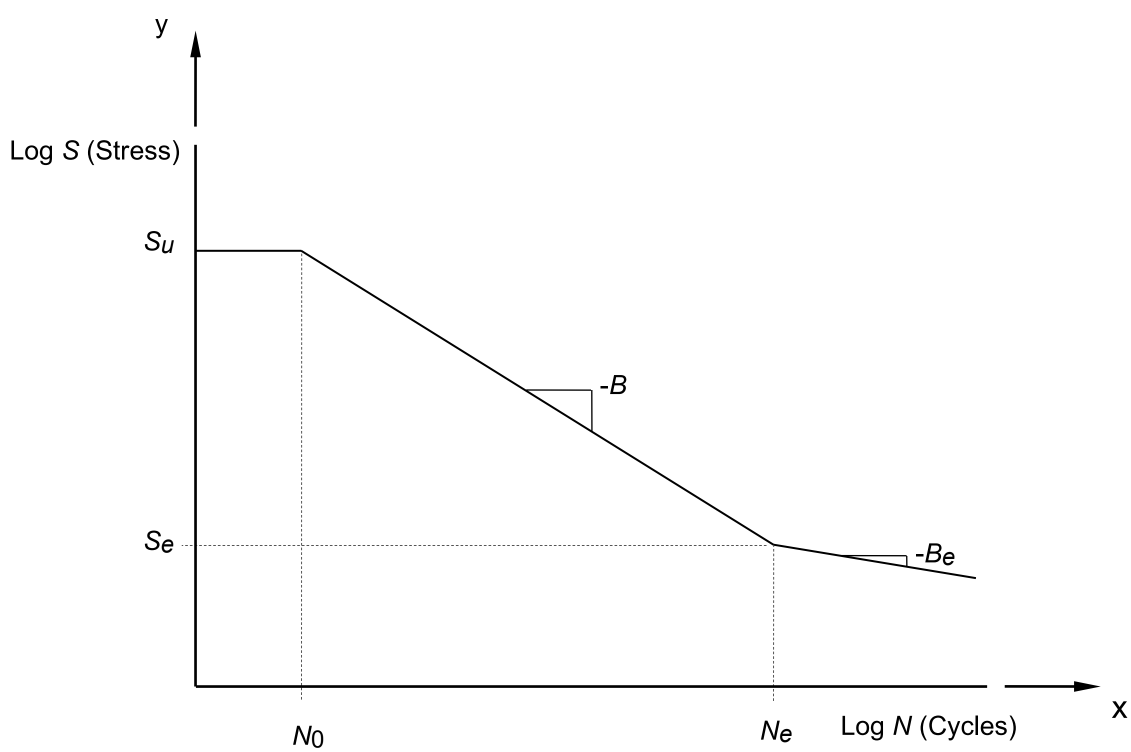


Figure 1. Stress-Life Curve Format.

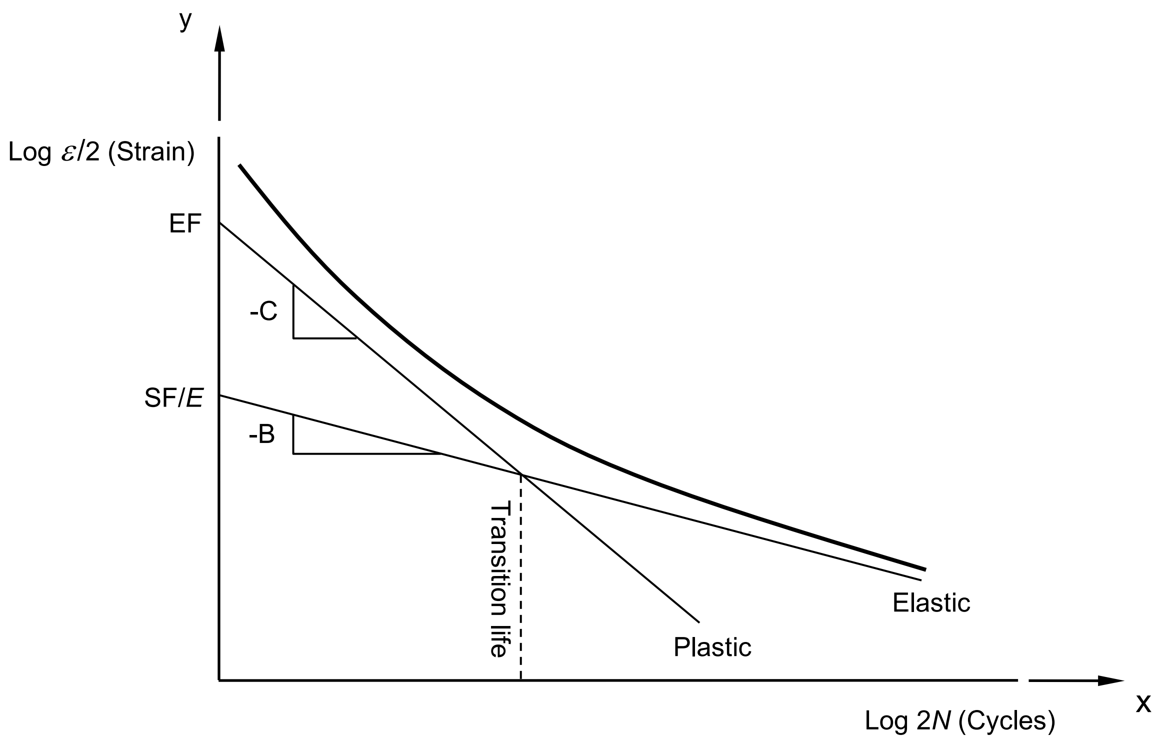


Figure 2. Strain-Life Curve Format.

FORCE Static Load

Description: Defines a static load 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	3	441	4	10.0	1.0	-1.0	0.0		

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
CID	Coordinate system identification number.	Integer ≥ 0 or blank	0
G	Grid point identification number.	Integer > 0	Required
F	Load vector scale factor.	Real	Required
N1, N2, N3	Load vector components of vector measured in the coordinate system defined by CID.	Real	Required; must have at least one nonzero component

Remarks:

1. The static load 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.
2. Load sets must be selected in the Case Control Section (LOAD = SID).
3. A CID of zero references the basic coordinate system.

FORCE1 — Static Load, Alternate Form 1

Description: Defines a static load at a grid point by specification of a value 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	3	141	-4.5	10	11				

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
G	Grid point identification number.	Integer > 0	Required
F	Load magnitude.	Real	Required
G1, G2	Grid point identification numbers.	Integer > 0; G1 ≠ G2	Required

Remarks:

1. The static load applied to grid point G is given by:

$$\vec{f} = F\vec{n}$$

where $\vec{n}$ is a unit vector parallel to a vector for G1 to G2.

2. Load sets must be selected in the Case Control Section (LOAD = SID).

FREQ Frequency List

Description: 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	5	1.5	2.05	15.8	21.6	24.3	27.8	30.1	
	23.1	28.4	15.3						

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
Fi	Frequency value in units of cycles per unit time.	Real ≥ 0.0	Required

Remarks:

1. FREQi entries must be selected with the Case Control command FREQUENCY = SID.
2. All FREQi entries with the same frequency set identification numbers will be used.
3. The DFREQ model parameter specifies the threshold for the elimination of duplicate frequencies. Duplicate frequencies will be ignored if,

$$|f_i - f_{i-1}| < \text{DFREQ} * |f_{MAX} - f_{MIN}|$$

where DFREQ is defaulted to 10^{-5} and f_{MAX} and f_{MIN} are the maximum and minimum solution frequencies of the combined FREQi entries. (See Section 5, *Parameters*, for more information on DFREQ.)

FREQ1 Frequency List, Alternate Form 1

Description: 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	8	2.2	0.4	15					

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
F1	First frequency in set.	Real ≥ 0.0	Required
DF	Frequency increment.	Real ≥ 0.0	Required
NDF	Number of frequency increments.	Integer > 0	Required

Remarks:

1. FREQi 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:

$$f_i = \mathrm{F1} + \mathrm{DF} * (i - 1)$$

4. The DFREQ model parameter specifies the threshold for the elimination of duplicate frequencies. Duplicate frequencies will be ignored if,

$$|f_i - f_{i-1}| < \mathrm{DFREQ} * |f_{MAX} - f_{MIN}|$$

where DFREQ is defaulted to $1.0\text{x}10^{-5}$ and f_{MAX} and f_{MIN} are the maximum and minimum solution frequencies of the combined FREQi entries. (See Section 5, *Parameters*, for more information on DFREQ.)

FREQ2 Frequency List, Alternate Form 2

Description: 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	1.+5	4					

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
F1	First frequency.	Real > 0.0	Required
F2	Last frequency.	Real > 0.0, F2 > F1	Required
NF	Number of logarithmic intervals	Integer > 0	1

Remarks:

1. FREQi 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:

$$f_i = \text{F1} * e^{(i-1)d}$$

where,

$$d = \frac{1}{\text{NF}} \ell n\left(\frac{\text{F2}}{\text{F1}}\right)$$

and,

$$i = 1, 2, \ldots, (\text{NF} + 1)$$

In the example above, the list of frequencies will be 1.0, 10.0, 100.0, 1000.0, and 10000.0 cycles per unit time.

4. The DFREQ model parameter specifies the threshold for the elimination of duplicate frequencies. Duplicate frequencies will be ignored if,

$$|f_i - f_{i-1}| < \text{DFREQ} * |f_{MAX} - f_{MIN}|$$

where DFREQ is defaulted to 10^{-5} and f_{MAX} and f_{MIN} are the maximum and minimum solution frequencies of the combined FREQi entries. (See Section 5, *Parameters*, for more information on DFREQ.)

FREQ3 — Frequency List, Alternate Form 3

Description: Defines a set of excitation frequencies for modal frequency response solutions by specifying the number of solution 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	5	10.0	100.0	LINEAR	10	2.0			

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
F1	Lower bound of modal frequency range in cycles per unit time.	Real > 0.0	Required
F2	Upper bound of modal frequency range in cycles per unit time.	Real > 0.0, F2 ≥ F1	F1
TYPE	Specifies the interpolation type between frequencies, one of the following character variables: LINEAR or LOG: LINEAR Linear interpolation between frequencies. LOG Logarithmic interpolation between frequencies.	Character	LINEAR
NEF	Number of solution frequencies within each subrange including the endpoints. 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 > 0.0	10
CLUSTER	Specifies clustering of the solution frequency near the endpoints of the range. See Remark 6.	Real > 0.0	1.0

Remarks:

1. FREQi entries must be selected with the Case Control command FREQUENCY = SID.
2. In the example above, there will be 10 frequencies in the interval between each set of modes within the bounds 10 and 1000, plus 10 frequencies between 10 and the lowest mode in the range, plus 10 frequencies between the highest mode in the range and 1000.
3. Since the forcing frequencies are near structural resonance, it is important that some amount of damping be specified.

4. The DFREQ model parameter specifies the threshold for the elimination of duplicate frequencies. Duplicate frequencies will be ignored if,

$$|f_i - f_{i-1}| < \text{DFREQ} * |f_{MAX} - f_{MIN}|$$

where DFREQ is defaulted to 10^{-5} and f_{MAX} and f_{MIN} are the maximum and minimum solution frequencies of the combined FREQi entries. (See Section 5, *Parameters*, for more information on DFREQ.)

5. CLUSTER is used to obtain better resolution near the modal frequencies where the response varies the most. CLUSTER $>$ 1.0 provides closer spacing of solution frequency spacing 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 TYPE is LINEAR then,

$$f_i = \frac{1}{2}(f_1 + f_2) + \frac{1}{2}(f_1 + f_2)|\xi|^{1/\text{CLUSTER}} * \text{SIGN}(\xi)$$

and,

$$\xi = -1 + 2(i - 1)/(\text{NEF} - 1)$$

where ξ is a parametric coordinate between –1 and 1 and i varies from 1 to NEF (i=1,2, …, NEF) and,

f_1 = is the lower limit of the frequency subrange

f_2 = is the upper limit of the frequency subrange

f_i = is the i-th solution frequency

FREQ4 Frequency List, Alternate Form 4

Description: Defines a set of frequencies used in the solution of modal frequency-response problems specifying the amount of “spread” around each natural frequency and the number of equally space excitation frequencies within the spread.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ4	SID	F1	F2	FSPD	NFM				

Example:

FREQ4	5	10.0	100.0	0.20	11				

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
F1	Lower bound of frequency range in cycles per unit time.	Real ≥ 0.0	0.0
F2	Upper bound of frequency range in cycles per unit time.	Real > 0.0, F2 ≥ F1	F1
FSPD	Frequency spread, +/- the factional amount specified for each mode which occurs in the frequency range F1 to F2.	0.0 < Real < 1.0	0.10
NFM	Number of evenly spaced frequencies per spread mode.	Integer > 0	3

Remarks:

1. FREQi entries must be selected with the Case Control command FREQUENCY = SID.
2. There will be NFM excitation frequencies between (1 – FSPD)$*f_i$ for each natural frequency in the range F1 to F2.
3. In the example above, the will be 11 equally spaced frequencies across a frequency band of 0.8$*f_i$ to 1.2$*f_i$ for each natural frequency that occurs between 10 and 1000.
4. The frequency spread can be used also to define the half-power bandwidth. The half-power bandwidth is given by $2*\zeta*f_i$ where ζ is the damping ratio. Therefore, if FSPD is specified equal to the damping ratio for the mode, NFM specifies the number of solution frequencies within the half-power bandwidth. See Figure 1 for the definition of half-power bandwidth.

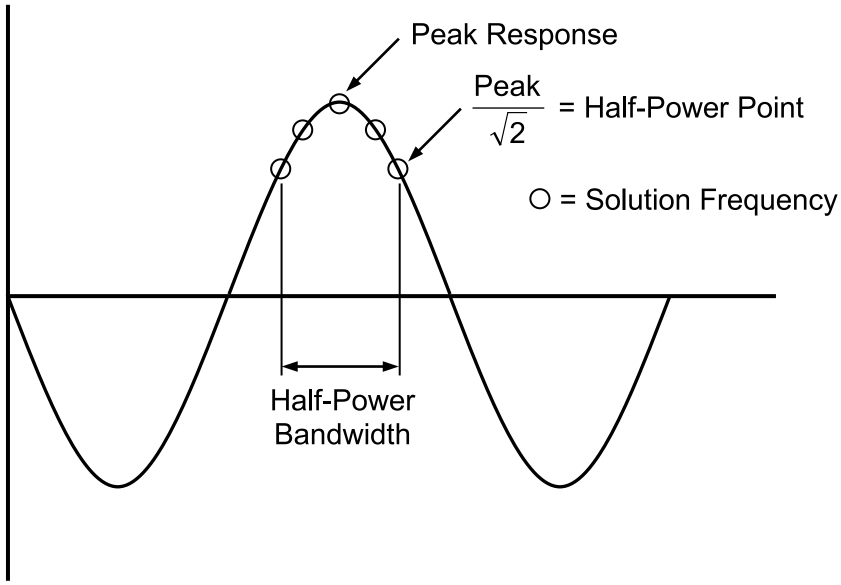


Figure 1. Half-Power Point and Bandwidth.

5. Since the forcing frequencies are near the structural resonance, it is important that some amount of damping be specified.
6. The DFREQ model parameter specifies the threshold for the elimination of duplicate frequencies. Duplicate frequencies will be ignored if,

$$|f_i - f_{i-1}| < \text{DFREQ} * |f_{MAX} - f_{MIN}|$$

where DFREQ is defaulted to 10^{-5} and f_{MAX} and f_{MIN} are the maximum and minimum solution frequencies of the combined FREQi entries. (See Section 5, *Parameters*, for more information on DFREQ.)

GENEL General Element

Description: 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 the 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	459		11	1	11	2	11	3	
	24	4	24	5	24	6			
	UD		6	1	6	2	6	3	
	6	4	6	5	6	6			
	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	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
UIi, CIi UDj, CDj	Identification numbers of coordinates in the UI or UD list, in sequence corresponding to the $[K]$, $[Z]$, and $[S]$ matrixes. UIi and UDi are grid point numbers, CIi and CDj are the component numbers.	Integer ≥ 0	
KZij	Values of the $[K]$ or $[Z]$ matrix ordered by columns from the diagonal, according to the UI list.	Real	Required
Sij	Values of the $[S]$ matrix ordered by rows according to the UD list.	Real	See Remark 1
UD, K, Z, and S	Character variables that indicate the start of data belonging to the UD list or the $[K]$, $[Z]$, or $[S]$ matrixes.	Character	

Remarks:

1. The stiffness approach:

$$\left\{\frac{f_i}{f_d}\right\} = \begin{bmatrix} K & -KS \\ -S^T K & S^T KS \end{bmatrix} \left\{\frac{u_i}{u_d}\right\}$$

The flexibility approach:

$$\begin{bmatrix}\frac{u_i}{f_d}\end{bmatrix} = \begin{bmatrix} Z & S \\ -S^T & O \end{bmatrix} \begin{bmatrix}\frac{f_i}{u_d}\end{bmatrix} \begin{bmatrix}\frac{u_i}{f_d}\end{bmatrix} = \begin{bmatrix} Z & S \\ -S^T & O \end{bmatrix} \begin{bmatrix}\frac{f_i}{u_d}\end{bmatrix} \begin{bmatrix}\frac{u_i}{f_d}\end{bmatrix} = \begin{bmatrix} Z & S \\ -S^T & O \end{bmatrix} \begin{bmatrix}\frac{f_i}{u_d}\end{bmatrix}$$

Where

$$\{u_i\} = [u_{i1}, u_{i2}, \ldots, u_{im}]^T \text{ and } \{u_d\} = [u_{d1}, u_{d2}, \ldots, u_{dn}]^T$$

$$[KZ] = [K] \text{or} [Z] = \begin{bmatrix} KZ11 & \ldots & \ldots & \ldots \\ KZ21 & KZ22 & \ldots & \ldots \\ KZ31 & KZ32 & \ldots & \ldots \\ \vdots & \vdots & \vdots & \vdots \\ KZ_{m1} & \ldots & \ldots & KZ_{mm} \end{bmatrix} \text{and } [KZ]^T = [KZ]$$

$$[S] = \begin{bmatrix} S11 & \ldots & S_{1n} \\ S21 & \ldots & \ldots \\ S31 & \ldots & \ldots \\ \vdots & \vdots & \vdots \\ S_{m1} & \ldots & S_{mn} \end{bmatrix}$$

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 $\{u_i\} = [S]\{u_d\}$.

2. When the stiffness matrix K is input, the number of significant digits should be the same for all terms.
3. The DMIG entry offers an alternative method for inputting large matrixes.
4. The general element entry in the example above defines the following:

$$\{u_i\} = [11\text{-}1,\ 11\text{-}2,\ 11\text{-}3,\ 24\text{-}4,\ 24\text{-}5,\ 24\text{-}6]^T$$

$$\{u_d\} = [6\text{-}1,\ 6\text{-}2,\ 6\text{-}3,\ 6\text{-}4,\ 6\text{-}5,\ 6\text{-}6]^T$$

where i-j means the j-th component of grid point i. Points 42 and 33 are scalar points.

$$[Z] = \begin{bmatrix} 1.0 & 2.0 & 3.0 & 4.0 \\ 2.0 & 5.0 & 6.0 & 7.0 \\ 3.0 & 6.0 & 8.0 & 9.0 \\ 4.0 & 7.0 & 9.0 & 10.0 \end{bmatrix} \qquad [S] = \begin{bmatrix} 1.5 & 2.5 \\ 3.5 & 4.5 \\ 5.5 & 6.5 \\ 7.5 & 8.5 \end{bmatrix}$$

GRAV Gravity Vector

Description: Used to define gravity vectors for use in determining gravity loading for the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	G	N1	N2	N3			
	TID1	TID2	TID3						

Example:

GRAV	3	1	4.5	0.0	0.5	-1.0			
	101	102	103						

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
CID	Coordinate system identification number.	Integer ≥ -1 or blank	0
G	Gravity vector scale factor.	Real	Required
N1, N2, N3	Gravity vector components measured in coordinate system defined by CID.	Real	Required; must have at least one nonzero component
TID1, TID2, TID3	TABLEDi set identification numbers that define position dependent scale factors in the x, y, and z directions of the basic coordinate system. See Remark 1.	Integer > 0 or blank	

Remarks:

1. The static load applied to grid point G is given by:

$$\vec{g} = G\,\vec{n}\,f(x, y, z)$$

   where $\vec{n}$ is the unit vector defined in fields 5, 6, and 7 and $f(x, y, z)$ is defined as the product of scale factors returned by tables defined in fields 2, 3, and 4 on the continuation entry.
2. A CID of zero references the basic coordinate system.
3. If CID = -1, the gravity vector components are in the local displacement coordinate system of the grid points.
4. Gravity loads may be combined with "simple loads" (e.g., FORCE, MOMENT). The SID on a GRAV entry may be the same as that on a simple load entry.
5. Load sets must be selected in the Case Control Section (LOAD = SID).

GRDSET — GRID Entry Defaults

Description: 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		1				2	3456		

Field	Definition	Type	Default
CP	Identification number of coordinate system in which the location of the grid point is defined.	Integer ≥ 0 or blank	0
CD	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors are all defined at the grid point.	Integer ≥ 0 or blank	0
PS	Permanent single-point constraints associated with grid point (any of the digits 1-6 with no imbedded blanks).	Integer ≥ 0 or blank	
SEID	Superelement identification number.	Integer ≥ 0 or blank	

Remarks:

1. The contents for fields 3, 7, 8, or 9 of this entry are assumed for the corresponding fields of any GRID entry whose field 3, 7, 8, or 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.
2. Only one GRDSET entry may appear in the Bulk Data Section.
3. The primary purpose of this entry is to minimize the burden of preparing data for problems with a large amount of repetition.

GRID — Grid Point

Description: 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	3	1	4.5	1.0	7.5	2			

Field	Definition	Type	Default
ID	Grid point identification number.	Integer > 0	Required
CP	Identification number of coordinate system in which the location of the grid point is defined.	Integer ≥ 0 or blank	0
X1, X2, X3	Location of the grid point in coordinate system CP.	Real	Required
CD	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors are all defined at the grid point.	Integer ≥ 0 or blank	0
PS	Permanent single-point constraints associated with grid point (any of the digits 1-6 with no imbedded blanks).	Integer ≥ 0 or blank	
SEID	Superelement identification number.	Integer ≥ 0 or blank	

Remarks:

1. All grid point identification numbers must be unique with respect to all other grid, scalar, and extra points.
2. The meaning of X1, X2 and X3 depend on the type of coordinate system, CP, as follows (see CORDi entry descriptions):

Type	X1	X2	X3
Rectangular	X	Y	Z
Cylindrical	R	θ (degrees)	Z
Spherical	R	θ (degrees)	ϕ (degrees)

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. A zero (or blank if the GRDSET entry is not specified) in the CP or CD fields refers to the basic coordinate system.

INCLUDE — Insert External File

Description: Inserts an external file into the Model Input File.

Format:

INCLUDE [d:] [path] filename[.ext]

Example:

The following INCLUDE statement shows how to fetch the Bulk Data from another file called Bolt.NAS:

```
TITLE = STATIC ANALYSIS
SPC = 1
LOAD = 2
BEGIN BULK
INCLUDE ‘BOLT.NAS’
ENDDATA
```

Remarks:

1. The INCLUDE statement may appear anywhere in the Model Input File.
2. Maximum file specification length is 72 characters.
3. INCLUDE statements cannot be nested (i.e., no INCLUDE statement can appear inside the external file).
4. Quotation marks on the file specification are optional.

LOAD Static Load Combination (Superposition)

Description: Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, PLOAD1, PLOAD2, PLOAD4, GRAV, and SPCD 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	131	0.2	1.0	3	7.5	2			

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
S	Scale factor.	Real	Required
Si	Scale factors.	Real	Required
Li	Load set identification numbers defined via entry types enumerated above.	Integer > 0; SID ≠ Li	Required

Remarks:

1. The load vector defined is given by:

$$\{P\} = S\sum_i Si\,\{P_{Li}\}$$

2. The Li must be unique.
3. Load sets must be selected in the Case Control Section with LOAD = SID.
4. A LOAD entry may not reference a set identification number defined by another LOAD entry.

LSEQ **Static Load Set Definition**

Description: Defines a sequence of static load sets used in transient response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	DAREA	LID	TID					

Example:

LSEQ	109	100	1000	1010					

Field	Definition	Type	Default
SID	Identification number of the LSEQ set.	Integer > 0	Required
DAREA	The DAREA set identification number assigned to this static load vector.	Integer > 0	Required
LID	Load set identification number of a set of static load entries (any entry that may be referenced by the LOAD Case Control command).	Integer > 0 or blank	See Remark 5
TID	Temperature set identification of a set of thermal load entries (any entry that may be referenced by the TEMP(LOAD) Case Control command).	Integer > 0 or blank	See Remark 5

Remarks:

1. LSEQ will not be used unless selected in the Case Control Section with the LOADSET command.
2. A static load vector will be created for each DAREA identification number referenced by a LSEQ entry.
3. The DAREA identification assigned to the static load vectors may be referenced by TLOAD1 and TLOAD2 entries.
4. Element data recovery for thermal loads is not currently implemented in transient response analysis.
5. LID and TID cannot both be blank.

MAT1 Isotropic Material Property Definition

Description: Defines the material properties for linear, temperature-independent, 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	FSM	CS	EC	GC	ALPHA0	
	SB	ERSF	GRSF	FT	NB				
		TERSF	TGRSF						

Example:

MAT1	13	1.+7		0.33	0.101				
	20.+4	15.+4	12.+4						

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required
E	Young's modulus.	Real ≥ 0.0 or blank	See Remarks 4, 5, and 6
G	Shear modulus.	Real ≥ 0.0 or blank	
NU	Poisson's ratio.	-1.0 < Real ≤ 0.5 or blank	
RHO	Mass density.	Real or blank	0.0
A	Thermal expansion coefficient.	Real or blank	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real or blank	0.0
GE	Structural element damping coefficient. See Remarks 10, 11, and 13.	Real or blank	0.0
ST, SC, SS	Allowable stresses in tension, compression, and shear, respectively. Required if composite element failure index is desired.	Real ≥ 0.0 or blank	0.0
FSM	Factor of safety calculation method, selected by one of the following values (see Remark 14). 0 = no calculation 1 = von Mises Stress 2 = Principal Stress	Integer	1

Field	Definition	Type	Default
CS	Honeycomb sandwich core cell size. Required if material defines the core of a honeycomb sandwich and dimpling stability index is desired (LAM = HCS on the PCOMP entry).	Real ≥ 0.0 or blank	0.0
EC	Honeycomb sandwich core Young's modulus used for stability index analysis.	Real ≥ 0.0 or blank	E
GC	Honeycomb sandwich core shear modulus used for stability index analysis.	Real ≥ 0.0 or blank	G
ALPHA0	Fracture angle for uniaxial transverse compression in degrees. Used in the NASA LaRC02 failure theory only (see LARC02 in PCOMP entry). See Remark 15.	0.0 ≤ Real ≤ 90.0	53.0
SB	Allowable inter-laminar shear stress of the composite laminate bonding material (allowable interlaminar shear stress). See Remark 16.	Real ≥ 0.0 or blank	See Remark 16
ERSF	Young's modulus reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 18.	0.0 ≤ Real ≤ 1.0	0.0
GRSF	Shear modulus reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 18.	0.0 ≤ Real ≤ 1.0	0.0
FT	Composite failure theory. The following theories are allowed. HILL for the Hill theory HOFF for the Hoffman theory TSAI for the Tsai-Wu theory STRESS for the maximum stress theory STRAIN for the maximum strain theory	Character or blank	
NB	Allowable inter-laminar normal stress of the composite laminate bonding material (allowable interlaminar normal stress). See Remark 17.	Real ≥ 0.0 or blank	See Remark 17
TERSF	Identification number of a TABLES1 or TABLEST entry which defines the extensional stress-strain relationship for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	
TGRSF	Identification number of a TABLES1 or TABLEST entry which defines the shear stress-strain relationship for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	

Remarks:

1. The material identification number must be unique for all MATi entries.
2. Either E or G must be specified (i.e. nonblank).
3. If any one of E, G, or NU is blank, it will be computed to satisfy the identity E = 2(1 + NU)G; otherwise, values supplied by the user will be used.
4. If E and NU or G and NU are both blank, they will both be given the value 0.0.

5. Implausible data on one or more MAT1 entries will result in a warning message. Implausible data is defined as any of E < 0.0, or G < 0.0, or NU > 0.5, or NU < 0.0, or $|1 - E / [2(1+NU)G]| > 0.01$.
6. It is strongly recommended that only two of the three values E, G, and NU be input. The three values may be input independently on the MAT2 entry.
7. MAT1 materials may be made temperature-dependent by use of the MATT1 entry. In STATIC solutions, linear elastic material properties will be updated as prescribed under the TEMPERATURE Case Control command.
8. The mass density, RHO, will be used to automatically compute mass for all structural elements.
9. 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.
10. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
11. TREF and GE are ignored if the MAT1 entry is referenced by a PCOMP entry.
12. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.
13. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
14. Factor of safety calculations are based on two methods: von Mises stress or principal stress. When FT is set to 1, the factor of safety is calculated using

$$FS = \frac{ST}{\sigma_{vm}}$$

and when FT is set to 2, the factor of safety is calculated using

$$FS = min\left[\left|\frac{ST}{\sigma_{max}}\right|,\left|\frac{SC}{\sigma_{min}}\right|\right]$$

where ST and SC come from fields 2 and 3 of the continuation entry, σ_{vm} is the von Mises stress, and σ_{max} and σ_{min} are the maximum and minimum principal stresses.
15. The default value for ALPHA0 has been found experimentally and is typical for fiber reinforced polymer laminates.
16. The allowable inter-laminar shear stress value SB corresponds to the top surface of the ply. The default value for SB is defined in the SB field of the PCOMP, PCOMPG, and PCOMPS entries and will be used when this field is blank.
17. The allowable inter-laminar normal stress value NB corresponds to the top surface of the ply. The default value for NB is defined in the NB field of the PCOMPS entry and will be used when this field is blank.
18. Recommended values for ERSF and GRSF are shown in the below table.

Variable	**Recommended Value**
ERSF	0.04
GRSF	0.20

MAT2 Shell Element Anisotropic Material Property Definition

Description: Defines the material properties for linear, temperature-independent, anisotropic materials for isoparametric shell 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	
		CS	EC	GC	ALPHA0				

Example:

MAT2	15	1.+4			3.+4		4.1+4	0.32	
	1.7-6	1.5-6	1.8-6			3.5+5	4.4+4		
		0.08							

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required
Gij	The material property matrix.	Real	Required
RHO	Mass density.	Real or blank	0.0
Ai	Thermal expansion coefficient vector.	Real or blank	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real or blank	0.0
GE	Structural element damping coefficient. See Remarks 8, 9, and 11.	Real or blank	0.0
ST, SC, SS	Allowable stresses in tension, compression, and shear, respectively. Required if composite element failure index is desired.	Real or blank	0.0
CS	Honeycomb sandwich core cell size. Required if material defines the core of a honeycomb sandwich and dimpling stability index is desired (LAM = HCS on the PCOMP entry).	Real ≥ 0.0 or blank	0.0
EC	Honeycomb sandwich core Young's modulus used for stability index analysis.	Real ≥ 0.0 or blank	E
GC	Honeycomb sandwich core shear modulus used for stability index analysis.	Real ≥ 0.0 or blank	G
ALPHA0	Fracture angle for uniaxial transverse compression in degrees. Used in the NASA LaRC02 failure theory only (see LARC02 in PCOMP entry). See Remark 12.	0.0 ≤ Real ≤ 90.0	53.0

Remarks:

1. The material identification number must be unique for all MATi entries.
2. The convention for the Gij in fields 3 through 8 are represented by the matrix relationship:

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} G11 & G12 & G13 \\ G12 & G22 & G23 \\ G13 & G23 & G33 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} - \begin{Bmatrix} A1 \\ A2 \\ A3 \end{Bmatrix} (T - TREF) \right)$$

3. If this entry is referenced by the MID3 field (transverse shear) on the PSHELL, then G13, G23, and G33 must be blank.
4. Unlike the MAT1 entry, data from the MAT2 entry is used directly, without adjustment of equivalent E, G, or NU values.
5. MAT2 materials may be made temperature-dependent by use of the MATT2 entry. In STATIC solutions, linear elastic material properties will be updated as prescribed under the TEMPERATURE Case Control command.
6. The mass density, RHO, will be used to automatically compute mass for all structural elements.
7. Weight density may be used in field 9 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.
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 MAT2 entry is referenced by a PCOMP entry.
10. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.
11. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
12. The default value for ALPHA0 has been found experimentally and is typical for fiber reinforced polymer laminates.

MAT3 Axisymmetric Solid Element Orthotropic Material Property Definition

Description: Defines the material properties for linear orthotropic materials for solid axisymmetric elements.

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.+7	1.1+7	1.2+7	0.3	0.25	0.27	1.-5	
			2.5+6	1.-4	1.-4	1.1-4	68.5	0.23	

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required
EX, ETH, EZ	Young's moduli in the x, θ, and z directions, respectively.	Real > 0.0	Required
NUXTH, NUTHZ, NUZX	Poisson's ratios (coupled strain ratios in the xθ, zθ, and zx direction, respectively).	Real	Required
RHO	Mass density.	Real or blank	0.0
GZX	Shear modulus.	Real > 0.0	Required
AX, ATH, AZ	Thermal expansion coefficients.	Real or blank	0.0
TREF	Reference temperature for the calculation of thermal loads. See Remark 8.	Real or blank	0.0
GE	Structural element damping coefficient. See Remarks 7 and 9.	Real or blank	0.0

Remarks:

1. The material identification number must be unique for all MATi entries.
2. All seven of the numbers EX, ETH, EZ, NUXTH, NUTHZ, NUZX, and GZX must be specified.
3. Material stability requires that

$$E_i > \upsilon_{ij}^2 E_j$$

$$1 - \upsilon_{x\theta}\upsilon_{\theta x} - \upsilon_{\theta z}\upsilon_{z\theta} - \upsilon_{zx}\upsilon_{xz} - 2\upsilon_{\theta x}\upsilon_{z\theta}\upsilon_{xz} > 0$$

   If either condition is not met a warning message will be issued.

4. MAT3 materials may only be referenced by the CTRIAX6 entry.

5. The x-axis lies along the material axis (see Figure 2 in the CTRIAX6 entry). The θ-axis lies in the azimuthal direction. The z-axis is normal to both.
6. The stress-strain relationship is:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_\theta \\ \varepsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{\text{EX}} & -\frac{\text{NUTHX}}{\text{ETH}} & -\frac{\text{NUZX}}{\text{EZ}} & 0 \\ -\frac{\text{NUXTH}}{\text{EX}} & \frac{1}{\text{ETH}} & -\frac{\text{NUZTH}}{\text{EZ}} & 0 \\ -\frac{\text{NUXZ}}{\text{EX}} & -\frac{\text{NUTHZ}}{\text{ETH}} & \frac{1}{\text{EZ}} & 0 \\ 0 & 0 & 0 & \frac{1}{\text{GZX}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_\theta \\ \sigma_z \\ \tau_{zx} \end{Bmatrix} + (T - \text{TREF}) \begin{Bmatrix} \text{AX} \\ \text{ATH} \\ \text{AZ} \\ 0 \end{Bmatrix}$$

where,

$$\frac{\text{NUXTH}}{\text{EX}} = \frac{\text{NUTHX}}{\text{ETH}}$$

$$\frac{\text{NUZX}}{\text{EZ}} = \frac{\text{NUXZ}}{\text{EX}}$$

$$\frac{\text{NUTHZ}}{\text{ETH}} = \frac{\text{NUZTH}}{\text{EZ}}$$

7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
8. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.
9. If PARAM, W4 is not specified, GE is ignored in transient analysis. (See Section 5, *Parameters*, for more information on W4.)

MAT4 Isotropic Thermal Material Properties Definition

Description: Defines the thermal material properties for temperature-independent, isotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K	CP	RHO	H	MU	HGEN	REFENTH	
	TCH	TDELTA	QLAT						

Example:

MAT4	1	150.	0.850	1800.					

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required
K	Thermal conductivity.	Real > 0	Required
CP	Heat capacity per unit mass at constant pressure (specific heat).	Real ≥ 0 or blank	0.0
RHO	Density.	Real ≥ 0 or blank	1.0
H	Free convection heat transfer coefficient.	Real ≥ 0 or blank	0.0
MU	Dynamic viscosity.	Real > 0 or blank	0.0
HGEN	Heat generation capability used with QVOL entries.	Real ≥ 0 or blank	1.0
REFENTH	Reference enthalpy.	Real or blank	0.0
TCH	Lower temperature limit at which phase change region is to occur.	Real or blank	0.0
TDELTA	Total temperature change range within which a phase change is to occur.	Real ≥ 0 or blank	0.0
QLAT	Latent heat of fusion per unit mass associated with the phase change.	Real > 0 or blank	0.0

Remarks:

1. The MID must be unique with respect to all other MAT4 and MAT5 entries.
2. 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.
3. Properties specified on the MAT4 entry may be defined as temperature-dependent by use of the MATT4 entry.

MAT5 Anisotropic Thermal Material Property Definition

Description: Defines the material properties for temperature-independent, 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	55	0.068			0.091		0.15	0.3	
	1.4								

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required
Kij	Thermal conductivity matrix.	Real	Required
CP	Heat capacity per unit mass.	Real ≥ 0.0 or blank	
RHO	Density.	Real > 0.0 or blank	1.0
HGEN	Heat generation capability used with QVOL entries.	Real ≥ 0 or blank	1.0

Remarks:

1. The thermal conductivity matrix has the following form:

$$K = \begin{bmatrix} K_{XX} & K_{XY} & K_{XZ} \\ K_{XY} & K_{YY} & K_{YZ} \\ K_{XZ} & K_{YZ} & K_{ZZ} \end{bmatrix}$$

2. The material identification number may be the same as a MAT1 or MAT2, 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.

MAT8 Shell Element Orthotropic Material Property Definition

Description: 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	CS	EC	GC	ALPHA0	SB	
	EF1	NUF12	MSMF	PNPT	PNPC		FT	NB	
	E3	NU23	NU31	E1RSF	E2RSF	G12RSF	G1ZRSF	G2ZRSF	
				TE1RSF	TE2RSF	TG12RSF	TG1ZRSF	TG2ZRSF	

Example:

MAT8	101	90.+6	1.+7	0.3	3.+5	7.+6	1.9+6	0.066	
	29.-6	1.1-6	175.0	1.+3	1.1+4	4.+2	2.+2	5.+3	
			1.0						

Field	Definition	Type	Default
MID	Material identification number. Referenced on a PSHELL or PCOMP entry only.	Integer > 0	Required
E1	Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction.	Real ≠ 0.0	Required
E2	Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction.	Real ≠ 0.0	Required
NU12	Poisson's ratio ($\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1-direction). Note that $\upsilon_{21} = \varepsilon_2/\varepsilon_1$ 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	Required
G12	In-plane shear modulus.	Real ≥ 0.0 or blank	0.0
G1Z	Transverse shear modulus for shear in 1-Z plane.	Real > 0.0 or blank	See Remark 2.
G2Z	Transverse shear modulus for shear in 2-Z plane.	Real > 0.0 or blank	See Remark 2.
RHO	Mass density.	Real or blank	0.0
Ai	Thermal expansion coefficient in i-direction.	Real or blank	0.0

Field	Definition	Type	Default
TREF	Reference temperature for the calculation of thermal loads.	Real or blank	0.0
Xt, Xc	Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. Required if composite element failure index is desired.	Real ≥ 0.0 or blank	Default value for Xc is Xt
Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the lateral direction. Required if composite element failure index is desired.	Real ≥ 0.0 or blank	Default value for Yc is Yt
S	Allowable stress or strain for in-plane shear	Real ≥ 0.0 or blank	0.0
GE	Structural element damping coefficient. See Remarks 7, 8, and 10.	Real or blank	0.0
F12	Interaction term in the tensor polynomial theory of Tsai-Wu. Required if composite element failure index by Tsai-Wu theory is desired and if value of F12 is different from 0.0. See Remark 11.	Real	0.0
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 allowable	Blank for stress allowable
CS	Honeycomb sandwich core cell size. Required if material defines the core of a honeycomb sandwich and dimpling stability index is desired (LAM = HCS on the PCOMP entry).	Real ≥ 0.0 or blank	0.0
EC	Honeycomb sandwich core Young's modulus used for stability index analysis.	Real ≥ 0.0 or blank	See Remark 12
GC	Honeycomb sandwich core shear modulus used for stability index analysis.	Real ≥ 0.0 or blank	See Remark 12
ALPHA0	Fracture angle for uniaxial transverse compression in degrees. Used in the NASA LaRC02 failure theory only (see LARC02 in PCOMP entry). See Remark 13.	0.0 < Real < 90.0	53.0
SB	Allowable inter-laminar shear stress of the composite laminate bonding material (allowable interlaminar shear stress). See Remark 14.	Real ≥ 0.0 or blank	See Remark 14
EF1	Modulus of elasticity of fiber. Used in the Puck PCP failure theory only (see PUCK in PCOMP entry). See Remark 15.	Real > 0.0 or blank	E1/0.6
NUF12	Poisson's ratio of fiber. Used in the Puck PCP failure theory only (see PUCK in PCOMP entry).	Real ≥ 0.0 or blank	0.3
MSMF	Mean stress magnification factor. Used in the Puck PCP failure theory only (see PUCK in PCOMP entry). See Remark 15.	Real ≥ 0.0 or blank	1.1
PNPT	Failure envelop slope parameter for transverse tension. Used in the Puck PCP failure theory only (see PUCK in PCOMP entry). See Remark 16.	Real ≥ 0.0 or blank	0.35

Field	Definition	Type	Default
PNPC	Failure envelop slope parameter for transverse compression. Used in the Puck PCP failure theory only (see PUCK in PCOMP entry). See Remark 17.	Real ≥ 0.0 or blank	0.3
FT	Composite failure theory. The following theories are allowed. HILL for the Hill theory HOFF for the Hoffman theory TSAI for the Tsai-Wu theory STRESS for the maximum stress theory STRAIN for the maximum strain theory MCT for the Multicontinuum Theory	Character or blank	
NB	Allowable inter-laminar normal stress of the composite laminate bonding material (allowable interlaminar normal stress). See Remark 15.	Real ≥ 0.0 or blank	See Remark 15
E3	Modulus of elasticity in thickness direction, also defined as the matrix direction or 3-direction. See Remark 17.	Real > 0.0	E2
NU23	Poisson's ratio ($\varepsilon_3/\varepsilon_2$ for uniaxial loading in 2-direction). Note that $\upsilon_{32} = \varepsilon_3/\varepsilon_2$ for uniaxial loading in 3-direction is related to υ_{23}, E_2, and E_3 by the relation $\upsilon_{23} E_3 = \upsilon_{32} E_2$. See Remarks 17 and 18.	Real	0.5∗E2/G2Z - 1
NU31	Poisson's ratio ($\varepsilon_1/\varepsilon_3$ for uniaxial loading in 3-direction). Note that $\upsilon_{13} = \varepsilon_1/\varepsilon_3$ for uniaxial loading in 1-direction is related to υ_{31}, E_1, and E_3 by the relation $\upsilon_{31} E_1 = \upsilon_{13} E_3$. See Remarks 17 and 18.	Real	NU12∗E3/E1
E1RSF	Longitudinal modulus of elasticity reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 19.	0.0 ≤ Real ≤ 1.0	1.0
E2RSF	Lateral modulus of elasticity reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 19.	0.0 ≤ Real ≤ 1.0	1.0
G12RSF	In-plane shear modulus reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 19.	0.0 ≤ Real ≤ 1.0	1.0
G1ZRSF	Transverse shear modulus reduction scale factor in 1-Z plane for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 19.	0.0 ≤ Real ≤ 1.0	G12RSF
G2ZRSF	Transverse shear modulus reduction scale factor in 2-Z plane for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 19.	0.0 ≤ Real ≤ 1.0	G12RSF
TE1RSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the longitudinal direction for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	

Field	Definition	Type	Default
TE2RSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the lateral direction for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	
TG12RSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the in-plane shear direction for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	
TG1ZRSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the 1-Z plane for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	TG12RSF
TG2ZRSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the 2-Z plane for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	TG12RSF

Remarks:

1. The material identification number must be unique for all MATi entries.
2. If test data is not available to accurately determine G1Z and G2Z an approximate value is the in-plane shear modulus G12 which is used by default when PARAM, SHELLTVSMATTYPE is set to FLEXIBLE. When PARAM, SHELLTVSMATTYPE is set to RIGID, G1Z and G2Z will be penalty values which approximate a rigid transverse shear stiffness. (See Section 5, *Parameters*, for more information on SHELLTVSMATTYPE.)
3. 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.
4. MAT8 materials may be made temperature-dependent by use of the MATT8 entry. In STATIC solutions, linear elastic material properties will be updated as prescribed under the TEMPERATURE Case Control command.
5. The mass density, RHO, will be used to automatically compute mass for all structural elements.
6. Weight density may be used in field 9 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.
7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
8. TREF and GE are ignored if the MAT8 entry is referenced by a PCOMP entry.
9. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.
10. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
11. The interaction term F12 is experimentally determined from test specimens under biaxial loading. This inconvenience along with the constraint that F12 satisfy a stability criterion of the form

$$\left(\frac{1}{x_t x_c}\right)\left(\frac{1}{y_t y_c}\right) - F_{12}^2 > 0$$

creates complications in the use of this theory. For this reason it is recommended that F12 be set to zero.

12. The default value for EC is the minimum value of E1 and E2. The default value for GC is the average of G1Z and G2Z unless these values are zero in which case G12 is then used.
13. The default value for ALPHA0 has been found experimentally and is typical for fiber reinforced polymer laminates. See the Autodesk Nastran User's Manual, Reference 5 for additional information.
14. The allowable inter-laminar shear stress value SB corresponds to the top surface of the ply. The default value for SB is defined in the SB field of the PCOMP, PCOMPG, and PCOMPS entries and will be used when this field is blank.
15. The allowable inter-laminar normal stress value NB corresponds to the top surface of the ply. The default value for NB is defined in the NB field of the PCOMPS entry and will be used when this field is blank.
16. The default values for MSMF, PNPT, and PNPC are for carbon fibers. See the Autodesk Nastran User's Manual, Reference 13 and the table below for additional materials.

Variable	Carbon Fiber	Glass Fiber
MSMF	1.10	1.30
PNPT	0.35	0.30
PNPC	0.30	0.25

17. When the MAT8 entry is used without reference to a PCOMP layer composite property, the presence of E3, NU23, and NU31 specify that a plane strain formulation should be used. The default is plane stress. When the MAT8 entry is referenced on a PCOMP which requires E3, NU23, and NU31, they will be used if specified with the default values determined assuming transverse isotropy.
18. Material stability requires that

$$E_i > \upsilon_{ij}^2 E_j$$

$$1 - \upsilon_{12}\upsilon_{21} - \upsilon_{23}\upsilon_{32} - \upsilon_{31}\upsilon_{13} - 2\upsilon_{21}\upsilon_{32}\upsilon_{13} > 0$$

If either condition is not met a warning message will be issued.
19. Recommended values for E1RSF, E2RSF, G12RSF, G1ZRSF, and G2ZRSF are shown in the below table.

Variable	Recommended Value
E1RSF	0.04
E2RSF	0.04
G12RSF	0.20
G1ZRSF	0.20
G2ZRSF	0.20

MAT9 Solid Element Anisotropic Material Property Definition

Description: Defines the material properties for linear temperature-independent, anisotropic materials for solid isoparametric elements.

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		
	ST	SC	SS						

Example:

MAT9	17	9.2+3						7.7+3	
					4.2+3				
	7.9+3			6.1+3		9.1+3	1.2	4.1-6	
	6.8-6					155.	0.005		
	10.5	9.2	5.4						

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required
Gij	Elements of the 6×6 symmetric material property matrix in the material coordinate system.	Real	Required
RHO	Mass density.	Real or blank	0.0
Ai	Thermal expansion coefficient vector.	Real or blank	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real or blank	0.0
GE	Structural element damping coefficient. See Remarks 7 and 9.	Real or blank	0.0
ST, SC, SS	Allowable stresses in tension, compression, and shear, respectively. Required if composite element failure index is desired.	Real ≥ 0.0 or blank	0.0

Remarks:

1. The material identification number must be unique for all MATi entries.
2. The third continuation entry is optional.

3. The subscripts 1 through 6 refer to x, y, z xy, yz, zx of the material coordinate system (see the MCID field on the PSOLID entry description). The stress-strain relationship is:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} G11 & G12 & G13 & G14 & G15 & G16 \\ & G22 & G23 & G24 & G25 & G26 \\ & & G33 & G34 & G35 & G36 \\ & & & G44 & G45 & G46 \\ & \text{Symmetric} & & & G55 & G56 \\ & & & & & G66 \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - \begin{Bmatrix} A1 \\ A2 \\ A3 \\ A4 \\ A5 \\ A6 \end{Bmatrix} (T - TREF) \right)$$

4. MAT9 materials may be made temperature-dependent by use of the MATT9 entry. In STATIC solutions, linear elastic material properties will be updated as prescribed under the TEMPERATURE Case Control command.
5. The mass density, RHO, will be used to automatically compute mass for all structural elements.
6. Weight density may be used in field 8 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.
7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
8. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.
9. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)

MAT12 Solid Element Orthotropic Material Property Definition

Description: Defines the material property for an orthotropic material for isoparametric solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT12	MID	E1	E2	E3	NU12	NU23	NU31	RHO	
	G12	G23	G31	A1	A2	A3	TREF	GE	
			FT	NB	Xt	Yt	Zt		
	S12	S23	S31	SB	Xc	Yc	Zc		
	F12	F23	F31	E1RSF	E2RSF	E3RSF	G12RSF	G23RSF	
	G31RSF			TE1RSF	TE2RSF	TE3RSF	TG12RSF	TG23RSF	
	TG31RSF								

Example:

MAT12	105	2.+7	2.+7	1.+4	0.1	0.0	0.0	0.066	
	4.5+5	2.5+5	2.5+5	1.1-6	1.1-6	0.0	70.0		
					1.1+5	1.1+5	2.+3		
	5.+4	2.+4	2.+4		8.+4	8.+4	1.+3		

Field	Definition	Type	Default
MID	Material identification number. Referenced on a PSHELL or PCOMP entry only.	Integer > 0	Required
E1	Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction.	Real > 0.0	Required
E2	Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction.	Real > 0.0	Required
E3	Modulus of elasticity in thickness direction, also defined as the matrix direction or 3-direction.	Real > 0.0	Required
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 $\upsilon_{12} E_2 = \upsilon_{21} E_1$. See Remark 3.	Real	Required
NU23	Poisson's ratio ($-\varepsilon_3/\varepsilon_2$ for uniaxial loading in 2-direction). Note that $\upsilon_{32} = -\varepsilon_2/\varepsilon_3$ for uniaxial loading in 3-direction is related to υ_{23}, E_2, and E_3 by the relation $\upsilon_{23} E_3 = \upsilon_{32} E_2$. See Remark 3.	Real	Required

Field	Definition	Type	Default
NU31	Poisson's ratio ($-\varepsilon_1/\varepsilon_3$ for uniaxial loading in 3-direction). Note that $\upsilon_{13} = -\varepsilon_3/\varepsilon_1$ for uniaxial loading in 1-direction is related to υ_{31}, E_1, and E_3 by the relation $\upsilon_{31} E_1 = \upsilon_{13} E_3$. See Remark 3.	Real	Required
RHO	Mass density.	Real or blank	0.0
G12	Shear modulus in plane 1-2.	Real > 0.0	Required
G23	Shear modulus in plane 2-3.	Real > 0.0	Required
G31	Shear modulus in plane 3-1.	Real > 0.0	Required
Ai	Thermal expansion coefficient in i-direction.	Real or blank	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real or blank	0.0
GE	Structural element damping coefficient. See Remarks 9, 10, and 12.	Real or blank	0.0
FT	Composite failure theory. The following theories are allowed. HILL for the Hill theory HOFF for the Hoffman theory TSAI for the Tsai-Wu theory STRESS for the maximum stress theory STRAIN for the maximum strain theory MCT for the Multicontinuum Theory	Character or blank	
NB	Allowable inter-laminar normal stress of the composite laminate bonding material (allowable interlaminar normal stress). See Remark 14.	Real ≥ 0.0 or blank	See Remark 14
Xt, Xc	Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. Required if composite element failure index is desired.	Real ≥ 0.0 or blank	Default value for Xc is Xt
Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the lateral direction. Required if composite element failure index is desired.	Real ≥ 0.0 or blank	Default value for Yc is Yt
Zt, Zc	Allowable stresses or strains in tension and compression, respectively, in the thickness direction. Required if composite element failure index is desired.	Real ≥ 0.0 or blank	Default value for Zc is Zt
S12	Allowable shear stress or strain for plane 1-2.	Real ≥ 0.0 or blank	0.0
S23	Allowable shear stress or strain for plane 2-3.	Real ≥ 0.0 or blank	0.0
S31	Allowable shear stress or strain for plane 3-1.	Real ≥ 0.0 or blank	0.0
F12	Interaction term in the tensor polynomial theory of Tsai-Wu. Required if composite element failure index by Tsai-Wu theory is desired and if value of F12 is different from 0.0. See Remark 13.	Real	0.0

Field	Definition	Type	Default
F23	Interaction term in the tensor polynomial theory of Tsai-Wu. Required if composite element failure index by Tsai-Wu theory is desired and if value of F23 is different from 0.0.	Real	0.0
F31	Interaction term in the tensor polynomial theory of Tsai-Wu. Required if composite element failure index by Tsai-Wu theory is desired and if value of F31 is different from 0.0.	Real	0.0
SB	Allowable inter-laminar shear stress of the composite laminate bonding material (allowable interlaminar shear stress). See Remark 15.	Real ≥ 0.0 or blank	See Remark 15
E1RSF	Longitudinal (1-direction) modulus of elasticity reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 16.	0.0 ≤ Real ≤ 1.0	1.0
E2RSF	Lateral (2-direction) modulus of elasticity reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 16.	0.0 ≤ Real ≤ 1.0	1.0
E3RSF	Through thickness (3-direction) modulus of elasticity reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 16.	0.0 ≤ Real ≤ 1.0	1.0
G12RSF	Plane 1-2 shear modulus reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 16.	0.0 ≤ Real ≤ 1.0	1.0
G23RSF	Plane 2-3 shear modulus reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 16.	0.0 ≤ Real ≤ 1.0	G12RSF
G31RSF	Plane 3-1 shear modulus reduction scale factor for nonlinear composite Progressive Ply Failure Analysis (PPFA). See Remark 16.	0.0 ≤ Real ≤ 1.0	G12RSF
TE1RSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the longitudinal direction for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	
TE2RSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the lateral direction for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	
TE3RSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the thickness direction for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	
TG12RSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the 1-2 plane for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	

TG23RSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the 2-3 plane for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	TG12RSF
TG31RSF	Identification number of a TABLES1 or TABLEST entry which defines the stress-strain relationship in the 3-1 plane for nonlinear composite Progressive Ply Failure Analysis (PPFA).	Integer ≥ 0 or blank	TG12RSF

Remarks:

1. The material identification number must be unique for all MATi entries.
2. An approximate value for G23 and G31 is the in-plane shear modulus G12. If test data is not available to accurately determine G23 and G31, the value to G12 may be supplied for G23 and G31.
3. Material stability requires that

$$E_i > \upsilon_{ij}^2 E_j$$

$$1 - \upsilon_{12}\upsilon_{21} - \upsilon_{23}\upsilon_{32} - \upsilon_{31}\upsilon_{13} - 2\upsilon_{21}\upsilon_{32}\upsilon_{13} > 0$$

If either condition is not met a warning message will be issued.

4. 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 (i.e., plane 1-2) and has different properties in the direction normal to this plane. In the plane of isotropy, the properties are reduced to

$$E_1 = E_2 = E_p$$

$$\upsilon_{31} = \upsilon_{32} = \upsilon_{np}$$

$$\upsilon_{13} = \upsilon_{23} = \upsilon_{pn}$$

$$G_{13} = G_{23} = G_n$$

with $\upsilon_{np}/E_n = \upsilon_{pn}/E_p$ and $G_p = \dfrac{E_p}{2(1+\upsilon_p)}$

There are five independent material constants for normal anisotropy (i.e., E_p, E_n, υ_p, υ_{np}, and 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 (i.e., E_1, E_2, E_3, υ_{12}, G_{12}, G_{23}, and G_{31}).

5. Xt, Yt, Zt, S12, S23, and S31 are required for composite element failure calculations when requested in the FT field of the PCOMP entry. Xc, Yc, and Zc are also used but not required.
6. MAT12 materials may be made temperature-dependent by use of the MATT12 entry. In STATIC solutions, linear elastic material properties will be updated as prescribed under the TEMPERATURE Case Control command.
7. The mass density, RHO, will be used to automatically compute mass for all structural elements.
8. Weight density may be used in field 9 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.
9. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
10. TREF and GE are ignored if the MAT12 entry is referenced by a PCOMP entry.

11. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.
12. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
13. The interaction terms F12, F23, and F31 are experimentally determined from test specimens under multiaxial loading. This inconvenience along with the constraint that F12, F23, and F31 satisfy stability criteria of the form

$$\left(\frac{1}{x_t x_c}\right)\left(\frac{1}{y_t y_c}\right) - F_{12}^2 > 0$$

$$\left(\frac{1}{y_t y_c}\right)\left(\frac{1}{z_t z_c}\right) - F_{23}^2 > 0$$

$$\left(\frac{1}{x_t x_c}\right)\left(\frac{1}{z_t z_c}\right) - F_{31}^2 > 0$$

creates complications in the use of this theory. For this reason it is recommended that F12, F23, and F31 be set to zero.
14. The allowable inter-laminar normal stress value NB corresponds to the top surface of the ply. The default value for NB is defined in the NB field of the PCOMPS entry and will be used when this field is blank.
15. The allowable inter-laminar shear stress value SB corresponds to the top surface of the ply. The default value for SB is defined in the SB field of the PCOMP, PCOMPG, and PCOMPS entries and will be used when this field is blank.
16. Recommended values for E1RSF, E2RSF, E3RSF, G12RSF, G1ZRSF, and G2ZRSF are shown in the below table.

Variable	Recommended Value
E1RSF	0.04
E2RSF	0.04
E3RSF	0.04
G12RSF	0.20
G23RSF	0.20
G31RSF	0.20

MATHP — Hyperelastic Material Properties, Polynomial Form

Description: Defines material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers) for isoparametric solid elements.

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	

Example:

MATHP	100	153.8	38.5	2.+5			70.0		

Field	Contents	Type	Default
MID	Material identification number.	Integer > 0	Required
Aij	Material constants related to distortional deformation.	Real	0.0
Di	Material constants related to volumetric deformation.	Real ≥ 0	$10^3 * (A10 + A01)$ for D1. 0.0 for D2 through D5
RHO	Mass density in original configuration.	Real	0.0
AV	Volumetric coefficient of thermal expansion.	Real	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real	0.0
GE	Structural element damping coefficient. See Remarks 7 and 9.	Real	0.0
NA	Order of the distortional strain energy polynomial function.	0 < Integer ≤ 5	1
ND	Order of the volumetric strain energy polynomial function.	0 < Integer ≤ 5	1

Field	Contents	Type	Default
TAB1	Table identification number of 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. If this convention is not followed the solution may fail to converge.	Integer > 0 or blank	
TAB2	Table identification number of 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 is the current value of the pressure and r_0, t_0 is the initial radius and thickness.	Integer > 0 or blank	
TAB3	Table identification number of 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 stress F/A_0.	Integer > 0 or blank	
TAB4	Table identification number of 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 $\lambda_1 = \ell/\ell_0$ 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 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 $\lambda = \ell/\ell_0$ 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 may be expressed as follows:

$$U\left(J,\bar{I}_1,\bar{I}_2\right)=\sum_{i+j=1}^{NA}Aij\left(\bar{I}_1-3\right)^i\left(\bar{I}_2-3\right)^j+\sum_{i=1}^{ND}Di\left(J-1-AV\left(T-T_0\right)\right)^{2i}$$

   where $\bar{I}_1$ and $\bar{I}_2$ are the first and second distortional strain invariants, respectively; $J = det\,F$ is the determinate of the deformation gradient; and 2D1 = *K* and 2(A10 + A01) = *G* at small strains, in which *K* is the bulk 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. Hyperelastic materials show a fully incompressible or nearly incompressible behavior. Full incompressibility is not presently available, while nearly incompressible behavior can be simulated using a large value of D1.
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, supersedes 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 estimation the material constants.
5. The mass density, RHO, will be used to automatically compute mass for all structural elements.
6. Weight density may be used in field 9 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.
7. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
8. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.
9. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)

MATHP1 Hyperelastic Material Properties, Ogden Form

Description: Defines material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers) for isoparametric solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
MATHP1	MID	MU1	ALPHA1	D1	RHO	AV	TREF	GE	
		NA	ND						
	MU2	ALPHA2	D2	MU3	ALPHA3	D3			
	D4								

Example:

MATHP1	100	0.3245	2.0	1.45+4			70.0		
		2	1						
	-0.2345	-2.0							

Field	Contents	Type	Default
MID	Material identification number.	Integer > 0	Required
MUi	Shear moduli related to distortional deformation.	Real	0.0
ALPHAi	Exponents related to distortional deformation.	Real	0.0
Di	Material constants related to volumetric deformation.	Real ≥ 0	See Remark 2
RHO	Mass density in original configuration.	Real	0.0
AV	Volumetric coefficient of thermal expansion.	Real	0.0
TREF	Reference temperature for the calculation of thermal loads.	Real	0.0
GE	Structural element damping coefficient. See Remarks 6 and 8.	Real	0.0
NA	Order of the distortional strain energy polynomial function.	0 < Integer ≤ 3	1
ND	Order of the volumetric strain energy polynomial function.	0 < Integer ≤ 4	1

Remarks:

1. The generalized Ogden strain energy may be expressed as follows:

$$U(\lambda_1,\lambda_2,\lambda_3,J)=\sum_{i=1}^{NA}\frac{\mu_i}{\alpha_i}\left[(\lambda_1)^{\alpha_i}+(\lambda_2)^{\alpha_i}+(\lambda_3)^{\alpha_i}-3\right]+\sum_{i=1}^{ND}Di\left(J-1-AV(T-T_0)\right)^{2i}$$

   where λ_1, λ_2 and λ_3 are principal stretches; $J = det\,F$ is the determinate of the deformation gradient; and 2D1 = *K* at small strains, where *K* is the bulk modulus. T is the current temperature and T_0 is the initial temperature.

2. The default for D1 is $\frac{1}{4}\left(\sum_{i=1}^{NA}\mu_i\alpha_i\right)*10^3$. The default for D2 through D4 is zero.
3. Hyperelastic materials show a fully incompressible or nearly incompressible behavior. Full incompressibility is not presently available, while nearly incompressible behavior can be simulated using a large value of D1.
4. The mass density, RHO, will be used to automatically compute mass for all structural elements.
5. Weight density may be used in field 9 if the value 1/g is entered on the PARAM, WTMASS entry, where g is the acceleration of gravity.
6. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
7. TREF is used only as the reference temperature for the calculation of thermal loads in linear solutions. If TEMPERATURE(INITIAL) is specified, TREF will be ignored.
8. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)

MATL8 — Shell Element Orthotropic Material Property Generation

Description: Specifies the material properties for the generation of a shell element orthotropic material using MCT or Halpin-Tsai theory.

Format:

1	2	3	4	5	6	7	8	9	10
MATL8	MID	MIDM	MIDF	MIDC	FVF	TYPE	METHOD	MCTMAT	
	LC	L	D	T	W		FBVF	WBVF	
	MIDX	MIDL	MIDW	MIDP					

Example:

MATL8	101	200	300	400	0.7	1			
	1.-2	1.-2	1.-3						

Field	Definition	Type	Default
MID	Material identification number. Referenced on a PSHELL or PCOMP entry only.	Integer > 0	Required
MIDM	Material identification number for the matrix material. See Remark 3.	Integer > 0	Required if METHOD = 1
MIDF	Material identification number for the reinforcement (fiber) material. See Remark 3.	Integer > 0	Required if METHOD = 1
MIDC	Material identification number for the composite material. See Remark 3.	Integer > 0	Required if METHOD = 2
FVF	Volume fraction of fiber.	0.3 ≤ Real ≤ 0.9	Required
TYPE	Reinforcement type, selected by one of the following values 1 = Aligned continuous fibers 2 = Spherical particles 3 = Oriented short fibers 4 = Oriented plates 5 = Oriented whiskers 6 = Plain weave fabrics (MCT only) See Remarks 3, 4, and 5.	Integer	1
METHOD	Calculation method, selected by one of the following values 1 = Halpin-Tsai 2 = MCT See Remarks 2, 3, 4, and 5.	Integer	1

Field	Definition	Type	Default
MCTMAT	MCT material input, selected by one of the following values 1 = Perform MCT optimization on input materials 2 = Use input materials without modification 3 = Use default Carbon/Epoxy fiber/matrix 4 = Use default Glass/Epoxy fiber/matrix 5 = Use default Kevlar/Epoxy fiber/matrix See Remarks 6, 7, and 9.	Integer	1
LC	Short fiber critical length.	Real > 0.0	Required if TYPE = 3
L	Fiber length.	Real > 0.0	Required if TYPE = 3, 4, or, 5
D	Fiber diameter.	Real > 0.0	Required if TYPE = 3 or 5
T	Fiber plate thickness.	Real > 0.0	Required if TYPE = 4
W	Fiber plate width.	Real > 0.0	Required if TYPE = 4
FBVF	Fill bundle volume fraction. See Remark 8.	0.2 ≤ Real ≤ 0.37	Required if TYPE = 6
WBVF	Warp bundle volume fraction. See Remark 8.	0.2 ≤ Real ≤ 0.37	FBVF
MIDX	Material identification number for the MCT fill-matrix material. See Remark 3.	Integer > 0	Required if TYPE = 6 and MCTMAT = 2
MIDL	Material identification number for the MCT fill material. See Remark 3.	Integer > 0	Required if TYPE = 6 and MCTMAT = 2
MIDW	Material identification number for the MCT warp material. See Remark 3.	Integer > 0	Required if TYPE = 6 and MCTMAT = 2
MIDP	Material identification number for the MCT matrix-pocket material. See Remark 3.	Integer > 0	Required if TYPE = 6 and MCTMAT = 2

Remarks:

1. The material identification number must be unique for all MATi entries.
2. The Halpin-Tsai method is based on a set of empirical relationships that enable the property of a composite material to be expressed in terms of the properties of the matrix and reinforcing phases together with their proportions and geometry. These equations were curve fitted to exact elasticity solutions and confirmed by experimental measurements. The parameter ζ depends on the particular elastic property being considered. Halpin-Tsai theory shows that the property of a composite P_c can be expressed in terms of the corresponding property of the matrix P_m and the reinforcing phase (or fiber) P_f using

$$P_c = P_m\left(\frac{1+\zeta\eta f}{1-\eta f}\right)$$

$$\eta = \frac{\left(\frac{P_f}{P_m}\right)-1}{\left(\frac{P_f}{P_m}\right)+\zeta}$$

The MCT (Multicontinuum Theory) method is a multiscale approach to composites analysis. Failure in the composite lamina is calculated by evaluating the stress state in either the fiber or matrix, rather than the homogenized composite lamina, allowing one to capture interactions between the two. The method is applicable to unidirectional and woven composites. High fidelity micromechanics models enable the generation/optimization of composite properties from properties of the matrix and fiber. MCT ply failure analysis is enabled by specifying MCT in the FT field of the PCOMP entry.

3. MIDM and MIDF may reference either a MAT1 or MAT8 entry for the Halpin-Tsai method and only a MAT8 entry for the MCT method. For MAT1 entries the E, G, and NU fields must be non-zero. The RHO, A, ST, SC, and SS fields are optional. For MAT8 entries the E1, E2, NU12, and G12 fields must be non-zero. The RHO, A1, A2, Xt, Xc, Yt, Yc, and S fields are optional. MIDC is required for the MCT method and optional for Halpin-Tsai. MIDC, MIDX, MIDL, MIDW, and MIDP must reference a MAT8 entry only. MIDC specifies properties for the generated MAT8 material that are not calculated. The tables below lists what orthotropic material properties are generated based on the fiber type selected for the Halpin-Tsai and MCT methods.

Halpin-Tsai Generated Orthotropic Material Property Output

TYPE	E1	E2	NU12	G12	RHO	A1	A2	Xt	Xc	Yt	Yc	S
1	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
2	✓	✓	✓	✓	✓	✓	✓					
3	✓	✓	✓	✓	✓	✓	✓					
4	✓	✓	✓	✓	✓	✓	✓					
5	✓	✓	✓	✓	✓	✓	✓					

MCT Generated Orthotropic Material Property Output

TYPE	E1	E2	NU12	G12	RHO	A1	A2	Xt	Xc	Yt	Yc	S
1	✓	✓	✓	✓	✓	✓	✓					
6	✓	✓	✓	✓	✓	✓	✓					

The material allowables (Xt, Xc, Yt, etc.) must be specified on the MAT8 referenced by MIDC if failure index/strength ratios are desired and

- METHOD = 1 and TYPE ≠ 1
- METHOD = 2

4. The TYPE field defines the fiber type. TYPE = 1 – 5 are applicable to Halpin-Tsai (METHOD = 1). TYPE = 1 or 6 is applicable to MCT (METHOD = 2). Fiber types are detailed in the following table.

TYPE	Description	Example
1	Aligned continuous fiber composite lamina. Individual continuous fibers oriented in a defined direction.	Unidirectional graphite fibers in an epoxy resin.
2	Spherical particle composite lamina. Particulate composite consisting of an aggregate material with roughly round filler particles.	Unreinforced concrete with a cement aggregate and sand filler.
3	Oriented short fiber composite lamina. Discontinuous short fibers oriented in a defined direction.	A glass fiber reinforced polymer.
4	Oriented plate composite lamina. Particulate composite consisting of an aggregate material with a flat filler sheet.	A phenolic thermoset polymer matrix with a glass filler.
5	Oriented whisker composite lamina. Discontinuous whisker-shaped fibers oriented in a defined direction.	SiC whisker-reinforced ceramic matrix composite.
6	Plain weave composite lamina. Woven fabric where fill and warp threads interlace alternately resulting in equal properties in each direction.	Graphite cloth in an epoxy resin.

5. The continuation entry is required based on TYPE and METHOD. For MCT (METHOD = 2) no continuation is required. For Halpin-Tsai (METHOD = 1), fiber parameters are required based on TYPE as shown below.

TYPE	FVF	LC	L	D	T	W
1	✓					
2	✓					
3	✓	✓	✓	✓		
4	✓		✓		✓	✓
5	✓		✓	✓		
6	✓					

6. The MCTMAT field is only applicable for MCT (METHOD = 2) and affects how material properties specified on MIDM, MIDF, and MIDC are processed. When MCTMAT is set to 1 (default) MIDM and MIDF properties are optimized using a very high fidelity micromechanics model resulting in generated MIDC values. When MCTMAT is set to 2, the MIDM, MIDF, and MIDC values are assumed already optimized and no adjustment in values is made. MCTMAT set to 3, 4, or 5 provide optimized default values for common materials.
7. MCT default material properties (MCTMAT = 3, 4, or 5) require that PARAM, UNITS be specified for the correct selection of default material units corresponding to the model input material property units (see Section 5, *Parameters*, for more information on UNITS).
8. Material stability requires that if FBVF ≠ WBVF, then FBVF + WBVF ≤ 0.68. If this condition is not met a fatal error will be issued.

9. MCT default fiber and matrix material properties (MCTMAT = 3, 4, or 5) are listed in the following table in metric units.

Variable	Carbon Fiber	Glass Fiber	Kevlar Fiber	Epoxy (Carbon)	Epoxy (Glass)	Epoxy (Kevlar)
E1	2.3E+11 Pa 3.3E+7 psi	8.0E+10 Pa 1.2E+7 psi	1.2E+11 Pa 1.7E+7 psi	3.5E+9 Pa 5.1E+5 psi	3.3E+9 Pa 4.9E+5 psi	3.5E+9 Pa 5.1E+5 psi
E2	1.5E+10 Pa 2.2E+6 psi	8.0E+10 Pa 1.2E+7 psi	6.9E+9 Pa 1.0E+6 psi	3.5E+9 Pa 5.1E+5 psi	3.3E+9 Pa 4.9E+5 psi	3.5E+9 Pa 5.1E+5 psi
E3	1.5E+10 Pa 2.2E+6 psi	8.0E+10 Pa 1.2E+7 psi	6.9E+9 Pa 1.0E+6 psi	3.5E+9 Pa 5.1E+5 psi	3.3E+9 Pa 4.9E+5 psi	3.5E+9 Pa 5.1E+5 psi
G12	1.5E+10 Pa 2.2E+6 psi	3.3E+10 Pa 4.8E+6 psi	2.8E+9 Pa 4.1E+5 psi	1.3E+9 Pa 1.9E+5 psi	1.2E+9 Pa 1.8E+5 psi	1.3E+9 Pa 1.9E+5 psi
G13	1.5E+10 Pa 2.2E+6 psi	3.3E+10 Pa 4.8E+6 psi	2.8E+9 Pa 4.1E+5 psi	1.3E+9 Pa 1.9E+5 psi	1.2E+9 Pa 1.8E+5 psi	1.3E+9 Pa 1.9E+5 psi
G23	6.3E+9 Pa 9.1E+5 psi	3.3E+10 Pa 4.8E+6 psi	2.8E+9 Pa 4.1E+5 psi	1.3E+9 Pa 1.9E+5 psi	1.2E+9 Pa 1.8E+5 psi	1.3E+9 Pa 1.9E+5 psi
NU12	0.20	0.20	0.36	0.35	0.35	0.35
NU23	0.20	0.20	0.36	0.35	0.35	0.35
NU31	0.01	0.20	0.01	0.35	0.35	0.35
A1	-5.5E-7 /°C -3.1E-7 /°F	4.9E-6 /°C 2.7E-6 /°F	-5.0E-6 /°C -2.8E-6 /°F	5.3E-5 /°C 2.9E-5 /°F	5.8E-5 /°C 3.2E-5 /°F	5.3E-5 /°C 2.9E-5 /°F
A2	1.0E-5 /°C 5.6E-6 /°F	4.9E-6 /°C 2.7E-6 /°F	4.1E-5 /°C 2.3E-5 /°F	5.3E-5 /°C 2.9E-5 /°F	5.8E-5 /°C 3.2E-5 /°F	5.3E-5 /°C 2.9E-5 /°F
A3	1.0E-5 /°C 5.6E-6 /°F	4.9E-6 /°C 2.7E-6 /°F	4.1E-5 /°C 2.3E-5 /°F	5.3E-5 /°C 2.9E-5 /°F	5.8E-5 /°C 3.2E-5 /°F	5.3E-5 /°C 2.9E-5 /°F

10. The model parameters TSAI2MCT, TSAI2MCTFVF, and TSAI2MCTBVF can be used to automatically generate MATL8 entries for MCT failure analysis. The TSAI2MCT ON setting will attempt to determine the composite material fiber material when the composite material property values are within 10% of the values listed in the below table.

	Composite Properties with Aligned Continuous Fibers			**Composite Properties with Plain Weave Fabric**		
Variable	**Carbon Fiber**	**Glass Fiber**	**Kevlar Fiber**	**Carbon Fiber**	**Glass Fiber**	**Kevlar Fiber**
FVF/BVF	0.6	0.52	0.6	0.373	0.373	0.373
E1	1.4E+11 Pa 2.0E+7 psi	4.3E+10 Pa 6.2E+6 psi	7.5E+10 Pa 1.1E+7 psi	5.2E+10 Pa 7.5E+6 psi	2.7E+10 Pa 3.9E+6 psi	2.8E+10 Pa 4.1E+6 psi
E2	8.0E+9 Pa 1.2E+6 psi	9.7E+9 Pa 1.4E+6 psi	5.5E+9 Pa 8.0E+5 psi	5.2E+10 Pa 7.5E+6 psi	2.7E+10 Pa 3.9E+6 psi	2.8E+10 Pa 4.8E+6 psi
G12	3.9E+9 Pa 5.7E+5 psi	3.5E+9 Pa 5.1E+5 psi	2.0E+9 Pa 2.9E+5 psi	4.0E+9 Pa 5.8E+5 psi	4.6E+9 Pa 6.7E+5 psi	2.0E+9 Pa 2.9E+5 psi
NU12	0.26	0.26	0.36	0.072	0.12	0.1
A1	6.4E-8 /°C 3.6E-8 /°F	7.2E-6 /°C 4.0E-6 /°F	-3.9E-6 /°C -2.2E-6 /°F	3.2E-6 /°C 1.8E-6 /°F	1.2E-5 /°C 6.7E-6 /°F	3.3E-6 /°C 6.1E-6 /°F
A2	3.3E-5 /°C 1.8E-5 /°F	3.5E-5 /°C 1.9E-5 /°F	5.4E-5 /°C 3.0E-5 /°F	3.2E-6 /°C 1.8E-6 /°F	1.2E-5 /°C 6.7E-6 /°F	3.3E-6 /°C 1.8E-6 /°F

If TSAI2MCTFVF or TSAI2MCTBVF are specified, TSAI2MCT must be set to either CARBON, GLASS, or KEVLAR as required. TSAI2MCT requires PARAM, UNITS to be specified. See Section 5, *Parameters*, for more information on TSAI2MCT, TSAI2MCTFVF, and TSAI2MCTBVF.

MATL12 — Solid Element Orthotropic Material Property Generation

Description: Specifies the material properties for the generation of a solid element orthotropic material using MCT or Halpin-Tsai theory.

Format:

1	2	3	4	5	6	7	8	9	10
MATL12	MID	MIDM	MIDF	MIDC	FVF	TYPE	METHOD	MCTMAT	
	LC	L	D	T	W		FBVF	WBVF	
	MIDX	MIDL	MIDW	MIDP					

Example:

MATL12	101	200	300	400	0.7	1			
	1.-2	1.-2	1.-3						

Field	Definition	Type	Default
MID	Material identification number. Referenced on a PSOLID or PCOMP entry only.	Integer > 0	Required
MIDM	Material identification number for the matrix material. See Remark 3.	Integer > 0	Required if METHOD = 1
MIDF	Material identification number for the reinforcement (fiber) material. See Remark 3.	Integer > 0	Required if METHOD = 1
MIDC	Material identification number for the composite material. See Remark 3.	Integer > 0	Required if METHOD = 2
FVF	Volume fraction of fiber.	$0.3 \le$ Real ≤ 0.9	Required
TYPE	Reinforcement type, selected by one of the following values 1 = Aligned continuous fibers 2 = Spherical particles 3 = Oriented short fibers 4 = Oriented plates 5 = Oriented whiskers 6 = Plain weave fabrics (MCT only) See Remarks 3, 4, and 5.	Integer	1
METHOD	Calculation method, selected by one of the following values 1 = Halpin-Tsai 2 = MCT See Remarks 2, 3, 4, and 5.	Integer	1

Field	Definition	Type	Default
MCTMAT	MCT material input, selected by one of the following values 1 = Perform MCT optimization on input materials 2 = Use input materials without modification 3 = Use default Carbon/Epoxy fiber/matrix 4 = Use default Glass/Epoxy fiber/matrix 5 = Use default Kevlar/Epoxy fiber/matrix See Remarks 6, 7, and 9.	Integer	1
LC	Short fiber critical length.	Real > 0.0	Required if TYPE = 3
L	Fiber length.	Real > 0.0	Required if TYPE = 3, 4, or, 5
D	Fiber diameter.	Real > 0.0	Required if TYPE = 3 or 5
T	Fiber plate thickness.	Real > 0.0	Required if TYPE = 4
W	Fiber plate width.	Real > 0.0	Required if TYPE = 4
FBVF	Fill bundle volume fraction. See Remark 8.	0.2 ≤ Real ≤ 0.37	Required if TYPE = 6
WBVF	Warp bundle volume fraction. See Remark 8.	0.2 ≤ Real ≤ 0.37	FBVF
MIDX	Material identification number for the MCT fill-matrix material. See Remark 3.	Integer > 0	Required if TYPE = 6 and MCTMAT = 2
MIDL	Material identification number for the MCT fill material. See Remark 3.	Integer > 0	Required if TYPE = 6 and MCTMAT = 2
MIDW	Material identification number for the MCT warp material. See Remark 3.	Integer > 0	Required if TYPE = 6 and MCTMAT = 2
MIDP	Material identification number for the MCT matrix-pocket material. See Remark 3.	Integer > 0	Required if TYPE = 6 and MCTMAT = 2

Remarks:

1. The material identification number must be unique for all MATi entries.

2. The Halpin-Tsai method is based on a set of empirical relationships that enable the property of a composite material to be expressed in terms of the properties of the matrix and reinforcing phases together with their proportions and geometry. These equations were curve fitted to exact elasticity solutions and confirmed by experimental measurements. The parameter ζ depends on the particular elastic property being considered. Halpin-Tsai theory shows that the property of a composite P_c can be expressed in terms of the corresponding property of the matrix P_m and the reinforcing phase (or fiber) P_f using

$$P_c = P_m\left(\frac{1+\zeta\eta f}{1-\eta f}\right)$$

$$\eta = \frac{\left(\frac{P_f}{P_m}\right)-1}{\left(\frac{P_f}{P_m}\right)+\zeta}$$

The MCT (Multicontinuum Theory) method is a multiscale approach to composites analysis. Failure in the composite lamina is calculated by evaluating the stress state in either the fiber or matrix, rather than the homogenized composite lamina, allowing one to capture interactions between the two. The method is applicable to unidirectional and woven composites. High fidelity micromechanics models enable the generation/optimization of composite properties from properties of the matrix and fiber. MCT ply failure analysis is enabled by specifying MCT in the FT field of the PCOMP entry.

3. MIDM and MIDF may reference a MAT1, MAT8, or MAT12 entry for the Halpin-Tsai method and only a MAT8 or MAT12 entry for the MCT method. For MAT1 entries the E, G, and NU fields must be non-zero. The RHO, A, ST, SC, and SS fields are optional. For MAT8 entries the E1, E2, NU12, and G12 fields must be non-zero. The RHO, A1, A2, Xt, Xc, Yt, Yc, and S fields are optional. For MAT12 entries the E1, E2, E3, NU12, NU23, NU31, G12, G23, and G31 fields must be non-zero. The RHO, A1, A2, A3, Xt, Xc, Yt, Yc, Zt, Zc, S12, S23, and S31 fields are optional. MIDC is required for the MCT method and optional for Halpin-Tsai. MIDC, MIDX, MIDL, MIDW, and MIDP must reference a MAT8 or MAT12 entry only. MIDC specifies properties for the generated MAT12 material that are not calculated. The tables below lists what orthotropic material properties are generated based on the fiber type selected for the Halpin-Tsai and MCT methods.

Halpin-Tsai Generated Orthotropic Material Property Output

TYPE	E1	E2	NU12	G12	RHO	A1	A2	Xt	Xc	Yt	Yc	S
1	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
2	✓	✓	✓	✓	✓	✓	✓					
3	✓	✓	✓	✓	✓	✓	✓					
4	✓	✓	✓	✓	✓	✓	✓					
5	✓	✓	✓	✓	✓	✓	✓					

MCT Generated Orthotropic Material Property Output

TYPE	E1	E2	E3	NU12	NU23	NU31	G12	G23	G31	RHO
1	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
6	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

MCT Generated Orthotropic Material Property Output

TYPE	A1	A2	A3	Xt	Xc	Yt	Yc	Zt	Zc	S12	S23	S31
1	✓	✓	✓									
6	✓	✓	✓									

The material allowables (Xt, Xc, Yt, etc.) must be specified on the MAT8 referenced by MIDC if failure index/strength ratios are desired and

- METHOD = 1 and TYPE ≠ 1
- METHOD = 2

4. The TYPE field defines the fiber type. TYPE = 1 – 5 are applicable to Halpin-Tsai (METHOD = 1). TYPE = 1 or 6 is applicable to MCT (METHOD = 2). Fiber types are detailed in the following table.

TYPE	Description	Example
1	Aligned continuous fiber composite lamina. Individual continuous fibers oriented in a defined direction.	Unidirectional graphite fibers in an epoxy resin.
2	Spherical particle composite lamina. Particulate composite consisting of an aggregate material with roughly round filler particles.	Unreinforced concrete with a cement aggregate and sand filler.
3	Oriented short fiber composite lamina. Discontinuous short fibers oriented in a defined direction.	A glass fiber reinforced polymer.
4	Oriented plate composite lamina. Particulate composite consisting of an aggregate material with a flat filler sheet.	A phenolic thermoset polymer matrix with a glass filler.
5	Oriented whisker composite lamina. Discontinuous whisker-shaped fibers oriented in a defined direction.	SiC whisker-reinforced ceramic matrix composite.
6	Plain weave composite lamina. Woven fabric where fill and warp threads interlace alternately resulting in equal properties in each direction.	Graphite cloth in an epoxy resin.

5. The continuation entry is required based on TYPE and METHOD. For MCT (METHOD = 2) no continuation is required. For Halpin-Tsai (METHOD = 1), fiber parameters are required based on TYPE as shown below.

TYPE	FVF	LC	L	D	T	W
1	✓					
2	✓					
3	✓	✓	✓	✓		
4	✓		✓		✓	✓
5	✓		✓	✓		
6	✓					

6. The MCTMAT field is only applicable for MCT (METHOD = 2) and affects how material properties specified on MIDM, MIDF, and MIDC are processed. When MCTMAT is set to 1 (default) MIDM and MIDF properties are optimized using a very high fidelity micromechanics model resulting in generated MIDC values. When MCTMAT is set to 2, the MIDM, MIDF, and MIDC values are assumed already optimized and no adjustment in values is made. MCTMAT set to 3, 4, or 5 provide optimized default values for common materials.
7. MCT default material properties (MCTMAT = 3, 4, or 5) require that PARAM, UNITS be specified for the correct selection of default material units corresponding to the model input material property units (see Section 5, *Parameters*, for more information on UNITS).
8. Material stability requires that if FBVF ≠ WBVF, then FBVF + WBVF ≤ 0.68. If this condition is not met a fatal error will be issued.
9. MCT (METHOD = 2) with aligned continuous fibers (TYPE = 1) requires that the orthotropic material referenced by MIDC be transversely isotropic where

$$E_3 = E_2$$

$$\upsilon_{23} = E_2/2G_{23} - 1$$

$$\upsilon_{31} = \upsilon_{12}E_3/E_1$$

$$G_{13} = G_{12}$$

$$Z_t = Y_t$$

$$Z_c = Y_c$$

10. MCT default fiber and matrix material properties (MCTMAT = 3, 4, or 5) are listed in the following table in metric units.

Variable	Carbon Fiber	Glass Fiber	Kevlar Fiber	Epoxy (Carbon)	Epoxy (Glass)	Epoxy (Kevlar)
E1	2.3E+11 Pa 3.3E+7 psi	8.0E+10 Pa 1.2E+7 psi	1.2E+11 Pa 1.7E+7 psi	3.5E+9 Pa 5.1E+5 psi	3.3E+9 Pa 4.9E+5 psi	3.5E+9 Pa 5.1E+5 psi
E2	1.5E+10 Pa 2.2E+6 psi	8.0E+10 Pa 1.2E+7 psi	6.9E+9 Pa 1.0E+6 psi	3.5E+9 Pa 5.1E+5 psi	3.3E+9 Pa 4.9E+5 psi	3.5E+9 Pa 5.1E+5 psi
E3	1.5E+10 Pa 2.2E+6 psi	8.0E+10 Pa 1.2E+7 psi	6.9E+9 Pa 1.0E+6 psi	3.5E+9 Pa 5.1E+5 psi	3.3E+9 Pa 4.9E+5 psi	3.5E+9 Pa 5.1E+5 psi
G12	1.5E+10 Pa 2.2E+6 psi	3.3E+10 Pa 4.8E+6 psi	2.8E+9 Pa 4.1E+5 psi	1.3E+9 Pa 1.9E+5 psi	1.2E+9 Pa 1.8E+5 psi	1.3E+9 Pa 1.9E+5 psi
G13	1.5E+10 Pa 2.2E+6 psi	3.3E+10 Pa 4.8E+6 psi	2.8E+9 Pa 4.1E+5 psi	1.3E+9 Pa 1.9E+5 psi	1.2E+9 Pa 1.8E+5 psi	1.3E+9 Pa 1.9E+5 psi
G23	6.3E+9 Pa 9.1E+5 psi	3.3E+10 Pa 4.8E+6 psi	2.8E+9 Pa 4.1E+5 psi	1.3E+9 Pa 1.9E+5 psi	1.2E+9 Pa 1.8E+5 psi	1.3E+9 Pa 1.9E+5 psi
NU12	0.20	0.20	0.36	0.35	0.35	0.35
NU23	0.20	0.20	0.36	0.35	0.35	0.35
NU31	0.01	0.20	0.01	0.35	0.35	0.35
A1	-5.5E-7 /°C -3.1E-7 /°F	4.9E-6 /°C 2.7E-6 /°F	-5.0E-6 /°C -2.8E-6 /°F	5.3E-5 /°C 2.9E-5 /°F	5.8E-5 /°C 3.2E-5 /°F	5.3E-5 /°C 2.9E-5 /°F
A2	1.0E-5 /°C 5.6E-6 /°F	4.9E-6 /°C 2.7E-6 /°F	4.1E-5 /°C 2.3E-5 /°F	5.3E-5 /°C 2.9E-5 /°F	5.8E-5 /°C 3.2E-5 /°F	5.3E-5 /°C 2.9E-5 /°F
A3	1.0E-5 /°C 5.6E-6 /°F	4.9E-6 /°C 2.7E-6 /°F	4.1E-5 /°C 2.3E-5 /°F	5.3E-5 /°C 2.9E-5 /°F	5.8E-5 /°C 3.2E-5 /°F	5.3E-5 /°C 2.9E-5 /°F

11. The model parameters TSAI2MCT, TSAI2MCTFVF, and TSAI2MCTBVF can be used to automatically generate MATL12 entries for MCT failure analysis. The TSAI2MCT ON setting will attempt to determine the composite material fiber material when the composite material property values are within 10% of the values listed in the below table.

	Composite Properties with Aligned Continuous Fibers			**Composite Properties with Plain Weave Fabric**		
Variable	**Carbon Fiber**	**Glass Fiber**	**Kevlar Fiber**	**Carbon Fiber**	**Glass Fiber**	**Kevlar Fiber**
FVF/BVF	0.6	0.52	0.6	0.373	0.373	0.373
E1	1.4E+11 Pa 2.0E+7 psi	4.3E+10 Pa 6.2E+6 psi	7.5E+10 Pa 1.1E+7 psi	5.2E+10 Pa 7.5E+6 psi	2.7E+10 Pa 3.9E+6 psi	2.8E+10 Pa 4.1E+6 psi
E2	8.0E+9 Pa 1.2E+6 psi	9.7E+9 Pa 1.4E+6 psi	5.5E+9 Pa 8.0E+5 psi	5.2E+10 Pa 7.5E+6 psi	2.7E+10 Pa 3.9E+6 psi	2.8E+10 Pa 4.1E+6 psi
E3	8.0E+9 Pa 1.2E+6 psi	9.7E+9 Pa 1.4E+6 psi	5.5E+9 Pa 8.0E+5 psi	8.2E+9 Pa 1.2E+6 psi	1.0E+10 Pa 1.5E+6 psi	5.8E+9 Pa 4.1E+6 psi
G12	3.9E+9 Pa 5.7E+5 psi	3.5E+9 Pa 5.1E+5 psi	2.0E+9 Pa 2.9E+5 psi	4.0E+9 Pa 5.8E+5 psi	4.6E+9 Pa 6.7E+5 psi	2.0E+9 Pa 2.9E+5 psi
G23	3.9E+9 Pa 5.7E+5 psi	3.5E+9 Pa 5.1E+5 psi	2.0E+9 Pa 2.9E+5 psi	2.7E+9 Pa 3.9E+5 psi	3.1E+9 Pa 4.5E+5 psi	1.9E+9 Pa 2.8E+5 psi
G31	2.9E+9 Pa 4.2E+5 psi	3.4E+9 Pa 4.9E+5 psi	2.0E+9 Pa 2.9E+5 psi	2.7E+9 Pa 3.9E+5 psi	3.1E+9 Pa 4.5E+5 psi	1.9E+9 Pa 2.8E+5 psi
NU12	0.26	0.26	0.36	0.072	0.12	0.1
NU23	0.26	0.26	0.36	0.4	0.36	0.45
NU31	0.38	0.42	0.37	0.4	0.36	0.45
A1	6.4E-8 /°C 3.6E-8 /°F	7.2E-6 /°C 4.0E-6 /°F	-3.9E-6 /°C -2.2E-6 /°F	3.2E-6 /°C 1.8E-6 /°F	1.2E-5 /°C 6.7E-6 /°F	3.3E-6 /°C 1.8E-6 /°F
A2	3.3E-5 /°C 1.8E-5 /°F	3.5E-5 /°C 1.9E-5 /°F	5.4E-5 /°C 3.0E-5 /°F	3.2E-6 /°C 1.8E-6 /°F	1.2E-5 /°C 6.7E-6 /°F	3.3E-6 /°C 1.8E-6 /°F
A3	3.3E-5 /°C 1.8E-5 /°F	3.5E-5 /°C 1.9E-5 /°F	5.4E-5 /°C 3.0E-5 /°F	5.2E-5 /°C 2.9E-6 /°F	4.4E-5 /°C 2.4E-6 /°F	7.3E-5 /°C 4.1E-6 /°F

If TSAI2MCTFVF or TSAI2MCTBVF are specified, TSAI2MCT must be set to either CARBON, GLASS, or KEVLAR as required. TSAI2MCT requires PARAM, UNITS to be specified. See Section 5, *Parameters*, for more information on TSAI2MCT, TSAI2MCTFVF, and TSAI2MCTBVF.

MATPFA — Helius PFA Material Property Definition

Description: Defines the material properties for composite materials used with Helius PFA.

Format:

1	2	3	4	5	6	7	8	9	10
MATPFA	MID	MIDH	PDIR	FT	MPSTIF/ MDE	FPSTIF/ FDE	DEVO	PRENL	
	TREF	MOIST	F/ALPHA	SBIAX	PFTYPE	HYDRO			

Example:

MATPFA	35	9001	1	MCT	0.1	0.1	ENERGY	ON	
	300	WET							

Field	Definition	Type	Default
MID	Material identification number.	Integer > 0	Required
MIDH	Helius PFA material identification number.	Integer	Required
PDIR	Principal material coordinate system.	Unidirectional → 1 2 Woven → 1, 2, 3	1
FT	Failure Theory.	Uni → MCT,STRESS, STRAIN, HILL, TSAI, CHRIS, HASH, PUCK, LaRC02, USER Woven → MCT, STRESS, STRAIN, USER	MCT
MPSTIF	Matrix Post Failure Stiffness	0.0 < value ≤ 1.0	Required
MDE	Matrix Degradation Energy	value > 0.0	
FPSTIF	Fiber Post Failure Stiffness	0.0 < value ≤ 1.0	Required
FDE	Fiber Degradation Energy	value > 0.0	
DEVO	Damage Evolution Method	INSTANT, ENERGY	INSTANT
PRENL	Pre-Failure Nonlinearity	OFF, ON	OFF

TREF	Temperature Flag	-1 (activate temperature dependence) value ≥ 0.0 (temperature corresponding to environment in mdata file)	
MOIST	Moisture Flag	AMBIENT, DRY, WET	AMBIENT
F	Cross-product Term in Tsai-Wu FC	Real Number	
ALPHA	Contribution of Longitudinal Shear Stress for Hashin FC	Real Number	
SBIAX	Equibiaxial Stress at Failure for Tsai-Wu FC	Real Number	
PFTYPE	Progressive Failure Analysis Type	Uni → 0 (off), 1 (on) Woven → 0 (off), 1 or 2 (on)	0
HYDRO	Hydrostatic Strengthening	OFF, ON	OFF

Refer to Appendix A of the *Helius PFA User's Guide for Autodesk Nastran* for more details on each MATPFA field.

Note: When PFTYPE = 1 or 2, a Helius PFA license is required.

MATS1 — Material Stress Dependence

Description: Specifies stress-dependent material properties for use in nonlinear analysis. This entry is used if a MAT1, MAT2, MAT8, MAT9, or MAT12 entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1	LIMIT2	

Example:

MATS1	25	100	PLASTIC		1	1	2.+4		

Field	Definition	Type	Default
MID	Identification number of a MAT1, MAT2, MAT8, MAT9, or MAT12 entry.	Integer > 0	Required
TID	Identification number of a TABLES1 or TABLEST entry. If H is given, then this field must be blank. See Remark 3	Integer ≥ 0 or blank	
TYPE	Type of material nonlinearity, one of the following character variables: NLELAST for nonlinear elastic or PLASTIC for elastic-plastic. See Remarks.	Character	Required
H	Work hardening slope (slope of stress vs. plastic strain) in units of stress. 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 1 = von Mises 2 = Tresca 3 = Mohr-Coulomb 4 = Drucker-Prager	Integer	von Mises
HR	Hardening rule, selected by one of the following values 1 = Isotropic 2 = Kinematic 3 = Combined isotropic and kinematic hardening	Integer	Isotropic
LIMIT1	Initial yield point. Y_1 for von Mises and Tresca yield criteria and 2 ∗ Cohesion, 2c (in units of stress).	Real	0.0
LIMIT2	Internal friction angle (measured in degrees) for the Mohr-Coulomb and Drucker-Prager yield criteria.	Real	0.0

Remarks:

1. If TYPE = NLELAST, then MID may refer to a MAT1 entry only. The TID in field three must be specified. The stress-strain data given in the TABLES1 entry will be used to determine the stress for a given value of strain. If specified, the values H, YF, and LIMIT will be ignored in this case.

   Thermoelastic analysis with temperature-dependent material properties is available for linear and nonlinear elastic isotropic materials (TYPE = NLELAST) and linear elastic orthotropic and anisotropic materials. Four options of constitutive relations exist. The relations appear in the table below along with the required Bulk Data entries.

Constitutive Relation	Require Bulk Data Entries
$\{\sigma\}=[G_e(T)]\{\varepsilon\}$	MATi and MATTi where i =1, 2, 8, or 9
$\{\sigma\}=\frac{\bar{E}(\sigma,\varepsilon)}{E}[G_e(T)]\{\varepsilon\}$	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 1, $\{\sigma\}$ 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, 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 $E_T=\frac{dY}{d\varepsilon}$ is the slope of the uniaxial stress-strain curve in the plastic region. See Figure 1.

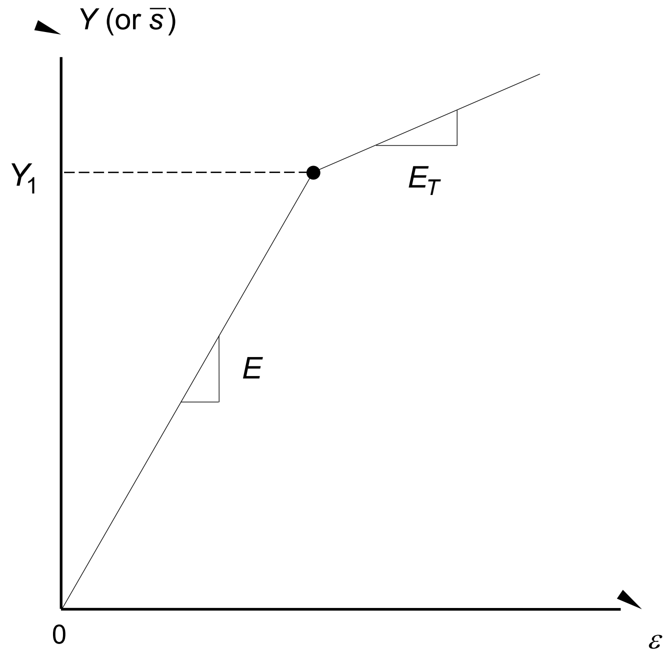


Figure 1. Stress-Strain Curve Definition When H is Specified in Field 5.

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 2):
   a) If TYPE = PLASTIC, the curve must be defined in the first quadrant. The first point must be at origin (X1 = 0, Y1 = 0) and the second point (X2, Y2) must be at the initial yield point (Y_1 or $2c$) specified on the MATS1 entry. The slope of the line joining the origin to the yield stress must be equal to the valued of *E*. Also, TID may not reference a TABLEST entry.
   b) If TYPE = NLELAST, the full stress-stress 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, Y1 = 0.0) and the compression properties will be assumed identical to tension properties.
4. Material nonlinear behavior requires a nonlinear solution.

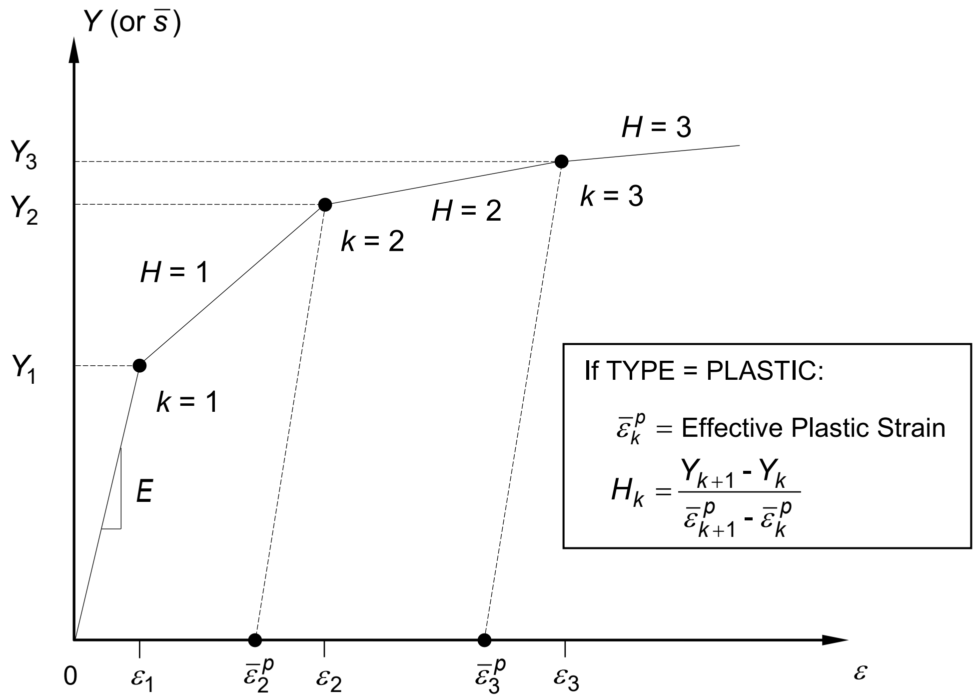


Figure 2. Stress-Strain Curve Definition When TID is Specified in Field 3.

MATS2 — Material Stress Dependence, Alternate Form

Description: Specifies stress-dependent material properties for use in nonlinear analysis. This entry is used if a MAT1 entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
MATS2	MID		TYPE	B	TY	SY	ALPHA		

Example:

MATS2	35		OHSAKI	1.4	2.+4		1.0		

Field	Definition	Type	Default
MID	Identification number of a MAT1 entry.	Integer > 0	Required
TYPE	Type of material nonlinearity, one of the following character variables: OHSAKI for hysteresis soil plasticity or RAMBERG for deformation plasticity. See Remarks.	Character	Required
B	Exponent.	Real > 0.0	Required
TY	Initial tensile yield strength.	Real > 0.0	See Remark 3
SY	Initial shear yield strength.	Real > 0.0	See Remark 3
ALPHA	Initial yield offset.	Real	0.0

Remarks:

10. If TYPE = OHSAKI, B is a soil type factor (1.6 for sand and 1.4 for clay) and TY is the initial tensile yield strength. If specified, ALPHA is ignored in this case. The constitutive relationship is given by

$$\frac{\varepsilon_{eff}}{M} = \frac{\sigma_{eqv}}{3G_0M}\left(1 + A\left|\frac{\sigma_{eqv}}{\mathrm{TY}\cdot M}\right|^{\mathrm{B}}\right)$$

where,

G_0 is the initial shear modulus

M is 1 for initial loading and 2 for unloading and reloading

ε_{eff} is the effective strain

σ_{eqv} is the equivalent stress

and

$$A = \frac{G_0}{100\mathrm{SY}} - 1$$

11. If TYPE = RAMBERG, B is a hardening exponent (normally, B > 5), TY is the initial tensile yield strength, and ALPHA is the initial yield offset. The constitutive relationship is given by

$$\frac{\varepsilon_{eff}}{M} = \frac{\sigma_{eqv}}{3G_0 M}\left(1 + \frac{3\text{ALPHA}G_0}{E}\left|\frac{\sigma_{eqv}}{\text{TY}\cdot M}\right|^{\text{B-1}}\right)$$

where,

E	is the reference elasticity modulus
G_0	is the initial shear modulus
M	is 1 for initial loading and 2 for unloading and reloading
ε_{eff}	is the effective strain
σ_{eqv}	is the equivalent stress

3. The relation between initial tensile yield strength and initial shear yield strength is given from the von Mises yield criterion: $\text{TY} = \sqrt{3}\text{SY}$. Either TY or SY must be given. When both are given, SY will be ignored.
4. When the loading direction changes, the effective stress and strain are calculated with respect to the last stress and strain locations of the previous load step (turning stress, σ_T and strain, ε_T). See Figure 1.
5. Material nonlinear behavior requires a nonlinear solution.

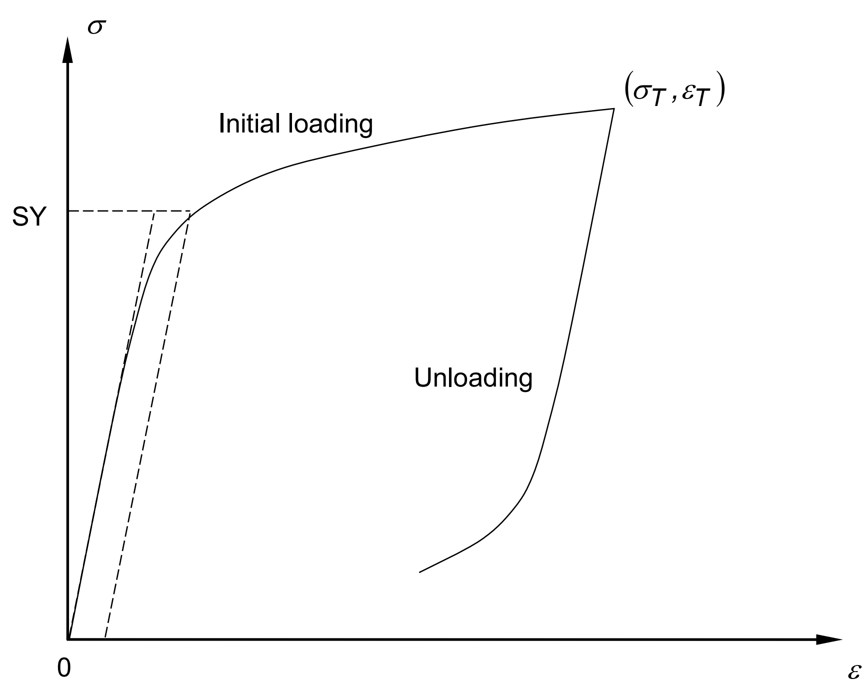


Figure 1. Stress-Strain Curve Definition for MATS2 Material.

MATST1 — Material Stress and Temperature Dependence

Description: Specifies temperature-dependent table references for MATS1 material properties. This entry is used if a MATS1 entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
MATST1	MID			T(H)			T(Y1)		

Example:

MATST1	55			100			110		

Field	Definition	Type	Default
MID	Material identification number that matches the MATS1 identification number.	Integer > 0	Required
T(H)	TABLEMi identification number for work hardening slope.	Integer ≥ 0 or blank	
T(Y1)	TABLEMi identification number for initial yield point.	Integer ≥ 0 or blank	

Remarks:

1. Temperature-dependent material properties are only calculated when a temperature distribution for materials is defined by using TEMPERATURE, TEMPERATURE(MATERIAL), or TEMPERATURE(BOTH) Case Control commands.
2. Fields 5 and 8 of this entry correspond, one-by-one, to fields 5 and 8 of the MATS1 entry referenced in field 2. The value in a particular field of the MATS1 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, H is modified by TABLEMi 100.
3. Any quantity modified by this entry must have a value on the MATS1 entry.
4. Table references must be present for each item that is temperature-dependent.
5. Material nonlinear behavior requires a nonlinear solution.

MATT1 — Isotropic Material Temperature Dependence

Description: Specifies temperature-dependent table references for MAT1 material properties. This entry is used if a MAT1 entry is specified with the same MID.

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	66	56				88			
	78	56							

Field	Definition	Type	Default
MID	Material identification number that matches the MAT1 identification number.	Integer > 0	Required
T(E)	TABLEMi identification number for Young's modulus.	Integer ≥ 0 or blank	
T(G)	TABLEMi identification number for shear modulus.	Integer ≥ 0 or blank	
T(NU)	TABLEMi identification number for Poisson's ratio.	Integer ≥ 0 or blank	
T(RHO)	TABLEMi identification number for mass density.	Integer ≥ 0 or blank	
T(A)	TABLEMi identification number for thermal expansion coefficient.	Integer ≥ 0 or blank	
T(GE)	TABLEMi identification number for damping coefficient.	Integer ≥ 0 or blank	
T(ST)	TABLEMi identification number for tensile stress limit.	Integer ≥ 0 or blank	
T(SC)	TABLEMi identification number for compressive stress limit.	Integer ≥ 0 or blank	
T(SS)	TABLEMi identification number for shear stress limit.	Integer ≥ 0 or blank	

Remarks:

1. Temperature-dependent material properties are only calculated when a temperature distribution for materials is defined by using TEMPERATURE, TEMPERATURE(MATERIAL), or TEMPERATURE(BOTH) Case Control commands.

2. Fields 3, 4, etc., of this entry correspond, one-by-one, 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 56. A blank or zero entry means no temperature dependence of that field on the MAT1 entry.
3. 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. If a table is specified for E and not for G, the E table reference will be used in the determination of G.
4. Table references must be present for each item that is temperature-dependent.

MATT2 Anisotropic Material Temperature Dependence

Description: Specifies temperature-dependent table references for MAT2 material properties. This entry is used if a MAT2 entry is specified with the same MID.

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	45	56				65			
	21	89							

Field	Definition	Type	Default
MID	Material property identification number that matches the MAT2 identification number.	Integer > 0	Required
T(Gij)	TABLEMi identification numbers for the terms in the material property matrix.	Integer ≥ 0 or blank	
T(RHO)	TABLEMi identification number for mass density.	Integer ≥ 0 or blank	
T(Ai)	TABLEMi identification number for the thermal expansion coefficient vector.	Integer ≥ 0 or blank	
T(GE)	TABLEMi identification number for damping coefficient.	Integer ≥ 0 or blank	
T(ST)	TABLEMi identification number for tensile stress limit.	Integer ≥ 0 or blank	
T(SC)	TABLEMi identification number for compressive stress limit.	Integer ≥ 0 or blank	
T(SS)	TABLEMi identification number for shear stress limit.	Integer ≥ 0 or blank	

Remarks:

1. Temperature-dependent material properties are only calculated when a temperature distribution for materials is defined by using TEMPERATURE, TEMPERATURE(MATERIAL), or TEMPERATURE(BOTH) Case Control commands.
2. Fields 3, 4, etc., of this entry correspond, one-by-one, 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. A blank or zero entry means no temperature dependence of that field on the MAT2 entry.

3. Any quantity modified by this entry must have a value on the MAT2 entry.
4. Table references must be present for each item that is temperature-dependent.

MATT4 Thermal Material Temperature Dependence

Description: Specifies table references for temperature-dependent MAT4 material properties. This entry is used if a MAT4 entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
MATT4	MID	T(K)	T(CP)		T(H)	T(μ)	T(HGEN)		

Example:

MATT4	2	10	11						

Field	Definition	Type	Default
MID	Identification number of a MAT4 entry, which is temperature-dependent.	Integer > 0	Required
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 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

Description: Specifies temperature-dependent table references for MAT5 material properties. This entry is used if a MAT5 entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
MATT5	MID	T(KXX)	(TKXY)	T(KXZ)	T(KYY)	T(KYZ)	T(KZZ)	T(CP)	
		T(HGEN)							

Example:

MATT5	2	10	11						

Field	Definition	Type	Default
MID	Material property identification number that matches the MAT5 identification number.	Integer > 0	Required
T(Kij)	Identification number of a TABLEMi entry that specify temperature dependence of the matrix term.	Integer ≥ 0 or blank	
T(CP)	Identification number of a TABLEMi entry that specifies the temperature dependence of the thermal heat capacity.	Integer ≥ 0 or blank	
T(HGEN)	Identification number of a TABLEMi entry that gives the temperature dependence of internal heat generation property for the QVOL entry.	Integer ≥ 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 independence of the referenced quantity on the basic MAT5 entry.

MATT8 Thermal Shell Element Orthotropic Material Dependence

Description: Specifies temperature-dependent table references for MAT8 material properties. This entry is used if a MAT8 entry is specified with the same MID.

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:

MATT8	101	122	145			124	22		
	202	202							
		220							

Field	Definition	Type	Default
MID	Material property identification number that matches the MAT8 identification number.	Integer > 0	Required
T(E1)	TABLEMi identification number for modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction.	Integer ≥ 0 or blank	
T(E2)	TABLEMi identification number for modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction.	Integer ≥ 0 or blank	
T(NU12)	TABLEMi identification number for Poisson's ratio ($\varepsilon 2/\varepsilon 1$ for uniaxial loading in 1-direction).	Integer ≥ 0 or blank	
T(G12)	TABLEMi identification number for in-plane shear modulus.	Integer ≥ 0 or blank	
T(G1Z)	TABLEMi identification number for transverse shear modulus for shear in 1-Z plane.	Integer ≥ 0 or blank	
T(G2Z)	TABLEMi identification number for transverse shear modulus for shear in 2-Z plane.	Integer ≥ 0 or blank	
T(RHO)	TABLEMi identification number for mass density.	Integer ≥ 0 or blank	
T(Ai)	TABLEMi identification number for thermal expansion coefficient in i-direction.	Integer ≥ 0 or blank	
T(Xt), T(Xc)	TABLEMi identification number for allowable stresses or strains in tension and compression, respectively, in the longitudinal direction.	Integer ≥ 0 or blank	

Field	Definition	Type	Default
T(Yt), T(Yc)	TABLEMi identification number for allowable stresses or strains in tension and compression, respectively, in the lateral direction.	Integer ≥ 0 or blank	
T(S)	TABLEMi identification number for allowable stress or strain for in-plane shear.	Integer ≥ 0 or blank	
T(GE)	TABLEMi identification number for damping coefficient.	Integer ≥ 0 or blank	
T(F12)	TABLEMi identification number for interaction term in the tensor polynomial theory of Tsai-Wu.	Integer ≥ 0 or blank	

Remarks:

1. Temperature-dependent material properties are only calculated when a temperature distribution for materials is defined by using TEMPERATURE, TEMPERATURE(MATERIAL), or TEMPERATURE(BOTH) Case Control commands.
2. Fields 3, 4, etc., of this entry correspond, one-by-one, 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. A blank or zero entry means no temperature dependence of that field on the MAT8 entry.
3. Any quantity modified by this entry must have a value on the MAT8 entry.
4. Table references must be present for each item that is temperature-dependent.

MATT9 Solid Element Anisotropic Material Temperature Dependence

Description: Specifies temperature-dependent table references for MAT9 material properties. This entry is used if a MAT9 entry is specified with the same MID.

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	56	66	68					34	
					78				
	41			124		88	90	23	
	101					54	44		

Field	Definition	Type	Default
MID	Material property identification number that matches the MAT9 identification number.	Integer > 0	Required
T(Gij)	TABLEMi identification number for elements of the 6×6 symmetric material property matrix.	Integer ≥ 0 or blank	
T(RHO)	TABLEMi identification number for mass density.	Integer ≥ 0 or blank	
T(Ai)	TABLEMi identification number for thermal expansion coefficient.	Integer ≥ 0 or blank	
T(GE)	TABLEMi identification number for damping coefficient.	Integer ≥ 0 or blank	

Remarks:

1. Temperature-dependent material properties are only calculated when a temperature distribution for materials is defined by using TEMPERATURE, TEMPERATURE(MATERIAL), or TEMPERATURE(BOTH) Case Control commands.
2. Fields 3, 4, etc., of this entry correspond, one-by-one, to fields 3, 4, etc., of the MAT9 entry referenced in field 2. The value in a particular field of the MAT9 entry is replaced or modified by the table referenced in the corresponding field of this entry. A blank or zero entry means no temperature dependence of that field on the MAT9 entry.
3. Any quantity modified by this entry must have a value on the MAT9 entry.
4. Table references must be present for each item that is temperature-dependent.

MATT12 Solid Element Orthotropic Material Temperature Dependence

Description: Specifies temperature-dependent table references for MAT12 material properties. This entry is used if a MAT12 entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
MATT12	MID	T(E1)	T(E2)	T(E3)	T(NU12)	T(NU23)	T(NU31)	T(RHO)	
	T(G12)	T(G23)	T(G31)	T(A1)	T(A2)	T(A3)		T(GE)	
					T(Xt)	T(Yt)	T(Zt)		
	T(S12)	T(S23)	T(S31)		T(Xc)	T(Yc)	T(Zt)		
	T(F12)	T(F23)	T(F31)						

Example:

MATT12	77	101	102	103					
	201	202	203						
					45	48	50		
	41	42	43		46	49	51		
	77	78	79						

Field	Definition	Type	Default
MID	Material property identification number that matches the MAT12 identification number.	Integer > 0	Required
T(E1)	TABLEMi identification number for modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction.	Integer ≥ 0 or blank	
T(E2)	TABLEMi identification number for modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction.	Integer ≥ 0 or blank	
T(E3)	TABLEMi identification number for modulus of elasticity in thickness direction, also defined as the matrix direction or 3-direction.	Integer ≥ 0 or blank	
T(NU12)	TABLEMi identification number for Poisson's ratio ($\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1-direction).	Integer ≥ 0 or blank	
T(NU23)	TABLEMi identification number for Poisson's ratio ($\varepsilon_3/\varepsilon_2$ for uniaxial loading in 2-direction).	Integer ≥ 0 or blank	
T(NU31)	TABLEMi identification number for Poisson's ratio ($\varepsilon_1/\varepsilon_3$ for uniaxial loading in 3-direction).	Integer ≥ 0 or blank	

Field	Definition	Type	Default
T(RHO)	TABLEMi identification number for mass density.	Integer ≥ 0 or blank	
T(G12)	TABLEMi identification number for shear modulus in plane 1-2.	Integer ≥ 0 or blank	
T(G23)	TABLEMi identification number for shear modulus in plane 2-3.	Integer ≥ 0 or blank	
T(G31)	TABLEMi identification number for shear modulus in plane 3-1.	Integer ≥ 0 or blank	
T(Ai)	TABLEMi identification number for thermal expansion coefficient in i-direction.	Integer ≥ 0 or blank	
T(GE)	TABLEMi identification number for damping coefficient.	Integer ≥ 0 or blank	
T(Xt), T(Xc)	TABLEMi identification number for allowable stresses or strains in tension and compression, respectively, in the longitudinal direction.	Integer ≥ 0 or blank	
T(Yt), T(Yc)	TABLEMi identification number for allowable stresses or strains in tension and compression, respectively, in the lateral direction.	Integer ≥ 0 or blank	
T(Zt), T(Zc)	TABLEMi identification number for allowable stresses or strains in tension and compression, respectively, in the thickness direction.	Integer ≥ 0 or blank	
T(S12)	TABLEMi identification number for allowable shear stress or strain for plane 1-2.	Integer ≥ 0 or blank	
T(S23)	TABLEMi identification number for allowable shear stress or strain for plane 2-3.	Integer ≥ 0 or blank	
T(S31)	TABLEMi identification number for allowable shear stress or strain for plane 3-1.	Integer ≥ 0 or blank	
T(F12)	TABLEMi identification number for F12 interaction term in the tensor polynomial theory of Tsai-Wu.	Integer ≥ 0 or blank	
T(F23)	TABLEMi identification number for F23 interaction term in the tensor polynomial theory of Tsai-Wu.	Integer ≥ 0 or blank	
T(F31)	TABLEMi identification number F31 for interaction term in the tensor polynomial theory of Tsai-Wu.	Integer ≥ 0 or blank	

Remarks:

1. Temperature-dependent material properties are only calculated when a temperature distribution for materials is defined by using TEMPERATURE, TEMPERATURE(MATERIAL), or TEMPERATURE(BOTH) Case Control commands.
2. Fields 3, 4, etc., of this entry correspond, one-by-one, to fields 3, 4, etc., of the MAT12 entry referenced in field 2. The value in a particular field of the MAT12 entry is replaced or modified by the table referenced in the corresponding field of this entry. A blank or zero entry means no temperature dependence of that field on the MAT12 entry.
3. Any quantity modified by this entry must have a value on the MAT12 entry.
4. Table references must be present for each item that is temperature-dependent.

MATVE

Viscoelastic Material Property Definition

Description: Specifies viscoelastic material properties for use in nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	GFUNC	KFUNC	RHO	A				
	SHIFT	C1	C2	T0					

Example:

MATVE	5	101	102	0.1					

Field	Definition	Type	Default
MID	Unique material identification number or identification number of a MAT1 entry.	Integer > 0	Required
GFUNC	Identification number of a TABVE entry. The TABVE table contains a series of shear modulii and decay coefficients to represent the shear modulus relaxation function of the material.	Integer > 0 or blank	
KFUNC	Identification number of a TABVE entry. The TABVE table contains a series of bulk modulii and decay coefficients to represent the bulk modulus relaxation function of the material.	Integer > 0 or blank	
RHO	Mass density.	Real or blank	0.0
A	Thermal expansion coefficient.	Real or blank	0.0
SHIFT	Time-temperature superposition shift law, selected by one of the following values 1 = WLF (William Landel-Ferry) 2 = Arrhenius	Integer	WLF
C1, C2	Material constants used by the WLF or Arrhenius shift function.	Real	0.0
T0	Reference temperature used by the WLF or Arrhenius shift function.	Real ≠ 0.0 if SHIFT = 2	Required if SHIFT = 2

Remarks:

1. This entry will be activated the NLPARM entry is prepared for viscoelastic analysis.
2. If a MAT1 entry with the same MID is used, the E, G, and NU fields will be used to define defaults when GFUNC and/or KFUNC are blank.

3. Viscoelasticity uses the Generalized Maxwell Model. The deviatoric stress is given by

$$s_{n+1} = \gamma_\infty s_{n+1}^0 + \sum_{i=1}^{N} \gamma_i h_{n+1}^{(i)}$$

where the stress at each relaxation component is calculated by

$$h_{n+1}^{(i)} = \mathrm{EXP}^{\left(\frac{\Delta t_n}{\tau_i}\right)} h_n^{(i)} + \mathrm{EXP}^{\left(\frac{\Delta t_n}{2\tau_i}\right)} (s_{n+1}^0 - s_n^0)$$

and s_n^0 and s_{n+1}^0 are deviatoric stresses without relaxation.

The viscoelastic relaxation occurs in the shear and/or bulk modulus. The modulus *E* in the following figure should be interpreted either shear modulus *G* or bulk modulus *K*.

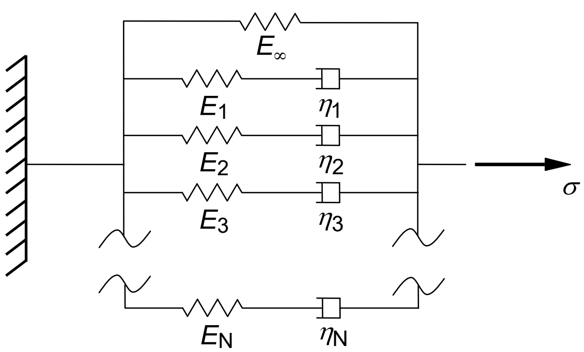


Figure 1. Viscoelastic Material Idealization.

where

E_∞ = stiffness at an infinite time

$E_0 = E_\infty + \sum E_i$ stiffness at the initial time

$\tau_i = \dfrac{\eta_i}{E_i}$ $\{i = 1, 2, 3, \ldots N\}$

$\gamma_i = \dfrac{E_i}{E_0}$ $\{i = 1, 2, 3, \ldots N\}$

$\gamma_\infty = \dfrac{E_\infty}{E_0}$ $0 \le \gamma_i < 1$

The γ_i and τ_i terms are defined using the TABVE Bulk Data entry where $0 \le N \le 120$.

4. Viscoelastic material properties strongly depend on temperature. Instead of estimating material properties at different temperatures, an assumption called thermorheological simplicity is used in which the relaxation curve at high temperature is identical to that at low temperature when the time is properly scaled. The relaxation times in the Prony series are scaled by the following equation:

$$\tau_i(T) = \frac{\tau_i(T_0)}{A(T, T_{0s})}$$

where two different scaling functions are supported.

William-Landel-Ferry:

$$\text{LOG}_{10}\left(A(T,T_0)\right) = -\frac{c_1(T - T_0)}{c_2 + T - T_0}$$

Arrhenius:

$$\text{LOG}_{10}\left(A(T,T_0)\right)A(T,T_0) = \begin{cases} c_1\left(\frac{1}{T} - \frac{1}{T_0}\right) & \text{if } T \geq T_0 \\ c_2\left(\frac{1}{T} - \frac{1}{T_0}\right) & \text{if } T < T_0 \end{cases}$$

MATXM Autodesk AME Material

Description: Specifies constants required to invoke Autodesk Advanced Material Exchange (AME).

Format:

1	2	3	4	5	6	7	8	9	10
MATXM	MID	MIDH							

Example:

MATXM	35	5							

Field	Definition	Type	Default
MID	Material identification number.	Integer	
MIDH	Identification number of the Autodesk Advanced Material Exchange material.	Integer	

Remarks:

1. When used with CHEXA elements, you must turn off internal nodes and reduced integration:
   - PARAM, HEXINODE, OFF
   - PARAM, HEXREDORD, OFF
2. When used with CTETRA elements, you must turn off reduced integration:
   - PARAM, TETREDORD, OFF
3. PSOLID must refer to the global material coordinate system. MCID = 0.
4. Make sure the material ID (MATID) of the PSOLID entry refers to the MATXM material.

MFLUID Fluid Volume Properties

Description: 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			
	RMAX								

Example:

MFLUID	53	12	5.8	1004.0	3				
	100.0								

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
CID	Identification number of rectangular coordinate system used to specify the orientation of the free surface (normal to the coordinate z-axis).	Integer ≥ 0 or blank	
ZFS	Intercept of the free surface on the z-axis of the coordinate system referenced by CID. See Remark 4.	Real	∞
RHO	Fluid density.	Real	Required
ELIST1	Identification number of an ELIST entry that lists the identification numbers 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 included. See Remarks 3, 4, and 5.	Integer ≥ 0 or blank	Required if ELIST2 is blank
ELIST2	Identification number of an ELIST entry that lists the identification numbers 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 included. See Remarks 3, 4, and 5.	Integer ≥ 0 or blank	Required if ELIST1 is blank
RMAX	Maximum element interaction distance. Interactions between elements with distance that is greater than RMAX will be ignored.	Real > 0.0	1.0E+10

Remarks:

5. The MFLUID entry must be selected with the Case Control command MFLUID = SID.
6. Several MFLUID entries corresponding to different fluid volumes can be used simultaneously.

7. The wetted side of an element in ELIST1 is determined by the presence or minus sign preceding the element 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.
8. The fluid volume may be finite (interior) or infinite (exterior) and may be bounded by an optional free surface defined by ZFS. The default free surface is located at an infinitely large positive ZFS value.
9. The ELIST entry may only reference CQUAD4/CQUADR and CTRIA3/CTRIAR elements.
10. The handling of special cases where adjacent element surfaces normals are more than 30 degrees from each other such as a corner is controlled using PARAM, VFMNORMTOL (see Section 5, *Parameters*, for more information on VFMNORMTOL).
11. PARAM, VFMADDMETHOD controls where in the solution sequence the virtual fluid mass is included in the global mass matrix (see Section 5, *Parameters*, for more information on VFMADDMETHOD).

MILLDIR Milling Direction List

Description: Defines a list of milling directions for 5-axis milling manufacturing constraints in design optimization analysis.

Format:

1	2	3	4	5	6	7	8	9	10
MILLDIR	LID	D1	D2	D3	D4	D5	D6	D7	
	E8	E9	E10	- etc.-					

Example:

MILLDIR	12	F1	F3	F4	F6	E12	E34	E41	
	E15	E26							

Field	Definition	Type	Default
LID	List identification number.	Integer > 0	Required
Di	Direction symbols. See Remarks 1 and 2.	Character	Required

Remarks:

1. At least one direction symbol is required. Specifying all 26 is equivalent to the default setting for 5-axis milling when a milling direction list is not specified on the TOPVAR Bulk Data entry.
2. The table below lists all 26 possible milling directions. C symbols are for corner directions, F symbols are for face directions, and E symbols are for edge directions. The cube shown in Figure 1 represents the voxelized model with a bounding box who’s corners numbered from 1 – 8. The milling directions are in the manufacturing coordinate system defined on the TOPVAR Bulk Data entry.

Direction Symbol	X-Component	Y-Component	Z-Component
C1	1.0	1.0	1.0
C2	-1.0	1.0	1.0
C3	-1.0	-1.0	1.0
C4	1.0	-1.0	1.0
C5	1.0	1.0	-1.0
C6	-1.0	1.0	-1.0
C7	-1.0	-1.0	-1.0
C8	1.0	-1.0	-1.0
F1	1.0	0.0	0.0
F2	0.0	-1.0	0.0
F3	0.0	0.0	-1.0
F4	-1.0	0.0	0.0
F5	0.0	1.0	0.0
F6	0.0	0.0	1.0
E12	0.0	1.0	1.0
E23	-1.0	0.0	1.0
E34	0.0	-1.0	1.0
E41	1.0	0.0	1.0
E56	0.0	1.0	-1.0
E67	-1.0	0.0	-1.0
E78	0.0	-1.0	-1.0
E85	1.0	0.0	-1.0
E15	1.0	1.0	0.0
E26	-1.0	1.0	0.0
E37	-1.0	-1.0	0.0
E48	1.0	-1.0	0.0

(Continued)

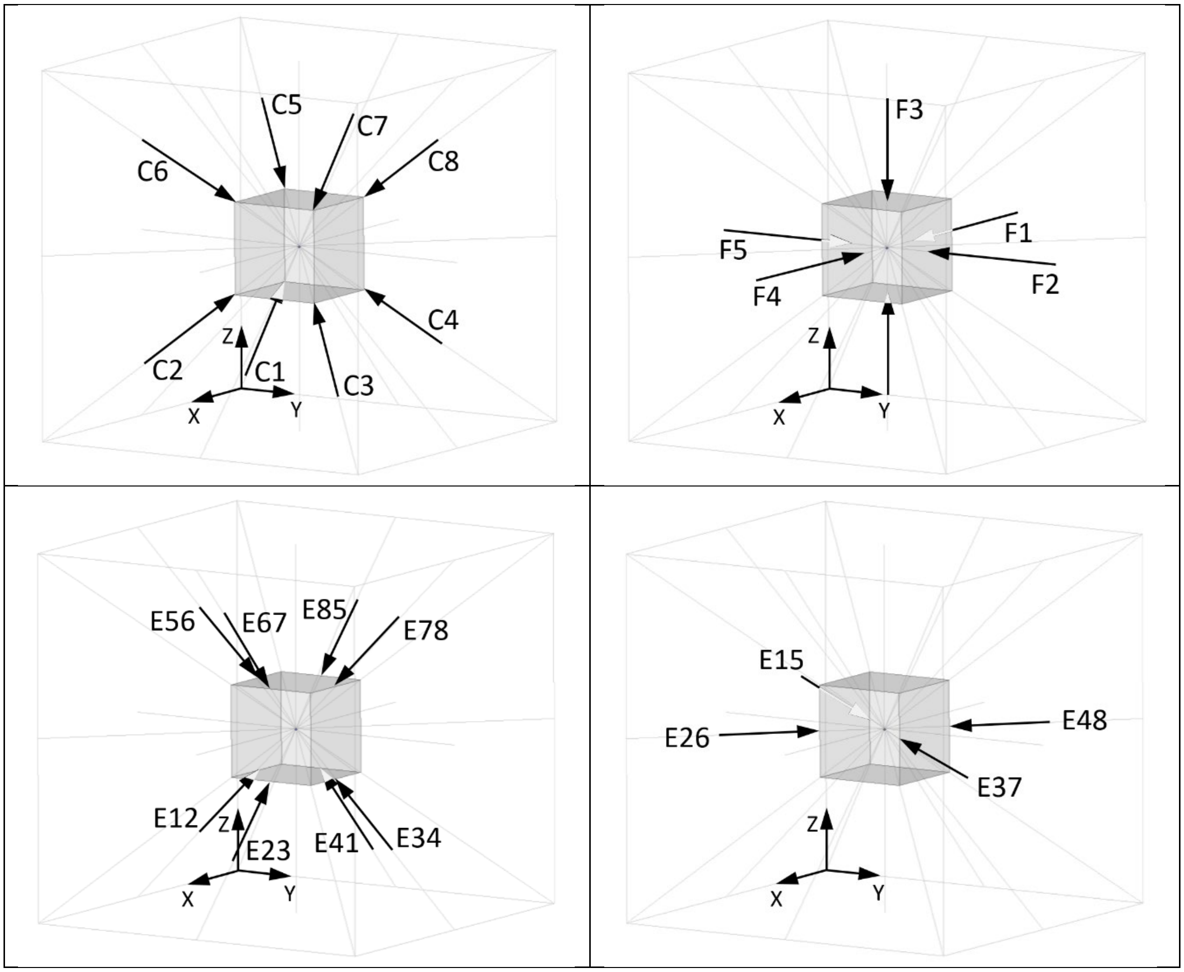


Figure 1. Milling Direction Symbol Definition.

(Continued)

MOMENT Static Moment

Description: Defines a static moment at a grid point by specifying a vector.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		

Example:

MOMENT	3	441	4	10.0	1.0	-1.0	0.0		

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
G	Grid point identification number.	Integer > 0	Required
CID	Coordinate system identification number.	Integer ≥ 0 or blank	0
M	Moment vector scale factor.	Real	Required
N1, N2, N3	Moment vector components measured in coordinate system defined by CID.	Real	Required; must have at least one nonzero component

Remarks:

1. The static load applied to grid point G is given by

$$\vec{m} = M\vec{N}$$

where $\vec{N}$ is the vector defined in fields 6, 7 and 8.

2. Load sets must be selected in the Case Control Section (LOAD = SID).
3. A CID of zero references the basic coordinate system.

MOMENT1 Static Moment, Alternate Form 1

Description: Defines a static moment at a grid point by specification of a value 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	3	141	-4.5	10	11				

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
G	Grid point identification number.	Integer > 0	Required
M	Moment magnitude.	Real	Required
G1, G2	Grid point identification numbers.	Integer > 0; G1 = G2	Required

Remarks:

1. The static load applied to grid point G is given by

$$\bar{m} = M\bar{n}$$

where $\bar{n}$ is a unit vector parallel to a vector for G1 to G2.

2. Load sets must be selected in the Case Control Section (LOAD = SID).

MPC — Multi Point Constraint

Description: Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

where u_j represents global degree of freedom Cj at grid 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	6	77	2	5.5	2	4	4.5		
		5	5	-2.91					

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
Gj	Grid point identification number.	Integer > 0	Required
Cj	Component number of global coordinate (any one of six unique digits may be placed in the field).	0 ≤ Integer ≤ 6	Required
Aj	Coefficient.	Real or blank	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. By default, a dependent degree of freedom assigned by one MPC entry cannot be assigned dependent by another MPC entry or rigid element and cannot be additionally constrained (e.g., single-point constraint). If this behavior is desired use PARAM, AUTOFIXRIGIDSPC which when set to ON will allow the constraint of dependent degrees of freedom (See Section 5, *Parameters*, for more information on AUTOFIXRIGIDSPC.)
3. Forces of multipoint constraint may be recovered with the MPCFORCE Case Control command.

MPCADD — Multipoint Constraint Set Combination

Description: 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	5	8	12	7	23			

Field	Definition	Type	Default
SID	Identification number of multipoint constraint set.	Integer > 0	Required
Sj	Identification numbers multipoint constraint sets defined via MPC entries.	Integer > 0	Required

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.

NITINOL — Nitinol Material Property Definition

Description: Defines material properties for use in shape memory alloys (Nitinol).

Format:

1	2	3	4	5	6	7	8	9	10
NITINOL	MID	ALPHA	ELMAX	CAS	CSA	TSAS	TFAS	TSSA	
	TFSA	BTAS	BTSA	SSAS	SFAS	SSSA	SFSA		

Example:

NITINOL	101	0.0	0.1	1.0	1.0	70.0	10.0	90.0	
	130.0	10.0	10.0	120.0	140.0	70.0	30.0		

Field	Definition	Type	Default
MID	Identification number of a MAT1 entry.	Integer > 0	Required
ALPHA	Pressure coefficient.	Real ≥ 0	0.1
ELMAX	Maximum residual strain.	Real ≥ 0	0.1
CAS	Conversion constant from austenite to martensite.	Real ≥ 0	1.0
CSA	Conversion constant from martensite to austenite.	Real ≥ 0	1.0
TSAS	Starting temperature of transformation from austenite to martensite.	Real	0.0 See Remark 1
TFAS	Ending temperature of transformation from austenite to martensite.	Real	0.0 See Remark 1
TSSA	Starting temperature of transformation from martensite to austenite.	Real	0.0 See Remark 1
TFSA	Ending temperature of transformation from martensite to austenite.	Real	0.0 See Remark 1
BTAS	Constant for exponential flow rule (austenite to martensite).	Real ≥ 0	0.0 See Remark 2
BTSA	Constant for exponential flow rule (martensite to austenite).	Real ≥ 0	0.0 See Remark 2
SSAS	Starting stress for transformation from austenite to martensite at reference temperature.	Real	0.0 See Remark 3
SFAS	Ending stress for transformation from austenite to martensite at reference temperature.	Real	0.0 See Remark 3

Field	Definition	Type	Default
SSSA	Starting stress for transformation from martensite to austenite at reference temperature.	Real	0.0 See Remark 3
SFSA	Ending stress for transformation from martensite to austenite at reference temperature.	Real	0.0 See Remark 3

Remarks:

1. The following relations must be satisfied between the four temperatures:

   TFAS < TSAS < TSSA < TFSA
2. When BTAS and BTSA are zero the material model is linear. When BTAS and BTSA are non-zero the material model is exponential with BTAS and BTSA as coefficients.
3. The transformation stresses and temperature can be combined such that the transformation stress can be calculated by
   - Starting stress for transformation from austenite to martensite = SSAS - CAS∗TSAS
   - Ending stress for transformation from austenite to martensite = SFAS - CAS∗TFAS
   - Starting stress for transformation from martensite to austenite = SSSA - CSA∗TSSA
   - Ending stress for transformation from martensite to austenite = SFSA - CSA∗TFSA

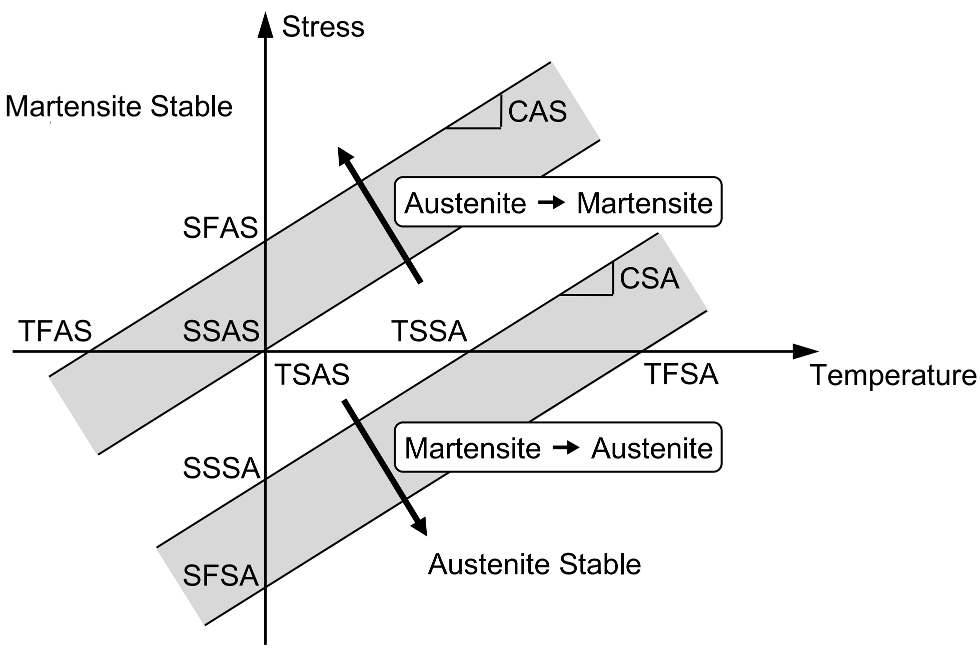


Figure 1. Stress-Temperature Transformation Variables.

NLPARM — Parameters for Nonlinear Static Analysis Control

Description: Defines a set of parameters for nonlinear static analysis.

Format:

1	2	3	4	5	6	7	8	9	10
NLPARM	ID	NINC	DT	KMETHOD	KSTEP	MAXITER	CONV	INTOUT	
	EPSU	EPSP	EPSW	MAXDIV	MAXUBIS	MAXLS	FSTRESS	LSTOL	
	MAXBIS	TDG	TDC	TDV	MAXR		RTOLB		
	INITINC	MININC	MAXINC	TTOTAL					

Example:

NLPARM	25	10					PW	YES	

Field	Definition	Type	Default
ID	Identification number.	Integer > 0	Required
NINC	Number of increments. See Remark 2.	Integer > 0	See Remark 2
DT	Incremental time interval for creep analysis. See Remark 3.	Real ≥ 0	0.0
KMETHOD	Method for controlling stiffness updates, one of the following character variables: AUTO, ITER, or SEMI. See Remark 4.	Character	AUTO
KSTEP	Number of iterations before stiffness update for the ITER method. See Remark 5.	Integer > 0	5
MAXITER	Limit on number of iterations for each load increment. See Remark 6.	Integer > 0 or AUTO	AUTO
CONV	Convergence criteria, one of the following character variables: U, P, or W, or any combination. See Remark 7.	Character	PW
INTOUT	Intermediate output request, one of the following character variables: YES, NO, or ALL or the load increment interval for output. See Remark 8.	Character or Integer > 0	NO
EPSU	Error tolerance for displacement (U) criterion.	Real > 0.0	See Remark 17
EPSP	Error tolerance for load (P) criterion.	Real > 0.0	See Remark 17
EPSW	Error tolerance for work (W) criterion.	Real > 0.0	See Remark 17
MAXDIV	Limit on probable divergence conditions per iteration before the solution is assumed to diverge. See Remark 9.	Integer > 0	3

Field	Definition	Type	Default
MAXUBIS	Maximum number of iterations for an upward load increment adjustment. Applicable when the load increment is bisected or the adaptive load increment/convergence method is used. See Remark 16.	Integer > 0	See Remark 16
MAXLS	Maximum number of line searches for each iteration. See Remark 10.	Integer ≥ 0	5
FSTRESS	Fraction of effective stress ($\bar{\sigma}$) used to limit the subincrement size in nonlinear material routines. See Remark 11.	0.0 < Real < 1.0	0.2
LSTOL	Line search tolerance. See Remark 10.	0.01 < Real ≤ 0.9	0.2
MAXBIS	Maximum number of bisections allowed for each load increment. See Remark 12.	Integer > 0	5
TDG	Terminate on displacement grid point identification number. See Remark 13.	Integer > 0	
TDC	Terminate on displacement component number. See Remark 13. MAXT Resultant of translation displacement components. MAXR Resultant of rotational displacement components.	0 ≤ Integer ≤ 6 or MAXT or MAXR	MAXT
TDV	Terminate on displacement value. See Remark 13.	Real	
MAXR	Maximum ratio for the adjusted arc-length increment relative to the initial value. See Remark 14.	1.0 ≤ Real ≤ 40.0	20.0
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 15.	Real > 0.0	20.0
INITINC	Initial load increment. See Remarks 2 and 16.	0.0 < Real < 1.0	1/NINC
MININC	Minimum load increment. See Remarks 2 and 16.	0.0 < Real < 1.0	INITINC
MAXINC	Maximum load increment. See Remarks 2 and 16.	0.0 < Real < 1.0	INITINC
TTOTAL	Total time for creep analysis. See Remark 3.	Real ≥ 0	0.0

Remarks:

1. The NLPARM entry must be selected with the Case Control command NLPARM = ID. Each solution subcase requires an NLPARM command.
2. In cases of static analysis (DT = 0.0) 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 creep analysis (DT > 0.0), NINC is the number of time step increments. When NINC is blank, the adaptive load increment/convergence method is used with INITINC, MININC, and MAXINC set to the following values:

Variable	Value
INITINC	1.0E-2
MININC	1.0E-3
MAXINC	0.3

3. The units for DT and TTOTAL must be consistent with the units used on the CREEP entry that defines the creep characteristics. TTOTAL specifies the total creep time for the subcase. When the fixed load increment/convergence method is used and TTOTAL is blank, DT is multiplied by NINC to determine total creep time for the subcase. When the adaptive load increment/convergence method is used and TTOTAL is blank, DT is multiplied by INITINC to determine total creep time for the subcase.
4. The stiffness update strategy is selected in the KMETHOD field.
   a) If the AUTO option is specified, 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) the estimated number of iterations to converge exceeds MAXITER or (ii) the solution diverges. See Remarks 7 and 9 for diverging solutions.
   b) 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 options.
   c) If the ITER option is selected, the program updates the stiffness matrix at every KSTEP iterations and on convergence if KSTEP ≤ MAXITER. However, if KSTEP > MAXITER, the 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, one of two actions is taken depending on the BISECT model parameter. If the BISECT model parameter is set to ON, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e. MAXBIS is attained), execution terminates with a fatal error. If the BISECT model parameter is set to OFF, the analysis is continued to the next load increment. (See Section 5, *Parameters*, for more information on BISECT.) The default AUTO setting uses an initial MAXITER value of 40 and automatically increases this value if the solution appears near convergence.
7. The symbols (U for displacement error, P for load equilibrium error, and W for 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.
8. INTOUT controls the output requests for displacements, element forces and stresses, etc. YES, ALL, or the load increment interval for output must be specified in order to output intermediate (incremental) results.

INTOUT	Output Processed
YES	For every computed load increment excluding bisected and quadsected load increments
NO	For the last load of the subcase
ALL	For every computed load increment including bisected and quadsected load increments
n	For computed load increments *n*, 2∗*n*, 3∗*n*,…, and the last converged increment

- For the Newton-Raphson iteration method (i.e., when no NLPCI Bulk Data entry 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 Bulk Data entry 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 iterations (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 ≥ MAXDIV. If the solution diverges and the load increment cannot be further bisected (i.e., MAXBIS is attained), execution terminates with a fatal error.

10. The line search is performed as required if MAXLS > 0. The line search procedure scales the displacement increment to minimize the energy error. The procedure is skipped if the absolute value of the relative energy error is less than the value specified by LSTOL.

11. The number of subincrements in the material routines is determined so that the subincrement size is approximately FSTRESS $*\ \bar{\sigma}$ (equivalent stress).

12. The number of bisections for a load increment is limited to MAXBIS. If the solution diverges, the stiffness is updated on the first divergence and the load is bisected on the second divergence.

13. When TDG, TDC, and TDV are specified the solution will proceed until either the entire load is applied or the specified displacement value (TDV) at grid point TDG in direction TDC is reached or exceeded. Displacements are in the displacement coordinate system of the TDG grid point.

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 using the relation:

$$\frac{1}{\text{MAXR}} \leq \frac{\Delta\ell_n}{\Delta\ell_o} \leq \text{MAXR}$$

where $\Delta\ell_n$ is the arc-length at step *n* and $\Delta\ell_o$ 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 load increment is bisected 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. INITINC, MININC, and MAXINC are used in the adaptive load increment/convergence method to define the overall upper and lower bounds on the load increment in the subcase. INITINC specifies the initial load increment and replaces the value determined using NINC. When MININC < INITINC < MAXINC, the load increment is adjusted up or down based on convergence and solution stability. MAXUBIS defines the maximum number of iterations for the load increment to be adjusted upward or downward. If the number of iterations in an increment is below this value the load increment is doubled and if greater than twice this value the load increment is halved. INITINC, MININC, and MAXINC are not applicable when arc-length methods are specified via the NLPCI Bulk Data entry. When adaptive loading is not used MAXUBIS defines the maximum number of iterations for the load increment to be adjusted upward during bisection.
17. Default tolerance sets are determined based on solution type, nonlinear behavior requested, and desired accuracy. Accuracy is under user control and can be specified using PARAM, NLTOL (see Section 5, *Parameters*, for more information on NLTOL). The NLTOL values are only used if one or more of the EPSU, EPSP and EPSW fields on the NLPARM entry are blank. The following tables show the tolerance values used depending on the NLTOL parameter setting specified.

Nonlinear Static Analysis without Contact and Material Nonlinearity

NLTOL	Level of Accuracy	EPSU	EPSP	EPSW
0	Very High	1.0E-3	1.0E-3	1.0E-6
1	High	1.0E-2	1.0E-2	1.0E-4
2	Engineering	1.0E-2	1.0E-2	1.0E-3
3	Preliminary Design	1.0E-1	1.0E-1	1.0E-2
Default	Engineering	1.0E-2	1.0E-2	1.0E-3

Nonlinear Static Analysis with Material Nonlinearity

NLTOL	Level of Accuracy	EPSU	EPSP	EPSW
0	Very High	1.0E-4	1.0E-4	1.0E-8
1	High	5.0E-4	5.0E-4	1.0E-8
2	Engineering	5.0E-4	5.0E-4	1.0E-7
3	Preliminary Design	1.0E-3	1.0E-3	1.0E-6
Default	Engineering	5.0E-4	5.0E-4	1.0E-7

Nonlinear Static Analysis with Contact

NLTOL	Level of Accuracy	EPSU	EPSP	EPSW
0	Very High	1.0E-3	1.0E-3	1.0E-6
1	High	1.0E-3	1.0E-3	1.0E-5
2	Engineering	5.0E-3	5.0E-3	1.0E-4
3	Preliminary Design	5.0E-3	5.0E-3	1.0E-4
Default	Engineering	5.0E-3	5.0E-3	1.0E-4

Nonlinear Steady State Heat Transfer

NLTOL	Level of Accuracy	EPSU	EPSP	EPSW
0	Very High	1.0E-3	1.0E-3	1.0E-6
1	High	1.0E-3	1.0E-3	1.0E-6
2	Engineering	1.0E-3	1.0E-3	1.0E-6
3	Preliminary Design	1.0E-3	1.0E-3	1.0E-6
Default	Engineering	1.0E-3	1.0E-3	1.0E-6

NLPCI — Parameters for Arc-Length Methods in Nonlinear Static Analysis

Description: Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis.

Format:

1	2	3	4	5	6	7	8	9	10
NLPCI	ID	TYPE	MINALR	MAXALR	SCALE	ALRITER	DESITER	MAXINC	
	ALROPT								

Example:

NLPCI	20	CRIS	1.0	1.0			9	20	

Option	Definition	Type	Default
ID	Identification number that matches an associated NLPARM entry.	Integer > 0	
TYPE	Constraint type. One of the following characters variables: CRIS, RIKS, or MRIKS. See Remark 2.	Character	CRIS
MINALR	Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3 and 4.	0.0 < Real ≤ 1.0	0.25
MAXALR	Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3 and 4.	Real ≥ 1.0	4.0
SCALE	Scale factor (w) for controlling the loading contribution in the arc-length constraint.	Real ≥ 0.0	0.0
ALRITER	Allowable arc-length adjustment ratio between iterations. See Remark 5.	Real ≥ 0	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	12
MAXINC	Maximum number of controlled increment steps allowed within a subcase. See Remark 6.	Integer > 0	40
ALROPT	Arc-length adjustment ratio method. One of the following characters variables: KRATIO, ITER, or BOTH. See Remark 7.	Character	BOTH

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. The NLPCI entry is not supported in creep analysis or heat transfer solutions.

2. The available constraint types are as follows:

   TYPE = CRIS:

$$\left\{u_n^i - u_n^0\right\}\left\{u_n^i - u_n^0\right\} + w^2\left(\mu^i - \mu^0\right)^2 = \Delta l_n^2$$

   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:

$$\left\{u_n^i - u_n^{i-1}\right\}^T \left\{u_n^{i-1} - u_n^0\right\} + w^2 \Delta\mu^i \left(\mu^{i-1} - \mu^0\right) = 0$$

   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

$$\text{MINALR} \le \frac{\Delta l_{new}}{\Delta l_{old}} \le \text{MAXALR}$$

   The arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence) and/or 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:

$$\Delta l_{new} = \sqrt{\frac{\text{DESITER} \;\; \Delta l_{old}}{I_{max}}}$$

5. The ALRITER field is used to limit the adjustment of the arc-length from one iteration to the next using

$$\frac{\Delta l_{old}}{\text{ALRITER}} \le \Delta l_{new} \le \Delta l_{old} * \text{ALRITER}$$

   The default ALRITER value of zero disables limiting the arc-length adjustment during iterations.

6. The MAXINC field is used to limit the number of controlled increment steps in case the solution never reaches the specified load. The default is the number of increments, NINC, specified on the corresponding NLPARM entry or 40 which ever is greater. This field is useful in limiting the number of increments computed for a collapse analysis.

7. When ALROPT is set to ITER, arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence). When ALROPT is set to KRATIO, adjustment is based on the change in stiffness. The default BOTH setting will consider both parameters.

NOLIN1 — Nonlinear Transient Load as a Tabular Function

Description: Defines nonlinear transient forcing functions of the form

Function of displacement: $P_i(t) = ST(u_j(t))$

Function of velocity: $P_i(t) = ST(\dot{u}_j(t))$

where $u_j(t)$ and $\dot{u}_j(t)$ are the displacement and velocity at point GJ in the direction CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN1	SID	GI	CI	S	GJ	CJ	TID		

Example:

NOLIN1	5	10	4	6.3	3	11	5		

Field	Definition	Type	Default
SID	Nonlinear load set identification number.	Integer > 0	Required
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied.	Integer > 0	Required
CI	Component number for GI.	0 ≤ Integer ≤ 6	Required
S	Scale factor.	Real	Required
GJ	Grid, scalar, or extra point identification number.	Integer > 0	Required
CJ	Component number for GJ.	0 ≤ Integer ≤ 6	Required
TID	Identification number of a TABLEDi entry.	Integer > 0	Required

Remarks:

1. Nonlinear loads must be selected with the Case Control Section (NONLINEAR = SID).
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. Nonlinear loads may be a function of displacement ($X = u$) or velocity ($X = \dot{u}$). Nonlinear loads as a function of velocity (equation 2 above) are denoted by component numbers ten times greater than the actual component number. For example, a component number of 11 is component 1 for velocity.

NOLIN2 Nonlinear Transient Load as the Product of Two Variables

Description: Defines nonlinear transient forcing functions of the form

$$P_i(t) = SX_j(t)X_k(t)$$

where $X_j(t)$ and $X_k(t)$ are the displacement and velocity at point GJ and GK in the direction of CJ and CK.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN2	SID	GI	CI	S	GJ	CJ	GK	GK	

Example:

NOLIN2	14	2	1	2.8	2	1	2		

Field	Definition	Type	Default
SID	Nonlinear load set identification number.	Integer > 0	Required
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied.	Integer > 0	Required
CI	Component number for GI.	0 ≤ Integer ≤ 6	Required
S	Scale factor.	Real	Required
GJ, GK	Grid, scalar, or extra point identification number.	Integer > 0	Required
CJ, CK	Component number for GJ, GK.	0 ≤ Integer ≤ 6	Required

Remarks:

1. Nonlinear loads must be selected with the Case Control Section (NONLINEAR = SID).
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. GI – CI, GJ – CJ, and CK – CK may be the same point.
4. Nonlinear loads may be a function of displacement ($X = u$) or velocity ($X = \dot{u}$). Nonlinear loads as a function of velocity (equation 2 above) are denoted by component numbers ten times greater than the actual component number. For example, a component number of 11 is component 1 for velocity.

NOLIN3 Nonlinear Transient Load as a Positive Variable Raised to a Power

Description: Defines nonlinear transient forcing functions of the form

$$P_i(t) = \begin{cases} S[X_j(t)]^A & ,X_j(t) > 0 \\ 0 & ,X_j(t) \le 0 \end{cases}$$

where $X_j(t)$ may be the 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	5	102		-6.1	2	15	-3.5		

Field	Definition	Type	Default
SID	Nonlinear load set identification number.	Integer > 0	Required
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied.	Integer > 0	Required
CI	Component number for GI.	0 ≤ Integer ≤ 6	Required
S	Scale factor.	Real	Required
GJ	Grid, scalar, or extra point identification number.	Integer > 0	Required
CJ	Component number for GJ.	0 ≤ Integer ≤ 6	Required
A	Exponent of the forcing function.	Real	Required

Remarks:

1. Nonlinear loads must be selected with the Case Control Section (NONLINEAR = SID).
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. Nonlinear loads may be a function of displacement ($X = u$) or velocity ($X = \dot{u}$). Nonlinear loads as a function of velocity (equation 2 above) are denoted by component numbers ten times greater than the actual component number. For example, a component number of 11 is component 1 for velocity.
4. Use a NOLIN4 entry for the negative range of $X_j(t)$.

NOLIN4 Nonlinear Transient Load as a Negative Variable Raised to a Power

Description: Defines nonlinear transient forcing functions of the form

$$P_i(t) = \begin{cases} -S\left[-X_j(t)\right]^A & ,X_j(t) > 0 \\ 0 & ,X_j(t) \le 0 \end{cases}$$

where $X_j(t)$ may be the 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	5	102		-6.1	2	15	-3.5		

Field	Definition	Type	Default
SID	Nonlinear load set identification number.	Integer > 0	Required
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied.	Integer > 0	Required
CI	Component number for GI.	0 ≤ Integer ≤ 6	Required
S	Scale factor.	Real	Required
GJ	Grid, scalar, or extra point identification number.	Integer > 0	Required
CJ	Component number for GJ.	0 ≤ Integer ≤ 6	Required
A	Exponent of the forcing function.	Real	Required

Remarks:

1. Nonlinear loads must be selected with the Case Control Section (NONLINEAR = SID).
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. Nonlinear loads may be a function of displacement ($X = u$) or velocity ($X = \dot{u}$). Nonlinear loads as a function of velocity (equation 2 above) are denoted by component numbers ten times greater than the actual component number. For example, a component number of 11 is component 1 for velocity.
4. Use a NOLIN3 entry for the positive range of $X_j(t)$.

OMIT — Omitted Analysis Set Degrees of Freedom

Description: 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	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

OMIT	15	4	17	123	7	6			

Field	Definition	Type	Default
Gi	Grid point identification number(s).	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required

Remarks:

1. In some cases it may be more convenient to use OMIT1, ASET, or ASET1 entries.

OMIT1 — Omitted Analysis Set Degrees of Freedom, Alternate Form

Description: 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	456	2	3	7	10	18	14	11	
	19	23							

Alternate Format and Example:

OMIT1	C	G1	THRU	G2					

OMIT1	123	15	THRU	512					

Field	Definition	Type	Default
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required
Gi	Grid point identification number(s).	Integer > 0; G1 < G2	Required

Remarks:

1. If the alternate form is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will be skipped.

PARAM

Parameter

Description: Specifies values for parameters used in solution sequences.

Format:

1	2	3	4	5	6	7	8	9	10
PARAM	N	V							

Example:

PARAM	EPZERO	1.-5							

Field	Definition	Type
N	Parameter name.	Character
V	Parameter value.	Integer, real, or character

Remarks:

1. Only parameters for which assigned values are allowed may be given values via the PARAM entry.
2. See Section 5, *Parameters*, for a list of parameter definitions.

PBAR — Bar Element Property

Description: Defines the properties of bar elements (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	C	F0				

Example:

PBAR	44	100	0.1	2.-3	0.12	1.-4			
	0.1	0.2	-0.1	-0.2					

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 2.	Integer > 0	Required
A	Area of bar cross-section.	Real	Required
I1, I2, I12	Area moments of inertia. ($I1 \geq 0.0$, $I2 \geq 0.0$, $I1*I2 > I12^2$)	Real or blank	0.0
J	Torsional constant.	Real or blank	0.0
NSM	Nonstructural mass per unit length.	Real or blank	0.0
Ci, Di, Ei, Fi	Stress recovery coefficients.	Real or blank	0.0
K1, K2	Area factors for shear.	Real or blank	See Remark 4
C	Coefficient to determine torsional stress.	Real or blank	See Remark 6
F0	Preload.	Real or blank	0.0

Remarks:

1. PBAR entries must all have unique property identification numbers.
2. For structural problems, PBAR entries may only reference MAT1 material entries.
3. See CBAR entry for a depiction of bar element geometry.
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 are infinite; in other words, the transverse shear flexibilities are set equal to zero. K1 and K2 are ignored if $I12 > 0.0$.
5. The stress recovery coefficients C1, 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.

6. A single von Mises stress value is determined is based on the maximum combined axial and bending stress, the transverse shear stress, and the torsional stress using

$$\tau_v = \left[\sigma_x^2 + 3\left(\tau_{xy}^2 + \tau_{xz}^2 + \tau^2\right)\right]^{\frac{1}{2}}$$

where the transverse shear stress is determined using

$$\tau_{\text{xy}} = \frac{V_y}{K_y A} \qquad \tau_{\text{xz}} = \frac{V_z}{K_z A}$$

and V_y and V_z are the element transverse shear forces and $K_y A = \text{K1} * \text{A}$ and $K_z A = \text{K2} * \text{A}$. The torsional stress is determined using

$$\tau = \frac{TC}{J}$$

where T is the torsional moment. The torsional stress coefficient, C, should be selected as the maximum wall thickness for open sections and the radius for circular sections.

PBARL Simple Beam Cross-Section Property

Description: 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		TYPE				F0	
	DIM1	DIM2	DIM3	DIM4	DIM5	DIM6	DIM7	DIM8	
	DIM9	-etc.-	NSM						

Example:

PBARL	40	5		BOX					
	0.9	0.7	0.1	0.05					

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 2.	Integer > 0	Required
TYPE	Cross-section type. Must be one of following character variables: BAR, BOX, BOX1, CHAN, CHAN1, CHAN2, CROSS, H, HAT, HEXA, I, I1, ROD, T, T1, T2, TUBE, or Z. See Remark 4.	Character	Required
F0	Preload.	Real or blank	0.0
DIMi	Cross-sectional dimensions.	Real > 0.0	Required
NSM	Nonstructural mass per unit length.	Real or blank	0.0

Remarks:

1. PID must be unique with respect to all other PBAR and PBARL property identification numbers.
2. For structural problems, PBARL entries must reference a MAT1 material entry.
3. A function of this entry is to derive equivalent an equivalent internal PBAR entry. This equivalent entry is given in the database definition section of the Model Results Output File and in the translated Bulk Data Output File.
4. 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 1. The origin of element coordinate system is centered at the shear center of the cross-section oriented as shown.

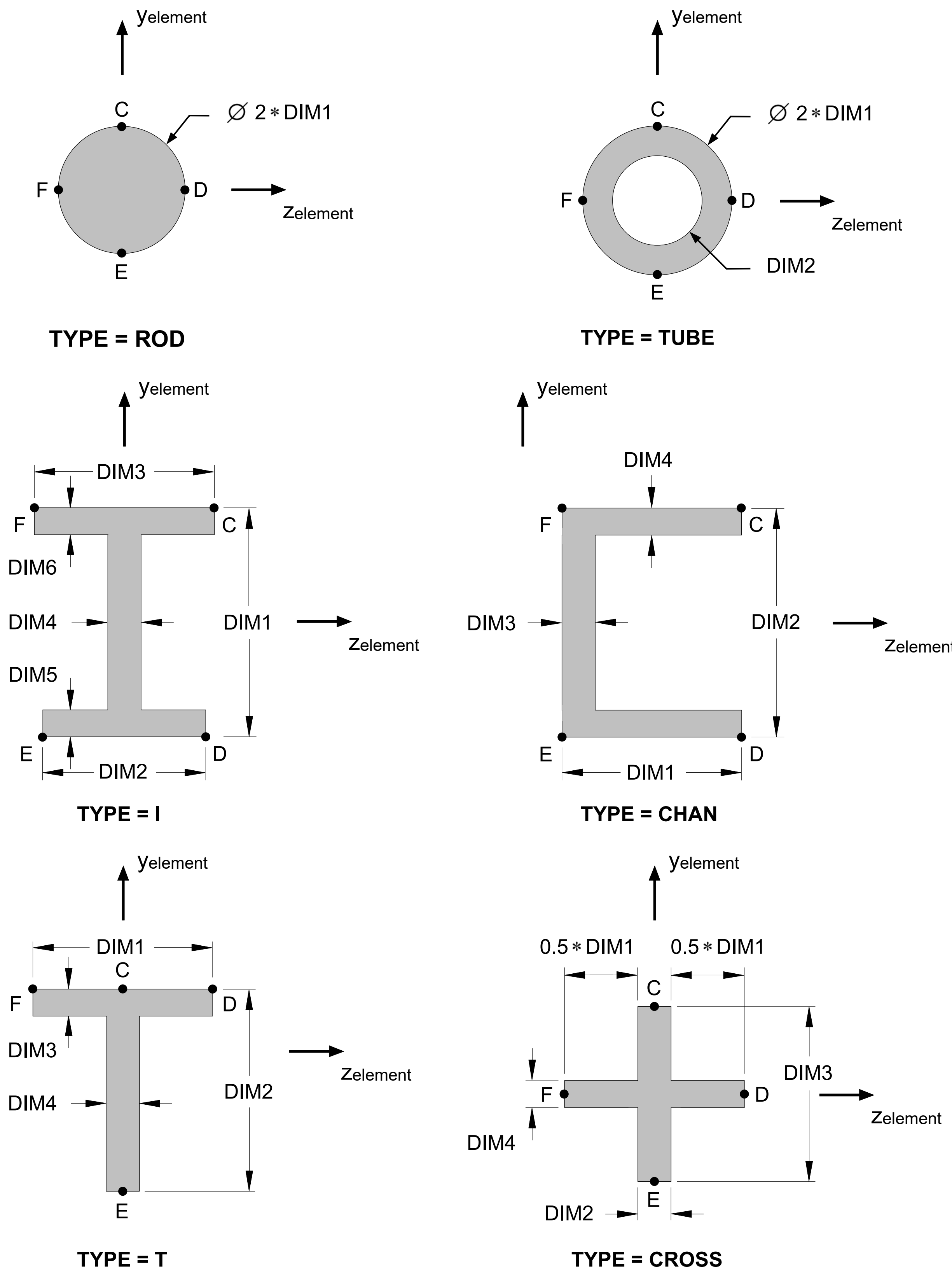


Figure 1a. Definition of Cross-Section Geometry and Stress Recovery Points.

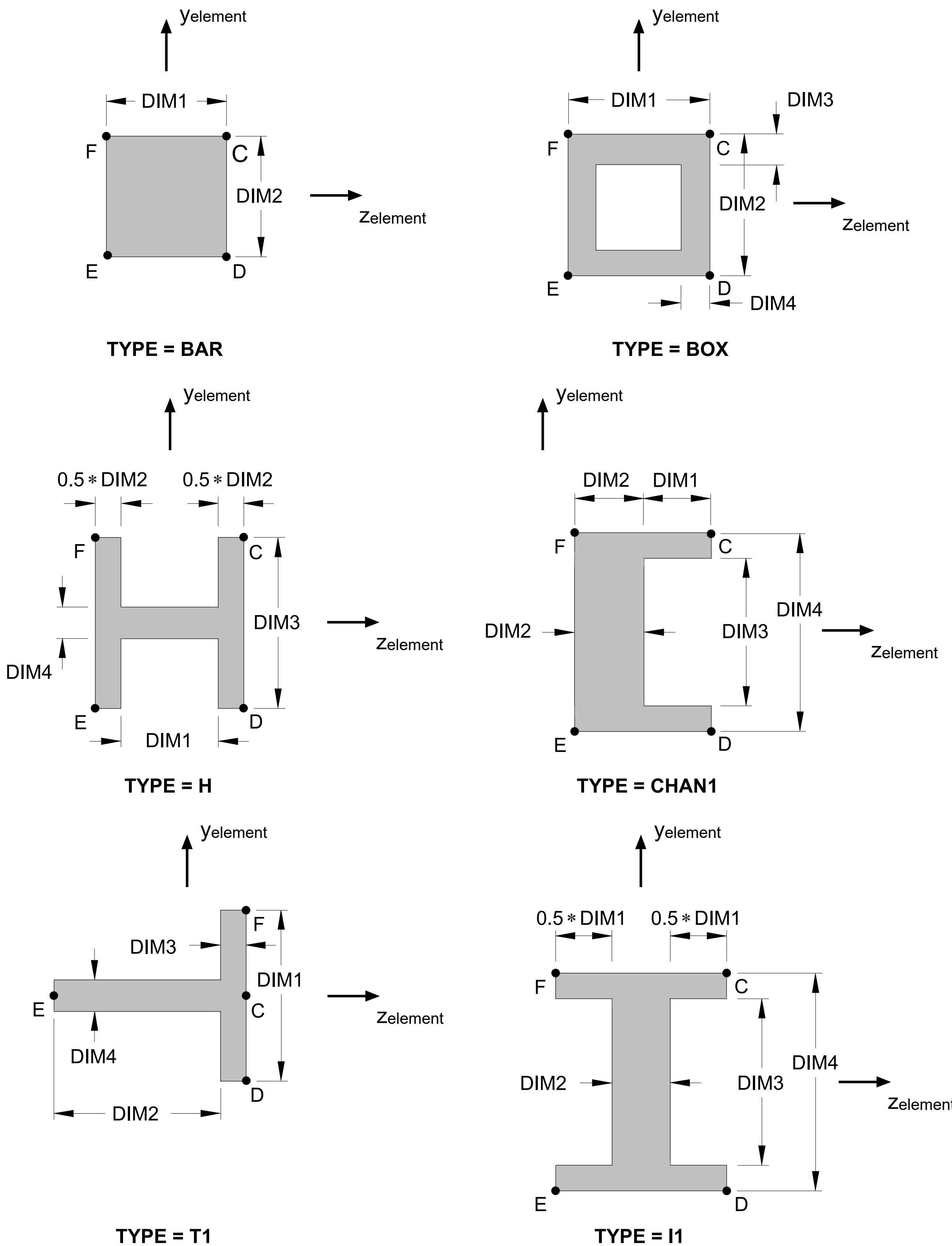


Figure 1b. Definition of Cross-Section Geometry and Stress Recovery Points.

(Continued)

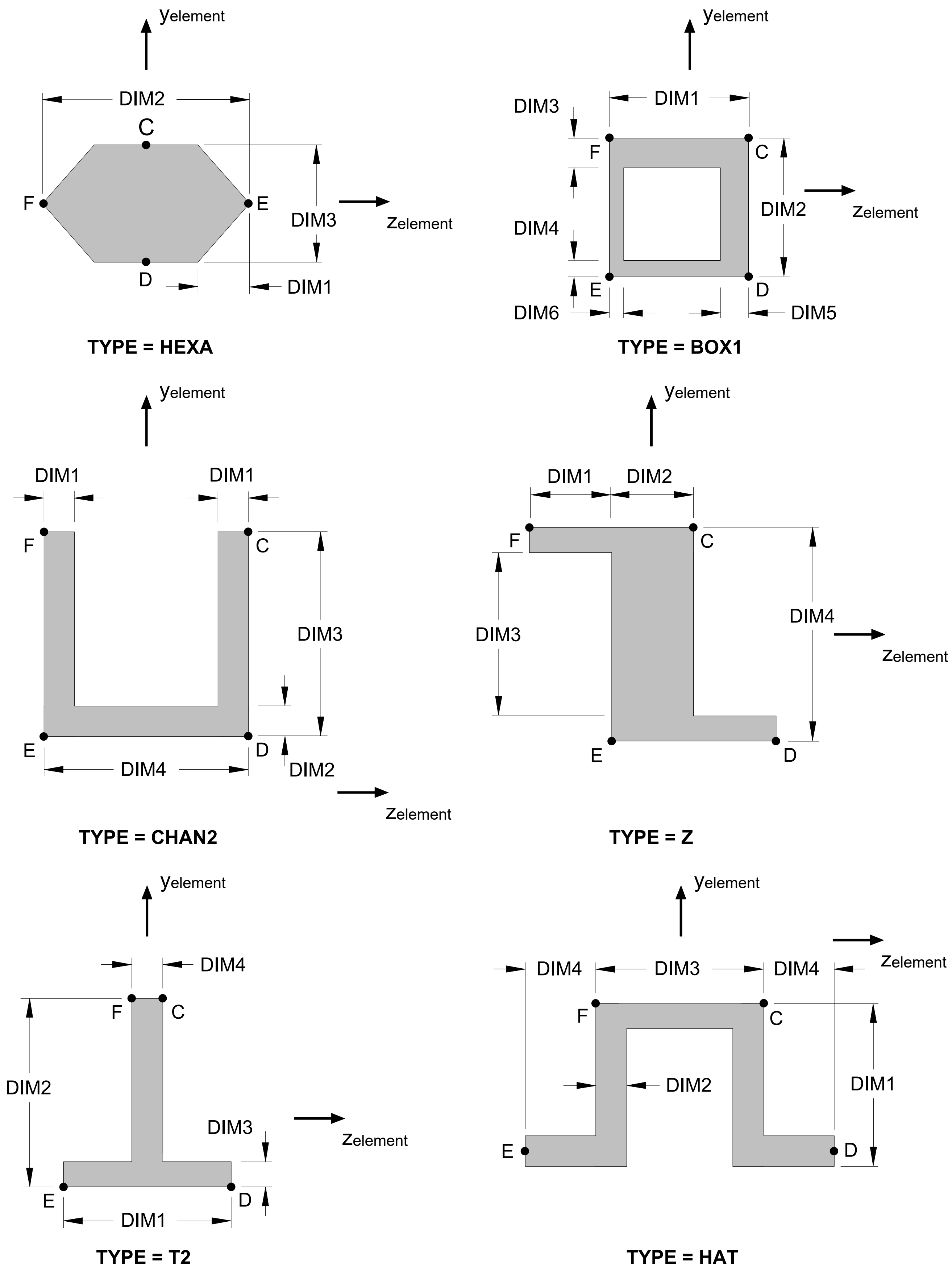


Figure 1c. Definition of Cross-Section Geometry and Stress Recovery Points.

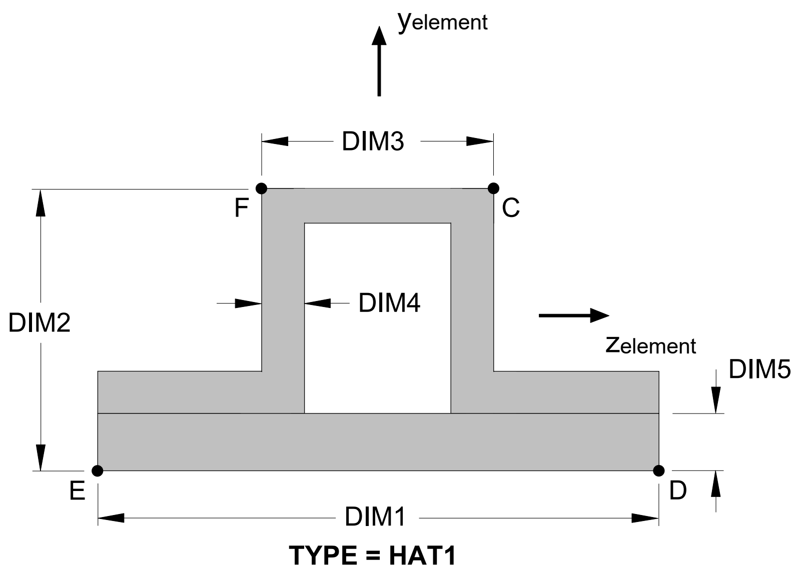


Figure 1d. Definition of Cross-Section Geometry and Stress Recovery Points.

PBEAM **Beam Element Property**

Description: Defines the properties of beam elements with optional taper (CBEAM entry).

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 5, 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 three continuations are:

	K1	K2	S1	S2	NSI(A)	NSI(B)			
	M1(A)	M1(B)	M2(A)	M2(B)	N1(A)	N2(A)	N1(B)	N2(B)	
	C	F0							

Example: Tapered beam with A = 4.5 at end A and A = 6.7 at end B.

PBEAM	40	5	4.5	2.9	5.45				
			1.5	-3.0					
	YES	1.0	6.7	25.4	37.8				
			3.5	-6.0					
			2.2		1.9				
					0.75		0.75		

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 2.	Integer > 0	Required
A(A)	Area of the beam cross-section at end A.	Real > 0.0	Required
I1(A)	Area moments of inertia for bending in plane 1 about the neutral axis. See Remark 8.	Real > 0.0	Required
I2(A)	Area moments of inertia at end A for bending in plane 2 about the neutral axis. See Remark 8.	Real > 0.0	Required

Field	Definition	Type	Default
I12(A)	Area product of inertia at end A ($I1*I2 > I12^2$). See Remark 8.	Real	0.0
J(A)	Torsional constant at end A. See Remark 9.	Real or blank	0.0
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 at end A for stress data recovery.	Real	0.0
SO	Stress output request option.	Character	Required
	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.		
X/XB	Distance from end A in the element coordinate system divided by the length of the element. See Remark 9.	Real > 0.0	See Remark 4
A, I1, I2, I12, J, NSM	Area moments of inertia, torsional stiffness parameter, and nonstructural mass for the cross-section located at x.	Real	See Remark 5
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 for the cross-section located at X/XB. The values are fiber locations for stress data recovery.	Real	See Remark 6
K1, K2	Area factors for shear for plane 1 and 2.	Real	See Remark 7
S1, S2	Shear relief coefficient due to taper for plane 1 and 2.	Real	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 Remark 9.	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 Remark 9.	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 Remark 9.	Real	0.0 (no offset from shear center), same values as end A
C	Coefficient to determine torsional stress.	Real or blank	See Remark 8
F0	Preload.	Real or blank	0.0

Remarks:

1. PBEAM entries must all have unique property identification numbers.
2. PBEAM entries may only reference MAT1 material entries.

3. 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.
4. 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.
   a) 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. One value of X/XB must be 1.0, corresponding to end B.
   b) 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.
5. 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.
6. If any fields 2 through 9 are blank on the continuation with the value of X/XB = 1.0, then the values Ci, Di, Ei, and Fi 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 2 through 9 are blank), a linear interpolation between the values at ends A and B is performed to obtain the missing stress recovery locations.
7. 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 are infinite; in other words, the transverse shear flexibilities are set equal to zero.
8. A single von Mises stress value is determined is based on the maximum combined axial and bending stress, the transverse shear stress, and the torsional stress using

$$\tau_v = \left[\sigma_x^2 + 3\left(\tau_{xy}^2 + \tau_{xz}^2 + \tau^2 \right) \right]^{\frac{1}{2}}$$

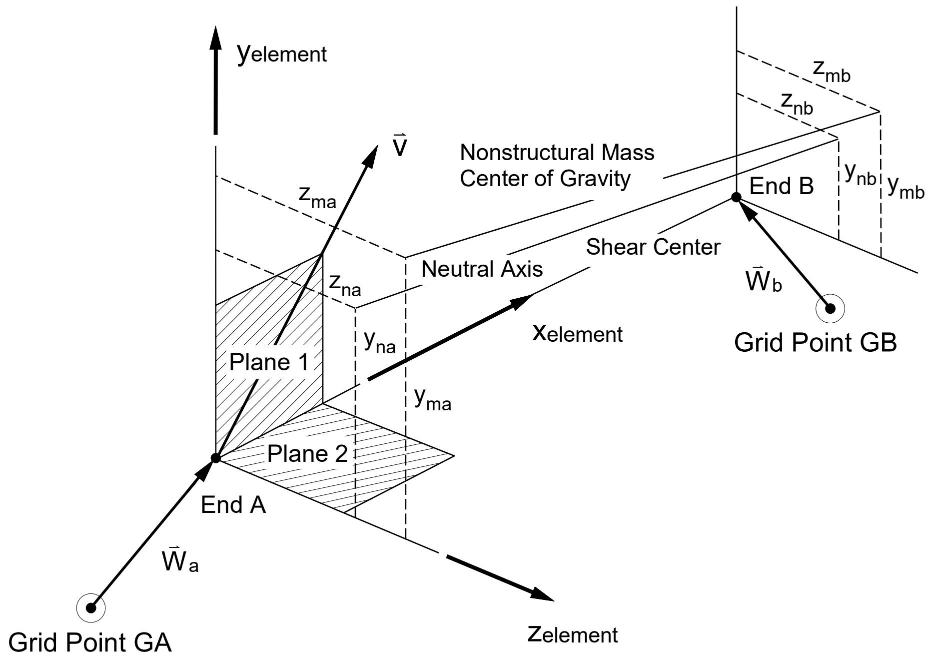
   where the transverse shear stress is determined using

$$\tau_{xy} = \frac{V_y}{K_y A} \qquad \tau_{xz} = \frac{V_z}{K_z A}$$

   and V_y and V_z are the element transverse shear forces and $K_y A = K1 * A$ and $K_z A = K2 * A$. The torsional stress is determined using

$$\tau = \frac{TC}{J}$$

   where T is the torsional moment. The torsional stress coefficient, C, should be selected as the maximum wall thickness for open sections and the radius for circular sections.
9. Figure 1 shows the PBEAM element coordinate system.



where:	$I1 = I_{(zz)elem}$	$N1(A) = y_{na}$	$N1(B) = y_{nb}$
	$I2 = I_{(yy)elem}$	$N2(A) = z_{na}$	$N2(B) = z_{nb}$
	$I12 = I_{(zy)elem}$	$M1(A) = y_{ma}$	$M1(B) = y_{mb}$
	$J = I_{(xx)elem}$	$M2(A) = z_{ma}$	$M2(B) = z_{mb}$

Figure 1. PBEAM Element Coordinate System.

PBEAML — Beam Cross-Section Property

Description: 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		TYPE				F0	
	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)	-etc.-	NSM(m)	SO(m)	X(m)/XB	DIM1(m)	-etc.-	
	DIMn(m)	NSM(m)	SO(B)	1.0	DIM1(B)	DIM2(B)	-etc.-	DIMn(B)	

Example:

PBEAML	99	21		T					
	12.0	14.8	2.5	2.6	0.5	NO	0.4	6.0	
	7.0	1.2	2.6		YES	0.6	6.0	7.8	
	5.6	2.3		YES					

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 2.	Integer > 0	Required
TYPE	Cross-section type. Must be one of following character variables: BAR, BOX, BOX1, CHAN, CHAN1, CHAN2, CROSS, H, HAT, HEXA, I, I1, L, ROD, T, T1, T2, TUBE, or Z. See Remark 5.	Character	Required
F0	Preload.	Real or blank	0.0
DIMi(A), DIMi(B)	Cross-sectional dimensions at end A and B.	Real > 0.0	Required
NSM(A), NSM(B)	Nonstructural mass per unit length.	Real or blank	0.0
SO(j), SO(B)	Stress output request option for intermediate station j and end B.	Character	YES
	YES — Stresses recovered at all points on the next continuation entry and shown in Figure 1 as C, D, E, and F.		
	NO — No stresses or forces are recovered.		

Field	Definition	Type	Default
X(j)/XB	Distance from end A to intermediate station j in the element coordinate system divided by the length of the element.	Real or blank	1.0
NSM(j)	Nonstructural mass per unit length at intermediate station j.	Real or blank	0.0
DIMi(j)	Cross-section dimensions at intermediate station j.	Real > 0.0	Required

Remarks:

1. PID must be unique with respect to all other PBEAM and PBEAML property identification numbers.
2. For structural problems, PBEAML entries must reference a MAT1 material entry.
3. See the PBEAM entry description for a discussion of beam-element geometry.
4. 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.
5. 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 1. The origin of element coordinate system is centered at the shear center of the cross-section oriented as shown.
6. A function of this entry is to derive equivalent an equivalent internal PBEAM entry. This equivalent entry is given in the database definition section of the Model Results Output File and in the translated Bulk Data Output File.

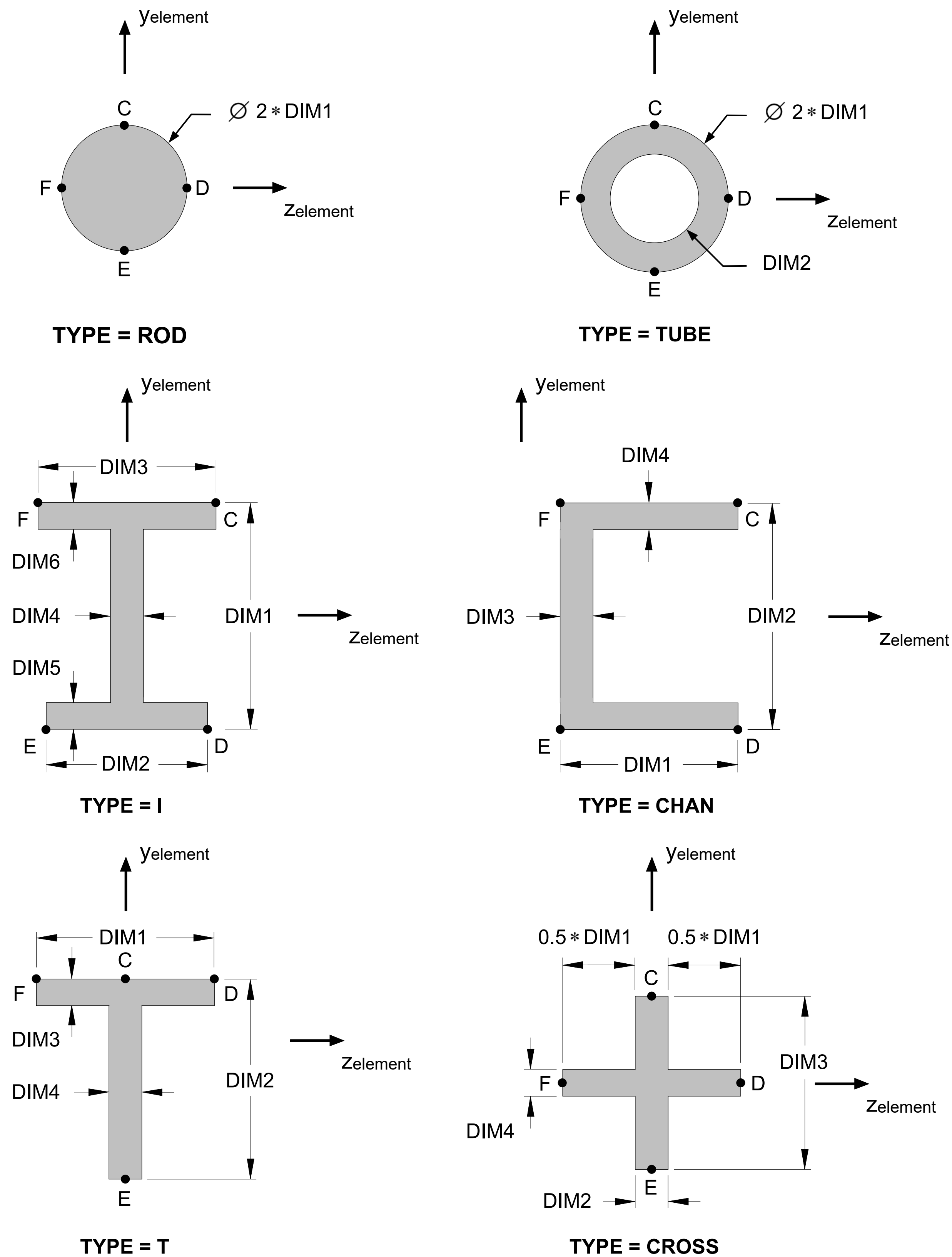


Figure 1a. Definition of Cross-Section Geometry and Stress Recovery Points.

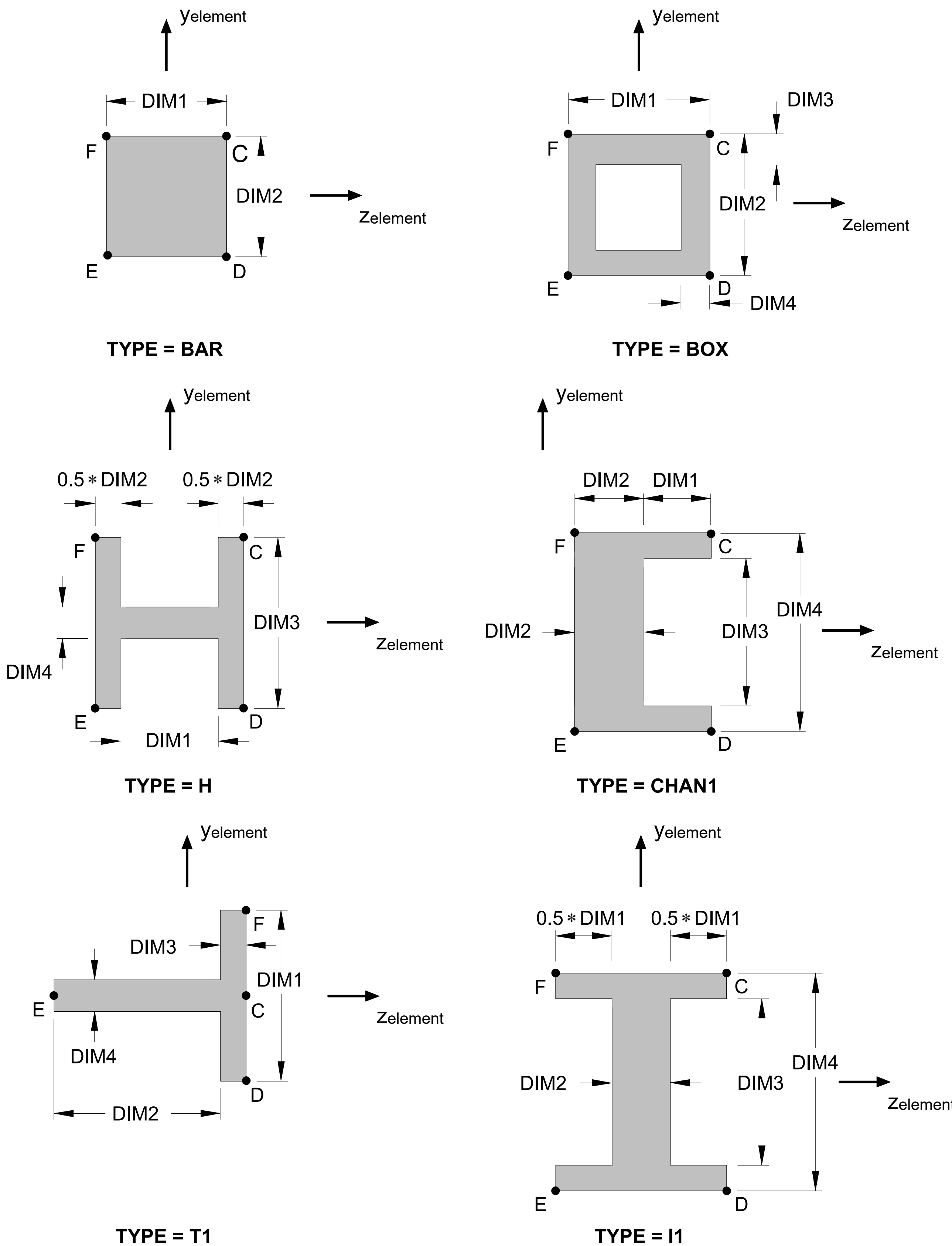


Figure 1b. Definition of Cross-Section Geometry and Stress Recovery Points.

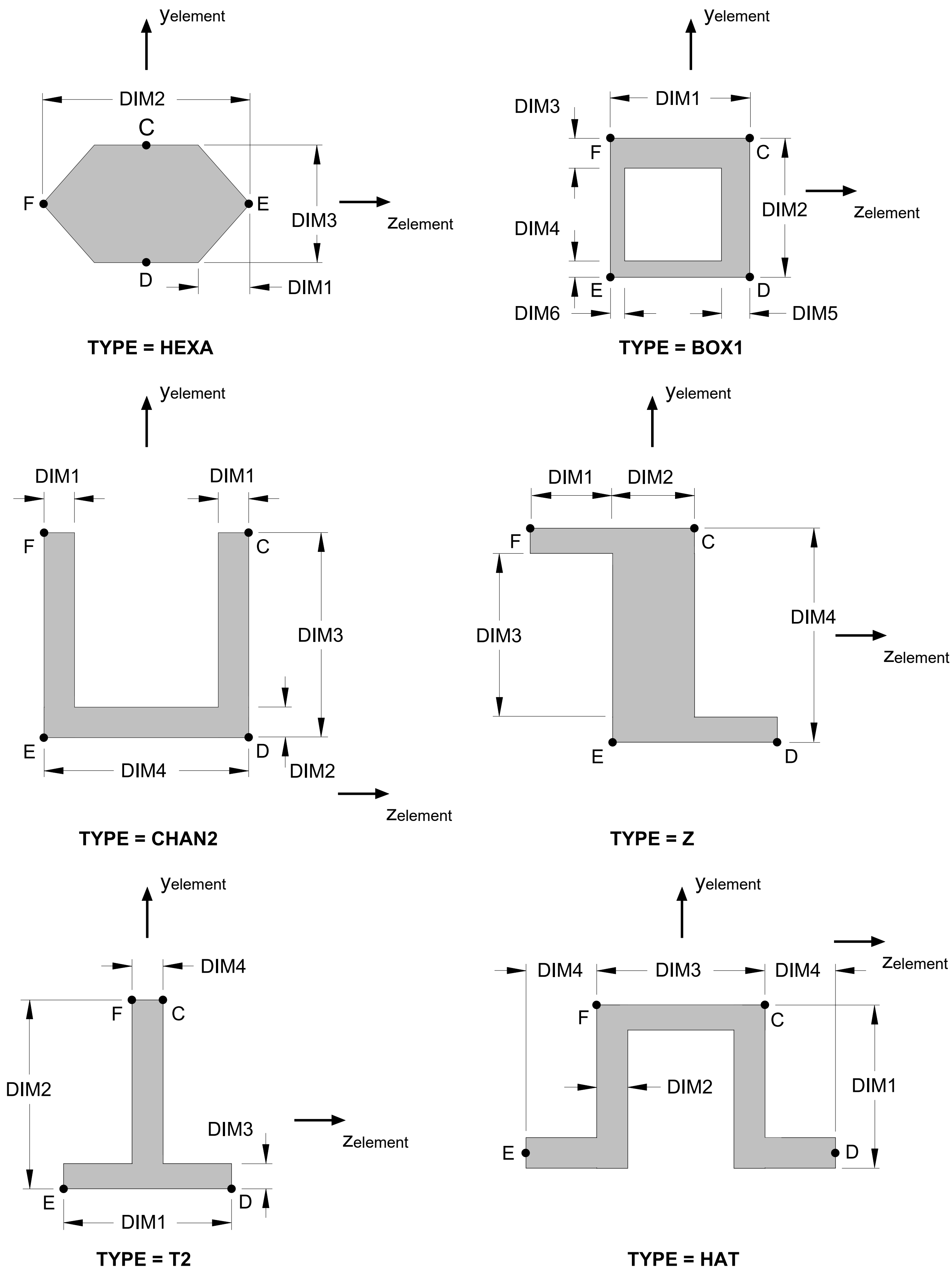
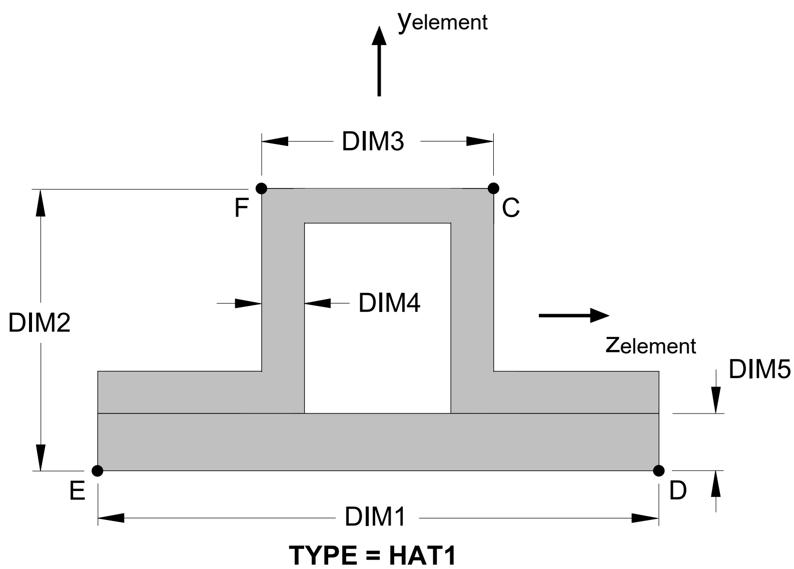


Figure 1c. Definition of Cross-Section Geometry and Stress Recovery Points.



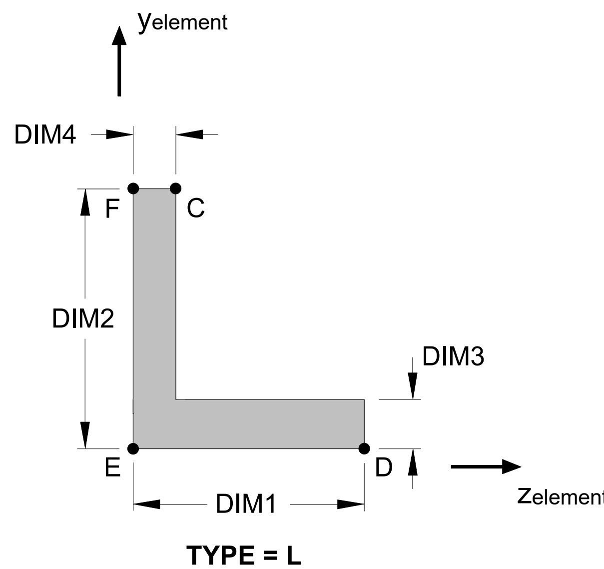


Figure 1d. Definition of Cross-Section Geometry and Stress Recovery Points.

PBUSH Generalized Spring and Damper Property

Description: 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						
		RCV	SA	ST	EA	ET			

Example:

PBUSH	40	K	2.55	2.55	5.05	1.5	1.5	3.1	
		GE	0.05						
		RCV	4.3	2.7					

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
K	Symbol indicating that the next 1 to 6 fields are stiffness values.	Character	Required
Ki	Nominal stiffness values in directions 1 through 6. See Remark 2 and 3.	Real	0.0
B	Symbol indicating that the next 1 to 6 fields are force per unit velocity damping.	Character	
Bi	Nominal damping coefficient in units of force per unit velocity. See Remark 3.	Real	0.0
GE	Symbol indicating that the next field is the structural damping constant.	Character	
GE1	Nominal structural element damping coefficient.	Real	0.0
RCV	Symbol indicating that the next 1 to 4 fields are stress coefficients.	Character	
SA	Stress recovery coefficient in the translational component direction 1 through 3.	Real	1.0
ST	Stress recovery coefficient in the rotational component direction 4 through 6.	Real	1.0
EA	Strain recovery coefficient in the translational component direction 1 through 3.	Real	1.0
ET	Strain recovery coefficient in the rotational component direction 4 through 6.	Real	1.0

Remarks:

1. Ki, Bi, or GE1 may be made frequency dependent for modal frequency response analysis and K may be made force dependent for nonlinear analysis by use of the PBUSHT entry.
2. 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, GE1 is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
4. The element stresses are computed by multiplying the stress coefficients with the recovered element forces.
5. The element strains are computed by multiplying the strain coefficients with the recovered element displacements.
6. The K, B, GE or RCV options may be specified in any order.

PBUSH1D — Rod Type Spring and Damper Property

Description: 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		
	SPRING		TID						
	DAMPER		TID						

Example:

PBUSH1D	15	1.+3	40.0	80.0					
	SPRING		100						
	DAMPER		110						

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
K	Stiffness. See Remark 1.	Real	
C	Viscous damping. See Remarks 1 and 2.	Real	
M	Total element mass.	Real	
SA	Stress recovery coefficient.	Real	1.0
SE	Strain recovery coefficient.	Real	1.0
SPRING	Character string specifying that the TID in field 4 defines a nonlinear elastic spring element in terms of a force versus displacement relationship. $F(u) = F_T(u)$ Tension is $u > 0$ and compression is $u < 0$.	Character	
DAMPER	Character string specifying that the TID in field 4 defines a nonlinear viscous element in terms of a force versus velocity relationship. $F(v) = F_T(v)$ Tension is $v > 0$ and compression is $v < 0$.	Character	
TID	Identification number of a TABLEDi entry for tension and compression.	Integer > 0	Required for SPRING or DAMPER

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 SPRING is specified, K is overwritten. When DAMPER is specified, C is overwritten. K and/or C should be non-zero if SPRING and/or DAMPER is specified otherwise the respective table will be ignored.
5. 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 $F(-u) = -F(u)$.
6. The element stresses are computed by multiplying the stress coefficient with the recovered element force.
7. The element strains are computed by multiplying the strain coefficient with the recovered element displacement.
8. The SPRING and DAMPER may be specified in any order.

PBUSHT Frequency Dependent Spring and Damper Property

Description: Defines the frequency or force 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						
		KN	TKNID1	TKNID2	TKNID3	TKNID4	TKNID5	TKNID6	

Example:

PBUSHT	45	K	70						
		B	25						

Field	Definition	Type	Default
PID	Property identification number that matches the identification number on a PBUSH entry.	Integer > 0	Required
K	Symbol 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 versus frequency relationship.	Integer ≥ 0	0
B	Symbol indicating that the next 1 to 6 fields are force per unit velocity frequency table identification numbers.	Character	
TBIDi	Identification number of a TABLEDi entry that defines the force per unit velocity damping versus frequency relationship.	Integer ≥ 0	0
GE	Symbol indicating that the next field is the structural element damping frequency table identification number.	Character	
TGEIDi	Identification number of a TABLEDi entry that defines the structural element damping versus frequency relationship.	Integer ≥ 0	0
KN	Symbol indicating that the next 1 to 6 entries are nonlinear force deflection table identification numbers.	Character	
TKNIDi	Identification number of a TABLEDi entry that defines the force versus deflection relationship.	Integer ≥ 0	0

Remarks:

1. The K, B, and GE entries are associated with same entries on the PBUSH entry.
2. PBUSHT may only be referenced by CBUSH elements.
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.
4. The K, B, GE or KN options may be specified in any order.
5. The PBUSHT entry is ignored in all solutions except frequency response and nonlinear analyses.

PCABLE — Cable Element Property

Description: Defines the properties of the cable element (CCABLE entry).

Format:

1	2	3	4	5	6	7	8	9	10
PCABLE	PID	MID	U0	T0	A	I	ST	PTYPE	

Example:

PCABLE	20	5	1.4		0.45				

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number.	Integer > 0	Required
U0	Initial cable slack. See Remark 1.	Real or blank	0.0
T0	Initial cable tension. See Remark 2.	Real ≥ 0.0 or blank	0.0
A	Area of cable cross-section.	Real > 0.0	Required
I	Area moment of inertia.	Real ≥ 0.0 or blank	See Remark 3
ST	Allowable tensile stress. See Remark 4.	Real ≥ 0.0 or blank	0.0
PTYPE	Preload option. If PTYPE = INIT, T0 is the initial tensile preload in the cable. If PTYPE = CONT, T0 is the actual cable tensile load and remains constant. See Remark 5.	Character or blank	INIT

Remarks:

1. The initial cable slack, U0, is the distance the cable must stretch before it will carry load.
2. The initial cable tension, T0, is the tensile preload in units of force that exists in the cable at the start of the nonlinear analysis. U0 and T0 should not be specified at the same time.
3. The default area moment of inertia is calculated using, A, and the formula for area moment of inertia for a circular cross-section,

$$I = \frac{A^2}{4\pi}$$

4. The allowable tensile stress, ST, is the stress above which the cable will no longer carry load.

5. The INIT setting will treat the T0 value as the initial tensile preload. This value will be continuously added to the element internal axial load generated from the displacement of the end nodes. The CONT setting will force the cable internal load to always be T0 regardless of the element nodal displacements. Use of the CONT setting may result in slower than normal nonlinear iteration convergence.
6. This element will default to a circular bar in linear solutions. A nonlinear solution must be selected for tension-only behavior.

PCOMP — Layered Composite Element Property

Description: 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:

PCOMP	190	-0.256	5.67	2500.0	HILL	70.0			
	200	0.065	0.0	YES	210	0.04	45.0		
	220	0.03	60.0						

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
Z0	Distance from the reference plane to the bottom surface.	Real	-1/2 element thickness
NSM	Nonstructural mass per unit area.	Real	0.0
SB	Allowable inter-laminar shear stress of the bonding material (allowable interlaminar shear stress). Required if bond shear failure index/strength ratio is desired.	Real > 0.0	
FT	Ply failure theory. The following theories are allowed. (If blank then no failure calculation is preformed) HILL for the Hill theory HOFF for the Hoffman theory TSAI for the Tsai-Wu theory STRESS for the maximum stress theory STRAIN for the maximum strain theory LARC02 for the NASA LaRC theory PUCK for the Puck PCP theory MCT for the Multicontinuum Theory	Character or blank	
TREF	Reference temperature. See Remark 3.	Real	0.0
GE	Structural element damping coefficient. See Remarks 12 and 13.	Real	0.0

Field	Definition	Type	Default
LAM	Laminate option, one of the following character variables: SYM, HCS, FCS, ACS, SME, or SMC. If LAM = 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 is desired with LAM = SYM then the center ply thickness (Ti) should be half the actual thickness. If LAM = HCS, LAM = FCS, or LAM = ACS a composite sandwich is defined for the purpose of facesheet stability index output. HCS specifies a honeycomb core material, FCS specifies a form core material, and ACS selects either HCS or FCS based on the core material specified. If LAM = SME, the ply effects are smeared and the stacking sequence is ignored. If LAM = SMC, a composite sandwich is defined using equivalent orthotropic properties. See Remarks 7 through 9.	Character or blank	If blank, all plies must be specified
MIDi	Material identification number 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, MAT4, MAT5, MAT8, or MAT12 Bulk Data entries. See Remark 11.	Integer > 0	MID1 required, see Remark 1
Ti	Ply thickness. See Remark 1.	Real or blank	T1 required
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 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 or blank	0.0
SOUTi	Stress or strain output request, one of the following character variables: YES or NO.	Character	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 present for a ply to exist. The minimum number of plies is one.
3. When PARAM, TEMPDEPCOMP is set to OFF (default is ON) the TREF given on the PCOMP entry will be used for all plies of the element and will override values supplied on material entries for individual plies. If the PCOMP entry references temperature-dependent material properties, then TREF given on the PCOMP will be used as the temperature to determine material properties and TEMPERATURE Case Control commands will be ignored for deriving the equivalent PSHELL and MAT1 entries used to describe the composite element. (See Section 5, *Parameters*, for more information on TEMPDEPCOMP.)
4. If PARAM, NOCOMPS is set to 1, or OFF, then composite element ply results will be output while the equivalent homogeneous element results will be suppressed. If PARAM, NOCOMPS is set to -1, 0 or ON, then composite element ply results will be suppressed while the equivalent homogeneous element results will be output.

5. When PARAM, COMPILSMETHOD is set to COMPONENT (default), the failure index for the bonding material is calculated as Failure Index = max(τ_{1z}, τ_{1z}). (See Section 5, *Parameters*, for more information on COMPILSMETHOD.) The Failure Index for the ply is calculated as shown in the table below.

Theory	Failure Index	Remarks
Hill	$\frac{\sigma_1^2}{x^2} - \frac{\sigma_1\sigma_2}{x^2} + \frac{\sigma_2^2}{y^2} + \frac{\tau_{12}^2}{s^2} = \text{F.I.}$	Orthotropic materials with equal strengths in tension and compression.
Hoffman	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} - \frac{\sigma_1\sigma_2}{x_t x_c} = \text{F.I.}$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
Tsai-Wu	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} + 2F_{12}\sigma_1\sigma_2 = \text{F.I.}$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
LaRC02	See the Autodesk Nastran User's Manual, Reference 5.	Orthotropic materials comprised of unidirectional plies under a general state of plane stress.
Puck	See the Autodesk Nastran User's Manual, References 12 and 13.	Orthotropic materials comprised of unidirectional plies under a general state of plane stress.
MCT	See the Autodesk Nastran User's Manual, References 20, 21, and 22.	Orthotropic materials comprised of unidirectional plies or plain weave fabric under a general state of plane stress.
Max Stress	$\text{Max}\left[\left(\frac{\sigma_1}{X_t}\right), \left(\frac{\sigma_2}{Y_t}\right), \left(\frac{\lvert\tau_{12}\rvert}{S}\right)\right]$	None
Max Strain	$\text{Max}\left[\left(\frac{\varepsilon_1}{X_t}\right), \left(\frac{\varepsilon_2}{Y_t}\right), \left(\frac{\lvert\gamma_{12}\rvert}{S}\right)\right]$	None

For LaRC02 and Puck failure theories the plies must reference an orthotropic, unidirectional material. Materials with stiffness or allowable ratios (axial/lateral) less than the value defined by the LARC02TSAITOL model parameter will automatically revert to the Tsai-Wu failure theory. (See Section 5, *Parameters*, for more information on LARC02TSAITOL.)

6. The STRENGTHRATIO model parameter is used to request the output of the Tsai Strength Ratio (R) instead of Failure Index. (See Section 5, *Parameters*, for more information on STRENGTHRATIO.)

7. The LAM field (FCS, HCS, or ACS options) can be used to define a composite sandwich laminate which consists of lower facesheet plies, followed by a single core ply (foam or honeycomb), and then upper facesheet plies. The number of plies defined must be greater than or equal to 3. When the total number of plies is greater than 3, the ply with the minimum equivalent material extensional stiffness is selected as the core ply automatically. Output includes facesheet stability indexes for three failure modes: wrinkling, dimpling, and crimping. Stability indexes are calculated using

$$\left(\frac{\sigma_1}{\sigma_a}\right)^3 + \left(\frac{\sigma_2}{\sigma_a}\right) = \text{S.I.}$$

Where σ_1 and σ_2 are the maximum and minimum facesheet principal stresses and σ_a is the facesheet allowable. If σ_1 is positive, the stability index is calculated using

$$\left(\frac{\sigma_2}{\sigma_a}\right) = \text{S.I.}$$

If σ_2 is positive, the stability index will be zero.

8. The SME and SMC options are used to define properties where the ply stacking sequence and membrane-bending coupling effects are ignored. The SME option smears the laminate material stiffness properties. The SMC option allows simplified modeling of a sandwich panel with equal face sheets and a central core. Output is for the equivalent homogeneous element and does not include individual ply results.
9. FCS, HCS, ACS, and SMC are all used to define sandwich laminate properties. FCS, HCS, and ACS define a composite laminate sandwich where the plies are specified in sequence from the bottom face sheet outer ply through to the top face sheet outer ply. Laminate properties and results are calculated the same as with the SYM or default laminate options with the addition of face sheet stability index output. SMC defines a simplified sandwich panel with equal face sheets and a central core. The facesheet plies are specified first followed by the core ply last. Stability index output is not available with the SMC option.
10. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. These equivalent entries are given in the database definition section of the Model Results Output File and in the translated Bulk Data Output File.
11. This entry may be used to define either a layered shell or solid element. For shell elements the MIDi fields may only reference MAT1, MAT2, or MAT8 entries. For solid elements the MIDi fields may only reference MAT1, MAT9, or MAT12 entries.
12. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
13. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
14. To compute a ply and/or bond failure index/strength ratio, the STRESS or STRAIN Case Control command must be present, SOUTi must be set to YES, and the following must be defined:
    a) For a ply stress or strain failure index/strength ratio:
        - FT on the PCOMP or the referenced MIDi entry
        - The stress or strain allowables on the referenced MIDi entry
    b) For a bond failure index/strength ratio:
        - The stress allowable SB on the PCOMP or referenced MIDi entry
15. Ply stress and strain results are always computed in the ply coordinate system.

PCOMPG — Layered Composite Element Property

Description: Defines the global plies and properties of an n-ply composite material laminate.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPG	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	GPLYIDi	MIDi	Ti	THETAi	SOUTi				

Example:

PCOMPG	190	-0.256	5.67	2500.0	HILL	70.0			
	2001	200	0.065	0.0	YES				
	1001	210	0.045	45.0	YES				
	2003	220	0.03	60.0					

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
Z0	Distance from the reference plane to the bottom surface.	Real	-1/2 element thickness
NSM	Nonstructural mass per unit area.	Real	0.0
SB	Allowable inter-laminar shear stress of the bonding material (allowable interlaminar shear stress). Required if bond shear failure index/strength ratio is desired.	Real > 0.0	
FT	Ply failure theory. The following theories are allowed. (If blank and not specified on the referenced MIDi entry then no failure calculation is preformed) HILL for the Hill theory HOFF for the Hoffman theory TSAI for the Tsai-Wu theory STRESS for the maximum stress theory STRAIN for the maximum strain theory LARC02 for the NASA LaRC theory PUCK for the Puck PCP theory MCT for the Multicontinuum Theory	Character or blank	
TREF	Reference temperature. See Remark 4.	Real	0.0
GE	Structural element damping coefficient. See Remarks 13 and 14.	Real	0.0

Field	Definition	Type	Default
LAM	Laminate option, one of the following character variables: SYM, HCS, FCS, ACS, SME, or SMC. If LAM = 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 is desired with LAM = SYM then the center ply thickness (Ti) should be half the actual thickness. If LAM = HCS, LAM = FCS, or LAM = ACS a composite sandwich is defined for the purpose of facesheet stability index output. HCS specifies a honeycomb core material, FCS specifies a form core material, and ACS selects either HCS or FCS based on the core material specified. If LAM = SME, the ply effects are smeared and the stacking sequence is ignored. If LAM = SMC, a composite sandwich is defined using equivalent orthotropic properties. See Remarks 8 through 10.	Character or blank	If blank, all plies must be specified
GPLYIDi	User defined global ply identification number.	Integer > 0	Ply number
MIDi	Material identification number 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, MAT4, MAT5, MAT8, or MAT12 Bulk Data entries. See Remark 12.	Integer > 0	MID1 required, see Remark 2
Ti	Ply thickness. See Remark 2.	Real or blank	T1 required
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 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 or blank	0.0
SOUTi	Stress or strain output request, one of the following character variables: YES or NO.	Character	NO

Remarks:

1. The global ply identification number should be unique with respect to all other global plies.
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 identification number (GPLYIDi) and at least one of the four values (MIDi, Ti, THETAi, SOUTi) must present for a ply to exist. The minimum number of plies is one.
4. When PARAM, TEMPDEPCOMP is set to OFF (default is ON) the TREF given on the PCOMP entry will be used for all plies of the element and will override values supplied on material entries for individual plies. If the PCOMP entry references temperature-dependent material properties, then TREF given on the PCOMP will be used as the temperature to determine material properties and TEMPERATURE Case Control commands will be ignored for deriving the equivalent PSHELL and MAT1 entries used to describe the composite element. (See Section 5, *Parameters*, for more information on TEMPDEPCOMP.)
5. If PARAM, NOCOMPS is set to 1, or OFF, then composite element ply results will be output while the equivalent homogeneous element results will be suppressed. If PARAM, NOCOMPS is set to -1, 0 or ON, then composite element ply results will be suppressed while the equivalent homogeneous element results will be output.

6. When PARAM, COMPILSMETHOD is set to COMPONENT (default), the failure index for the bonding material is calculated as Failure Index = max(τ_{1z}, τ_{1z}). (See Section 5, *Parameters*, for more information on COMPILSMETHOD.) The Failure Index for the ply is calculated as shown in the table below.

Theory	Failure Index	Remarks
Hill	$\frac{\sigma_1^2}{x^2} - \frac{\sigma_1\sigma_2}{x^2} + \frac{\sigma_2^2}{y^2} + \frac{\tau_{12}^2}{s^2} = \text{F.I.}$	Orthotropic materials with equal strengths in tension and compression.
Hoffman	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} - \frac{\sigma_1\sigma_2}{x_t x_c} = \text{F.I.}$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
Tsai-Wu	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} + 2F_{12}\sigma_1\sigma_2 = \text{F.I.}$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
LaRC02	See the Autodesk Nastran User's Manual, Reference 5.	Orthotropic materials comprised of unidirectional plies under a general state of plane stress.
Puck	See the Autodesk Nastran User's Manual, References 12 and 13.	Orthotropic materials comprised of unidirectional plies under a general state of plane stress.
MCT	See the Autodesk Nastran User's Manual, References 20, 21, and 22.	Orthotropic materials comprised of unidirectional plies or plain weave fabric under a general state of plane stress.
Max Stress	$\text{Max}\left[\left(\frac{\sigma_1}{X_t}\right), \left(\frac{\sigma_2}{Y_t}\right), \left(\frac{\lvert\tau_{12}\rvert}{S}\right)\right]$	None
Max Strain	$\text{Max}\left[\left(\frac{\varepsilon_1}{X_t}\right), \left(\frac{\varepsilon_2}{Y_t}\right), \left(\frac{\lvert\gamma_{12}\rvert}{S}\right)\right]$	None

For LaRC02 and Puck failure theories the plies must reference an orthotropic, unidirectional material. Materials with stiffness or allowable ratios (axial/lateral) less than the value defined by the LARC02TSAITOL model parameter will automatically revert to the Tsai-Wu failure theory. (See Section 5, *Parameters*, for more information on LARC02TSAITOL.)

7. The STRENGTHRATIO model parameter is used to request the output of the Tsai Strength Ratio (R) instead of Failure Index. (See Section 5, *Parameters*, for more information on STRENGTHRATIO.)

8. The LAM field (FCS, HCS, or ACS options) can be used to define a composite sandwich laminate which consists of lower facesheet plies, followed by a single core ply (foam or honeycomb), and then upper facesheet plies. The number of plies defined must be greater than or equal to 3. When the total number of plies is greater than 3, the ply with the minimum equivalent material extensional stiffness is selected as the core ply automatically. Output includes facesheet stability indexes for three failure modes: wrinkling, dimpling, and crimping. Stability indexes are calculated using

$$\left(\frac{\sigma_1}{\sigma_a}\right)^3 + \left(\frac{\sigma_2}{\sigma_a}\right) = \text{S.I.}$$

Where σ_1 and σ_2 are the maximum and minimum facesheet principal stresses and σ_a is the facesheet allowable. If σ_1 is positive, the stability index is calculated using

$$\left(\frac{\sigma_2}{\sigma_a}\right) = \text{S.I.}$$

If σ_2 is positive, the stability index will be zero.

9. The SME and SMC options are used to define properties where the ply stacking sequence and membrane-bending coupling effects are ignored. The SME option smears the laminate material stiffness properties. The SMC option allows simplified modeling of a sandwich panel with equal face sheets and a central core. Output is for the equivalent homogeneous element and does not include individual ply results.
10. FCS, HCS, ACS, and SMC are all used to define sandwich laminate properties. FCS, HCS, and ACS define a composite laminate sandwich where the plies are specified in sequence from the bottom face sheet outer ply through to the top face sheet outer ply. Laminate properties and results are calculated the same as with the SYM or default laminate options with the addition of face sheet stability index output. SMC defines a simplified sandwich panel with equal face sheets and a central core. The facesheet plies are specified first followed by the core ply last. Stability index output is not available with the SMC option.
11. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. These equivalent entries are given in the database definition section of the Model Results Output File and in the translated Bulk Data Output File.
12. This entry may be used to define either a layered shell or solid element. For shell elements the MIDi fields may only reference MAT1, MAT2, or MAT8 entries. For solid elements the MIDi fields may only reference MAT1, MAT9, or MAT12 entries.
13. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
14. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
15. To compute a ply and/or bond failure index/strength ratio, the STRESS or STRAIN Case Control command must be present, SOUTi must be set to YES, and the following must be defined:
    a) For a ply stress or strain failure index/strength ratio:
        - FT on the PCOMPG or the referenced MIDi entry
        - The stress or strain allowables on the referenced MIDi entry
    b) For a bond failure index/strength ratio:
        - The stress allowable SB on the PCOMPG or referenced MIDi entry
16. Ply stress and strain results are always computed in the ply coordinate system.

PCOMPS — Layered Composite Solid Element Property

Description: Defines the global plies and properties of an n-ply composite material laminate for CHEXA and CPENTA solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPS	PID	MCID	XZDIR	SB	NB	TREF	GE		
	GPLYIDi	MIDi	Ti	THETAi	PLYFTi	ILFTi	SOUTi		

Example:

PCOMPS	40			1000.0					
	2	1	0.03	0.0	TSAI		YES		
	3	2	0.04	90.0	HILL		YES		

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	
MCID	Identification number of the material coordinate system. See Remarks 5 and 6.	Integer ≥ -1 or blank	See Remark 5
XZDIR	Ply orientation reference material axis and element orientation, one of the following character variables: 12, 13, 21, 23, 31, or 32. See Remark 6.	Integer	13
SB	Allowable inter-laminar shear stress of the bonding material (allowable interlaminar shear stress). Required if bond shear failure index/strength ratio is desired.	Real > 0.0	
NB	Allowable inter-laminar normal stress of the bonding material.	Real > 0.0	
TREF	Reference temperature. See Remark 4.	Real	0.0
GE	Structural element damping coefficient. See Remarks 12 and 13.	Real	0.0
GPLYIDi	User defined global ply identification number.	Integer > 0	Ply number
MIDi	Material identification number of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT8, MAT9, or MAT12 Bulk Data entries. See Remark 11.	Integer > 0	
Ti	Ply thickness. See Remarks 2 and 7.	Real or blank	T1 required
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element. The plies are 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. See Remark 6.	Real or blank	0.0

Field	Definition	Type	Default
PLYFTi	Ply failure theory. The following theories are allowed. (If blank and not specified on the referenced MIDi entry then no failure calculation is preformed) HILL for the Hill theory HOFF for the Hoffman theory TSAI for the Tsai-Wu theory STRESS for the maximum stress theory STRAIN for the maximum strain theory MCT for the Multicontinuum Theory	Character or blank	
ILFTi	Inter-laminar failure theory. The following theories are allowed. (If blank then both calculations are performed) SB for maximum transverse shear stress theory NB for maximum normal stress theory	Character or blank	Both
SOUTi	Stress or strain output request, one of the following character variables: YES or NO.	Character	NO

Remarks:

1. The global ply identification number should be unique with respect to all other global plies.
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 identification number (GPLYIDi) and at least one of the four values (MIDi, Ti, THETAi, SOUTi) must present for a ply to exist. The minimum number of plies is one.
4. When PARAM, TEMPDEPCOMP is set to OFF (default is ON) the TREF given on the PCOMP entry will be used for all plies of the element and will override values supplied on material entries for individual plies. If the PCOMP entry references temperature-dependent material properties, then TREF given on the PCOMP will be used as the temperature to determine material properties and TEMPERATURE Case Control commands will be ignored for deriving the equivalent PSHELL and MAT1 entries used to describe the composite element. (See Section 5, *Parameters*, for more information on TEMPDEPCOMP.)
5. See the CHEXA, CPENTA, CPYRA, or CTETRA entry for the definition of the element coordinate system. The material coordinate system (MCID) may be the basic system (0), any defined system (Integer $>$ 0), or the element coordinate system (-1 or blank). The default for MCID is the element coordinate system.
6. The ply orientation is relative to the element material x-direction similar to that of a composite shell element. By default the element material x-direction is defined by projecting the MCID x-axis onto a surface defined by the element z-axis. The MCID y-axis or z-axis may be specified using the first component number of the XZDIR field. The element z-axis can be reoriented using the second component number of the XZDIR field. The element z-axis also defines the element thickness direction. Only CHEXA and CPENTA elements may be referenced if the property defines a layered solid element.
7. The laminate thickness is adjusted at the corners to coincide with the distance between grid points. The thickness of each ply in the laminate is adjusted proportionally.
8. When PARAM, COMPILSMETHOD is set to COMPONENT (default), the failure index for the bonding material is calculated as Failure Index = max(τ_{1z}, τ_{1z}). (See Section 5, *Parameters*, for more information on COMPILSMETHOD.) The Failure Index for the ply is calculated as shown in the table on the following page.

Theory	Failure Index	Remarks
Hill	$\frac{\sigma_1^2}{x^2} - \frac{\sigma_1\sigma_2}{x^2} + \frac{\sigma_2^2}{y^2} + \frac{\tau_{12}^2}{s^2} = F.I.$	Orthotropic materials with equal strengths in tension and compression.
Hoffman	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} - \frac{\sigma_1\sigma_2}{x_t x_c} = F.I.$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
Tsai-Wu	$\left(\frac{1}{x_t} - \frac{1}{x_c}\right)\sigma_1 + \left(\frac{1}{y_t} - \frac{1}{y_c}\right)\sigma_2 + \frac{\sigma_1^2}{x_t x_c} + \frac{\sigma_2^2}{y_t y_c} + \frac{\tau_{12}^2}{s^2} + 2F_{12}\sigma_1\sigma_2 = F.I.$	Orthotropic materials under a general state of plane stress with unequal tensile and compressive strengths.
MCT	See the Autodesk Nastran User's Manual, References 20, 21, and 22.	Orthotropic materials comprised of unidirectional plies or plain weave fabric under a general state of plane stress.
Max Stress	$\mathrm{Max}\left[\left(\frac{\sigma_1}{X_t}\right), \left(\frac{\sigma_2}{Y_t}\right), \left(\frac{\lvert\tau_{12}\rvert}{S}\right)\right]$	None
Max Strain	$\mathrm{Max}\left[\left(\frac{\varepsilon_1}{X_t}\right), \left(\frac{\varepsilon_2}{Y_t}\right), \left(\frac{\lvert\gamma_{12}\rvert}{S}\right)\right]$	None

9. The STRENGTHRATIO model parameter is used to request the output of the Tsai Strength Ratio (R) instead of Failure Index. (See Section 5, *Parameters*, for more information on STRENGTHRATIO.)
10. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. These equivalent entries are given in the database definition section of the Model Results Output File and in the translated Bulk Data Output File.
11. This entry may only be used to define a layered solid element. The MIDi fields may only reference MAT1, MAT8, MAT9, or MAT12 entries.
12. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
13. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)
14. To compute a ply and/or bond failure index/strength ratio, the STRESS or STRAIN Case Control command must be present, SOUTi must be set to YES, and the following must be defined:
    a) For a ply stress or strain failure index/strength ratio:
        - PLYFTi on the PCOMPS or referenced MIDi entry
        - The stress or strain allowables on the referenced MIDi entry
    b) For a bond failure index/strength ratio:
        - The stress allowables SB and/or NB on the PCOMPS or referenced MIDi entry
15. Ply stress and strain results are always computed in the ply coordinate system.

PCONV Convection Property Definition

Description: Specifies the free convection boundary condition properties of a surface element used for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
PCONV	PID	MID	FORM		CTID1	CTID2	CTID3	ATID1	
	ATID2	ATID3							

Example:

PCONV	5	10							

Field	Definition	Type	Default
PID	Convection property identification number.	Integer > 0	Required
MID	Material property identification number.	Integer > 0	Required
FORM	Film temperature option if film grid point is not specified. See Remark 3.	1 ≤ Integer ≤ 3	1
CTID1, CTID2, CTID3	TABLEDi set identification numbers that define control point position dependent scale factors in the x, y, and z directions of the basic coordinate system. See Remark 1.	Integer > 0 or blank	
ATID1, ATID2, ATID3	TABLEDi set identification numbers that define ambient point position dependent scale factors in the x, y, and z directions of the basic coordinate system. See Remark 1.	Integer > 0 or blank	

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).
3. The FORM field specifies how temperatures are averaged to determine film temperature. The options are described as follows:

FORM	Description
1	Film temperature is the average of average surface and average ambient temperatures
2	Film temperature is the average of surface temperatures
3	Film temperature is the average of ambient temperatures

4. The basic exchange relationship can be expressed in one of the following forms:

   a) $q = \mathrm{H} * \mathrm{u}_{\mathrm{CNTRLND}}\, c\,(x, y, z)\left[\mathrm{T} - \mathrm{TAMB}\, a\,(x, y, z)\right]$, CNTRLND ≠ 0

   b) $q = \mathrm{H} * \left[\mathrm{T} - \mathrm{TAMB}\, a\,(x, y, z)\right]$, CNTRLND = 0

   where $c(x, y, z)$ is defined as the product of scale factors returned by tables defined in fields 6, 7, and 8, and $a(x, y, z)$ is defined as the product of scale factors returned by tables defined in field 9 and fields 2 and 3 on the continuation entry.

PDAMP Scalar Damper Property

Description: Specifies the damping value of a damper element (CDAMP1 or CDAMP3 entry).

Format:

1	2	3	4	5	6	7	8	9	10
PDAMP	PID1	B1	PID2	B2	PID3	B3	PID4	B4	

Example:

PDAMP	14	3.2	16	4.0					

Field	Definition	Type	Default
PIDi	Property identification number.	Integer > 0	Required
Bi	Force per unit velocity	Real	Required

Remarks:

1. PDAMP entries must all have unique property identification numbers.
2. Up to four damping properties may be defined on a single entry.

PDAMPT Frequency-Dependent Damper Property

Description: 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	14	40							

Field	Definition	Type	Default
PIDi	Identification number of a PDAMP entry.	Integer > 0	Required
TBID1	Identification number of a TABLEDi entry that defines the damping force per-unit velocity versus frequency relationship.	Integer > 0	Required

Remarks:

1. PDAMPT may only be referenced by CDAMP1 or CDAMP3 elements.
2. The PDAMPT entry is ignored in all solution sequences except for frequency response analysis.

PELAS Elastic Element Property

Description: Specifies the stiffness and stress coefficient of a 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	24	1.+3		0.9					

Field	Definition	Type	Default
PIDi	Property identification number.	Integer > 0	Required
Ki	Elastic property value	Real	Required
GEi	Structural element damping coefficient. See Remark 4.	Real or blank	0.0
Si	Stress coefficient	Real or blank	0.0

Remarks:

1. PELAS entries must all have unique property identification numbers.
2. K and GE may be made frequency dependent for modal frequency response analysis and K may be made force dependent for nonlinear analysis by use of the PELAST entry.
3. The use of negative spring values may result in fatal errors.
4. One or two elastic spring properties may be defined on a single entry.
5. To obtain the damping coefficient GE, multiply the critical damping ratio C/C_0, by 2.0.
6. If PARAM, W4 is not specified, GE is ignored in transient response analysis. (See Section 5, *Parameters*, for more information on W4.)

PELAST — Frequency Dependent Elastic Property

Description: Defines the frequency or force 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	24	40							

Field	Definition	Type	Default
PID	Identification number of a PELAS entry.	Integer > 0	Required
TKID	Identification number of a TABLEDi entry that defines the force per unit displacement versus frequency relationship.	Integer > 0	See Remark 3
TGEID	Identification number of a TABLEDi entry that defines the nondimensional structural damping coefficient versus frequency relationship.	Integer > 0	See Remark 3
TKNID	Identification number of a TABLEDi entry that defines the nonlinear force per unit displacement versus frequency relationship.	Integer > 0	See Remark 3

Remarks:

1. PELAST may only be referenced by CELAS1 or CELAS3 elements.
2. For frequency dependent modal frequency response the modes are calculated using nominal Ki values as specified on the PELAS entry.
3. The following table summarizes the usage of the 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

4. The PELAST is ignored in all solutions except frequency response and nonlinear analysis.

PGAP

Gap Element Property

Description: Defines the properties of gap elements (CGAP entry).

Format:

1	2	3	4	5	6	7	8	9	10
PGAP	PID	U0	F0	KA	KB	KT	MUY	MUZ	
	TMAX	MAR	TRMIN						

Example:

PGAP	10	0.015		1.+6			0.2	0.2	

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
U0	Initial gap opening. See Figure 2 and Remark 1.	Real or AUTO	0.0
F0	Preload. See Figure 2.	Real	0.0
KA	Axial stiffness for the closed gap (i.e. $U_A - U_B > U0$. See Figure 2.	Real > 0.0	Required
KB	Axial stiffness for the open gap (i.e. $U_A - U_B < U0$. See Figure 2 and Remark 3.	Real ≥ 0.0 or blank	$10^{-10} * KA$
KT	Transverse stiffness when the gap is closed. See Figure 3. It is recommended that KT ≥ (0.1 ∗ KA).	Real ≥ 0.0	MUY ∗ KA
MUY	Coefficient of friction in the y transverse direction (μ_y). See Remark 4.	Real ≥ 0.0	0.0
MUZ	Coefficient of friction in the z transverse direction (μ_z). See Remark 4.	Real ≥ 0.0	0.0
TMAX	Maximum allowable penetration used in the adjustment of penalty values. A positive value activates the penalty value adjustment. See Remark 5.	Real	0.0
MAR	Maximum allowable adjustment ratio for adaptive penalty values KA and KT. See Remark 6.	Real > 1.0	100.0
TRMIN	Fraction of TMAX defining the lower bound for the allowable penetration. See Remark 7.	0.0 ≤ Real ≤ 1.0	0.001

Remarks:

1. The default initial gap opening is zero. If AUTO or -1.0 is specified, the initial gap opening will be set to the initial element length. This is particularly useful when defining multiple gap elements over an uneven surface.
2. Figures 1 through 3 show the gap element and the force-displacement curves used in the stiffness and force computations for the element.
3. 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 results in inaccurate results. The value is adjusted as necessary if TMAX $>$ 0.0.
4. 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.
5. There are two types of gap elements: adaptive gap and nonadaptive gap. If TMAX $\geq$ 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 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.
6. The maximum adjustment ratio MAR is used only for the adaptive gap element. Upper and lower bounds of the adjusted penalty are defined by

$$\frac{K_{initial}}{\text{MAR}} \leq K \leq K_{initial} * \text{MAR}$$

   where $K_{initial}$ is either KA or KT.
7. 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 $*$ TMAX. The penalty values are decreased if the penetration is below the lower bound.
8. This element will default to a linear spring in linear solutions with an axial stiffness equal to KA and a transverse stiffness equal to KT. A nonlinear solution must be selected for general contact behavior.

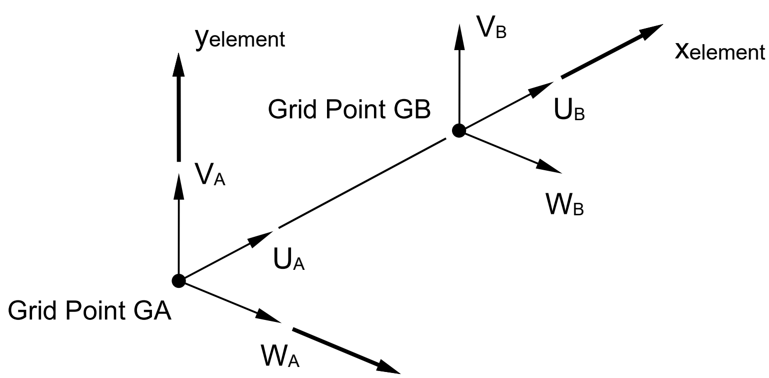


Figure 1. GAP Element Coordinate System.

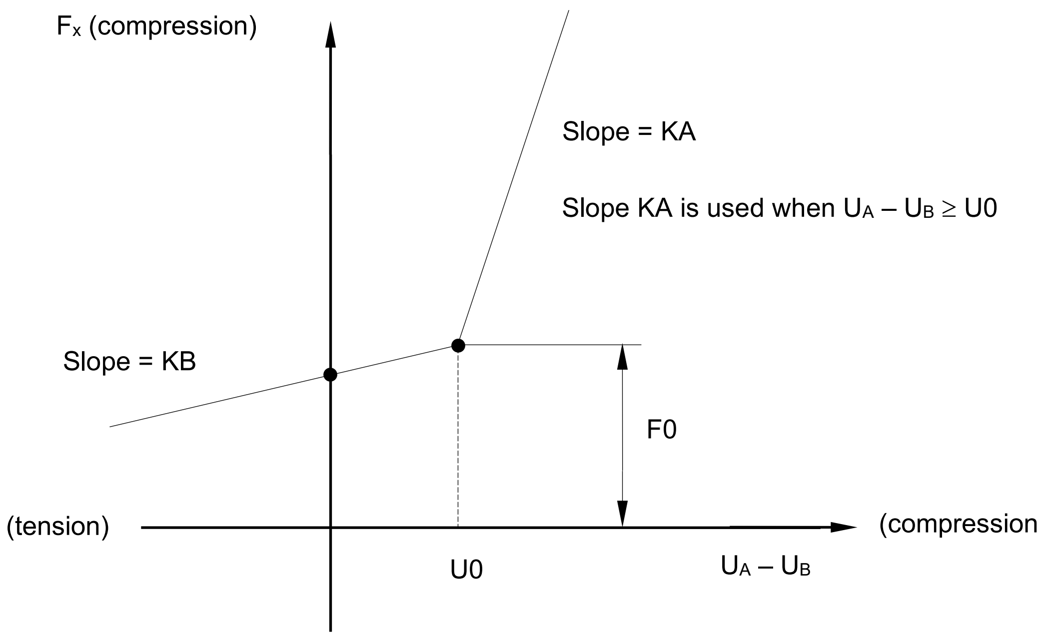


Figure 2. GAP Element Force-Deflection Curve for Nonlinear Analysis.

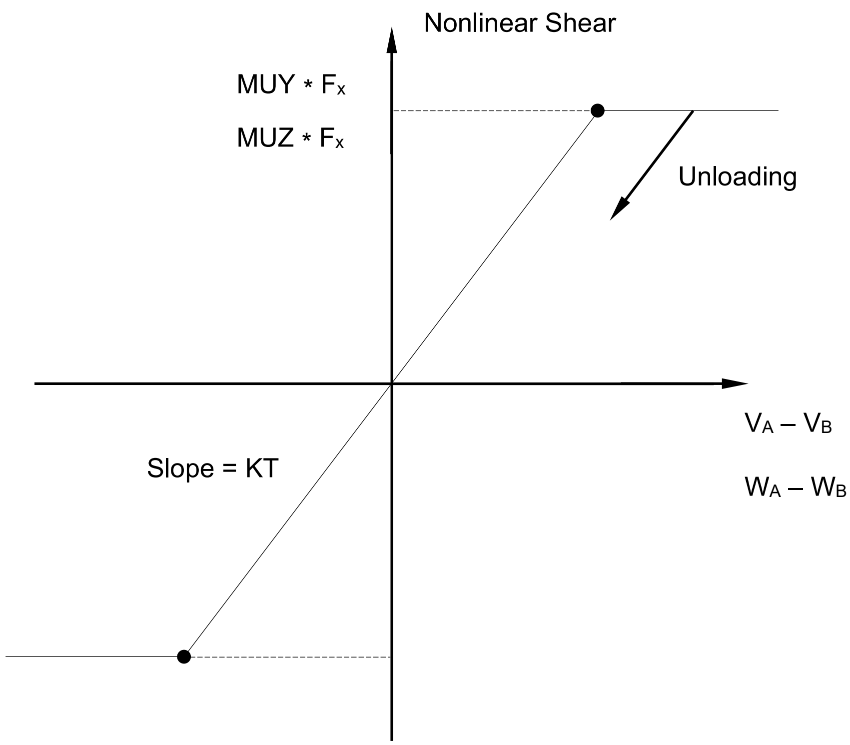


Figure 3. GAP Element Shear Forces for Nonlinear Analysis.

PHBDY CHBDYP Geometric Element Definition

Description: Referenced by CHBDYP entries to give additional geometric information for boundary condition surface elements.

Format:

1	2	3	4	5	6	7	8	9	10
PHBDY	PID	AF							

Example:

PHBDY	5	0.01							

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
AF	Area factor of the surface used only for CHBDYP elements with surface types: POINT and LINE.	Real > 0.0 or blank	Required

Remarks:

1. All PHBDY property entries must have unique identification numbers.
2. The PHBDY entry is used with CHBDYP entries.
3. AF is the area for POINT-type surfaces and the effective width for LINE-type surfaces.

PLOAD — Static Pressure Load

Description: 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	5	-3.5	15	12	19				

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
P	Pressure value.	Real	Required
Gi	Grid point identification numbers.	Integer > 0; G4 may be blank	Required

Remarks:

1. Load sets must be selected in the Case Control Section (LOAD = SID).
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 1. The total load on the surface 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.
5. The follower force effects due to loads from this entry are not included in the stiffness in all linear solution sequences that calculate a differential stiffness.

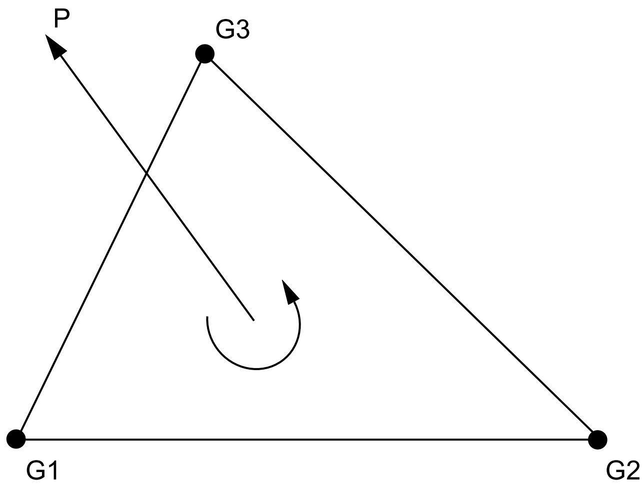


Figure 1. Pressure Convention for Triangular Surface.

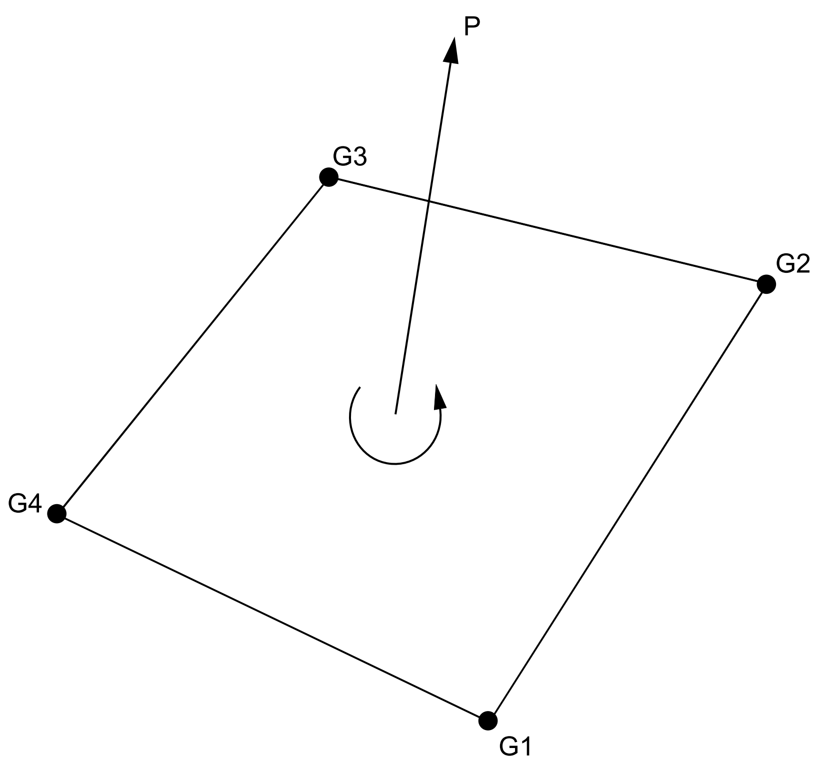


Figure 2. Pressure Convention for Quadrilateral Surface.

PLOAD1 — Applied Loads on Bar and Beam Elements

Description: Defines concentrated, uniformly distributed, or linearly distributed applied loads to CBAR and CBEAM elements at user chosen points along the axis.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	

Example:

PLOAD1	4	102	MYE	FRPR	0.1	2.5+3	0.8	1.5+2	

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
EID	Element identification number.	Integer > 0	Required
TYPE	Load type, one of the following character variables: FX, FY, FZ, FXE, FYE, FZE, MX, MY, MZ, MXE, MYE, and MZE.	Character	Required
SCALE	Determines scale factor for X1, X2. Must be one of following character variables: LE, FR, LEPR, or FRPR.	Character	Required
X1, X2	Distances along element axis from end A.	X2 ≥ X1 ≥ 0.0; X2 may be real or blank	
P1, P2	Load factors at positions X1, X2.	Real or blank	

Remarks:

1. If X2 ≠ 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 7 and 10 below.
2. If X2 is blank or equal to X1, a concentrated load of value P1 will be applied at position X1.
3. 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 7 and 10 below.
4. Load TYPE symbols are 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 coordinate system.
    - MXE, MYE, or MZE: Moment in the x, y, or z direction of the element coordinate system.
5. If SCALE = LE (length), the Xi values are actual distances along the bar x-axis, and (if X1 ≠ X2) Pi are load intensities per unit length of the bar.

6. If SCALE = FR (fractional), the Xi values are ratios of the distance along the axis to the total length, and (if X2 ≠ X1) Pi are load intensities per unit length of the element.
7. If SCALE = LEPR (length projected), the Xi values are actual distance along the bar x-axis and (if X2 ≠ X1) the distributed load is input in terms of the projected length of the bar.

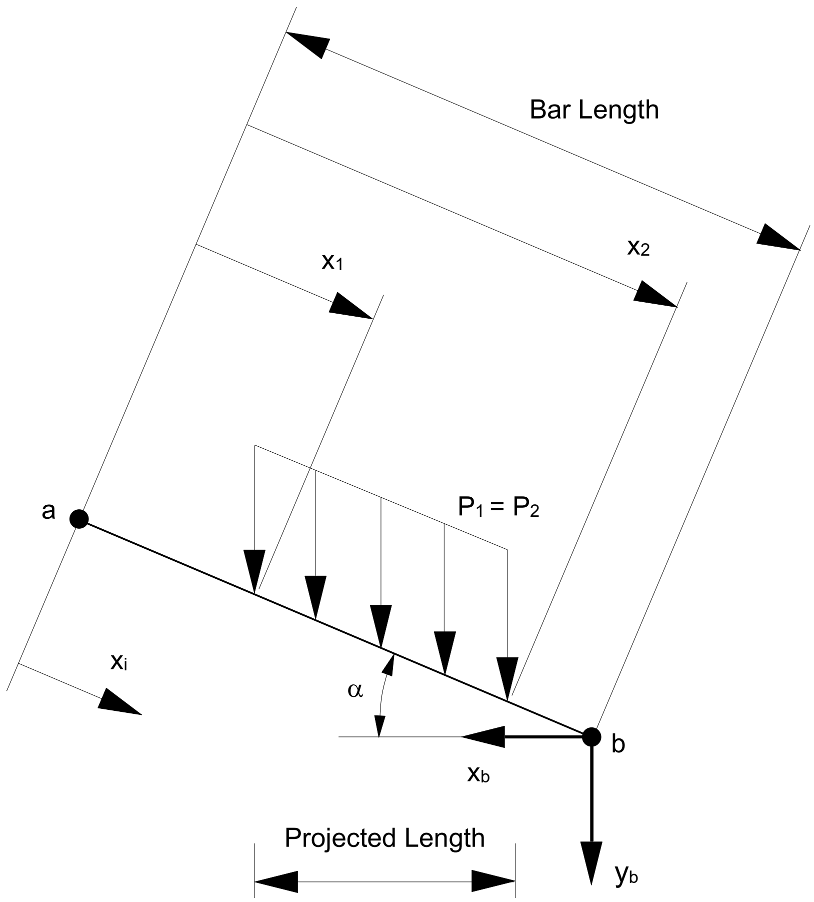


Figure 1. PLOAD1 Convention on Bar and Beam Elements.

8. If SCALE = LE, the total load applied to the bar is P1(X2 – X1) in the y_b direction.
9. If SCALE = LEPR, the total load applied to the bar is P1(X2 – X1)COS(α) in the y_b direction.
10. If SCALE = FRPR (fractional projected), the Xi values are ratios of the actual distance to the length of the bar and (X1 ≠ X2) the distributed load is input in terms of the projected length of the bar.
11. Load sets must be selected in the Case Control Section (LOAD = SID).

PLOAD2 Pressure Load on Shell Elements

Description: Defines a uniform static pressure load applied to shell elements. Only CQUAD4, CQUADR, CSHEAR, CTRIA3, or CTRIAR elements may have a pressure load applied to them via this entry.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	

Example:

PLOAD2	30	-1.3	106	222	21				

Alternate Format and Example:

PLOAD2	SID	P	EID1	THRU	EID2				

PLOAD2	40	12.0	16	THRU	122				

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
P	Pressure value.	Real or blank	
EIDi	Element identification number(s).	Integer > 0; EID1 < EID2	Required

Remarks:

1. Load sets must be selected in the Case Control Section (LOAD = SID).
2. At least one EID must be present on each PLOAD2 entry.
3. If the alternate form is used, all elements EID1 through EID2 that are not compatible or do not exist will be skipped.
4. Elements must not be specified more than once.
5. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element entry.
6. All elements directly referenced must exist.
7. Continuations are not allowed.

PLOAD4 Pressure Loads on Face of Shell and Solid Elements

Description: Defines a load on a face of a shell or solid element. Only CQUAD4, CQUADR, CTRIA3, CTRIAR, CHEXA, CPENTA, CPYRA, and CTETRA elements may have a pressure load applied to them via this entry.

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					

Example:

PLOAD4	2	1405	1.0	1.5	1.5	1.0			

Alternate Format and Example:

PLOAD4	SID	EID1	P1	P2	P3	P4	THRU	EID2	
	CID	N1	N2	N3					

PLOAD4	2	1106	10.0	8.0	5.0		THRU	1143	
	6	0.0	1.0	0.0					

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
EID	Element identification number.	Integer > 0	Required
P1, P2, P3, P4	Load per unit surface area (pressure) at the corners of the face of the element.	Real or blank	P1 is the default for P2, P3, and P4
G1	Identification number of a grid point connected to a corner of the face.	Integer > 0 or blank	Required for solid elements
G3	Identification number of a grid point connected to a corner diagonally opposite to G1 on the same face of a CHEXA, CPENTA, or CPYRA element. Required for the quadrilateral faces of CHEXA, CPENTA, and CPYRA elements. Must be omitted for a triangular face on a CPENTA or CPYRA element.	Integer > 0 or blank	Required for CHEXA and CPENTA elements
G4	Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded.	Integer > 0 or blank	Required for CTETRA elements

Field	Definition	Type	Default
CID	Coordinate system identification number.	Integer ≥ 0 or blank	See Remark 2
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.	Real	Required if CID is not blank and must have at least one non-zero component

Remarks:

1. Load sets must be selected in the Case Control Section (LOAD = SID).
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 shell elements, the direction of positive pressure (default 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.
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 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, CTRIAR, CQUAD4, and CQUADR elements.
8. The alternate format is available only for CTRIA3, CTRIAR, CQUAD4, 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 CPYRA elements, G1 must be a grid point on the quadrilateral face. For faces of CTETRA elements, G1 is the 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 element has only four corner points, G4 will be unique and different for each of the four faces of a CTETRA element.

PLOADG — Pressure Load at a Grid Point

Description: Defines a pressure load at a grid point by specifying a vector.

Format:

1	2	3	4	5	6	7	8	9	10
PLOADG	SID	G	CID	P	N1	N2	N3		

Example:

PLOADG	3	110		10.0	0.0	1.0	0.0		

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
G	Grid point identification number	Integer > 0	Required
CID	Coordinate system identification number.	Integer ≥ 0 or blank	0
P	Load per unit surface area (pressure).	Real	Required
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.	Real	Required; must have at least one nonzero component

Remarks:

1. This entry can only be used for input to the LOADINTERPOLATE Case Control command.
2. The TRSLPRESDATA directive may be used to convert PLOAD2 and PLOAD4 pressures to PLOADG. (See Section 2, *Initialization*, for more information on TRSLPRESDATA.)

PLOADX1 Pressure Load on Axisymmetric Elements

Description: Defines surface tractions to be used with solid axisymmetric elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOADX1	SID	EID	PA	PB	GA	GB	THETA		

Example:

PLOADX1	4	102	MYE	FRPR	0.1	2.5+3	0.8	1.5+2	

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
EID	Element identification number.	Integer > 0	Required
PA, PB	Surface tractions at grid points GA and GB.	Real	PA is the default for PB
GA, GB	Corner grid points. GA and GB are any two adjacent corner grid points of the element.	Integer > 0	Required
THETA	Angle between surface traction and inward normal to the line segment.	Real	0.0

Remarks:

1. Load sets must be selected in the Case Control Section (LOAD = SID).
2. PLOADX1 is intended only for the CTRIAX6 element.
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 1. Positive pressure is in the direction of inward normal to the line segment.

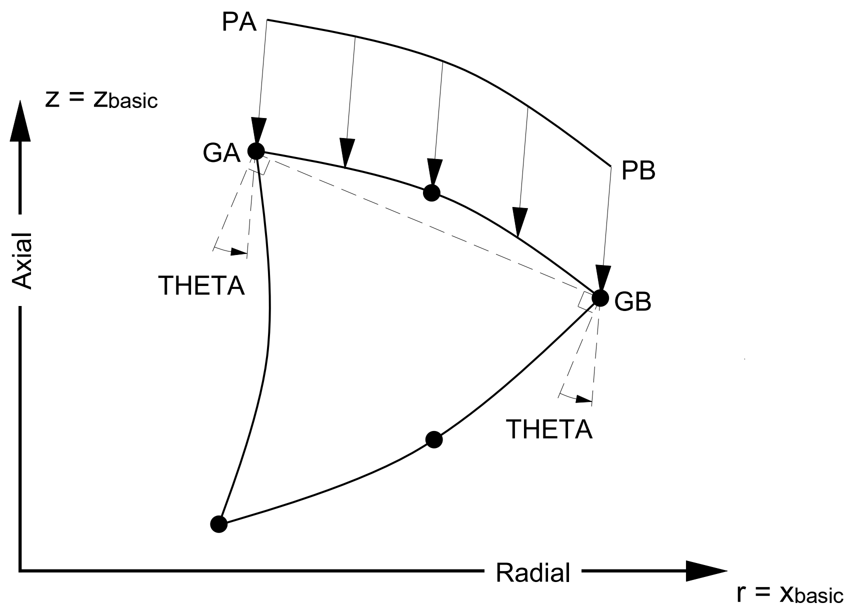


Figure 1. PLOADX1 Convention on Axisymmetric Elements.

PLSOLID Nonlinear Large Strain Solid Element Property

Description: Defines a nonlinear large strain solid element property (CHEXA, CPENTA, CPYRA, and CTETRA elements only).

Format:

1	2	3	4	5	6	7	8	9	10
PLSOLID	PID	MID	MCID						

Example:

PLSOLID	2	100	6						

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Identification number of a MAT1, MAT9, MAT12, MATHP, or MATHP1 entry.	Integer > 0	Required
MCID	Identification number of the material coordinate system. See Remarks 3 and 4.	Integer ≥ -1 or blank	See Remark 3

Remarks:

1. PLSOLID entries must have unique identification numbers.
2. Isotropic (MAT1), anisotropic (MAT9), or orthotropic (MAT12) material properties may be referenced.
3. See the CHEXA, CPENTA, CPYRA, or CTETRA entry for the definition of the element coordinate system. The material coordinate system (MCID) may be the basic system (0), any defined system (Integer > 0), or the element coordinate system (-1 or blank). The default for MCID is the element coordinate system.
4. If MID references a MAT9 entry, then MCID defines the material property coordinate system for Gij on the MAT9 entry. If MID references a MAT12 entry, then MCID defines the material property coordinate system for the Ei, Gi, and NUij on the MAT12 entry.

PMASS — Scalar Mass Property

Description: Specifies the mass value of a scalar mass element (CMASS1 entries).

Format:

1	2	3	4	5	6	7	8	9	10
PMASS	PID1	M1	PID2	M2	PID3	M3	PID4	M4	

Example:

PMASS	5	7.26	4	17.8					

Field	Definition	Type	Default
PIDi	Property identification number.	Integer > 0	Required
Mi	Mass value.	Real	Required

Remarks:

1. PMASS entries must all have unique property identification numbers.
2. The use of negative mass values may result in fatal errors.
3. Up to four mass values may be defined by this entry.

PMOUNT — Nonlinear Shock and Vibration Element Property

Description: Specifies the nonlinear properties of a shock and vibration element (CBUSH1D entries).

Format:

1	2	3	4	5	6	7	8	9	10
PMOUNT	PID	TFKID	TFBID	TFCID				F0	

Example:

PMOUNT	10	100	110	120				-1050.6	

Alternate Format and Example:

PMOUNT	PID	TFKID	TFBID	TFCID1	TFCID2	TFCID3	TFCID4	F0	

PMOUNT	10	100	110	120	130	140	150		

Field	Definition	Type	Default
PID	Property identification number that matches the identification number on a PBUSH1D entry.	Integer > 0	Required
TFKID	Identification number of a TABLEDi entry that defines a nonlinear elastic spring element in terms of a force versus displacement relationship. $F(u) = F_T(u)$ Tension is $u > 0$ and compression is $u < 0$.	Integer > 0	Required
TFBID	Identification number of a TABLEDi entry that defines a nonlinear viscous element in terms of a force versus velocity relationship. $F(v) = F_T(v)$ Tension is $v > 0$ and compression is $v < 0$.	Integer > 0	Required
TFCID	Identification number of a TABFV entry that defines stiffness-damping coupling in terms force versus displacement tables at constant velocity. See Remark 1. $F(u,v) = F_T(u,v)$	Integer ≥ 0	0
TFCIDi	Identification numbers of TABLED4 entries that define stiffness-damping coupling in the form of a power series. See Remark 1.	Integer ≥ 0	0

Field	Definition	Type	Default
F0	Preload.	Real	0.0

Remarks:

1. There are two displacement/velocity-dependent forms that may be used to define the nonlinear stiffness and damping characteristics of this element. In each the displacement u and velocity v are the relative displacement and relative velocity with respect to grid point GA. In the first form the force versus velocity/displacement relationship is given by

$$F(u,v) = [F_k(u)]u + [F_b(v)]v + [F_c(u,v)]u$$

where the force due to stiffness $F_k(u)$ is given by TFKID, the force due to damping $F_b(v)$ is given by TFBID, and the force due to stiffness-damping coupling $F_c(u,v)$ is given by TFCID which defines force versus displacement data for a constant velocity.

Term	Field	Table Type
$F_k(u)$	TFKID	TABLEDi
$F_b(v)$	TFBID	TABLEDi
$F_c(u,v)$	TFCID	TABFV

In the second form the force versus velocity/displacement relationship is given by a power series of the form

$$F(u,v) = \mathrm{B}_1\left(u + \frac{v}{\mathrm{A}}\right) + \mathrm{B}_2\left(u + \frac{v}{\mathrm{A}}\right)^2 + \mathrm{B}_3\left(u + \frac{v}{\mathrm{A}}\right)^3 + \mathrm{B}_4\left(u + \frac{v}{\mathrm{A}}\right)^4 + \mathrm{B}_5\left(u + \frac{v}{\mathrm{A}}\right)^5$$

which may be further reduced to

$$F(u,v) = [F_k(u)]u + [F_b(v)]v + [F_c(u,v)]u$$

where

$$F_k(u) = \left(\mathrm{B}_1 + \mathrm{B}_2 u + \mathrm{B}_3 u^2 + \mathrm{B}_4 u^3 + \mathrm{B}_5 u^5\right)$$

$$F_b(v) = \left(\mathrm{C}_1 + \mathrm{C}_2 v + \mathrm{C}_3 v^2 + \mathrm{C}_4 v^3 + \mathrm{C}_5 v^5\right)$$

$$F_c(u,v) = \left(\mathrm{D}_1 v + \mathrm{E}_1 v^2 + \mathrm{F}_1 v^3\right) + \left(\mathrm{D}_2 v + \mathrm{E}_2 v^2 + \mathrm{F}_2 v^3\right)u + \left(\mathrm{D}_3 v + \mathrm{E}_3 v^2\right)u^2 + \left(\mathrm{D}_4 v\right)u^3$$

and

$$\mathrm{C}_1 = \frac{\mathrm{B}_1}{\mathrm{A}},\ \mathrm{C}_2 = \frac{\mathrm{B}_1}{\mathrm{A}^2},\ \mathrm{C}_3 = \frac{\mathrm{B}_1}{\mathrm{A}^3},\ \mathrm{C}_4 = \frac{\mathrm{B}_1}{\mathrm{A}^4},\ \mathrm{C}_5 = \frac{\mathrm{B}_1}{\mathrm{A}^5}$$

$$\mathrm{D}_1 = 2\frac{\mathrm{B}_2}{\mathrm{A}},\ \mathrm{D}_2 = 3\frac{\mathrm{B}_3}{\mathrm{A}},\ \mathrm{D}_3 = 4\frac{\mathrm{B}_4}{\mathrm{A}},\ \mathrm{D}_4 = 5\frac{\mathrm{B}_5}{\mathrm{A}}$$

$$\mathrm{E}_1 = 3\frac{\mathrm{B}_3}{\mathrm{A}^2},\ \mathrm{E}_2 = 6\frac{\mathrm{B}_4}{\mathrm{A}^2},\ \mathrm{E}_3 = 10\frac{\mathrm{B}_5}{\mathrm{A}^2}$$

$$\mathrm{F}_1 = 4\frac{\mathrm{B}_4}{\mathrm{A}^3},\ \mathrm{F}_2 = 6\frac{\mathrm{B}_5}{\mathrm{A}^3}$$

Term	Field	Table Type
$F_k(u)$	TFKID	TABLED4
$F_b(v)$	TFBID	TABLED4
$F_c(u,v)$	TFCIDi	TABLED4

2. 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 $F(-u) = -F(u)$.

PPIPE — Pipe Element Property

Description: Defines the properties of pipe elements (CPIPE entry).

Format:

1	2	3	4	5	6	7	7	9	10
PPIPE	PID	MID	OD	T	P	EC	NSM		

Example:

PPIPE	50	30	1.2	0.1	100.5				

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number.	Integer > 0	Required
OD	Pipe outer diameter.	Real > 0.0	Required
T	Pipe wall thickness.	0.0 < Real ≤ OD/2.0	Required
P	Internal pressure.	Real or blank	0.0
EC	End condition, one of the following character variables: CLOSED or OPEN: CLOSED Both ends are closed. OPEN Both ends are open.	Character	CLOSED
NSM	Nonstructural mass per unit length.	Real or blank	0.0

Remarks:

1. PPIPE entries must all have unique property identification numbers.
2. For structural problems, PPIPE entries may only reference MAT1 material entries.
3. Hoop stress due to internal pressure and longitudinal, shear, and torsional stress due end forces and moments are combined to generate invariant stresses as follows

Maximum shear stress:

$$\tau_{max} = \left[\left(\frac{\sigma_L - \sigma_H}{2}\right)^2 + (\sigma_T)^2\right]^{\frac{1}{2}}$$

Maximum principal stress:

$$\sigma_{max} = \left(\frac{\sigma_L + \sigma_H}{2}\right) + \tau_{max}$$

Octahedral shear stress:

$$\tau_o = \left[\frac{2}{9}\left(\frac{\sigma_L + \sigma_H}{2}\right)^2 + 3(\sigma_H)^2\right]^{\frac{1}{2}}$$

PROD — Rod Element Property

Description: Defines the properties of rod elements (CROD entry).

Format:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	NSM			

Example:

PROD	44	100	0.1	2.-3	0.12				

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number.	Integer > 0	Required
A	Area of rod cross-section.	Real	Required
J	Torsional constant.	Real or blank	0.0
C	Coefficient to determine torsional stress.	Real or blank	0.0
NSM	Nonstructural mass per unit length.	Real or blank	0.0

Remarks:

1. PROD entries must all have unique property identification numbers.
2. For structural problems, PROD entries may only reference MAT1 material entries.
3. The formula used to compute torsional stress is

$$\tau = \frac{TC}{J}$$

where T is the torsional moment.

PSHEAR — Shear Panel Property

Description: Defines the properties of shear elements (CSHEAR entry).

Format:

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM	F1	F2	F3	F4	

Example:

PSHEAR	44	100	0.1	0.72	3.24	0.5			

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number.	Integer > 0	Required
T	Thickness of shear panel.	Real	Required
NSM	Nonstructural mass per unit area.	Real	0.0
F1	Effectiveness factor for extensional stiffness along edge 1-2. See Remark 3.	Real ≥ 0.0	0.0
F2	Effectiveness factor for extensional stiffness along edge 2-3. See Remark 3.	Real ≥ 0.0	0.0
F3	Effectiveness factor for extensional stiffness along edge 3-4. See Remark 3.	Real ≥ 0.0	F1
F4	Effectiveness factor for extensional stiffness along edge 4-1. See Remark 3.	Real ≥ 0.0	F2

Remarks:

1. All PSHEAR entries must have unique identification numbers.
2. PSHEAR entries may reference only MAT1 material entries when PARAM, SHEARELEMTYPE is set to NASTRAN.
3. The effective extensional area is defined by means of equivalent rods on the perimeter of the element. If F1 ≤ 1.01, the area of the rod on edge 1-2 is set equal to (F1∗T∗PA)/(L12+L34) where PA is the panel surface area and L12, L34 are the lengths of sides 1-2 and 3-4. Thus, if F1 = F3 = 1.0, the panel is fully effective for extension in the 1-2 direction. If F1 > 1.01, the area of the rod on edge 1-2 is set equal to $0.5 * F1 * T^2$. In the case of an orthotropic material (MAT8) E1 will be used for F1 and F3 and E2 for F2 and F4.
4. Poisson's ratio coupling for extensional effects is ignored.

PSHELL — Shell Element Property

Description: Defines the membrane, bending, and transverse shear properties of shell elements (CTRIA3, CTRIAR, CQUAD4, and CQUADR entries).

Format:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12I/T3	MID3	TS/T	NSM	
	Z1	Z2	MID4	THETA/MCID	SDIR	SC	SFACTCX	SFACTCY	
	RTYPE	F1	F2	F3	F4				

Example:

PSHELL	44	100	0.1	0.72					

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID1	Material identification number for the membrane.	Integer > 0 or blank	
T	Default membrane thickness for Ti on the connection entry.	Real or blank	
MID2	Material identification number for bending.	Integer ≥ -1 or blank	See Remark 18
12I/T3	Bending stiffness parameter.	Real or blank	1.0
MID3	Material identification number for transverse shear.	Integer > 0 or blank, must be blank unless MID2 > 0	See Remark 18
TS/T	Transverse shear thickness divided by the membrane thickness.	Real or blank	0.833333
NSM	Nonstructural mass per unit area.	Real or blank	0.0
Z1, Z2	Fiber distances for stress computation. The right-hand rule and the order in which the grid points are listed on the connection entry determine the positive direction.	Real or blank	See Remark 10
MID4	Material identification number for membrane-bending coupling. See Remark 6.	Integer > 0 or blank, must be blank unless MID1 > 0, MID2 > 0, and MID3 > 0, may not equal MID1, MID2, or MID3	

Field	Definition	Type	Default
THETA	Material property orientation angle in degrees.	Real or blank	See Remark 12
MCID	Material coordinate system identification number.	Integer ≥ 0	See Remark 12
SDIR	Element stress component direction for tension-only element. See Remark 13.	0 ≤ Integer ≤ 4	0
SC	Compression allowable. Stress components, as defined by SDIR, less than this value will degenerate the element membrane stiffness to a shear panel.	Real	0.0
SFACTCX	Compression stiffness scale factor in the element x-direction. See Remark 14.	Real	1.0E-10
SFACTCY	Compression stiffness scale factor in the element y-direction. See Remark 14.	Real	SFACTCX
RTYPE	Reversion element type. See Remarks 15 and 16. 1 = Revert to a tension-only shell element 2 = Revert to a full shear panel element	1 ≤ Integer ≤ 2	1
F1	Effectiveness factor for extensional stiffness along edge 1-2. See Remark 17.	Real ≥ 0.0	0.0
F2	Effectiveness factor for extensional stiffness along edge 2-3. See Remark 17.	Real ≥ 0.0	0.0
F3	Effectiveness factor for extensional stiffness along edge 3-4. See Remark 17.	Real ≥ 0.0	F1
F4	Effectiveness factor for extensional stiffness along edge 4-1. See Remark 17.	Real ≥ 0.0	F2

Remarks:

1. All PSHELL property entries must have unique identification numbers.
2. The translational structural mass is computed from the membrane material density and rotational mass from the bending material density.
3. PSHELL entries may reference MAT1, MAT2, or MAT8 material property entries. If element reversion to a full shear panel element is specified in a nonlinear solution and a MAT2 or MAT8 material is referenced, PARAM, SHEARELEMTYPE must be set to NORAN or AUTO or a fatal error will be issued.
4. If the transverse shear material, MID3, references a MAT2 data entry, then G13, G23, and G33 must be zero or blank.
5. The results of leaving an MID field blank are:

   MID1 No membrane or coupling stiffness.
   MID2 No bending or coupling stiffness
   MID3 No transverse shear stiffness or coupling stiffness
   MID4 No membrane-bending coupling unless ZOFFS is specified on the connection entry. See Remark 6.

   Note: MID1, MID2, and MID3 must be specified if the ZOFFS field is also specified on the connection entry.

6. The MID4 field should be left blank if the material properties are symmetric with respect to the mid-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.
7. If MID3 references an isotropic material via a MAT1 entry:

$$\begin{Bmatrix} \tau_{zx} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} G & 0 \\ 0 & G \end{bmatrix} \begin{Bmatrix} \gamma_{zx} \\ \gamma_{yz} \end{Bmatrix}$$

8. If MID3 references an anisotropic material via a MAT2 entry:

$$\begin{Bmatrix} \tau_{zx} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} \\ G_{12} & G_{22} \end{bmatrix} \begin{Bmatrix} \gamma_{zx} \\ \gamma_{yz} \end{Bmatrix}$$

9. If MID3 references an orthotropic material via a MAT8 entry:

$$\begin{Bmatrix} \tau_{zx} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} G_{1z} & 0 \\ 0 & G_{2z} \end{bmatrix} \begin{Bmatrix} \gamma_{zx} \\ \gamma_{yz} \end{Bmatrix}$$

10. 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 thickness' at connected grid points, if they are input on connection entries.
11. For plane strain analysis, set MID2 = -1 and set MID1 to reference a MAT1 entry.
12. THETA/MCID is used only if field 8 of the CQUAD4 or CQUADR, or field 7 of the CTRIA3 or CTRIAR entry is blank. If field 5 of the PSHELL continuation is also blank, then THETA = 0.0 is assumed when a non-isotropic material is referenced.
13. The SDIR field specifies which element stress component direction should be used when determining if the element has failed and should revert to a shear panel.

SDIR	Description
0	Standard tension/compression shell element
1	Use the element membrane normal-x stress
2	Use the element membrane normal-y stress
3	Use either the element membrane normal-x or normal-y stress
4	Standard shear panel element

14. The SFACTCi scale factors are applicable when RTYPE = 1 and are used to reduce the membrane stiffness of the reverted tension-only element. The default value will revert the element membrane contribution to tension-only while a value of 1.0 will result in no change in behavior (standard shell element). Intermediate values provide for some compressive load carrying capability in the element. Values greater than 1.0 are normalized relative to the element width (CQUAD4/CQUADR elements only). For example in the element x-direction the scale factor used is $SFACTCx * T/w_y$.

15. The RTYPE setting determines the element type after reversion. The default value of 1 will revert the membrane portion of the element to tension-only behavior with limited compressive load carrying capability determined by the SFACTCi settings. If PARAM, FIXNLTOQUAD is set to OFF and the element load state changes back to tension, the element will revert back to a normal shell element. When RTYPE is set to 2, the element reverts to a full shear panel with the extensional stiffness defined by the effectiveness factors, Fi and no bending or transverse shear capability. With this setting once the element has reverted it will remain a shear panel regardless of the PARAM, FIXNLTOQUAD setting.
16. Element reversion to tension-only behavior requires a nonlinear solution. Tension-only behavior may be disabled by setting PARAM, NLTOQUAD to OFF (default is ON).
17. The Fi effectiveness factors are applicable when RTYPE = 2 and the element has reverted to a full shear panel element (CSHEAR). The effective extensional area is defined by means of equivalent rods on the perimeter of the element. If F1 $\leq$ 1.01, the area of the rod on edge 1-2 is set equal to (F1∗T∗PA)/(L12+L34) where PA is the panel surface area and L12, L34 are the lengths of sides 1-2 and 3-4. Thus, if F1 = F3 = 1.0, the panel is fully effective for extension in the 1-2 direction. If F1 $>$ 1.01, the area of the rod on edge 1-2 is set equal to $0.5 * F1 * T^2$. The rod material used is the same as the parent element. In the case of an orthotropic material (MAT8) E1 will be used for F1 and F3 and E2 for F2 and F4.
18. The default for the MID2 and MID3 fields is MID1 when MID1 is a nonlinear material.

PSOLID Solid Element Property

Description: Defines the properties of solid elements (CHEXA, CPENTA, CPYRA, and CTETRA entries).

Format:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	MCID	PCPID					

Example:

PSOLID	2	100	6						

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Identification number of a MAT1, MAT9, MAT12, or MATHP, or MATHP1 entry.	Integer > 0	Required
MCID	Identification number of the material coordinate system. See Remarks 3 and 4.	Integer ≥ -1 or blank	See Remark 3
PCPID	Identification number of a PCOMP entry. See Remark 5.	Integer > 0	Required

Remarks:

1. PSOLID entries must have unique identification numbers.
2. Isotropic (MAT1), anisotropic (MAT9), or orthotropic (MAT12) material properties may be referenced.
3. See the CHEXA, CPENTA, CPYRA, or CTETRA entry for the definition of the element coordinate system. The material coordinate system (MCID) may be the basic system (0), any defined system (Integer > 0), or the element coordinate system (-1 or blank). The default for MCID is the element coordinate system.
4. If MID references a MAT9 entry, then MCID defines the material property coordinate system for Gij on the MAT9 entry. If MID references a MAT12 entry, then MCID defines the material property coordinate system for the Ei, Gi, and NUij on the MAT12 entry.
5. A non-zero PCPID value in field 5 specifies a layered solid element where the ply definitions are given on the referenced PCOMP Bulk Data entry. The ply orientation is relative to the element material x-direction similar to that of a composite shell element. The element material x-direction is defined by projecting the MCID x-axis onto a surface defined by the element z-axis. The element z-axis also defines the element thickness direction. Only CHEXA and CPENTA elements may be referenced if the property defines a layered solid element.

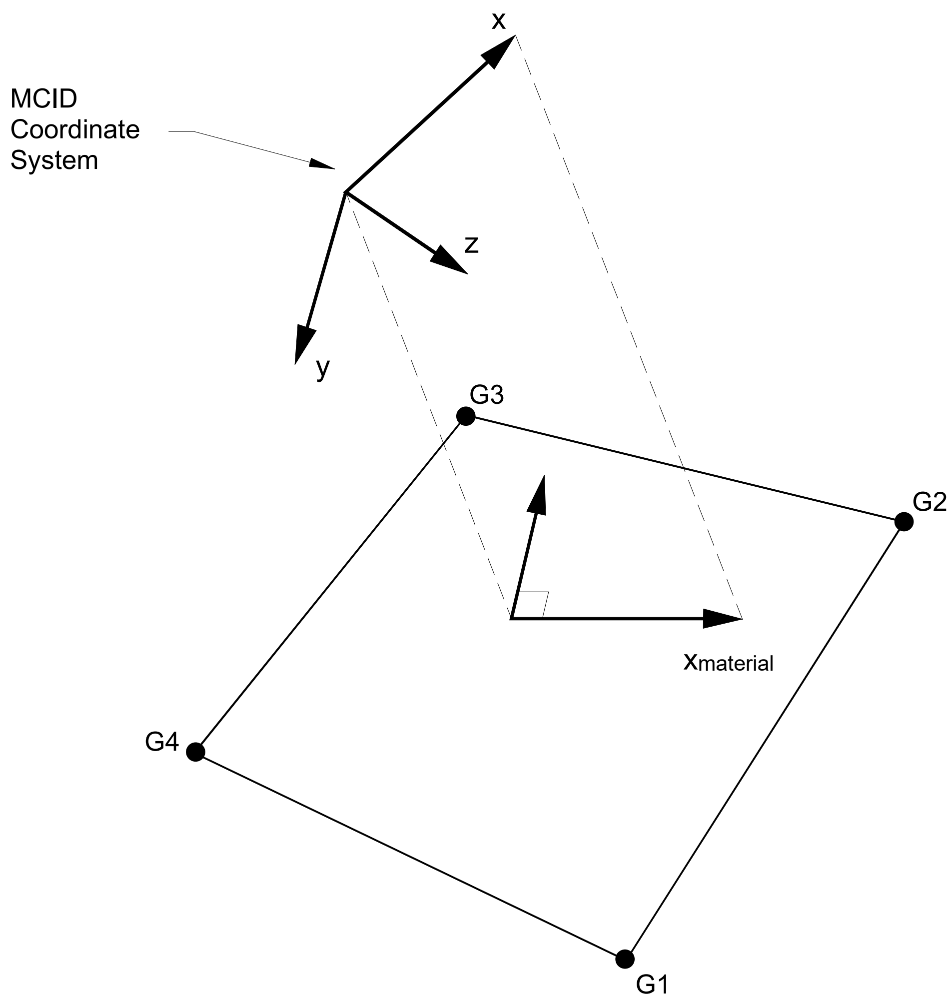


Figure 1. Layered Solid Element MCID Coordinate System Definition.

PTUBE

Tube Element Property

Description: Defines the properties of a cylindrical tube element (CTUBE entry).

Format:

1	2	3	4	5	6	7	8	9	10
PTUBE	PID	MID	OD	T	NSM				

Example:

PTUBE	50	30	1.2	0.1					

Field	Definition	Type	Default
PID	Property identification number.	Integer $>$ 0	Required
MID	Material identification number.	Integer $>$ 0	Required
OD	Tube outer diameter.	Real $>$ 0.0	Required
T	Tube wall thickness.	0.0 $<$ Real $\leq$ OD/2.0	Required
NSM	Nonstructural mass per unit length.	Real or blank	0.0

Remarks:

1. PTUBE entries must all have unique property identification numbers.
2. For structural problems, PTUBE entries may only reference MAT1 material entries.

PVISC — Viscous Damping Element Property

Description: Defines the properties of a 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	4	5.3	2.57						

Field	Definition	Type	Default
PIDi	Property identification number.	Integer > 0	Required
CE1, CE2	Viscous damping values for extension in units of force per unit velocity.	Real or blank	0.0
CR1, CR2	Viscous damping values for rotation in units of moment per unit velocity.	Real or blank	0.0

Remarks:

1. PVISC entries must all have unique property identification numbers.
2. Viscous properties are material (temperature) independent.
3. One or two viscous element properties may be defined on a single entry.

PWELD — WELD Element Property

Description: Defines the properties of a connector element (CWELD entry).

Format:

1	2	3	4	5	6	7	8	9	10
PWELD	PID	MID	D					CTYPE	

Example:

PWELD	200	5	1.5						

Field	Definition	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 2.	Integer > 0	Required
CTYPE	Weld connection type, one of the following character variables: SPOT or GENERAL. See Remark 3. SPOT — Weld type connection. GENERAL — General connection.	Character	GENERAL
D	Diameter of the connector. See Remark 2.	Real > 0.0	Required

Remarks:

1. PWELD entries must all have unique property identification numbers.
2. Material MID, diameter D and the length are used to calculate the stiffness of the connector in all 6 component directions. MID can only refer to the MAT1 Bulk Data entry. The length is the distance of GA to GB as shown in Figure 1.
3. For CTYPE = SPOT and FTYPE = ELEMID on the CWELD entry, the effective length for the stiffness of the weld element is set to $\ell_e = (t_A + t_B)/2$ regardless of the distance GA to GB. t_A and t_B are the shell thicknesses of SHIDA and SHIDB on the CWELD entry. For all other cases, the effective length of the weld element is equal to the true length, the distance of GA to GB, provided the ratio of length to diameter is in the range $0.2 \leq L/D \leq 5.0$. If L is below this range, the effective length is set to $\ell_e = 0.2D$ and if L is above this range, the effective length is set to $\ell_e = 5.0D$.

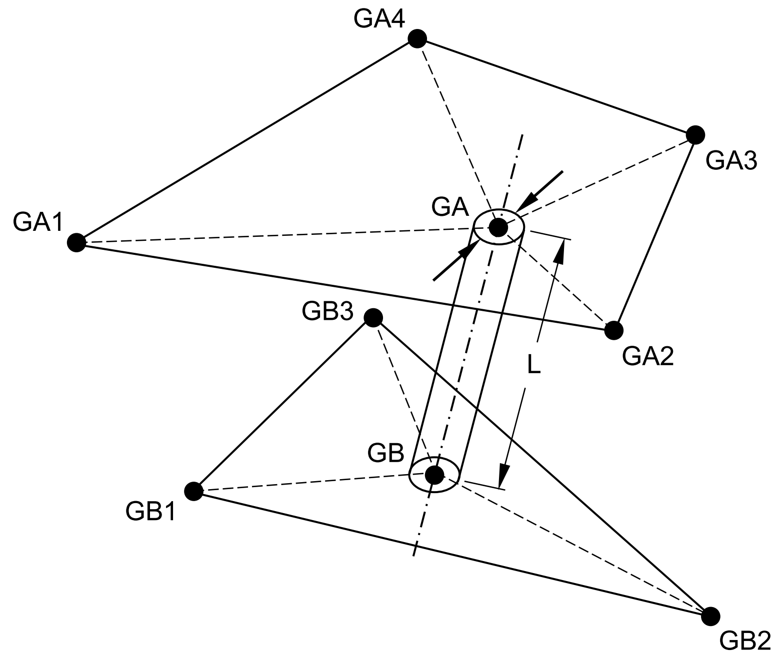


Figure 1. Length and Diameter of the Weld Connector.

QBDY1 Boundary Heat Flux Load for CHBDYj Elements

Description: 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	103	2.-4	25						

Alternate Format and Example:

QBDY1	SID	Q0	EID1	THRU	EID2				

QBDY1	10	5.4	16	THRU	122				

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
Q0	Heat flux into element.	Real	Required
EIDi	CHBDYj element identification number(s).	Integer > 0; EID2 > EID1	Required

Remarks:

1. QBDY1 entries must be selected with the Case Control command LOAD = SID in order to be used in steady state heat transfer analysis.
2. The total power into an element is given by the equation:

$$P_{in} = (\text{Effective area}) * Q0$$

3. Q0 is positive for heat input.
4. At least one EID must be present on each QBDY1 entry.
5. If the alternate form is used, all elements EID1 through EID2 that are not compatible or do not exist will be skipped.
6. Elements must not be specified more than once.
7. All elements directly referenced must exist.
8. Continuations are not allowed.

QBDY2 Boundary Heat Flux Load for CHBDYj Elements, Form 2

Description: 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	15	120	1.-5						

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
EID	Element identification number of a CHBDYj element.	Integer > 0	Required
Q0i	Heat flux at the i-th grid point on the referenced CHBDYj element.	Real	Required

Remarks:

1. QBDY2 entries must be selected with the Case Control command LOAD = SID in order to be used in steady state heat transfer analysis.
2. The total power into each point i on an element is given by the equation:

$$P_{in} = \text{Area}_i * \text{Q0}$$

3. Q0i is positive for heat flux input to the element.

QBDYG Heat Flux Load at a Grid Point

Description: Defines a heat flux load at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
QBDYG	SID	G	Q0						

Example:

QBDYG	5	120	10.0						

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
G	Grid point identification number	Integer > 0	Required
Q0	Heat flux into grid point.	Real	Required

Remarks:

1. This entry can only be used for input to the LOADINTERPOLATE Case Control command.

QHBDY Boundary Heat Flux Load

Description: Defines a uniform heat flux into a set of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
QHBDY	SID	TYPE	Q0	AF	G1	G2	G3	G4	
	G5	G6	G7	G8					

Example:

QHBDY	5	AREA4	14.5		10	11	12	13	

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
TYPE	Surface type, one of the following character variables: POINT, LINE, AREA3, AREA4, AREA6, or AREA8. See Remark 2.	Character	Required
Q0	Magnitude of thermal flux into face.	Real	Required
AF	Area factor depends on type.	Real > 0.0 or blank	0.0
Gi	Grid point identification of connected grid points.	Integer > 0 or blank	Required

Remarks:

1. QHBDY entries must be selected with the Case Control command LOAD = SID in order to be used in steady state heat transfer analysis.
2. The heat flux applied to the area is transformed to loads on the points. These points need not correspond to an HBDY surface element.
3. The total power into each point i is given by the equation:

$$P_{in} = \text{Area}_i * Q0$$

4. The number of connect points for the types are 1 (POINT), 2 (LINE), 3 (AREA3), 4 (AREA4), 4-6(AREA6), 5-8 (AREA8).
5. 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.
6. The type of face (TYPE) defines a surface in the same manner as the CHBDYi data entry. For descriptions of the geometry involved, see the CHBDYG discussion.
7. The continuation entry is optional.

QSET Generalized Degree of Freedom

Description: 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	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

QSET	15	1	17	456	7	4			

Field	Definition	Type	Default
Gi	Grid point identification number(s).	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \le$ Integers ≤ 6	Required

Remarks:

1. Degrees of freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
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).

QSET1 Generalized Degree of Freedom, Alternate Form

Description: 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
QSET1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	- etc.-					

Example:

QSET1	123456	6	3	7	10	18	14	11	
	19	23							

Alternate Format and Example:

QSET1	C	G1	THRU	G2					

QSET1	1	15	THRU	512					

Field	Definition	Type	Default
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks).	$1 \leq$ Integers ≤ 6	Required
Gi	Grid point identification number(s).	Integer > 0; G1 < G2	Required

Remarks:

1. If the alternate form is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will be skipped.
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).

QVOL — Volume Heat Addition

Description: 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		EID1	EID2	EID3	EID4	EID5	
	EID6	- etc.-							

Example:

QVOL	4	7.3		23	45	14	8		

Alternate Format and Example:

QVOL	SID	QVOL		EID1	THRU	EID2			

QVOL	40	12.0		101	THRU	221			

Field	Definition	Type	Default
SID	Load set identification.	Integer > 0	Required
QVOL	Power input per unit volume produced by a heat conduction element.	Real	Required
EIDi	Element identification number(s).	Integer > 0; EID2 > EID1	Required

Remarks:

1. QVOL entries must be selected with the Case Control command LOAD = SID in order to be used in steady state heat transfer analysis.
2. EIDi references material properties (MAT4 and MAT5) that include HGEN, the element material property for heat generation, which may be temperature-dependent. If HGEN is temperature-dependent, it is based on the average element temperature.
3. The total power into an element is given by the equation:

$$P_{in} = Volume * \mathrm{HGEN} * \mathrm{QVOL}$$

4. At least one EID must be present on each QVOL entry.
5. If the alternate form is used, all elements EID1 through EID2 that are not compatible or do not exist will be skipped.
6. Elements must not be specified more than once.

7. All elements directly referenced must exist.
8. The continuation entry is optional.

RADBC — Space Radiation Specification

Description: Specifies a CHBDYi element face for application of radiation boundary conditions.

Format:

1	2	3	4	5	6	7	8	9	10
RADBC	AMBND	FAMB	CNTRLND	EID1	EID2	EID3	EID4	EID5	

Example:

RADBC	4	1.0		5					

Alternate Format and Example:

RADBC	AMBND	FAMB	CNTRLND	EID1	THRU	EID2	BY	INC	

RADBC	4	1.0		100	THRU	220	BY	10	

Field	Definition	Type	Default
AMBND	Ambient point for radiation exchange.	Integer > 0	Required
FAMB	Radiation view factor between the face and the ambient point.	Real ≥ 0.0	Required
CNTRLND	Control point for free convection boundary condition.	Integer ≥ 0 or blank	0
EIDi	CHBDYi element identification number(s).	Integer > 0; EID2 > EID1	Required
INC	Element number increment.	Integer or blank	1

Remarks:

1. The basic exchange relationship can be expressed in one of the following forms:

   a) $q = \sigma * \text{FAMB} * u_{\text{CNTRLND}} * \left(\epsilon T^4 - \alpha \text{TAMB}^4\right)$, CNTRLND ≠ 0

   b) $q = \sigma * \text{FAMB} * \left(\epsilon T^4 - \alpha \text{TAMB}^4\right)$, CNTRLND = 0

2. AMBND is treated as a black body with its own ambient temperature for radiation exchange between the surface element and space.

3. Two PARAM entries are required when for radiation heat transfer:
   - TABS defines the absolute temperature scale factor used to convert temperature to absolute. (See Section 5, *Parameters*, for more information on TABS.)
   - SIGMA (σ) is the Stefan-Boltzmann constant. (See Section 5, *Parameters*, for more information on SIGMA.)
4. RADBC allows for surface radiation to space. The emissivity and absorptivity are supplied from a RADM entry.

RADCAV Radiation Cavity Identification

Description: Identifies the characteristics of each radiant enclosure.

Format:

1	2	3	4	5	6	7	8	9	10
RADCAV	ICAVITY	ELEAMB	SHADOW	SCALE					

Example:

RADCAV	1	1							

Field	Definition	Type	Default
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. See Remark 1.	Integer > 0, Unique among all CHBDYi elements	
SHADOW	Flag to control third body shading calculation during view factor calculation for each identified cavity, one of the following character variables: YES or NO. See Remark 2.	Character	YES
SCALE	View factor that the enclosure sum will be set to if a view factor is greater than 1.0. See Remark 3.	$0.0 \le$ Real ≤ 1.0	0.0

Remarks:

1. For the surface 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 in the VIEW Bulk Data entry as well as reduces the calculation time.
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.

RADM — Radiation Boundary Material Property

Description: Defines the radiation property of a boundary element for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RADM	RADMID	ABSORP	EMISIV						

Example:

RADM	12	0.8	0.8						

Field	Definition	Type	Default
RADMID	Material identification number.	Integer > 0	Required
ABSORP	Surface absorptivity.	$0.0 \leq$ Real ≤ 1.0	Required
EMISIV	Surface emissivity.	$0.0 \leq$ Real ≤ 1.0	Required

Remarks:

1. The RADM entry is directly referenced only by a CHBDYG or CHBDYP surface element entry.
2. Two PARAM entries are required when for radiation heat transfer:
   - TABS defines the absolute temperature scale factor used to convert temperature to absolute. (See Section 5, *Parameters*, for more information on TABS.)
   - SIGMA (σ) is the Stefan-Boltzmann constant. (See Section 5, *Parameters*, for more information on SIGMA.)

RADMT Radiation Boundary Material Property Temperature Dependence

Description: Specifies table references for temperature-dependent radiation boundary properties.

Format:

1	2	3	4	5	6	7	8	9	10
RADMT	RADMID	T(α)	T(ε)						

Example:

RADMT	11	10	20						

Field	Definition	Type	Default
RADMID	Material identification number	Integer > 0	Required
T(α)	TABLEMj identifier for surface absorptivity.	Integer $\geq$ 0 or blank	Required
T(ε)	TABLEMj identifier for surface emissivity.	Integer $\geq$ 0 or blank	Required

Remarks:

1. The basic quantities on the RADM entry of the RADMID are always multiplied by the corresponding tabular function.
2. Tables T(α) and T(ε) 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.

RADSET — Identifies a Set of Radiation Cavities

Description: 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	10	1	2	3					

Field	Definition	Type	Default
ICAVITYi	Unique identification number for a radiation cavity to be considered for enclosure radiation analysis.	Integer > 0	

Remarks:

1. For multiple radiation cavities, RADSET specifies which cavities are to be included in the analysis.

RANDPS — Power Spectral Density Specification

Description: 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	10	6	14	2.5	2.0	1			

Field	Definition	Type	Default
SID	Random analysis set identification number.	Integer > 0	Required
J	Subcase identification number of the excited load set.	Integer > 0	Required
K	Subcase identification number of the applied load set.	Integer > 0 or blank, K ≥ J	Required
X, Y	Components of complex number.	Real	0.0
TID	Identification number of a TABRND1 card which defines *G*(*F*)	Integer ≥ 0 or blank	See Remark 4

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 uncoupled power spectral density functions (i.e., no J < K entries) any number of J = K entries are allowed with unique values of J. For coupled power spectral density functions (i.e., some J < K entries) a maximum of four entries may be specified.
4. For TID=0 or blank, *G*(*F*)=1.0.
5. RANDPS Bulk Data entries may not reference subcases in a different loop. Loops are defined by a change in the FREQUENCY command.

RANDT1 — Autocorrelation Function Time Lag

Description: 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	Definition	Type	Default
SID	Random analysis set identification number.	Integer > 0	Required
N	Number of time lag intervals.	Integer > 0	Required
T0	Starting time lag.	Real ≥ 0.0	0.0
TMAX	Maximum time lag.	Real > T0	Required

Remarks:

1. Time lags 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:

$$T_i = T0 + \frac{TMAX - T0}{N}(i-1), \quad i = 1, \; N+1$$

RBAR Rigid Bar

Description: 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		

Example:

RBAR	12	3	7	123456					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
GA, GB	Grid point identification number of connection points.	Integer > 0	Required
CNA, CNB	Component numbers of independent degrees of freedom in the global coordinate system for the element at grid points GA and GB. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6 or blank	See Remark 1
CMA, CMB	Component numbers of dependent degrees of freedom in the global coordinate system assigned by the element at grid points GA and GB. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6 or blank	See Remark 2 and 3

Remarks:

1. The total number of components in CNA and CNB must equal six; for example, CNA = 1235, CNB = 34. Furthermore, they must jointly be capable of representing any general rigid body motion of the element.
2. If both CMA and CMB are zero or blank, all of the degrees of freedom not in CNA and CNB will be made dependent.
3. The dependent degrees of freedom specified on this entry may not additionally constrained by other rigid elements or single-point constraints.
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. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
6. Forces of multipoint constraint may be recovered with the MPCFORCE Case Control command.

RBE1 Rigid Body Element, Form 1

Description: 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						
		GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	- etc.-					

Example:

RBE1	67	58	123456						
		61	123	77	23	105	3		

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
GNi	Identification number of grid point to which independent degrees of freedom for the element are assigned.	Integer > 0	Required
CNi	Independent degrees of freedom in the global coordinate system for the rigid element at grid points GNi. See Remark 1. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required
GMj	Grid point identification numbers at which dependent degrees of freedom are assigned.	Integer > 0	
CMj	Dependent degrees of freedom in the global coordinate system at grid points GMj. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	

Remarks:

1. A dependent degree of freedom assigned by one element cannot be assigned dependent by another rigid element or MPC entry and cannot be additionally constrained (e.g., single-point constraint).
2. By default, a dependent degree of freedom assigned by one element cannot be assigned dependent by another rigid element or MPC entry and cannot be additionally constrained (e.g., single-point constraint). If this behavior is desired use PARAM, AUTOFIXRIGIDSPC which when set to ON will allow the constraint of dependent degrees of freedom (See Section 5, *Parameters*, for more information on AUTOFIXRIGIDSPC.)
3. 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.
4. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
5. Forces of multipoint constraint may be recovered with the MPCFORCE Case Control command.

RBE2 Rigid Body Element, Form 2

Description: Defines a rigid body whose independent degrees of freedom are specified at a single grid point and whose dependent degrees of freedom 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	GM9	- etc.-	A			

Example:

RBE2	12	2	123	15	18	22	25	27	
	34								

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
GN	Identification number of grid point to which all six independent degrees of freedom for the element are assigned.	Integer > 0	Required
CM	Component numbers of dependent degrees of freedom in the global coordinate system of grid point GN at grid points GMi. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required
GMi	Grid point identification numbers at which dependent degrees of freedom are assigned.	Integer > 0	
A	Thermal expansion coefficient.	Real or blank	0.0

Remarks:

1. The components indicated by CM are made dependent at all grid points GMi.
2. By default, a dependent degree of freedom assigned by one element cannot be assigned dependent by another rigid element or MPC entry and cannot be additionally constrained (e.g., single-point constraint). If this behavior is desired use PARAM, AUTOFIXRIGIDSPC which when set to ON will allow the constraint of dependent degrees of freedom (See Section 5, *Parameters*, for more information on AUTOFIXRIGIDSPC.)
3. 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.
4. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
5. Forces of multipoint constraint may be recovered with the MPCFORCE Case Control command.

RBE3 Interpolation Constraint Element

Description: 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	
	- etc. -	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	GM6	CM6		

Example:

RBE3	20		101	1234	1.0	123	1	3	
	5	4.5	1	2	4	6	6.1	2	
	7	8	9	8.3	1	12	17		
	UM	1	2						

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
REFGRID	Reference grid point identification number.	Integer > 0	Required
REFC	Component numbers at the reference grid point. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required
WTi	Weighting factor for components of motion at grid points Gi,j.	Real	Required
Ci	Component numbers with weighting factor WTi at grid points Gi,j. (Up to three unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required
Gi,j	Grid points whose components Ci have weighting factor WTi in the averaging equations.	Integer > 0	Required
GMi	Grid points whose components CMi are to be made dependent. See Remark 7.	Integer > 0	Required
CMi	Component numbers of GM. (Up to six unique digits may be placed in the field with no embedded blanks.) See Remark 7.	1 ≤ Integers ≤ 6	Required

Remarks:

1. Components Ci at grid points Gi,j must be able to react rigid body motion resulting from REFC. For most applications components 123 can be specified for Ci, except when Gi,j is collinear. In the latter case, only the inplane components should be specified.
2. Blank spaces may be left at the end of a Gi,j sequence.
3. By default, a dependent degree of freedom assigned by one element cannot be assigned dependent by another rigid element or MPC entry and cannot be additionally constrained (e.g., single-point constraint). If this behavior is desired use PARAM, AUTOFIXRIGIDSPC which when set to ON will allow the constraint of dependent degrees of freedom (See Section 5, *Parameters*, for more information on AUTOFIXRIGIDSPC.)
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. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
6. Forces of multipoint constraint may be recovered with the MPCFORCE Case Control command.
7. The purpose of the GMi and CMi fields are to replace dependent reference degrees of freedom with independent ones which can be either assigned dependent by another rigid element or MPC entry or be additionally constrained (e.g., single-point constraint). Specification of these degrees of freedom can result in the generation of invalid MPC equations and subsequent fatal errors. The preferred method is the use of PARAM, AUTOFIXRIGIDSPC which when set to ON will allow the constraint of dependent degrees of freedom (See Section 5, *Parameters*, for more information on AUTOFIXRIGIDSPC.)

RFORCE — Rotational Force

Description: Defines static loading resulting from angular velocity and/or acceleration.

Format:

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	G	CID	A	R1	R2	R3		
	RACC								

Example:

RFORCE	5	66		-4.2	0.0	0.0	1.0		
	2.5								

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
G	Grid point identification number.	Integer ≥ 0 or blank	0
CID	Coordinate system identification number.	Integer ≥ 0 or blank	0
A	Scale factor of the angular velocity in revolutions per unit time.	Real	Required
R1, R2, R3	Rectangular component of rotation vector $\vec{R}$. The vector defined will pass through point G.	Real	Required; must have at least one nonzero component
RACC	Scale factor of the angular acceleration in revolutions per unit time squared.	Real	0.0

Remarks:

1. The force vector at grid point Gi in Figure 1, is given by:

$$\{\vec{F}\}_i = [m]_i \left[\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)\right]$$

where,

angular velocity is given by $\vec{\omega} = 2\pi A * \vec{R}$ (radians/unit time)

angular acceleration is given by $\vec{\alpha} = 2\pi RACC * \vec{R}$ (radians/unit time squared)

$[m]_i$ is the translational mass matrix at grid point Gi

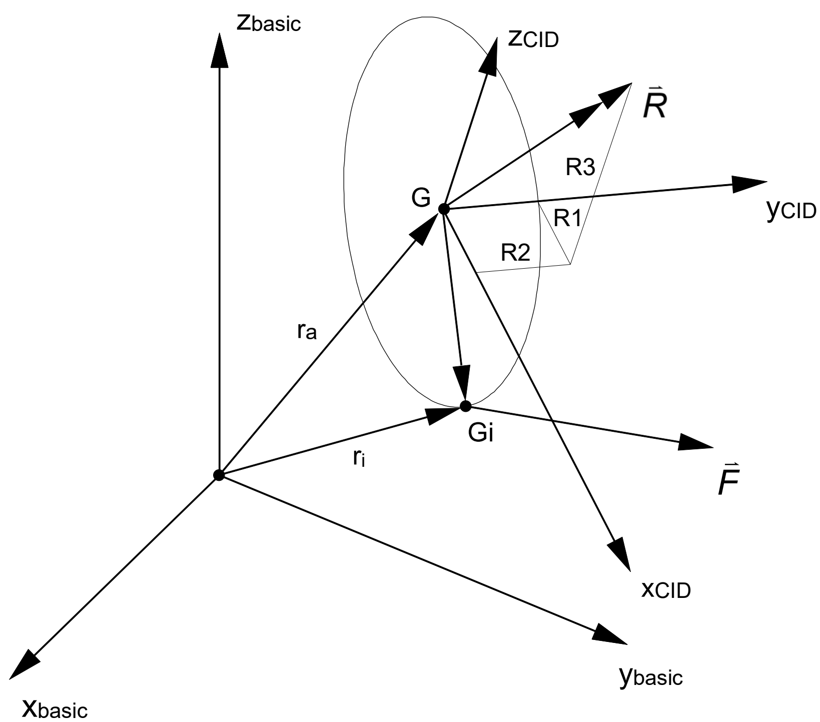


Figure 1. RFORCE Vector at Grid Point Gi.

2. Load sets must be selected in the Case Control Section (LOAD = SID).
3. G = 0 indicates that the rotation vector acts through the origin of the basic coordinate system.
4. A CID of zero references the basic coordinate system.
5. The continuation entry is optional.

RLOAD1 — Frequency Response Dynamic Load, Form 1

Description: 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	12			2				

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
EXCITEID	DAREA or SPCD entry set identification number that defines *A*.	Integer > 0	Required
DELAY	DELAY set identification number that defines τ.	Integer > 0 or blank	
DPHASE	DPHASE set identification number that defines θ.	Integer > 0 or blank	
TC	TABLEDi set identification number that defines $C(f)$.	Integer > 0 or blank	
TD	TABLEDi set identification number that defines $D(f)$.	Integer > 0 or blank	
TYPE	Defines the nature of the dynamic excitation. See Remark 2.	0 ≤ Integer ≤ 3 or character	0

Remarks:

1. Dynamic load sets must be selected with the Case Control command DLOAD=SID.

2. The nature of the dynamic excitation is defined in the following table:

TYPE	Type of Dynamic Excitation
0, L, or LOAD	Applied load (force or moment) (default)
1, D, or DISP	Enforced displacement using SPCD
2, V, or VELO	Enforced velocity using SPCD
3, A, or ACCE	Enforced acceleration using SPCD

3. The TYPE field determines the manner in which the EXCITEID field is used as described below
   a) Excitation specified by TYPE is an applied load
      - If there *is no* LOADSET request in the Case Control then EXCITEID may directly reference DAREA, static, and thermal load set entries.
      - If there *is a* LOADSET request in the Case Control then the model will reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID.
   b) Excitation specified by TYPE is an enforced motion
      - If there *is no* LOADSET request in the Case Control then EXCITEID will reference SPCD entries.
      - If there *is a* LOADSET request in Case Control then the model will reference SPCD entries specified by the LID field in the selected LSEQ entries corresponding to the EXCITEID.
4. If any of DELAY, DPHASE, TC, or TD fields are blank, the corresponding τ, θ, $C(f)$, and $D(f)$ will be zero. Either TC or TD may be blank, but not both.

RLOAD2 Frequency Response Dynamic Load, Form 2

Description: Defines a frequency-dependent dynamic load of the form

$$P(f) = AB(f)e^{i[\phi(f) + \theta - 2\pi f\tau]}$$

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	12	4		3					

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
EXCITEID	DAREA or SPCD entry set identification number that defines *A*.	Integer > 0	Required
DELAY	DELAY set identification number that defines τ.	Integer > 0 or blank	
DPHASE	DPHASE set identification number that defines θ.	Integer > 0 or blank	
TB	TABLEDi set identification number that defines $B(f)$.	Integer > 0 or blank	
TP	TABLEDi set identification number that defines $\phi(f)$.	Integer > 0 or blank	
TYPE	Defines the nature of the dynamic excitation. See Remark 2.	0 ≤ Integer ≤ 3 or character	0

Remarks:

1. Dynamic load sets must be selected with the Case Control command DLOAD=SID.

2. The nature of the dynamic excitation is defined in the following table:

TYPE	Type of Dynamic Excitation
0, L, or LOAD	Applied load (force or moment) (default)
1, D, or DISP	Enforced displacement using SPCD
2, V, or VELO	Enforced velocity using SPCD
3, A, or ACCE	Enforced acceleration using SPCD

3. The TYPE field determines the manner in which the EXCITEID field is used as described below
   a) Excitation specified by TYPE is an applied load
      - If there *is no* LOADSET request in the Case Control then EXCITEID may directly reference DAREA, static, and thermal load set entries.
      - If there *is a* LOADSET request in the Case Control then the model will reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID.
   b) Excitation specified by TYPE is an enforced motion
      - If there *is no* LOADSET request in the Case Control then EXCITEID will reference SPCD entries.
      - If there *is a* LOADSET request in Case Control then the model will reference SPCD entries specified by the LID field in the selected LSEQ entries corresponding to the EXCITEID.
4. If any of DELAY, DPHASE, or TP fields are blank, the corresponding τ, θ, $\phi(f)$ will be zero.

RROD Rigid Pin-Ended Element Connection

Description: 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				

Example:

RROD	15	1	2	2					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
GA, GB	Grid point identification number of connection points.	Integer > 0	Required
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.	1 ≤ Integer ≤ 3	Either CMA or CMB has a single value, the other must be blank

Remarks:

1. A dependent degree of freedom assigned by one element cannot be assigned dependent by another rigid element or MPC entry and cannot be additionally constrained (e.g., single-point constraint).
2. 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.
3. Rigid elements, unlike MPCs are not selected through the Case Control Section.
4. Forces of multipoint constraint may be recovered with the MPCFORCE Case Control command.

RSPLINE — Interpolation Constraint Element

Description: 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	65		30	31	123456	32		33	
	123	70	123	71					

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
D/L	Ratio of the diameter of the elastic tube to the sum of the lengths of all segments.	Real > 0.0	0.1
Gi	Grid point identification number.	Integer > 0	Required
Ci	Components to be constrained. See Remark 2.	1 ≤ Integers ≤ 6	Required

Remarks:

1. Displacements are interpolated from the equations of an elastic beam passing through the grid points.
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. 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.
4. The independent degrees of freedom that are the rotation components most nearly parallel to the line joining the regions should not normally be constrained.
5. A dependent degree of freedom assigned by one element cannot be assigned dependent by another rigid element or MPC entry and cannot be additionally constrained (e.g., single-point constraint).
6. 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.
8. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
9. Forces of multipoint constraint may be recovered with the MPCFORCE Case Control command.

RTRPLT Rigid Triangular Plate

Description: 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						

Example:

RTRPLT	5	1	2	3	123456				

Field	Definition	Type	Default
EID	Element identification number.	Integer > 0	Required
GA, GB	Grid point identification number of connection points.	Integer > 0	Required
CNA, CNB, CNC	Independent degrees of freedom in the global coordinate system for the element at grid points GA, GB, and GC. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6 or blank	See Remark 1
CMA, CMB, CMC	Component numbers of dependent degrees of freedom in the global coordinate system assigned by the element at grid points GA and GB. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6 or blank	

Remarks:

1. The total number of components in CNA, CNB, and CNC must equal six; for example, CNA = 1235, CNB = 3, and CNC = 3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element.
2. If CMA, CMB, and CMC are all zero blank or if the continuation entry is omitted, all of the degrees of freedom not in CNA, CNB, or CNC will be made dependent.
3. A dependent degree of freedom assigned by one element cannot be assigned dependent by another rigid element or MPC entry and cannot be additionally constrained (e.g., single-point constraint).
4. Rigid elements, unlike MPCs, are not selected through the Case Control Section.
5. Forces of multipoint constraint may be recovered with the MPCFORCE Case Control command.

RVDOF — Degrees of Freedom Specification for Residual Vectors

Description: Defines 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	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

RVDOF	25	3	13	456	19	4			

Field	Definition	Type	Default
Gi	Grid point identification number(s).	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required

Remarks:

1. In some cases it may be more convenient to use RVDOF1.

RVDOF1 Degrees of Freedom Specification for Residual Vectors, Alternate Form

Description: Defines 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	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	- etc.-					

Example:

RVDOF1	123	6	3	2	15	19	14	21	
	29	43							

Alternate Format and Example:

RVDOF1	C	G1	THRU	G2					

RVDOF1	456	15	THRU	512					

Field	Definition	Type	Default
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required
Gi	Grid point identification number(s).	Integer > 0; G1 < G2	Required

Remarks:

1. If the alternate form is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will be skipped.

SEELT — Superelement Interior Element Definition

Description: Defines interior elements for a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
SEELT	SEID	EID1	EID2	EID3	EID4	EID5	EID6	EID7	

Example:

SEELT	6	15	17	39	122				

Alternate Format and Example:

SEELT	SEID	EID1	THRU	EID2					

SEELT	6	15	THRU	26					

Field	Definition	Type	Default
SEID	Superelement identification number.	Integer > 0	Required
EIDi	Element identification number(s).	Integer > 0; EID1 < EID2	Required

Remarks:

1. SEELT defines elements to be included in a superelement. SEELT may be used as the primary means of defining superelements or it may be used in combination with SESET or field 9 of the GRID Bulk Data entry which define grid points interior to a superelement.
2. EIDi may appear on an SEELT entry only once.
3. If the alternate form is used, elements in the sequence EID1 through EID2 are not required to exist. Elements that do not exist will be skipped.
4. All degrees of freedom for grid points attached to EIDi that are interior to the superelement boundary are placed in the omit set (o-set) of the superelement.

SELABEL Superelement Output Label

Description: Defines a label or name to be displayed in the superelement output headings.

Format:

1	2	3	4	5	6	7	8	9	10
SELABEL	SEID	LABEL							

Example:

SELABEL	10	ENGINE SECTION WITH SOLID ROCKET MOTORS							

Field	Definition	Type	Default
SEID	Superelement identification number.	Integer > 0	Required
LABEL	Label associated with superelement SEID for output headings.	Character	

Remarks:

1. Only one SELABEL per superelement may be specified.
2. The label will appear in all superelement output headings.

SESET Superelement Interior Point Definition

Description: 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	2	5	7	29	122				

Alternate Format and Example:

SESET	SEID	G1	THRU	G2					

SESET	2	55	THRU	126					

Field	Definition	Type	Default
SEID	Superelement identification number.	Integer > 0	Required
Gi	Grid point identification number(s).	Integer > 0; G1 < G2	Required

Remarks:

1. Interior grid points may also be defined via field 9 of the GRID Bulk Data entry. SESET defines grid 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. If the alternate form is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will be skipped.
4. All degrees of freedom for Gi are placed in the omit set (o-set) of the superelement.

SLOAD Static Scalar Load

Description: 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	33	5	6.5	15	-2.5	17	-4.7		

Field	Definition	Type	Default
SID	Load set identification.	Integer > 0	Required
Si	Scalar or grid point identification number.	Integer > 0	Required
Fi	Load magnitude.	Real	0.0

Remarks:

1. SLOAD is only supported in heat transfer analysis and must be selected with the Case Control command LOAD = SID.
2. Up to three loads may be defined on a single entry.
3. 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).

SNDATA — Stress-Life Method Material Fatigue Data

Description: Specifies material property data needed for fatigue analysis. This entry is used if a MAT1, MAT2, MAT8, MAT9, or MAT12 entry is specified with the same MID.

Format:

1	2	3	4	5	6	7	8	9	10
SNDATA	MID	B	SU	N0	KF	BE	SE		

Example:

SNDATA	200	0.16	4.5+3		0.9				

Field	Definition	Type	Default
MID	Identification number of a MAT1, MAT2, MAT8, MAT9, or MAT12 entry.	Integer > 0	Required
B	S-N curve slope. See Remark 3.	Real > 0.0	See Remark 2.
SU	Intercept stress level. Typically taken as the material ultimate stress. See Remark 3.	Real > 0.0	See Remark 2.
N0	Intercept cycles. See Remark 3.	Integer > 0	1000
KF	Factor applied to compensate for life reduction effects such as finish, corrosion, and notch effects. See Remark 3.	Real > 0.0	1.0
BE	Slope after endurance limit. See Remark 4.	Real > 0.0	0.1∗B
SE	Endurance limit. See Remark 3.	Real ≥ 0.0	0.2∗SU

Remarks:

1. SNDATA entries must all have unique set identification numbers.
2. VFATIGUE and FATIGUE entries provide defaults to SNDATA. Values not specified on SNDATA entries will be replaced with ones from the VFATIGUE or FATIGUE entry STRESS continuation.
3. The S-N curve shown in Figure 1 is characterized by the following equations

If $S_i \geq S_e$

$$N_f = \mathrm{N0}\left(\frac{\mathrm{SU}}{\mathrm{KF} * S_i}\right)^{\frac{1}{\mathrm{B}}}$$

If $S_i < S_e$

$$N_f = N_e\left(\frac{\mathrm{SE}}{\mathrm{KF} * S_i}\right)^{\frac{1}{\mathrm{BE}}}$$

where,

N_f is the number of cycles to failure

S_i is the amplitude of input stress (S_{max} – S_{min})/2

N_e is the number of failure cycles at the endurance limit

4. A small slope is required to prevent infinite life. See Figure 1.

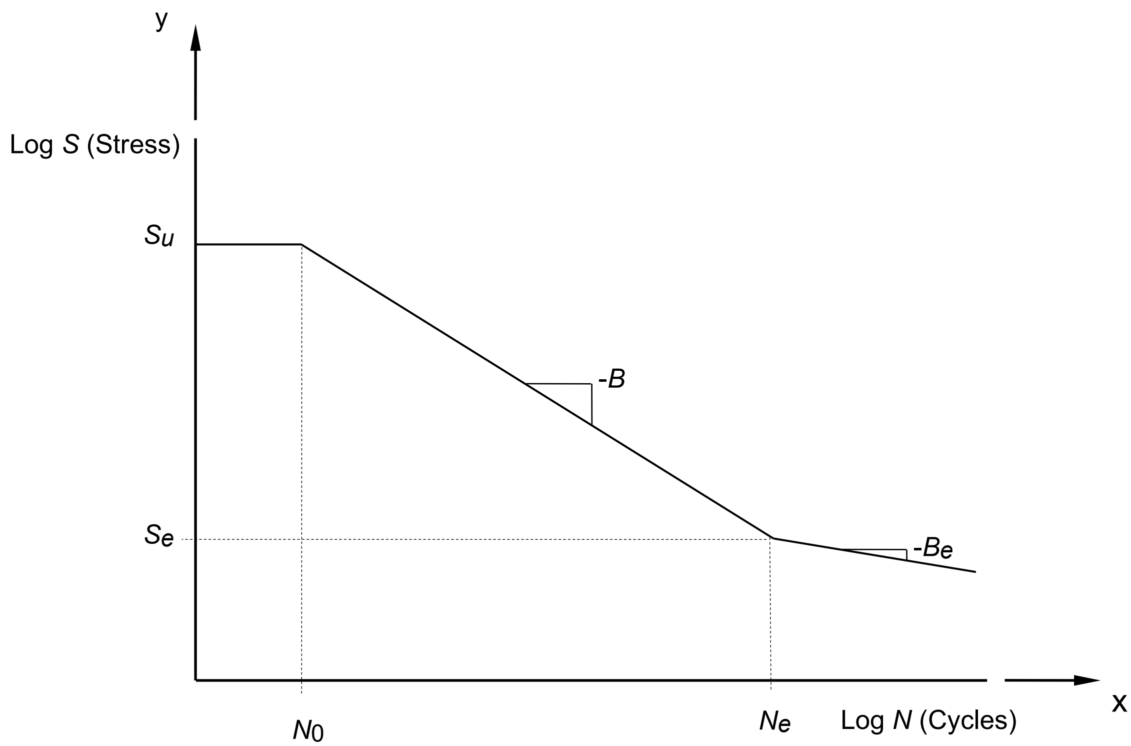


Figure 1. Stress-Life Curve Format.

SPC Single Point Constraint

Description: Defines sets of single-point constraints and enforced displacements.

Format:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1	D1	G2	C2	D2		

Example:

SPC	2	32	436	2.5					

Field	Definition	Type	Default
SID	Identification number of single point constraint set.	Integer ≥ 0	Required
Gi	Grid point identification number.	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required
Di	Enforced displacement for all coordinates designated by G and C.	Real or blank	0.0

Remarks:

1. Single-point constraint sets must be selected in the Case Control Section (SPC = SID).
2. From one to twelve degrees of freedom may be defined on a single entry.
3. Continuations are not allowed.
4. The SPCD entry is the preferred method for applying enforced displacements, rather than the “D” field described above when multiple subcases with different enforced displacement conditions are applied.
5. Single-point constraint sets with SID set to zero will be applied to all subcases.

SPC1 Single Point Constraint, Alternate Form

Description: Defines sets 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	2	123	436	432	455	460	470		

Alternate Format and Example:

SPC1	SID	C	G1	THRU	G2				

SPC1	2	246	2	THRU	122				

Field	Definition	Type	Default
SID	Identification number of single-point constraint set.	Integer ≥ 0	Required
Gi	Grid point identification number(s).	Integer > 0; G1 < G2	Required
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	1 ≤ Integers ≤ 6	Required

Remarks:

1. Note that enforced displacements are not available via this entry.
2. Single-point constraint sets must be selected in the Case Control Section (SPC = SID) to be used.
3. If the alternate form is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will be skipped.
4. Single-point constraint sets with SID set to zero will be applied to all subcases.

SPCADD Single Point Constraint Set Combination

Description: Defines a single-point constraint set as a union of single-point constraint sets defined via 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	2	4	5	6	8				

Field	Definition	Type	Default
SID	Identification number of single point constraint set.	Integer > 0	Required
Si	Identification numbers of single-point constraint sets defined via SPC or by SPC1 entries.	Integer > 0; SID ≠ Si	Required

Remarks:

1. The Si values must be unique.
2. Single-point constraint sets must be selected in the Case Control Section (SPC = SID) to be used.
3. No Si may be the identification number of a single-point constraint set defined by another SPCADD entry.

SPCD — Enforced Displacement Value

Description: Defines an enforced displacement value for static analysis, which is requested as a LOAD.

Format:

1	2	3	4	5	6	7	8	9	10
SPCD	SID	G1	C1	D1	G2	C2	D2		

Example:

SPCD	2	523	246	1.6					

Field	Definition	Type	Default
SID	Identification number of single load set.	Integer ≥ 0	Required
Gi	Grid point identification number.	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required
D	Enforced displacement for all coordinates designated by G and C.	Real or blank	0.0

Remarks:

1. A global coordinate (G and C) referenced on this entry must also be referenced on a SPC or SPC1 Bulk Data entry and selected by the SPC Case Control command.
2. Values of D will override the values specified on an SPC Bulk Data entry, if the SID is selected on the LOAD Case Control command.
3. SPCD loads may be combined with other loads using the LOAD Bulk Data entry.
4. This is the preferred method for applying enforced displacements, rather than the “D” field of the SPC entry when multiple subcases with different enforced displacement conditions are applied.
5. SPCD loads with SID set to zero will be applied to all subcases.

SPOINT Scalar Point Definition

Description: 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	5	22	2	7	45	6			

Alternate Format and Example:

SPOINT	ID1	THRU	ID2						

SPOINT	8	THRU	345						

Field	Definition	Type	Default
IDi	Scalar point identification number(s).	Integer > 0; ID2 $>$ ID1	Required

Remarks:

1. All scalar point identification numbers must be unique with respect to all other grid, scalar, and extra points.
2. At least one ID must be present on each SPOINT entry.
3. If the alternate form is used, all points ID1 through ID2 that do not exist will be skipped.
4. Scalar points must not be specified more than once.
5. Continuations are not allowed.

STRAIN Element Initial Strain

Description: Defines the shell and solid element initial strain state for use in nonlinear analysis.

Format:

1	2	3	4	5	6	7	8	9	10
STRAIN	SID	EID	S1	S2	S3	S4	S5	S6	
	S7	S8	- etc.-						

Example:

STRAIN	15	23	1.075-5	-2.364-5	4.006-8	2.235-4	-2.096-7	1.084-9	
	S7	S8	- etc.-						

Alternate Format and Example:

STRAIN	SID	EID1	S1	S2	S3	S4	S5	S6	
	THRU	ED2							

STRAIN	15	23	1.075-5	-2.364-5	4.006-8	2.235-4	-2.096-7	1.084-9	
	THRU	55							

Field	Definition	Type	Default
SID	Load set identification number.	Integer > 0	Required
EID	Element identification number.	Integer > 0	Required
Si	Strain component values. See Remark 5.	Real	0.0

Remarks:

1. Initial strain sets must be selected in the Case Control Section (INITSTRAIN = SID).
2. If the alternate form is used, all elements EID1 through EID2 that are not compatible or do not exist will be skipped.
3. Elements must not be specified more than once.
4. All elements directly referenced must exist.

5. Strain vectors are specified either at the element centroid or the corner nodes. When corner data is input the strain vector repeats the number of corner nodes minus one times. For shell elements the input format is:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \text{Membrane} = \begin{Bmatrix} \text{S1} \\ \text{S2} \\ \text{S3} \end{Bmatrix} \quad \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \text{Bending} = \begin{Bmatrix} \text{S4} \\ \text{S5} \\ \text{S6} \end{Bmatrix} \quad \begin{Bmatrix} \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} \text{Transverse Shear} = \begin{Bmatrix} \text{S7} \\ \text{S8} \end{Bmatrix}$$

For solid elements the format is:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{Bmatrix} \text{S1} \\ \text{S2} \\ \text{S3} \\ \text{S4} \\ \text{S5} \\ \text{S6} \end{Bmatrix}$$

6. STRAIN Bulk Data entries can be exported using the TRSLSTRNDATA Model Initialization directive. (See Section 2, *Initialization*, for more information on TRSLSTRNDATA.)

SUPORT

Spectrum Input Location

Description: Specifies input spectrum degrees of freedom for response spectrum analysis.

Format:

1	2	3	4	5	6	7	8	9	10
SUPORT	GID	C							

Example:

SUPORT	6	3							

Field	Definition	Type	Default
GID	Grid point identification number.	Integer > 0	Required
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required

Remarks:

1. Note that SUPORT is spelled with one P.

TABDMP1 Modal Damping Table

Description: Defines model damping as a tabular function of 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								
	1.4	0.03068	2.6	0.04372	ENDT				

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
TYPE	Type of damping units, one of the following character variables: G, CRIT, or Q.	Character	G
fi	Frequency value in cycles per unit time.	Real ≥ 0.0	Required
gi	Damping value.	Real	Required

Remarks:

1. Modal damping tables must be selected with the Case Control command SDAMPING = TID.
2. The frequency values, fi must be in either ascending or descending order, but not both.
3. Discontinuities may be specified between any two points. If g is evaluated at a discontinuity, then the average value of g is used. In Figure 1, the value of g at f = f3 is g = (g3 + g4)/2.
4. At least one continuation entry must be specified.
5. Any fi-gi pair may be ignored by placing SKIP in either of the two fields.
6. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
7. TABDMP1 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 endpoints, see Figure 1. No warning messages are given if table data is input incorrectly.

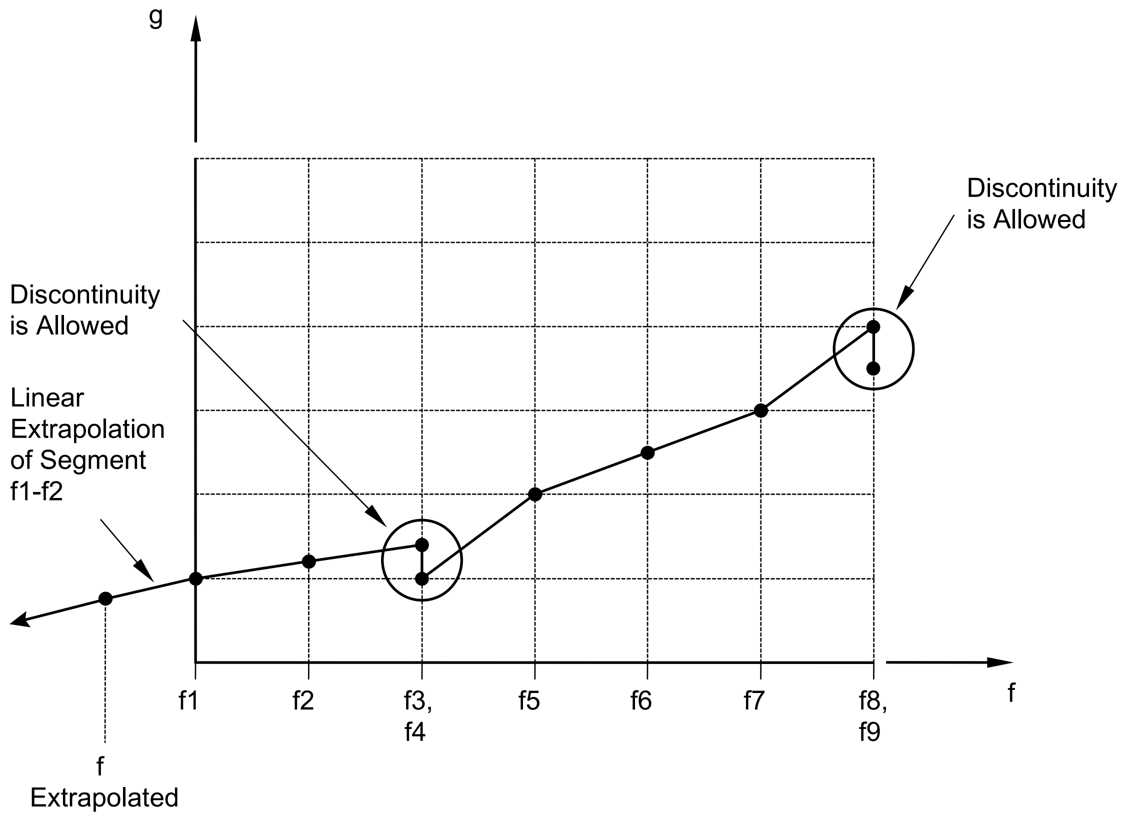


Figure 1. Example of Table Extrapolation and Discontinuity.

8. This form of damping is only used in modal formulations of complex eigenvalue, transient response, or frequency response analysis. The type of damping used depends on the solution sequence (structural damping is displacement-dependent and viscous damping is velocity-dependent).
9. PARAM, KDAMP may be used in solution sequences that perform modal frequency and modal complex eigenvalue analysis to select the type of damping. (See Section 5, *Parameters*, for more information on KDAMP.)
10. If TYPE is G or blank, the damping values gi, etc., are in units of equivalent viscous dampers, as follows:

$$b_i = \frac{g_i}{\omega_i} K_i$$

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:

$$C/C_0 = g/2$$

$$Q = \begin{Bmatrix} 1/(2C/C_0) \\ 1/g \end{Bmatrix}$$

TABFV Stiffness Velocity-Dependence Table

Description: Specifies the force versus displacement tables for a nonlinear shock and vibration element (CBUSH1D) which references a PMOUNT property.

Format:

1	2	3	4	5	6	7	8	9	10
TABFV	TID								
	V1	TID1	V2	TID2	V3	TID3	- etc.-		

Example:

TABFV	105								
	130.0	20	195.0	40	ENDT				

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
Vi	Velocity values.	Real	Required
TIDi	Table identification numbers of TABLED1 entries.	Integer > 0	Required

Remarks:

1. TIDi must be unique with respect to all TABLED1 and TABFV table identification numbers.
2. Velocity 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. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
4. This table is referenced only by PMOUNT entries that define a nonlinear shock and vibration element (CBUSH1D).

TABLED1 — Dynamic Load Tabular Function, Form 1

Description: Defines a tabular function for use in generating 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.-		

Example:

TABLED1	32								
	-2.0	8.0	1.9	6.5	3.1	7. 6	ENDT		

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis, one of the following character variables: LINEAR or LOG.	Character	LINEAR
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis, one of the following character variables: LINEAR or LOG.	Character	LINEAR
xi, yi	Tabular values.	Real	Required

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points. If y is evaluated at a discontinuity, then the average value of y is used. In Figure 1, the value of y at x = x3 is y = (y3 + y4)/2. If the y-axis is a LOG axis the jump at the discontinuity is evaluated as $y = \sqrt{y3y4}$.
3. At least one continuation entry must be specified.
4. Placing SKIP in either of the two fields may ignore any xi-yi pair.
5. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
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 linear interpolation within the table and linear extrapolation outside the table using the two starting or endpoints, see Figure 1. No warning messages are given if table data is input incorrectly. The algorithms used for interpolation or extrapolation are:

XAXIS	YAXIS	y(x)
LINEAR	LINEAR	$\frac{x_{i+1}-x}{x_{i+1}-x_i}y_i+\frac{x-x_i}{x_{i+1}-x_i}y_{i+1}$
LOG	LINEAR	$\frac{\ln(x_{i+1}/x)}{\ln(x_{i+1}/x_i)}y_i+\frac{\ln(x/x_i)}{\ln(x_{i+1}/x_i)}y_{i+1}$
LINEAR	LOG	$\exp\left[\left(\frac{x_{i+1}-x}{x_{i+1}-x_i}\ln y_i+\frac{x-x_i}{x_{i+1}-x_i}\ln y_{i+1}\right)\right]$
LOG	LOG	$\exp\left[\frac{\ln(x_{i+1}/x)}{\ln(x_{i+1}/x_i)}\ln y_i+\frac{\ln(x-x_i)}{\ln(x_{i+1}-x_i)}\ln y_{i+1}\right]$

where $x_i < x < x_{i+1}$

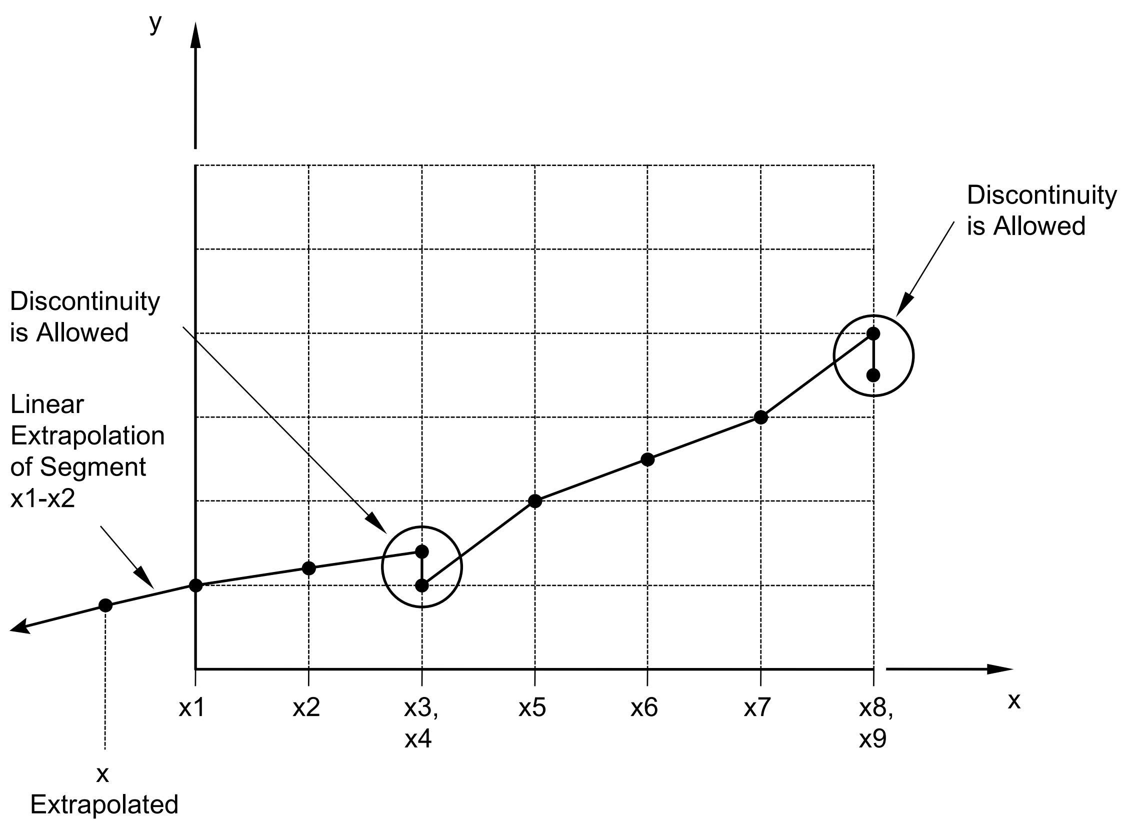


Figure 1. Example of Table Extrapolation and Discontinuity.

7. Tabular values on an axis if XAXIS or YAXIS equals LOG must be positive.
8. For frequency dependent loads xi is measured in cycles per unit time.

TABLED2 Dynamic Load Tabular Function, Form 2

Description: Defines a parametric tabular function for use in generating time-dependent dynamic loads.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED2	TID	X1							
	x1	y1	x2	y2	x3	y3	- etc.-		

Example:

TABLED2	16	-12.5							
	2.0	-3.5	3.0	-5.2	4.0	5.9	8.0	6.4	
	SKIP	SKIP	10.0	6.7	ENDT				

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
X1	Table parameter.	Real	0.0
xi, yi	Tabular values.	Real	Required

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points. If y is evaluated at a discontinuity, then the average value of y is used. In Figure 1, 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. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
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 endpoints, see Figure 1. No warning messages are given if table data is input incorrectly.
7. For frequency dependent loads, X1 and xi are measured in cycles per unit time.

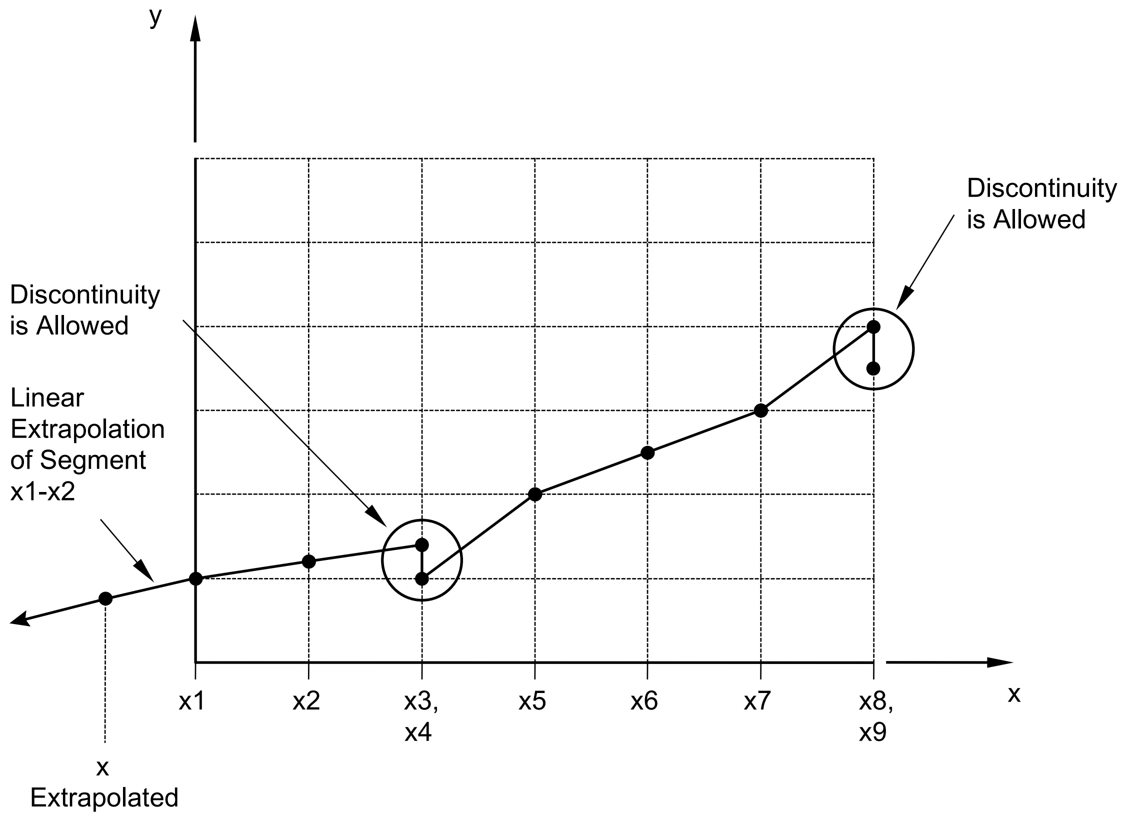


Figure 1. Example of Table Extrapolation and Discontinuity.

TABLED3 — Dynamic Load Tabular Function, Form 3

Description: Defines a parametric tabular function for use in generating time-dependent dynamic loads.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	- etc.-		

Example:

TABLED3	75	123.9	29.0						
	2.8	3.1	3.3	4.65	5.1	6.2	ENDT		

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
X1	Table parameter.	Real	0.0
X2	Table parameter.	Real ≠ 0.0	Required
xi, yi	Tabular values.	Real	Required

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 endpoints. For example, in Figure 1 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 1 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. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
6. TABLED3 uses the algorithm

$$y = y_T\left(\frac{x - X1}{X2}\right)$$

   where x is input to the table, y is returned, and is supplied from the MAT1 entry. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or endpoints. See Figure 1. No warning messages are issued if table data is input incorrectly.
7. 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.

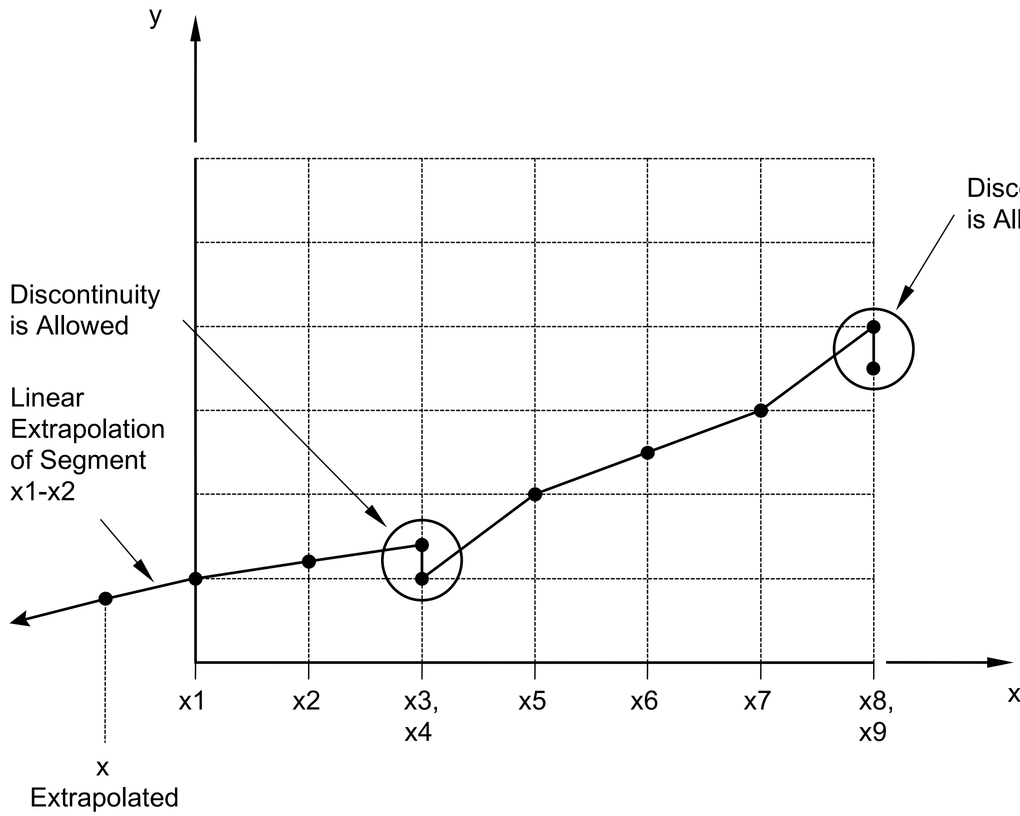


Figure 1. Example of Table Extrapolation and Discontinuity.

TABLED4 — Dynamic Load Tabular Function, Form 4

Description: Defines coefficients of a power series used in generating time-dependent dynamic loads.

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	35	0.0	1.0	0.0	200.				
	5.42	-0.0647	7.89-3	0.0	-2.9-7	ENDT			

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
X1	Table parameter.	Real	0.0
X2	Table parameter.	Real ≠ 0.0	Required
X3	Table parameter.	Real, X3 < X4	Required
X4	Table parameter.	Real	Required
Ai	Coefficients.	Real	Required

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. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
3. TABLED4 uses the algorithm

$$y = \sum_{i=0}^{N} \text{Ai}\left(\frac{x - \text{X1}}{\text{X2}}\right)^{i}$$

   where x is input to the table, y is returned. Whenever x < X3, X3 is used for x and whenever x > X4, X4 is used for x. There are $N + 1$ entries in the table. No warning messages are issued if table data is input incorrectly.
4. For frequency dependent loads, xi are measured in cycles per unit time.

TABLEM1 Material Property Table, Form 1

Description: 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	XAXIS	YAXIS						
	x1	y1	x2	y2	x3	y3	- etc.-		

Example:

TABLEM1	55								
	-500.0	10.15+6	0.0	8.54+6	1000.0	5.32+6	ENDT		

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis, one of the following character variables: LINEAR or LOG.	Character	LINEAR
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis, one of the following character variables: LINEAR or LOG.	Character	LINEAR
xi, yi	Tabular values.	Real	Required

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points. If y is evaluated at a discontinuity, then the average value of y is used. In Figure 1, the value of y at x = x3 is y = (y3 + y4)/2. If the y-axis is a LOG axis the jump at the discontinuity is evaluated as $y = \sqrt{y3y4}$.
3. At least one continuation entry must be specified.
4. Placing SKIP in either of the two fields may ignore any xi-yi pair.
5. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
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 endpoints, see Figure 1. No warning messages are given if table data is input incorrectly. The algorithms used for interpolation or extrapolation are:

X-AXIS	Y-AXIS	y(x)
LINEAR	LINEAR	$\frac{x_{i+1} - x}{x_{i+1} - x_i} y_i + \frac{x - x_i}{x_{i+1} - x_i} y_{i+1}$
LOG	LINEAR	$\frac{\ln(x_{i+1}/x)}{\ln(x_{i+1}/x_i)} y_i + \frac{\ln(x/x_i)}{\ln(x_{i+1}/x_i)} y_{i+1}$
LINEAR	LOG	$\exp\left[\left(\frac{x_{i+1} - x}{x_{i+1} - x_i} \ln y_i + \frac{x - x_i}{x_{i+1} - x_i} \ln y_{i+1}\right)\right]$
LOG	LOG	$\exp\left[\frac{\ln(x_{i+1}/x)}{\ln(x_{i+1}/x_i)} \ln y_i + \frac{\ln(x - x_i)}{\ln(x_{i+1} - x_i)} \ln y_{i+1}\right]$

where $x_i < x < x_{i+1}$

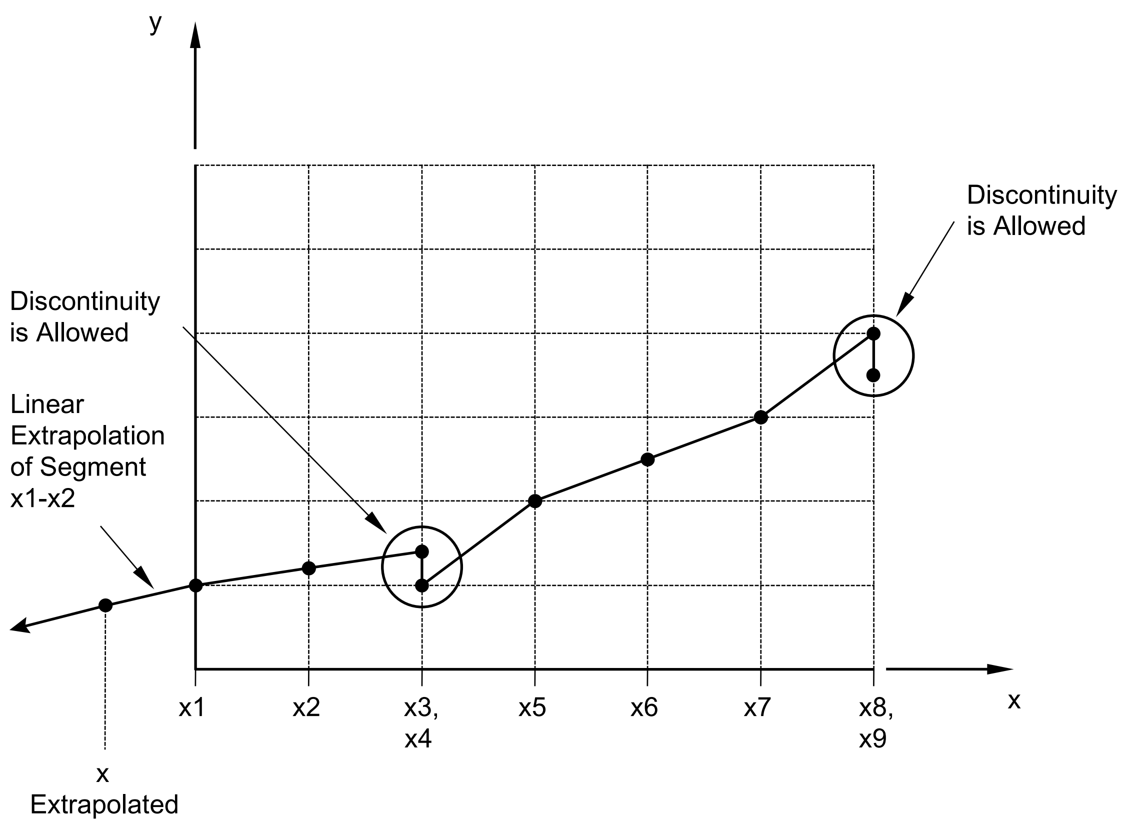


Figure 1. Example of Table Extrapolation and Discontinuity.

7. Tabular values on an axis if XAXIS or YAXIS equals LOG must be positive.

TABLEM2 — Material Property Table, Form 2

Description: Defines a parametric tabular function for use in generating temperature-dependent material properties.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM2	TID	X1							
	x1	y1	x2	y2	x3	y3	- etc.-		

Example:

TABLEM2	15	-10.5							
	-250.0	0.75	0.0	1.05	SKIP	SKIP	1000.0	1.245	
	1500.0	1.432	2000.0	2.976	ENDT				

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
X1	Table parameter.	Real	0.0
xi, yi	Tabular values.	Real	Required

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points. If y is evaluated at a discontinuity, then the average value of y is used. In Figure 1, 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. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
6. TABLEM2 uses the algorithm

$$y = z\, y_T(x - X1)$$

where x is input to the table, y is returned, and z is supplied from the MATi entry using the specific property value for the term being evaluated. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or endpoints, see Figure 1. No warning messages are given if table data is input incorrectly.

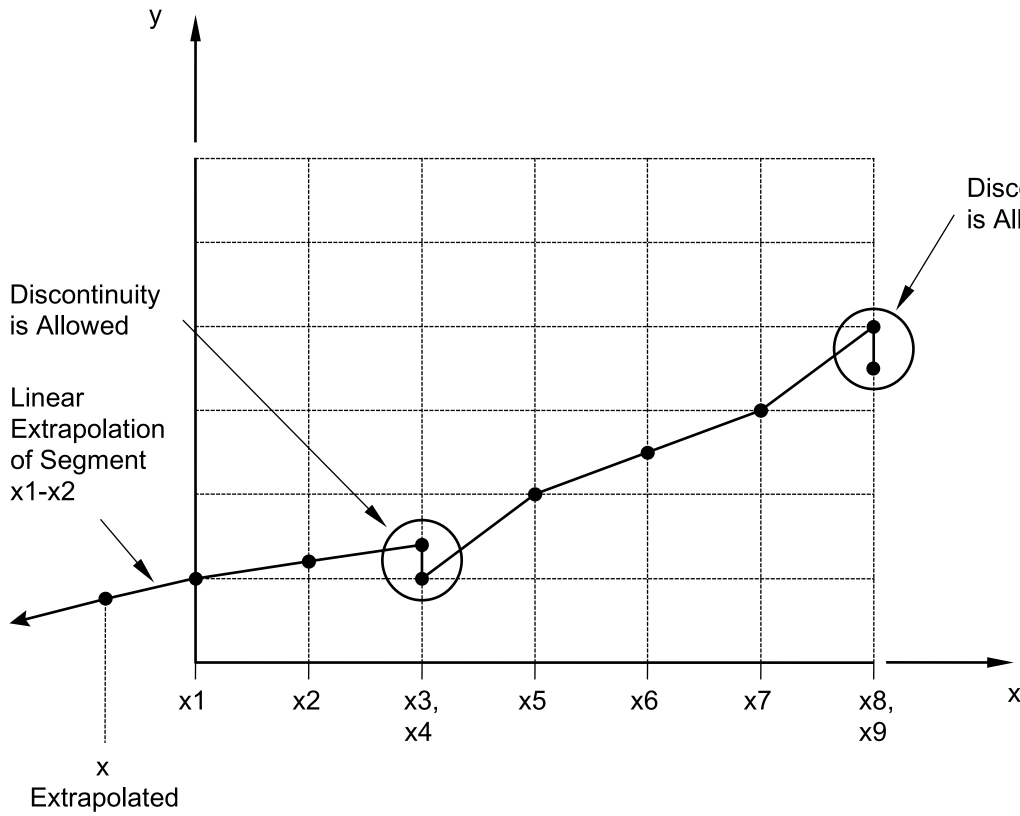


Figure 1. Example of Table Extrapolation and Discontinuity.

TABLEM3 — Material Property Table, Form 3

Description: Defines a parametric tabular function for use in generating temperature-dependent material properties.

Format:

1	2	3	4	5	6	7	8	9	10
TABLEM3	TID	X1	X2						
	x1	y1	x2	y2	x3	y3	- etc.-		

Example:

TABLEM3	66	156.9	50.0						
	2.8	2.9	3.3	5.5	5.8	11.2	ENDT		

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
X1	Table parameter.	Real	0.0
X2	Table parameter.	Real ≠ 0.0	Required
xi, yi	Tabular values.	Real	Required

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 endpoints. For example, in Figure 1 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 1 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. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
6. TABLEM3 uses the algorithm

$$y = z\, y_T\left(\frac{x - X1}{X2}\right)$$

where x is input to the table, y is returned, and z is supplied from the MATi entry using the specific property value for the term being evaluated. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or endpoints. See Figure 1. No warning messages are issued if table data is input incorrectly.

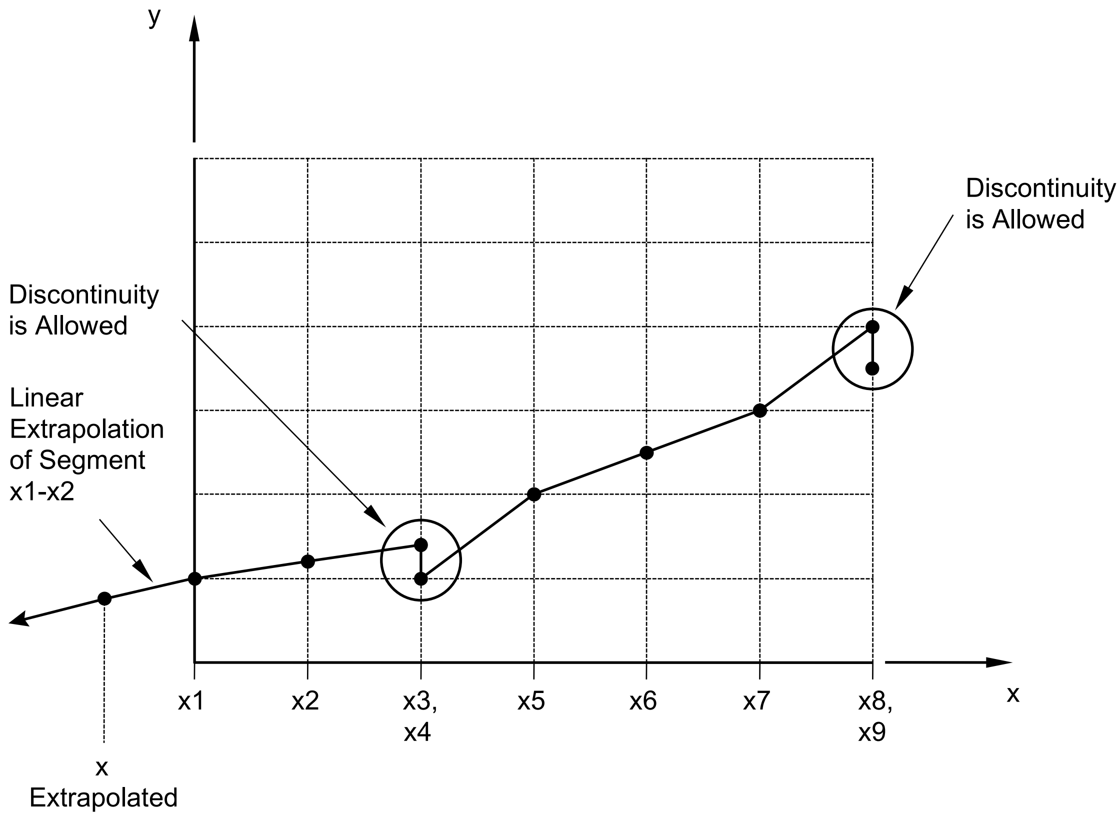


Figure 1. Example of Table Extrapolation and Discontinuity.

TABLEM4 Material Property Table, Form 4

Description: Defines coefficients of a power series used in generating temperature-dependent material properties.

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	45	0.0	1.0	0.0	50.				
	2.45	-0.0543	7.87-5	0.0	-8.4-8	ENDT			

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
X1	Table parameter.	Real	0.0
X2	Table parameter.	Real ≠ 0.0	Required
X3	Table parameter.	Real, X3 > X4	Required
X4	Table parameter.	Real	Required
Ai	Coefficients.	Real	Required

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. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
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 using the specific property value for the term being evaluated. Whenever x < X3, then X3 is used for x and whenever x > X4, X4 is used for x. There are N + 1 entries in the table. No warning messages are issued if table data is input incorrectly.

TABLES1 — Material Property Table, Form 1

Description: Defines a tabular function for stress-dependent material properties such as the stress-strain curve.

Format:

1	2	3	4	5	6	7	8	9	10
TABLES1	TID	XAXIS	YAXIS						
	x1	y1	x2	y2	x3	y3	- etc.-		

Example:

TABLES1	45								
	0.0	0.0	0.02	1.+4	0.04	1.4+4	ENDT		

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis, one of the following character variables: LINEAR or LOG.	Character	LINEAR
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis, one of the following character variables: LINEAR or LOG.	Character	LINEAR
xi, yi	Tabular values.	Real	0.0

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points. If y is evaluated at a discontinuity, then the average value of y is used. In Figure 1, the value of y at x = x3 is y = (y3 + y4)/2.
3. At least one continuation entry must be specified.
4. Placing SKIP in either of the two fields may ignore any xi-yi pair.
5. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
6. TABLES1 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 endpoints, see Figure 1. No warning messages are given if table data is input incorrectly. The algorithms used for interpolation or extrapolation are:

X-AXIS	Y-AXIS	y(x)
LINEAR	LINEAR	$\frac{x_{i+1} - x}{x_{i+1} - x_i} y_i + \frac{x - x_i}{x_{i+1} - x_i} y_{i+1}$
LOG	LINEAR	$\frac{\ln(x_{i+1}/x)}{\ln(x_{i+1}/x_i)} y_i + \frac{\ln(x/x_i)}{\ln(x_{i+1}/x_i)} y_{i+1}$
LINEAR	LOG	$\exp\left[\left(\frac{x_{i+1} - x}{x_{i+1} - x_i} \ln y_i + \frac{x - x_i}{x_{i+1} - x_i} \ln y_{i+1}\right)\right]$
LOG	LOG	$\exp\left[\frac{\ln(x_{i+1}/x)}{\ln(x_{i+1}/x_i)} \ln y_i + \frac{\ln(x - x_i)}{\ln(x_{i+1} - x_i)} \ln y_{i+1}\right]$

where $x_i < x < x_{i+1}$

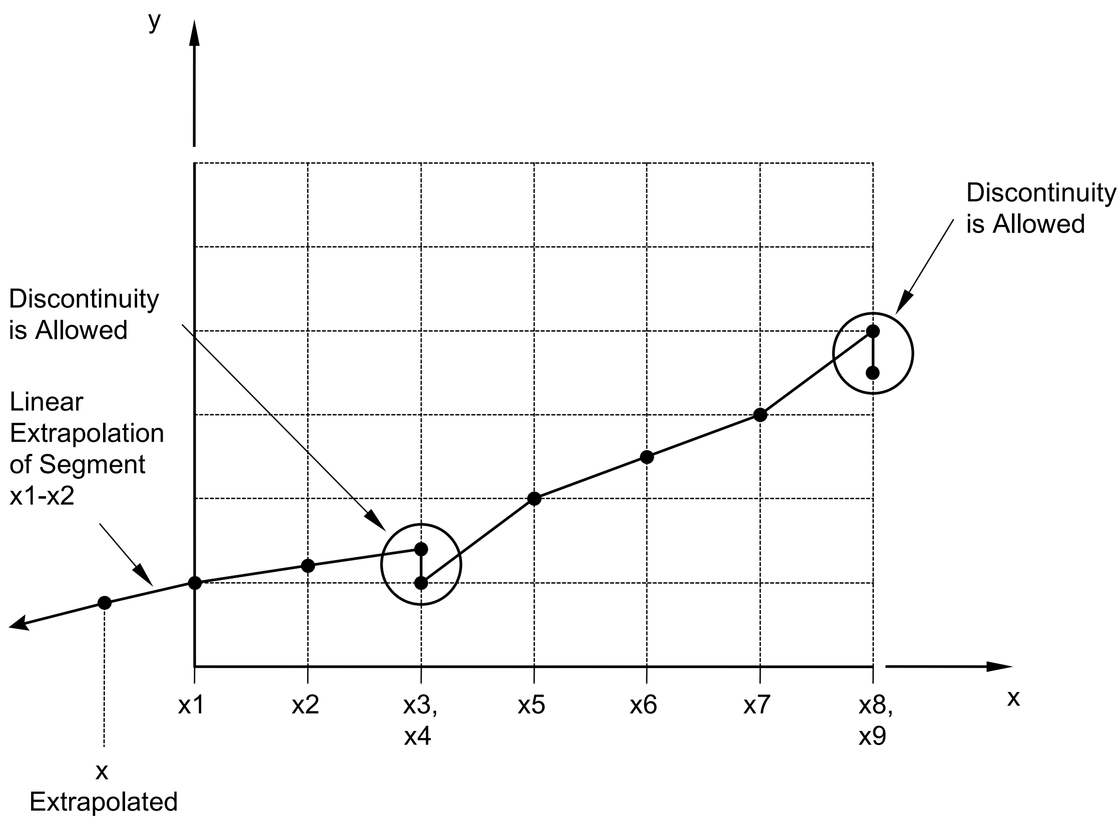


Figure 1. Example of Table Extrapolation and Discontinuity.

7. Tabular values on an axis if XAXIS or YAXIS equals LOG must be positive.

(Continued)

TABLEST

Material Property Temperature-Dependence Table

Description: 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	TID3	- etc.-		

Example:

TABLEST	105								
	130.0	20	195.0	40	ENDT				

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
Ti	Temperature values.	Real	Required
TIDi	Table identification numbers of TABLES1 entries.	Integer > 0	Required

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. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
4. This table is referenced only by MATS1 entries that define nonlinear elastic (TYPE = NLELAST) materials.

TABRND1 Power Spectral Density Table

Description: 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	5								
	3.1	0.01095	56.5	0.0543	ENDT				

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
XAXIS	Specifies a linear or logarithmic interpolation for the x-axis, one of the following character variables: LINEAR or LOG.	Character	LINEAR
YAXIS	Specifies a linear or logarithmic interpolation for the y-axis, one of the following character variables: LINEAR or LOG.	Character	LINEAR
fi	Frequency value in cycles per unit time.	Real ≥ 0.0	Required
gi	Power spectral density.	Real	Required

Remarks:

1. fi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points. If g is evaluated at a discontinuity, then the average value of g is used. In Figure 1, the value of g at f = f3 is g = (g3 + g4)/2. If the y-axis is a LOG axis the jump at the discontinuity is evaluated as $y = \sqrt{y3y4}$.
3. At least two entries must be present.
4. At least one continuation entry must be specified.
5. Placing SKIP in either of the two fields may ignore any fi-gi pair.
6. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.

7. 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 1. No warning messages are given if table data is input incorrectly. The algorithms used for interpolation or extrapolation 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	$\exp\left[\left(\frac{f_{i+1} - f}{f_{i+1} - f_i} \ln g_i + \frac{f - f_i}{f_{i+1} - f_i} \ln g_{i+1}\right)\right]$
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 $f_i < f < f_{i+1}$

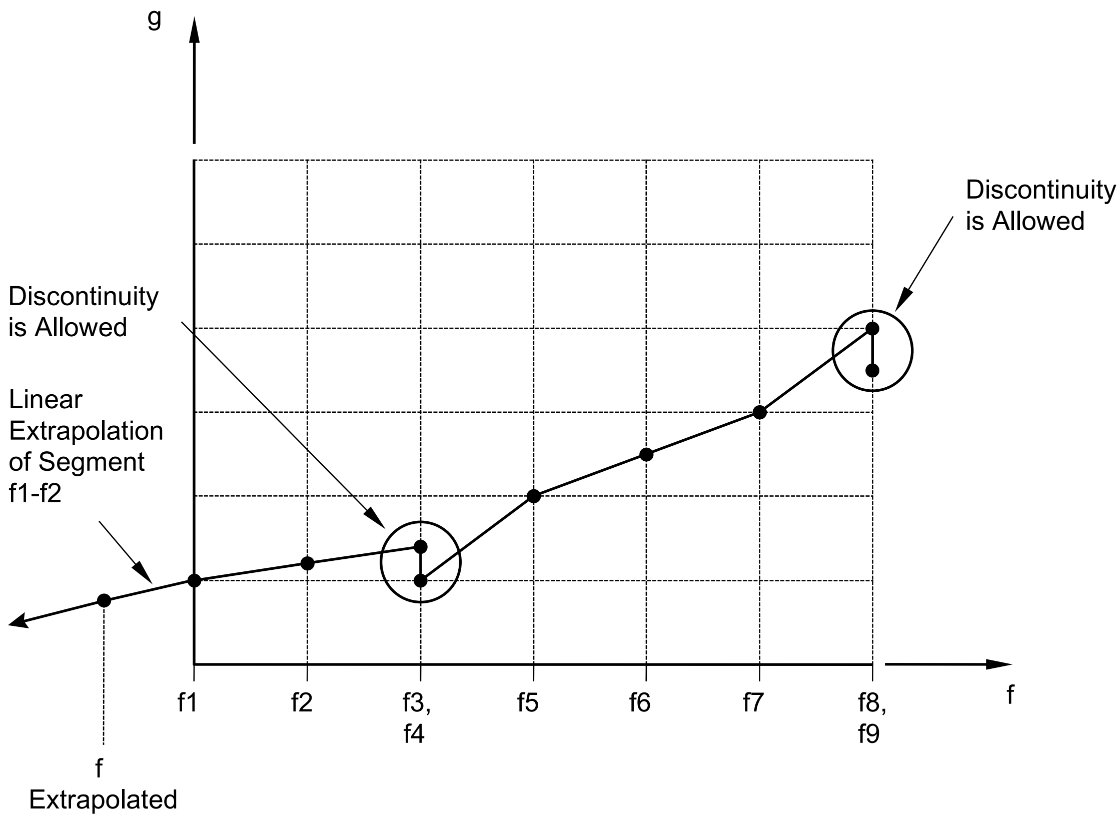


Figure 1. Example of Table Extrapolation and Discontinuity.

8. For auto spectral density, the value of g returned must be greater than or equal to zero.
9. Tabular values on an axis if XAXIS or YAXIS equals LOG must be positive.

TABVE — Viscoelastic Material Coefficient

Description: Defines a series of modulii and decay coefficients used for viscoelastic material definition.

Format:

1	2	3	4	5	6	7	8	9	10
TABVE	TID	MOD0							
	mod1	decay1	mod2	decay2	mod3	decay3	- etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
TABVE	101	0.0							
	38456.1	3.5-2	48122.2	0.22	ENDT				

Field	Definition	Type	Default
TID	Table identification number.	Integer > 0	Required
MOD0	The 0-th term of the modulus representation.	Real	0.0
modi	The optional i-th term of the modulus in the Prony series.	Real	
decayi	The optional i-th term of the decay coefficient in the Prony series.	Real	

Remarks:

1. At least one continuation entry must be specified.
2. Any xi-yi pair may be ignored by placing SKIP in either of the two fields.
3. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. Any continuations that follow the entry containing the end-of-table flag ENDT will be ignored.
4. The maximum number of terms allowed is 120. Exceeding this value will result in a fatal error.

TEMP Grid Point Temperature Field

Description: Defines temperature at grid points for determination of thermal and 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	Definition	Type	Default
SID	Temperature set identification number.	Integer > 0	Required
Gi	Grid point identification number.	Integer > 0	Required
T_i	Temperature value.	Real	Required

Remarks:

1. Set ID must be unique with respect to all other LOAD type entries.
2. From one to three grid point temperatures may be defined on a single entry.
3. 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.
4. Grid point temperatures are obtained by averaging element temperatures at the grid point. If no element temperature is specified then the temperature defined by the above entry is used.
5. Equivalent grid point loads are computed by numerical integration using isoparametric shape functions. Note that a uniform temperature will not necessarily result in equivalent grid point loads.

TEMPBC

Grid Point Temperature

Description: Defines transient and steady state temperature boundary conditions for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPBC	SID	TYPE	TEMP1	G1	TEMP2	G2	TEMP3	G3	

Example:

TEMPBC	5	STAT	50.0	1	100.0	2	150.0	3	

Alternate Format and Example:

TEMPBC	SID	TYPE	TEMP1	G1	THRU	G2	BY	INC	

TEMPBC	10	STAT	100.0	5	THRU	60	BY	5	

Field	Definition	Type	Default
SID	Temperature set identification number.	Integer > 0	Required
TYPE	Type of temperature boundary, one of the following character variables: STAT for a constant temperature boundary condition or TRAN for a time-varying temperature boundary condition.	Character	Required
TEMPi	Temperature value.	Real	Required
Gi	Grid point identification number(s).	Integer > 0; G2 $>$ G1	Required
INC	Grid point number increment.	Integer or blank	1

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 and transient analysis.
2. For a time-varying boundary condition (TYPE = TRAN), SID is referenced by a TLOADi Bulk Data entry through the DAREA specification. TYPE = TRAN is permitted only in transient analysis. A function of time $F(t - \tau)$ defined on the TLOADi entry multiplies the general load where τ provides any required time delay. The load set identifier on the TLOADi entry must be selected in the Case Control (DLOAD = SID) for use in transient analysis.

TEMPD — Grid Point Temperature Field Default

Description: Defines a temperature value for all grid points of the structural model that has 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	Definition	Type	Default
SIDi	Temperature set identification number.	Integer > 0	Required
Ti	Temperature value.	Real	Required

Remarks:

1. SIDi must be unique with respect to all other LOAD type entries.
2. From one to four grid point temperatures may be defined on a single entry.
3. 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.
4. Grid point temperatures are obtained by averaging element temperatures at the grid point. If no element temperature is specified then the temperature defined by the above entry is used.
5. Equivalent grid point loads are computed by numerical integration using isoparametric shape functions. Note that a uniform temperature will not necessarily result in equivalent grid point loads.

TEMPP1 — Shell Element Temperature Field, Form 1

Description: Defines a temperature field for shell elements (by an average temperature and a thermal gradient through the thickness) for determination of thermal loading and stress recovery.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPP1	SID	EID1	T	T′	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:

TEMPP1	EID2	THRU	EIDi	EIDj	THRU	EIDk			

TEMPP1	1	THRU	10	30	THRU	61			

Field	Definition	Type	Default
SID	Temperature set identification number.	Integer > 0	Required
EIDi, EIDj, EIDk	Element identification number(s).	Integer > 0; EID2 < EIDi, EIDj < EIDk	Required
T	Average temperature through the thickness. Assumed constant over area.	Real	Required
T′	Effective linear thermal gradient through thickness. Assumed constant over area.	Real	Required
T1, T2	Temperatures used to determine average temperature through the thickness and linear thermal gradient, if not specified in fields 4 and 5.	Real	Required if T and T′ are not specified

Remarks:

1. SET ID must be unique with respect to all other LOAD type entries if TEMPERATURE is specified in the Case Control Section.
2. Only CQUAD4, CQUADR, CTRIA3, or CTRIAR elements may have a temperature field applied to them via this entry.
3. If continuation entries are present, EID1 and elements specified on the continuation entry are used.
4. 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. For temperature field other than a constant gradient, the "effective gradient" for a homogeneous plate is:

$$T' = \frac{1}{I}\int_{z} T(z) \cdot z \cdot dz$$

where *I* is the bending inertia and *z* is the distance from the neutral surface in the positive normal direction.

7. The average temperature for a homogeneous plate is:

$$T = \frac{1}{Volume}\int_{Volume} T \cdot dVolume$$

TEMPRB — Rod and Bar Element Temperature Field

Description: Defines a temperature field for CROD and CBAR elements for determination of thermal loading and stress recovery.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPRB	SID	EID1	TA	TB	T'1A	T'1B	T'2A	T'2B	
	TCA	TDA	TEA	TFA	TCB	TDB	TEB	TFB	
	EID2	EID3	EID4	EID5	EID6	EID7	- etc.-		

Example:

TEMPRB	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			

	2	THRU	4	10	THRU	14			

Field	Definition	Type	Default
SID	Temperature set identification number.	Integer > 0	Required
EIDi, EIDj, EIDk	Element identification number(s).	Integer > 0; EID2 < EIDi, EIDj < EIDk	
TA, TB	Average temperature over the area at end A and end B.	Real	
T'ij	Effective linear gradient in direction i on end j (CBAR only).	Real	
Tij	Temperature at point i as defined on the PBAR entry at end j.	Real	

Remarks:

1. SID must be unique with respect to all other LOAD type entries.

2. If field 6 and/or 7 is blank, the effective linear thermal gradient is calculated using the stress recovery temperatures (fields 2 through 9 on the continuation entry) and stress recovery coefficients (fields 2 through 9 on the PBAR continuation entry. For example the equation at end A is:

$$T'_{1A} = \frac{\Delta T}{\text{Depth}}$$

where,

$$\Delta T = \frac{(T_{CA} + T_{FA}) - (T_{DA} + T_{EA})}{2}$$

$$\text{Depth} = \frac{(C_1 + F_1) - (D_1 + E_1)}{2}$$

Note: It is assumed that all four stress recovery coefficients are specified and that they are ordered as follows: C(+,+), D(-,+), E(-,-), F(+,-) in the y-z coordinate system.

3. The linear temperature gradients, not the Tij values, are used for stress recovery.
4. If the second (and succeeding) continuation is present, EID1 and elements specified on the second (and succeeding) continuations are used.
5. Elements must not be specified more than once.
6. 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.
7. The effective thermal gradients in the element coordinate system for the bar element are defined by the following integrals over the cross-section. For end A (end B is similar):

$$T'_{1A} = \frac{1}{I_1} \int_A T_A(y,z)\, y\, dA'$$

$$T'_{2A} = \frac{1}{I_2} \int_A T_A(y,z)\, z\, dA'$$

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 and I2 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 T'1A, T'1B, T'2A, and T'2B are the actual gradients.

TIC Transient Initial Condition

Description: Defines values for initial conditions of variables used in transient response analysis. Displacement, velocity, and acceleration may be specified at independent degrees of freedom.

Format:

1	2	3	4	5	6	7	8	9	10
TIC	SID	G	C	U0	V0	A0			

Example:

TIC	10	25	2	12.5	-5.0				

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
Pi	Grid point identification number.	Integer > 0	Required
Ci	Component number of global coordinate (up to six unique digits may be placed in the field with no embedded blanks).	0 ≤ Integer ≤ 6	Required
U0	Initial displacement.	Real	0.0
V0	Initial velocity.	Real	0.0
A0	Initial acceleration.	Real	0.0

Remarks:

1. Transient initial condition sets must be selected with the Case Control command IC = SID.
2. If no TIC set id selected in the Case Control Section, all initial conditions are assumed zero.
3. Initial conditions for coordinates not specified on TIC cards will be assumed zero.

TLOAD1 — Transient Response Dynamic Load, Form 1

Description: Defines a time-dependent dynamic load or enforced motion of the form

$$P(t) = A * F(t - \tau)$$

for use in transient response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	EXCITEID	DELAY	TYPE	TID				

Example:

TLOAD1	10	100			205				

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
EXCITEID	DAREA or SPCD entry set identification number that defines *A*.	Integer > 0	Required
DELAY	DELAY set identification number that defines τ.	Integer > 0 or blank	
TYPE	Defines the nature of the dynamic excitation. See Remark 2.	0 ≤ Integer ≤ 3 or character	0
TID	TABLEDi set identification number that defines *F*(*t*).	Integer > 0	Required

Remarks:

1. Dynamic load sets must be selected with the Case Control command DLOAD = SID.
2. The nature of the dynamic excitation is defined in the following table:

TYPE	Type of Dynamic Excitation
0, L, or LOAD	Applied load (force or moment) (default)
1, D, or DISP	Enforced displacement using large mass or SPCD
2, V, or VELO	Enforced velocity using large mass or SPCD
3, A, or ACCE	Enforced acceleration using large mass or SPCD

3. The TYPE field determines the manner in which the EXCITEID field is used as described below
   a) Excitation specified by TYPE is an applied load
      - If there *is no* LOADSET request in the Case Control then EXCITEID may directly reference DAREA, static, and thermal load set entries.
      - If there *is a* LOADSET request in the Case Control then the model will reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID.

   b) Excitation specified by TYPE is an enforced motion
      - If there *is no* LOADSET request in the Case Control then EXCITEID will reference SPCD entries. If these entries indicate null enforced motion, Autodesk Nastran 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.
      - If there *is a* LOADSET request in Case Control then the model will reference SPCD entries specified by the LID field in the selected LSEQ entries corresponding to the EXCITEID. If these entries indicate null enforced motion, Autodesk Nastran will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries corresponding to the DAREA entry in the selected LSEQ entries, just as in the case of applied load excitation.

4. EXCITEID may reference sets containing QHBDY, QBDYi, and QVOL entries in heat transfer analysis.
5. If DELAY is blank or zero, τ will be set to zero.

TLOAD2 Transient Response Dynamic Load, Form 2

Description: Defines a time-dependent dynamic load or enforced motion of the form

$$P(t) = \begin{cases} 0, & t < (T1+\tau) \text{ or } t > (T2+\tau) \\ A\,\tilde{t}^{B} e^{C\tilde{t}} \cos(2\pi F \tilde{t} + P), & (T1+\tau) \le t \le (T2+\tau) \end{cases}$$

for use in transient response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	EXCITEID	DELAY	TYPE	T1	T2	F	P	
	C	B							

Example:

TLOAD2	25	55			1.0	4.9	10.5		
	3.0								

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
EXCITEID	DAREA or SPCD entry set identification number that defines *A*.	Integer > 0	Required
DELAY	DELAY set identification number that defines τ.	Integer > 0 or blank	
TYPE	Defines the nature of the dynamic excitation. See Remark 2.	0 ≤ Integer ≤ 3 or character	0
T1	Time constant.	Real ≥ 0.0	Required
T2	Time constant.	Real; T2 > T1	Required
F	Frequency in cycles per unit time.	Real ≥ 0.0	0.0
P	Phase angles in degrees.	Real	0.0
C	Exponential coefficient.	Real	0.0
B	Growth coefficient.	Real	0.0

Remarks:

1. Dynamic load sets must be selected with the Case Control command DLOAD = SID.

2. The nature of the dynamic excitation is defined in the following table:

TYPE	Type of Dynamic Excitation
0, L, or LOAD	Applied load (force or moment) (default)
1, D, or DISP	Enforced displacement using large mass or SPCD
2, V, or VELO	Enforced velocity using large mass or SPCD
3, A, or ACCE	Enforced acceleration using large mass or SPCD

3. The TYPE field determines the manner in which the EXCITEID field is used as described below
   a) Excitation specified by TYPE is an applied load
      - If there *is no* LOADSET request in the Case Control then EXCITEID may directly reference DAREA, static, and thermal load set entries.
      - If there *is a* LOADSET request in the Case Control then the model will reference static and thermal load set entries specified by the LID or TID field in the selected LSEQ entries corresponding to the EXCITEID.
   b) Excitation specified by TYPE is an enforced motion
      - If there *is no* LOADSET request in the Case Control then EXCITEID will reference SPCD entries. If these entries indicate null enforced motion, Autodesk Nastran 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.
      - If there *is a* LOADSET request in Case Control then the model will reference SPCD entries specified by the LID field in the selected LSEQ entries corresponding to the EXCITEID. If these entries indicate null enforced motion, Autodesk Nastran will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries corresponding to the DAREA entry in the selected LSEQ entries, just as in the case of applied load excitation.
4. EXCITEID may reference sets containing QHBDY, QBDYi, and QVOL entries in heat transfer analysis.
5. If DELAY is blank or zero, τ will be set to zero.
6. The continuation entry is optional.

TOPVAR

Topological Design Variable

Description: Defines 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	MCID	MSi	MSi	MSi	CS	NCS	GSTID	
	EXT	MCID	EDi			TOOLD			
	ALM	MCID	ADi	ALPHA					
	MILL	MCID	MDi	MTYPE	TOOLL	TOOLD		NLEVEL	
	CAST	MCID	DDi	DIE					
	TDMIN	TV							

Example:

1	2	3	4	5	6	7	8	9	10
TOPVAR	1	DR02	PSOLID	0.4				100	
	SYM	10	XY	ZX					

Field	Definition	Type	Default
ID	Topology design region identification number.	Integer > 0	Required
LABEL	Label associated with design region used for output headings.	Character	
PTYPE	Property type. Used with PID to identify the elements to be designed, one of the following character variables: PSOLID, PSHELL, or PCOMP.	Character	Required
XINIT	Initial value for design variable. Typically XINIT is defined to match the mass target constraint, so the initial design does not have violated constraints.	XLB ≤ XINIT	0.5
XLB	Lower bound for design variable to prevent the singularity of the stiffness matrix.	Real > 0.0	1.0E-03
DELXV	Fractional change allowed for the design variable during design iteration. See Remark 3.	Real > 0.0	0.2
POWER	A penalty factor used in the relation between topology design variables and element Young's modulus. The range between 2.0 ≤ POWER ≤ 5.0 is recommended. See Remark 3.	Real > 1.0	3.0

Field	Definition	Type	Default
PID	Property identification number. Must be unique with respect to the PID values specified in other TOPVAR entries as design regions cannot share the same element. See Remark 1.	Integer > 0	Required
SYM	Symbol indicating that this line defines symmetry constraints.	Character	
MCID	Coordinate system identification number to define symmetric planes. See Remark 4.	Integer > 0 or blank	
MSi	Mirror symmetry planes, one of the following character variables: XY, YZ, or ZX. See Remark 4.	Character	
CS	Cyclic symmetry axis, one of the following character variables: X, Y, or Z. See Remark 10.	Character	
NCS	Number of cyclic symmetry segments in 360 degrees. See Remark 10.	Integer > 0 or blank	
GSTID	Reference topology design region identification number for global symmetry option. See Remark 11.	Integer > 0 or blank or AUTO	No global symmetry
EXT	Symbol indicating that this line defines extrusion constraints (i.e., enforced constant cross-section).	Character	
EDi	Extrusion direction, one of the following character variables: X, Y, Z, -X, -Y, -Z, +X, +Y, +Z, where -X, -Y, -Z indicates the opposite direction of X, Y, Z and +X, +Y, +Z the same direction. See Remarks 4 and 5.	Character	Required
TDMIN	Symbol indicating that minimum member size is specified.	Character	
TV	Minimum member size. See Remark 6.	Character	Required
ALM	Symbol indicating that this line defines additive layer manufacturing constraints. See Remark 7.		
ADi	Print direction, one of the following character variables: X, Y, Z, -X, -Y, -Z, +X, +Y, +Z, where -X, -Y, -Z indicates the opposite direction of X, Y, Z and +X, +Y, +Z the same direction. See Remarks 4 and 5.	Character	Required
ALPHA	Maximum overhang angle (measured in degrees).	Real	45.0
MILL	Symbol indicating that this line defines milling manufacturing constraints. See Remark 7.		
MDi	Milling direction for 5-axis, 3-axis and 2.5-axis milling. For 5-axis the identification number of a MILLDIR milling direction data entry. For 3-axis one of the following character variables: X, Y, Z, -X, -Y, -Z, +X, +Y, +Z, XY, YZ, ZX, -XY, -YZ, -ZX, +XY, +YZ, +ZX, XYZ, -XYZ, +XYZ where -X, -Y, -Z indicates the opposite direction of X, Y, Z and +X, +Y, +Z the same direction. For 2.5-axis one of the following character variables: X, Y, Z, -X, -Y, -Z, +X, +Y, +Z. The absence of a sign specifies both positive and negative directions. See Remarks 4 and 5.	Character/ Integer > 0	Required
MTYPE	Milling type. The following milling types are allowed: 3AXIS for 3-axis milling 5AXIS for 5-axis milling 2.5AXIS for 2.5-axis milling		3AXIS

Field	Definition	Type	Default
	2AXIS for 2-axis milling		
TOOLL	Milling tool length.	Real	
TOOLD	Milling tool bit diameter.	Real	
NLEVEL	Number of 2.5 axis milling levels.	$1 \leq$ Integer ≤ 20 or blank	Required for MILL=2.5AXIS
CAST	Symbol indicating that this line defines casting constraints (i.e., die draw direction constraints). See Remarks 7, 8, and 9.		
DDi	Draw direction, one of the following character variables: X, Y, Z, -X, -Y, -Z, +X, +Y, +Z, where -X, -Y, -Z indicates the opposite direction of X, Y, Z and +X, +Y, +Z the same direction. See Remarks 4 and 5.	Character	
DIE	Die option selected by one of the following values 1 = Single die 2 = Two dies When a single die is specified the die slides in the given draw direction (i.e., material grows from the bottom in the draw direction). When two dies are specified 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).	Blank or Integer 1 or 2	1

Remarks:

1. The topologically designable element properties include PSHELL, PCOMP, and PSOLID. Multiple TOPVAR entries are allowed in a single file. Those elements whose PID is not specified in TOPVAR entries are considered to be non-designable elements; that is, they are considered to be fully filled by the material and are not changed during topology optimization.
2. When X is the topology design variable of an element, the Young's modulus of the element is calculated by

$$E = X^{\text{POWER}} E_0$$

where,

E_0 is Young's modulus of the material

3. A blank in field 8 uses an initial value for POWER of 1.0 and then increments POWER each design iteration to a maximum value of 4.0.
4. One, two, or three different mirror symmetry planes can be present (such as MS1 = XY, MS2 = YZ, and MS3 = ZX). When the mesh is regular and parallel to the coordinate system MCID, all elements on the positive coordinate side are considered to have independent design variables, and elements on the negative side are considered dependent design. When the mesh is not regular or not parallel to the coordinate system MCID, an element in the negative coordinate side is considered dependent if the element is moved to the mirror plane and if there is an independent element on the positive side within the distance specified by the model parameter TOPTELEMSYMTOL (see Section 5, *Parameters*, for more information on TOPTELEMSYMTOL).
5. Some symmetry constraint types can be combined with extrusion, casting, milling, and additive manufacturing constraints. The referenced coordinate system CID must be the same for the combined constraints. For example with extrusion some possible combinations are: (EDi = X, MSi = XY, and/or ZX), (ED = Y, MSi = YZ, and/or XY), (ED = Z, MSi = ZX, and/or YZ).
6. TDMIN is a dimensional quantity with a guideline that it be set to at least three times a representative element dimension. Without a TDMIN continuation line, the minimum member size is set to 3 levels of adjacent elements. Minimum member size constraints can be used with all other manufacturing constraint types.

7. Casting, milling, and additive manufacturing constraints cannot be combined with extrusion constraints for the same TOPVAR entry.
8. For two dies option (DIE = 2), the splitting plane is the surface closest to origin of MCID. For a single die DIE = 1, the parting plane is the bottom surface of the designed part in the draw direction.
9. 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.
10. The first symmetry segment starts at the X-axis when CS = Z (at the Z-axis when CS = Y, and at the Y-axis when CS = X). One cyclic symmetry manufacturing constraint can be combined with one planar symmetry constraint when the axis of cyclic symmetry is normal to the planar symmetry plane (e.g., MSi = YZ with CS = X).
11. Global symmetry allows multiple TOPVAR regions to use a common set of symmetry and extrude constraints. When a GSTID identification number is specified, that region's manufacturing constraint data is used (SYM and EXT options only). The ALM, MILL, and CAST manufacturing constraints are not compatible with this option.

TSTEP — Transient Time Step

Description: Defines time step intervals at which a solution will be generated and output in transient response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TSTEP	SID	N1	DT1	NO1	ADJUST	MSTEP	RB	MAXR	
		N2	DT2	NO2					
		- etc.-							

Example:

TSTEP	25	100	0.005	5					
		50	0.001	3					

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
Ni	Number of time steps of values DTi.	Integer ≥ 1	Required
DTi	Time increment.	Real > 0	Required
NOi	Skip factor for output. Every NOi-th step will be output.	Integer > 0	1
ADJUST	Time step skip factor for automatic time step adjustment. See Remark 3.	Integer ≥ 0	5
MSTEP	Number of steps to obtain the dominant period response. See Remark 4.	10 ≤ Integer ≤ 200	Variable between 20 and 40.
RB	Bounds for maintaining the same time step for the stepping function. See Remark 4.	0.1 ≤ Real ≤ 1.0	0.75
MAXR	Maximum ratio for the adjusted incremental time relative to DT allowed for time step adjustment. See Remark 5.	1.0 ≤ Real ≤ 32.0	16.0

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 100 time steps of value 0.005, which is then followed by 50 time steps of value 0.001. Results will be output for t = 0.0, 0.005, 0.01, 0.015, 0.02, etc. This feature is not supported in direct transient solutions. To change the time step size in a direct transient solution use multiple subcases each referencing a different TSTEP Bulk Data entry.

3. ADJUST controls the automatic time stepping when PARAM, ADAPTTIMESTEP is set to ON and the solution type is direct transient (see Section 5, *Parameters*, for more information on ADAPTTIMESTEP). 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. 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.
   a) If ADJUST = 0, then the automatic adjustment is deactivated. This is recommended when the loading consists of short duration pulses.
   b) If ADJUST > 0, the time increment is continually adjusted for the first few steps until a good value is obtained. After this initial adjustment, the time increment is adjusted every ADJUST time step only.
   c) If ADJUST is one order greater than NDT, then automatic adjustment is deactivated after the initial adjustment.
4. MSTEP and RB are used to adjust the time increment during analysis when PARAM, ADAPTTIMESTEP is set to ON and the solution type is direct transient. The recommended value of MSTEP is 20.

   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,

$$r = \frac{1}{\text{MSTEP}}\left(\frac{2\pi}{\omega_n}\right)\left(\frac{1}{\Delta t_n}\right)$$

   and,

$f = 0.25$	for	$r < 0.5*RB$
$f = 0.5$	for	$0.5*RB \le r < RB$
$f = 1.0$	for	$RB \le r < 2.0$
$f = 2.0$	for	$2.0 \le r < 3.0/RB$
$f = 4.0$	for	$r \ge 3.0/RB$

5. MAXR is used to define the upper and lower bounds for adjusted time step size such that

$$\text{MIN}\left(\frac{\text{DT}}{2^{\text{MAXBIS}}}, \frac{\text{DT}}{\text{MAXR}}\right) \le \Delta t \le \text{MAXR} * \text{DT}$$

TSTEPNL

Parameters for Nonlinear Transient Analysis

Description: Defines a set of parameters for nonlinear transient analysis.

Format:

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO	METHOD	KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXUBIS	MAXLS	FSTRESS	LSTOL	
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB		
		TDG	TDC	TDV					

Example:

TSTEPNL	120	200	0.001	5	ADAPT		15	PW	

Field	Definition	Type	Default
ID	Identification number	Integer > 0	Required
NDT	Number of time steps of value DT. See Remark 2.	Integer > 0	Required
DT	Time increment. See Remark 2.	Real > 0.0	Required
NO	Time step interval for output. Every NOi-th step will be output. See Remark 3.	Integer > 0	1
METHOD	Method for controlling stiffness updates, one of the following character variables: AUTO, TSTEP, or ADAPT. See Remark 4.	Character	ADAPT
KSTEP	Number of time steps before stiffness update for the TSTEP method. See Remark 4.	Integer > 0	5
MAXITER	Limit on number of iterations for each time step. See Remark 5.	Integer > 0 or AUTO	AUTO
CONV	Convergence criteria, one of the following character variables: U, P, or W, or any combination. See Remark 6.	Character	PW
EPSU	Error tolerance for displacement (U) criterion.	Real > 0.0	See Remark 17
EPSP	Error tolerance for load (P) criterion.	Real > 0.0	See Remark 17
EPSW	Error tolerance for work (W) criterion.	Real > 0.0	See Remark 17
MAXDIV	Limit on probable divergence conditions per iteration before the solution is assumed to diverge. See Remark 7.	Integer > 0	3
MAXUBIS	Maximum number of iterations for an upward load increment adjustment. Applicable when the load increment is bisected.	Integer > 0	7

Field	Definition	Type	Default
MAXLS	Maximum number of line searches for each iteration. See Remark 8.	Integer ≥ 0	4
FSTRESS	Fraction of effective stress ($\bar{\sigma}$) used to limit the subincrement size in nonlinear material routines. See Remark 9.	$0.0 <$ Real < 1.0	0.2
LSTOL	Line search tolerance. See Remark 8.	$0.01 <$ Real ≤ 0.9	0.5
MAXBIS	Maximum number of bisections allowed for each time step. See Remark 10.	Integer > 0	5
ADJUST	Time step skip factor for automatic time step adjustment. See Remark 11.	Integer ≥ 0	5
MSTEP	Number of steps to obtain the dominant period response. See Remark 12.	$10 \leq$ Integer ≤ 200	Variable between 20 and 40.
RB	Bounds for maintaining the same time step for the stepping function. See Remark 12.	$0.1 \leq$ Real ≤ 1.0	0.75
MAXR	Maximum ratio for the adjusted incremental time relative to DT allowed for time step adjustment. See Remark 13.	$1.0 \leq$ Real ≤ 32.0	16.0
UTOL	Tolerance on displacement or temperature increment below which a special provision is made for numerical stability. See Remark 14.	$0.001 <$ Real ≤ 1.0	0.1
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 15.	Real > 2.0	20.0
TDG	Terminate on displacement grid point identification number. See Remark 16.	Integer > 0	
TDC	Terminate on displacement component number. See Remark 16. MAXT Resultant of translation displacement components. MAXR Resultant of rotational displacement components.	$0 \leq$ Integer ≤ 6 or MAXT or MAXR	MAXT
TDV	Terminate on displacement value. See Remark 16.	Real	

Remarks:

1. The TSTEPNL Bulk Data entry must be selected by the Case Control command TSTEPNL = ID. Each solution subcase requires a TSTEPNL command 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. Initial conditions specified using the IC Case Control command apply only to the first subcase.
2. NDT is used to define the total duration for analysis, which is NDT∗DT. Since the adaptive time integration method uses a variable time increment, the actual number of time steps will usually not be equal to NDT. Also, DT is used only as an initial value for the time increment.

3. Results output is generated at time steps 1, NO, 2∗NO, 3∗NO,…, and the last converged step. The Case Control command OTIME may also be used to control the output times.
4. The stiffness update strategy is selected in the METHOD field.
   a) If the AUTO option is specified, the stiffness matrix is automatically updated based on convergence.
   b) If the TSTEP option is selected, the stiffness matrix is updated at every KSTEP increment of time.
   c) If the ADAPT option is selected, the time step is automatically adjusted based on the severity of the nonlinearity and a stiffness matrix update is performed.

   In all methods the stiffness matrix is always updated for new subcase.
5. The number of iterations for a time increment is limited to MAXITER. If the solution does not converge in MAXITER iterations, one of two actions is taken depending on the BISECT model parameter. If the BISECT model parameter is set to ON, the time increment is bisected and the analysis is repeated. If the time increment cannot be bisected (i.e. MAXBIS is attained), execution terminates with a fatal error. If the BISECT model parameter is set to OFF, the analysis is continued to the next load increment. (See Section 5, *Parameters*, for more information on BISECT.) The default AUTO setting uses an initial MAXITER value of 40 and automatically increases this value if the solution appears near convergence.
6. The symbols (U for displacement error, P for load equilibrium error, and W for 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.
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 $\geq$ MAXDIV. If the solution diverges and the load increment cannot be further bisected (i.e., MAXBIS is attained), execution terminates with a fatal error.
8. The line search is performed as required if MAXLS $>$ 0. The line search procedure scales the displacement increment to minimize the energy error. The procedure is skipped if the absolute value of the relative energy error is less than the value specified by LSTOL.
9. The number of subincrements in the material routines is determined so that the subincrement size is approximately FSTRESS $*\ \bar{\sigma}$ (equivalent stress).
10. The number of bisections for a load increment is limited to MAXBIS. If the solution diverges, the stiffness is updated on the first divergence and the load is bisected on the second divergence.
11. ADJUST controls the automatic time stepping for METHOD = ADAPT. 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. 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.
    a) If ADJUST = 0, then the automatic adjustment is deactivated. This is recommended when the loading consists of short duration pulses.
    b) If ADJUST $>$ 0, the time increment is continually adjusted for the first few steps until a good value is obtained. After this initial adjustment, the time increment is adjusted every ADJUST time step only.
    c) 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 for METHOD = ADAPT. 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 changes in the global stiffness matrix.

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,

$$r = \frac{1}{\text{MSTEP}}\left(\frac{2\pi}{\omega_n}\right)\left(\frac{1}{\Delta t_n}\right)$$

and,

$f = 0.25$ for	$r < 0.5*RB$	
$f = 0.5$	for	$0.5*RB \le r < RB$
$f = 1.0$	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 such that

$$\text{MIN}\left(\frac{\text{DT}}{2^{\text{MAXBIS}}}, \frac{\text{DT}}{\text{MAXR}}\right) \le \Delta t \le \text{MAXR} * \text{DT}$$

14. UTOL is a tolerance used to filter undesirable time step adjustments such that

$$\frac{\|\dot{U}_n\|}{\|\dot{U}\|_{max}} < \text{UTOL}$$

Under this condition no time step adjustment is performed.

15. The load increment is bisected 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. When TDG, TDC, and TDV are specified the solution will proceed until either all load is applied or the specified displacement value (TDV) at grid point TDG in direction TDC is reached or exceeded. Displacements are in the displacement coordinate system of the TDG grid point.
17. Default tolerance sets are determined based on solution type, nonlinear behavior requested, and desired accuracy. Accuracy is under user control and can be specified using PARAM, NLTOL (see Section 5, *Parameters*, for more information on NLTOL). The NLTOL values are only used if one or more of the EPSU, EPSP and EPSW fields on the TSTEPNL entry are blank. The following tables show the tolerance values used depending on the NLTOL model parameter setting specified.

Nonlinear Transient Dynamic Analysis without Contact and Material Nonlinearity

NLTOL	Level of Accuracy	EPSU	EPSP	EPSW
0	Very High	1.0E-3	1.0E-3	1.0E-6
1	High	1.0E-3	1.0E-3	1.0E-5
2	Engineering	5.0E-3	5.0E-3	1.0E-5
3	Preliminary Design	1.0E-2	1.0E-2	1.0E-4
Default	Engineering	5.0E-3	5.0E-3	1.0E-5

Nonlinear Transient Dynamic Analysis with Material Nonlinearity

NLTOL	Level of Accuracy	EPSU	EPSP	EPSW
0	Very High	1.0E-4	1.0E-4	1.0E-8
1	High	5.0E-4	5.0E-4	1.0E-8
2	Engineering	5.0E-4	5.0E-4	1.0E-7
3	Preliminary Design	1.0E-3	1.0E-3	1.0E-6
Default	Engineering	5.0E-4	5.0E-4	1.0E-7

Nonlinear Transient Dynamic Analysis with Contact

NLTOL	Level of Accuracy	EPSU	EPSP	EPSW
0	Very High	1.0E-3	1.0E-3	1.0E-6
1	High	1.0E-3	1.0E-3	1.0E-5
2	Engineering	5.0E-3	5.0E-3	1.0E-5
3	Preliminary Design	1.0E-2	1.0E-2	1.0E-4
Default	Engineering	5.0E-3	5.0E-3	1.0E-5

Nonlinear Transient Heat Transfer

NLTOL	Level of Accuracy	EPSU	EPSP	EPSW
0	Very High	1.0E-3	1.0E-3	1.0E-6
1	High	1.0E-3	1.0E-3	1.0E-6
2	Engineering	1.0E-3	1.0E-3	1.0E-6
3	Preliminary Design	1.0E-3	1.0E-3	1.0E-6
Default	Engineering	1.0E-3	1.0E-3	1.0E-6

VFATIGUE Vibration Fatigue Data

Description: Defines data needed for vibration fatigue analysis.

Format:

1	2	3	4	5	6	7	8	9	10
VFATIGUE	SID	APRCH	METHOD			DT	TCF		
	STRESS	B	SU	N0	KF	BE	SE		
	STRAIN	SF	EF	B	C				

Example:

VFATIGUE	200	STRAIN	1			1.5+3			
	STRESS	0.16	4.5+3		0.9				
	STRAIN	1.7+9	0.83	0.095	0.65				

Field	Definition	Type	Default
SID	Set identification number.	Integer > 0	Required
APRCH	Fatigue life approach, one of the following character variables: STRESS or STRAIN.	Character	See Remark 2.
METHOD	Life calculation method, selected by one of the following values 1 = von Mises stress/strain 2 = Maximum principal stress/strain 3 = Maximum shear stress/strain	Integer	2
DT	Event duration used to determine life. See Remark 5.	Real > 0.0	Required
TCF	Factor to convert DT and life output to units other than seconds. See Remark 5.	Real > 0.0	1.0
B	S-N curve slope. See Remark 3.	Real > 0.0	See Remark 2.
SU	Intercept stress level. Typically taken as the material ultimate stress. See Remark 3.	Real > 0.0	See Remark 2.
N0	Intercept cycles. See Remark 3.	Integer > 0	1000
KF	Factor applied to compensate for life reduction effects such as finish, corrosion, and notch effects. See Remark 3.	Real > 0.0	1.0
BE	Slope after endurance limit. See Remark 3.	Real > 0.0	0.1∗B
SE	Endurance limit. See Remark 3.	Real ≥ 0.0	0.2∗SU
SF	Coefficient of fatigue strength. See Remark 4.	Real > 0.0	See Remark 2

Field	Definition	Type	Default
EF	Coefficient of fatigue ductility. See Remark 4.	Real > 0.0	See Remark 2
B	Exponent of fatigue strength. See Remark 4.	Real > 0.0	See Remark 2
C	Exponent of fatigue ductility. See Remark 4.	Real > 0.0	See Remark 2

Remarks:

1. VFATIGUE entries must all have unique set identification numbers.
2. The APRCH field is required when neither the SNDATA nor ENDATA Bulk Data entries are included. The data provided on the continuation entries serve as default values for properties normally defined on these entries. Values not specified on SNDATA entries will be replaced with ones from the STRESS continuation and values not specified on the ENDATA will be replaced with ones from the STRAIN continuation.
3. The S-N curve shown in Figure 1 is characterized by the following equations

   If $S_i \geq S_e$

   $$N_f = \text{N0}\left(\frac{\text{SU}}{\text{KF} * S_i}\right)^{\frac{1}{\text{B}}}$$

   If $S_i < S_e$

   $$N_f = N_e\left(\frac{\text{SE}}{\text{KF} * S_i}\right)^{\frac{1}{\text{BE}}}$$

   where,

   N_f is the number of cycles to failure

   S_i is the amplitude of input stress $(S_{max} - S_{min})/2$

   N_e is the number of failure cycles at the endurance limit

   and the slope B is shown in Figure 1 is calculated by

   $$B = \frac{\log(\text{SU}) - \log(\text{SE})}{\log(N_e) - \log(\text{N0})}$$

4. The ε-N curve shown in Figure 2 is characterized by the equation

   $$\frac{\varepsilon}{2} = \frac{\text{SF}}{E}(2N_f)^{\text{-B}} + \text{EF}(2N_f)^{\text{-C}}$$

   where,

   ε is the range of strain ($\varepsilon_{max} - \varepsilon_{min}$)

   $2N_f$ is the number of cycles to failure

   E is the modulus of elasticity

5. The default value for DT is determined using the difference between the largest and smallest TABLEDi times (time range). If the specified DT is smaller that this time range, it is set equal to it. DT is useful when the event duration is different from the time range due to idling time. TCF is a time conversion factor that is typically used to convert a default DT time from seconds to another set of units such as hours. Life output will be in the same units as DT where life is defined using

   $$Life = \frac{\text{DT} * \text{TCF}}{Damage}$$

   where,

   Damage is the ratio of applied cycles over cycles to failure.

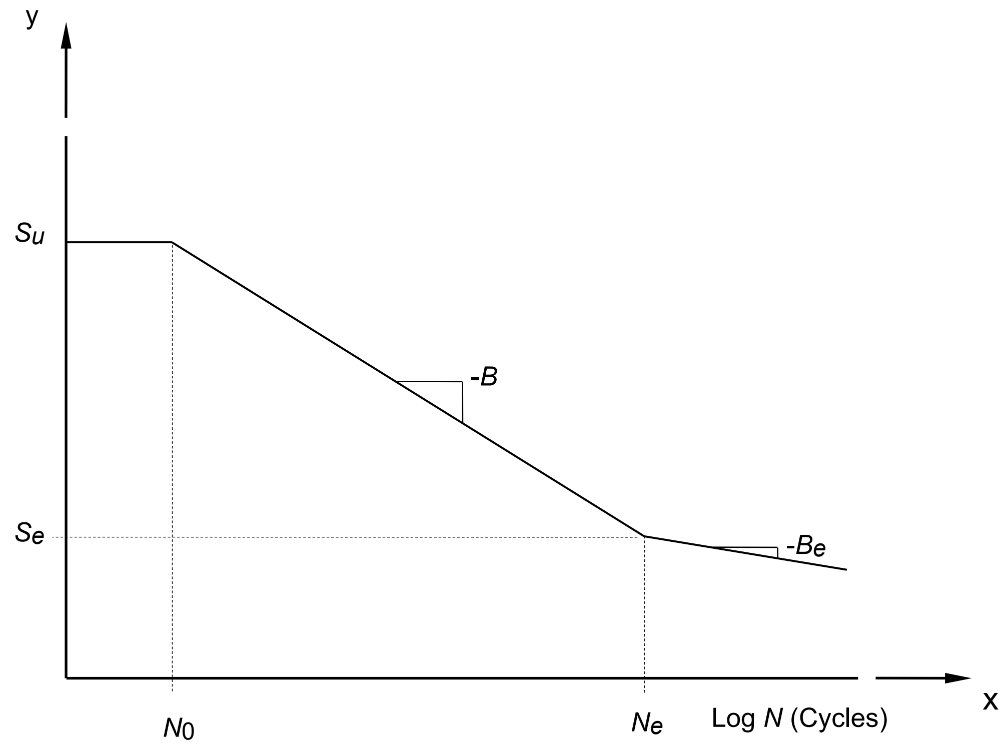


Figure 1. Stress-Life Curve Format.

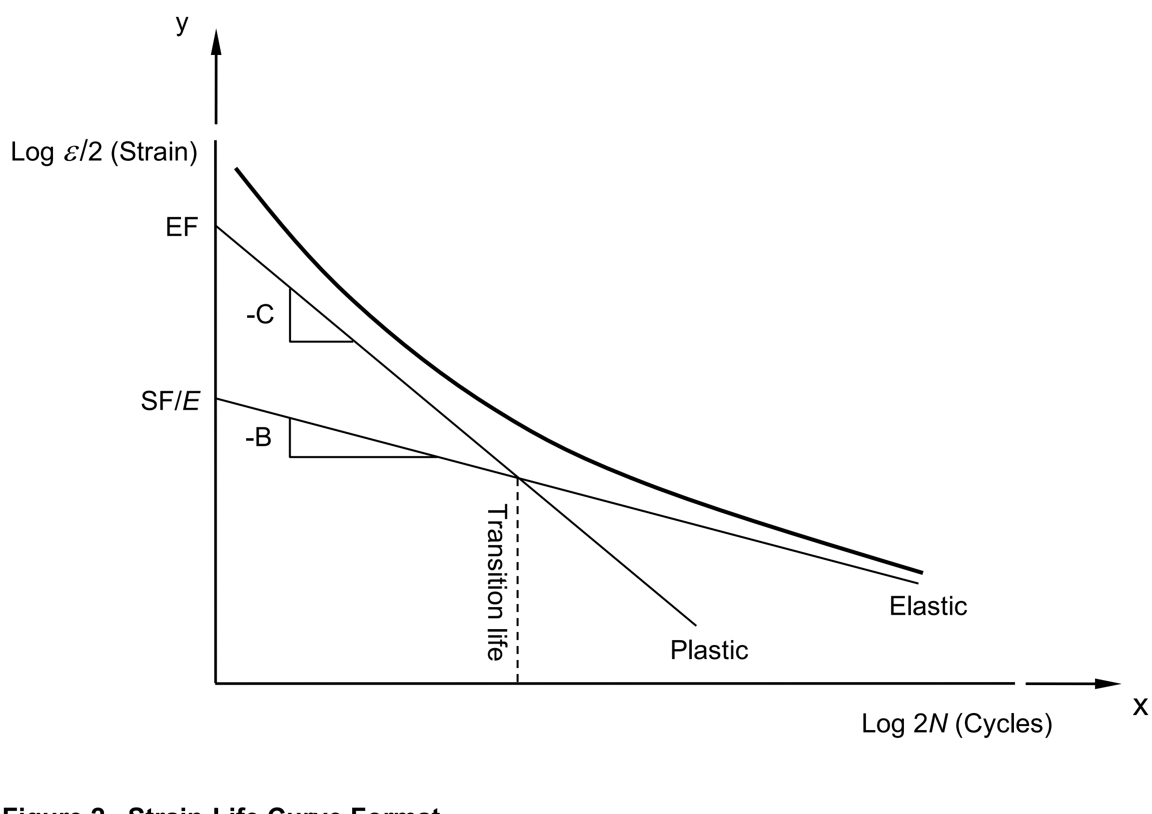


Figure 2. Strain-Life Curve Format.

VIEW — View Factor Definition

Description: 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						

Example:

VIEW	1	1	BOTH						

Field	Definition	Type	Default
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. One of the following characters variables: NONE, KSHD, KBSHD, BOTH: NONE The face can neither shade nor be shaded by other faces KSHD The the face can shade other faces KBSHD The face can be shaded by other faces BOTH The face can both shade and be shaded by other faces	Character	BOTH

Remarks:

1. VIEW must be referenced by 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. SHADE references shadowing for CHBDYi elements participating in a radiation cavity, the VIEW calculation can involve shadowing.

VIEW3D View Factor Definition – Gaussian Integration Method

Description: Defines parameters to control view factor calculation for a specified cavity.

Format:

1	2	3	4	5	6	7	8	9	10
VIEW3D	ICAVITY	MAXRU	MAXRO	MINRO	ITOL	ZTOL		VFDOUT	

Example:

VIEW3D	1	1	2	4		1.0E-6			

Field	Definition	Type	Default
ICAVITY	Radiant cavity identification number on RADCAV entry.	Integer > 0	
MAXRU	Maximum number of recursions used in computing unobstructed view factors. See Remark 1.	Integer > 0	8
MAXRO	Maximum number of recursions used in computing obstructed view factors. See Remark 1.	Integer > 0	8
MINRO	Minimum number of recursions used in computing obstructed view factors. See Remark 2.	Integer ≥ 0	0
ITOL	Integration convergence tolerance for both adaptive integration and view obstruction calculations. See Remark 3.	Real > 0.0	1.0E-5
ZTOL	View factor calculation zero tolerance. Value below which computed view factors are considered to be zero.	Real ≥ 0.0	1.0E-10
VFDOUT	View factor diagnostic output, one of the following character variables: YES or NO. When set to YES the following view factor calculation information is output to the Model Results Output File: • Area • View factor • Area-View factor product • Error estimate • Third-body showing • Enclosure summation	Character	YES

Remarks:

1. Limiting the maximum number of unobstructed recursions (MAXRU) or obstructed recursions (MAXRO) can reduce analysis time but may prevent reaching the specified convergence (ITOL). The default value provides a compromise between accuracy and analysis time.

2. The default minimum number of obstructed recursions (MINRO) may miss an obstruction. Increasing the default value of 0 to 1 or 2 can prevent this but at the cost of increased analysis time. Typically increasing MINRO is not necessary except when very accurate view factors are desired.
3. The value specified for ITOL is not an exact measure of the accuracy of the computed view factors, but smaller values will typically lead to more precise values. Values less than 1.0E-6 may not lead to improved accuracy.

XSET External Data Set Definition

Description: Defines degrees of freedom used with the XSETGENERATE Case Control command to generate the reduced eigendata set (e-set) used in Modal Assurance Criterion (MAC) analysis.

Format:

1	2	3	4	5	6	7	8	9	10
XSET	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

XSET	15	3	17	456	7	4			

Field	Definition	Type	Default
Gi	Grid point identification number(s).	Integer > 0	Required
Ci	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required

Remarks:

1. The XSET is used in the automated generation of the ESET using the XSETGENERATE Case Control command.

XSET1 — External Data Set Definition, Alternate Form

Description: Defines degrees of freedom used with the XSETGENERATE Case Control command to generate the reduced eigendata set (e-set) used in Modal Assurance Criterion (MAC) analysis.

Format:

1	2	3	4	5	6	7	8	9	10
XSET1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	- etc.-					

Example:

XSET1	123	6	3	7	10	18	14	11	
	19	23							

Alternate Format and Example:

XSET1	C	G1	THRU	G2					

XSET1	456	15	THRU	512					

Field	Definition	Type	Default
C	Component number of global coordinate. (Up to six unique digits may be placed in the field with no embedded blanks.)	$1 \leq$ Integers ≤ 6	Required
Gi	Grid point identification number(s).	Integer > 0; G1 < G2	Required

Remarks:

1. The XSET is used in the automated generation of the ESET using the XSETGENERATE Case Control command.
2. If the alternate form is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will be skipped.

Section 5

PARAMETERS

Parameter Descriptions

Parameters are used for input of scalar values and for requesting special features. Parameters can be specified in the Case Control and the Bulk Data Sections of the Model Input File, in the Model Initialization File, or on the Nastran command line. Note the following examples:

Model Input File

Case Control Section

```
PARAM, STIFFRATIOTOL, 1.0E-8
PARAM, AUTOSPC, ON
```

Bulk Data Section

```
PARAM, STIFFRATIOTOL, 1.0E-8
PARAM, AUTOSPC, ON
```

Model Initialization File

```
STIFFRATIOTOL = 1.0E-8
AUTOSPC = ON
```

Nastran Command Line

```
NASTRAN filename.NAS STIFFRATIOTOL=1.0E-8 AUTOSPC=ON
```

Parameters in the Case Control Section of the Model Input File use 16 character fields. Parameters in the Bulk Data Section use 8 character fields. Parameters specified in the Model Initialization File and on the Nastran command line use directive format (i.e., directive = option).

Model Translator Parameters:

Parameter	Description	Type	Default
ALIGNEDGENODE	When set to ON, will correct bad parabolic solid element geometry due to excessive curvature. PARAM, EDGENODETOL is used to specify the tolerance for repositioning nodes and is given in degrees of the curved edge relative to a straight one. When EDGENODETOL set to AUTO, any solid element with a non-positive Jacobian will have all curved edges aligned.	ON/OFF	OFF
AUTOFIXELEMGEOM	Option for automatically correcting elements that are singular due to an incorrect ordering of the element grid points.	ON/OFF	ON
AUTOFIXRIGIDELEM	When set to ON, will automatically correct improperly defined RBE3 elements by adding rotational degrees of freedom to averaging grid points as needed to prevent rigid body motion.	ON/OFF	ON
AUTOFIXRIGIDSPC	When set to ON, will automatically correct the following rigid element, interpolation element, and MPC equation issues by adding a near rigid spring at the dependent degrees of freedom: • A rigid element, interpolation element, or MPC equation dependent degree of freedom is constrained. • One or more rigid elements, interpolation elements, or MPC equations reference the same dependent degree of freedom. • A series of rigid elements, interpolation elements, and/or MPC equations are connected in a continuous link. • An RBE2 element is defined with the independent grid point located at the origin of a cylindrical coordinate system and rigidity is desired only in the R or T component direction. When AUTOFIXRIGIDSPC is set to OFF, behavior will be that of a rigid element defined in the Cartesian rectangular system which defined the specified cylindrical system. When AUTOFIXRIGIDSPC is set to ON and a translational or rotational component is missing, the local grid coordinate system at each independent grid point defines that dependent/independent segment. The spring element stiffness is defined by the KRIGIDELEM model parameter. See KRIGIDELEM below.	ON/OFF	OFF
COMPCONNECTOUT	When set to ON will check for disconnected components and output the number of model components and assemblies. Additionally, it will output the Model Component Definition and Model Component Connectivity Matrix to the Model Results Output File	ON/OFF	OFF
CYSYMGEN	Option for automatically generating cyclic symmetric boundary conditions on an axisymmetric model. When set to a valid cylindrical coordinate system id, boundary conditions are automatically generated which force cyclic symmetric behavior. Grid points are automatically identified at each r-z boundary plane based on the specified near tolerance, CYSYMTOL. See CYSYMTOL below.	Integer > 0	0
CYSYMTOL	Near tolerance used to identify boundary grid points for the application of cyclic symmetric boundary conditions. The actual tolerance is derived using CYSYMTOL and a model reference dimension. Each r-z boundary is identified as all grid points within this tolerance at the minimum and maximum θ values of the model.	Real	1.0E-10
EDGENODETOL	See ALIGNEDGENODE above.	Real AUTO	AUTO
FLOATINZERO	Character input floating point zero tolerance. Input real data less than FLOATINZERO will be set to zero. Material property data will not be zeroed.	Real	1.0E-15

Model Translator Parameters (Continued):

Parameter	Description	Type	Default
KRIGIDELEM	Stiffness value assigned to bush elements generated from converted RBE2 rigid elements. The AUTO setting will determine the optimum value based on model dimensions and the largest Young's modulus specified. See RIGIDELEM2ELAS and RIGIDELEMTYPE below.	Real AUTO	AUTO
MAXADJEDGE	This option is used to adjust storage space when using slide line and/or surface contact elements or when either the QUADEGRID, TRIEGRID, HEXEGRID, PENTEGRID, PYREGRID, TETEGRID, SHELLEGRID or SOLIDEGRID Model Initialization directives are set to ON resulting in a T2222 fatal error. A starting value between 10 and 100 is recommended but may need to be increased further if another T2222 error occurs. The AUTO setting will set MAXADJEDGE to 50 if SLINEMAXACTDIST is set to AUTO and zero if set otherwise.	Integer > 0 AUTO	AUTO
RIGIDELEM2ELAS	Rigid element to spring element conversion option. When RIGIDELEM2ELAS is set to ON, rigid elements (RBE2) will be converted to the element type specified by the RIGIDELEMTYPE model parameter. The AUTO setting enables rigid element thermal expansion effects when a non-modal solution type is selected and a coefficient of thermal expansion is specified on a RBE2 Bulk Data entry. See KRIGIDELEM above and RIGIDELEMTYPE below.	ON/OFF AUTO	AUTO
RIGIDELEMCORD	Rigid and interpolation element individual coordinate system option. When set to ON or AUTO will allow rigid or interpolation elements or MPC equations which are linked to be in separate coordinate systems through internally generated collocated spring elements whose stiffness is specified by KRIGIDELEM. The OFF setting will select the dominant coordinate system of all connected elements as the common element coordinate system.	ON/OFF AUTO	AUTO
RIGIDELEMTYPE	Rigid element conversion element type: BAR – Selects a bar element form to replace RBE2 elements for large displacement nonlinear analysis and thermal expansion effects when a coefficient of thermal expansion is specified on the RBE2 Bulk Data entry. The bar element stiffness is controlled by the KRIGIDELEM model parameter. If a dependent grid point is collocated with an independent grid point, the RBE form will be selected automatically. ELAS – Selects a bush element form to replace RBE2 elements with one dependent grid point specified. RBE – Selects the default rigid element which will result in the generation of equivalent multipoint constraint equations. See also KRIGIDELEM and RIGIDELEM2ELAS above.	BAR/ELAS/ RBE	RBE
WARNING	Option for disabling output of warning messages.	ON/OFF	ON

Geometry Processor Parameters:

Parameter	Description	Type	Default
AUTOFIXELEMSING	When set to ON will check for and remove singular elements in the Geometry Processor Module before matrix assembly	ON/OFF	OFF
CB1, CB2	Used to specify scale factors for the total damping matrix. The total damping matrix is given by $$[B_{GLB}] = \text{CB1} * [B_1] + \text{CB2} * [B_2]$$ where $[B_2]$ is selected via the Case Control command B2GG and $[B_1]$ comes from viscous and structural damping terms. These parameters are effective only if B2GG is selected in the Case Control Section.	Real	1.0
CHECKRUN	Model check run option. When set to ON the analysis will run up to and including the geometry processor module and then terminate providing a check run for translator and geometry processor diagnostics.	ON/OFF	OFF
CHECKOUT	See CHECKRUN above.	ON/OFF	OFF
CK1, CK2	Used to specify scale factors for the total stiffness matrix. The total stiffness matrix is given by $$[K_{GLB}] = \text{CK1} * [K_1] + \text{CK2} * [K_2]$$ where $[K_2]$ is selected via the Case Control command K2GG and $[K_1]$ is generated from structural element entries in the Bulk Data. These parameters are effective only if K2GG is selected in the Case Control.	Real	1.0
CM1, CM2	Used to specify scale factors for the total mass matrix. The total mass matrix is given by $$[M_{GLB}] = \text{CM1} * [M_1] + \text{CM2} * [M_2]$$ where $[M_2]$ is selected via the Case Control command M2GG and $[M_1]$ is generated from mass element entries in the Bulk Data. These parameters are effective only if M2GG is selected in the Case Control.	Real	1.0
CONVMATRIX	Convection matrix formulation option. When set to ON, requests the generation of convection boundary condition matrix off diagonal terms.	ON/OFF	OFF
COUPMASS	COUPMASS $>$ 0 or ON requests the generation of coupled rather than diagonal mass matrices for elements with coupled mass capability. This option applies to both structural and nonstructural mass for the following elements: CBEAM, CBAR, CROD, CQUAD4, CQUADR, CTRIA3, CTRIAR, CHEXA, CPENTA, CPYRA, and CTETRA. A negative value or OFF causes the generation of diagonal mass matrices for all of the above elements. The diagonal mass matrix is formed by scaling the diagonal terms of the coupled mass matrix for the correct element mass and setting the off-diagonal terms to zero. Note that the diagonal mass matrix formulation includes rotary inertia terms. The AUTO setting (default) will use the coupled mass formulation when rigid elements or multipoint constraints are specified in the model.	Integer ON/OFF AUTO	AUTO

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
CP1, CP2	Used to specify scale factors for the total load vector. The load vectors are generated from the equation $$\{P_{GLB}\} = \text{CP1} * \{P_1\} + \text{CP2} * \{P_2\}$$ where $\{P_2\}$ is selected via the Case Control command P2G and $\{P_1\}$ comes from Bulk Data static load entries.	Real	1.0
DMIPDIAG	When set to ON, will add DMIGP diagonal terms at the DMIGG assembly point.	ON/OFF	ON
ELEMGEOMCHECKS	Element geometry check option. When set to ON, shell and solid element Jacobian determinant, aspect ratio, skew angle, taper ratio, and warping angle will be calculated. When set to OFF, element geometry checks will be skipped and no warning messages will be output for highly distorted elements.	ON/OFF	ON
ELEMGEOMFATAL	Option to handle certain geometry warnings as fatal errors. When set to ON will terminate execution if an element geometry related warning occurs (warnings: T2217-T2221 and G3007-G3017).	ON/OFF	OFF
ELEMGEOMOUT	Option to output individual element geometry statistics. When ELEMGEOMOUT is set to ON, the following statistics are output to the Model Results Output File for each element: • Aspect ratio • Taper ratio • Skew angle • Warping angle • Normalized Jacobian The data is sorted based on normalized Jacobian determinant, skew angle, and aspect ratio in ascending order for each element type. If ELEMGEOMOUT is set to ASPECTRATIO, then the sort will be in descending order and only based on element aspect ratio. If ELEMGEOMOUT is set to SKEWANGLE, then the sort will be in descending order and only based on element skew angle. If ELEMGEOMOUT is set to JACOBIAN1, then the sort will be in ascending order and only based on the total Jacobian determinant normalized using element volume. If ELEMGEOMOUT is set to JACOBIAN2, then the sort will be in ascending order and only based on the minimum Jacobian determinant at each corner node normalized using adjacent element edge lengths.	ON/OFF ASPECTRATIO/ SKEWANGLE/ JACOBIAN1/ JACOBIAN2	OFF
GPWEIGHT	See GRDPNT below.		

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
GRDPNT	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. The following weight and balance information is output to the Model Results Output File following the execution of the grid point weight generator: • Total mass • Location of center of gravity • Mass moment of inertia • 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 • 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	Integer	-1
GRIDCOLTOL	Grid collocation tolerance. A warning message will be given if the distance between any two grid points on an element is less than or equal to the specified value.	Real	0.0
HEXARTOL	Hex element aspect ratio tolerance. A warning message will be given if a hex element has an aspect ratio greater than or equal to the specified value.	Real	100.0
HEXENODE	Hex element edge node option. Setting HEXENODE and HEXINODE to ON will sometimes give better results when hex elements are used as thin plates with highly distorted initial geometry.	ON/OFF	OFF
HEXFACEMAXIATOL	Hex element face maximum interior angle tolerance. A warning message will be given if a hex element has a face interior angle greater than or equal to the specified value.	Real	165.0
HEXFACEMINIATOL	Hex element face minimum interior angle tolerance. A warning message will be given if a hex element has a face interior angle less than or equal to the specified value.	Real	25.0
HEXFACESKEWTOL	Hex element face skew angle tolerance. A warning message will be given if a hex element has a face skew angle greater than or equal to the specified value.	Real	65.0
HEXFACETAPERTOL	Hex element face taper ratio tolerance. A warning message will be given if a hex element has a face taper ratio greater than or equal to the specified value.	Real	0.75

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
HEXFACEWARPTOL	Hex element face warping angle tolerance. A warning message will be given if a hex element has a face warping angle greater than or equal to the specified value.	Real	45.0
HEXINODE	Hex element internal node option. When set to ON, hex elements will produce more accurate results with a small performance degradation. The AUTO setting (default) will use the ON setting for stiffness matrix and stress calculations for models less than DECOMPAUTOSIZE or nonlinear solutions. For models greater than DECOMPAUTOSIZE and AUTO, only the stiffness matrix assembly phase will use the ON setting. The AUTO setting is recommended and provides optimal performance with accuracy.	ON/OFF AUTO	AUTO
HEXMAXEPADTOL	Hex element maximum edge-point angular deviation tolerance. A warning message will be given if a hex element has an edge-point angular deviation greater than or equal to the specified value.	Real	30.0
HEXMINEPLRTOL	Hex element minimum edge-point length ratio tolerance. A warning message will be given if a hex element has an edge-point length ratio less than or equal to the specified value.	Real	0.5
HEXREDORD	Hex element reduced order integration option. When set to ON, hex elements will produce more accurate results by minimizing shear and Poisson's ratio locking. When set to OFF, the element may be too stiff and under predict results.	ON/OFF	ON
J4ROT	Specifies the stiffness to be added to the torsional degree of freedom of bar and beam elements when a torsional constant is not supplied. The AUTO setting determines a value sufficient to suppress singularities due to incomplete element stiffness.	Real AUTO	AUTO
K6ROT	Specifies the stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements. This is an alternate method to suppress the grid point singularities. The default AUTO setting will use a value of 100.0 except for modal solutions where a value of 1.0E+4 is used. The K6ROT setting may affect convergence in nonlinear and eigenvalue solutions if values other that AUTO are specified. This parameter is ignored for CQUADR and CTRIAR elements.	Real AUTO	AUTO
MAXELEMGEOMMSG	Limits the number of warning/fatal error messages output for element geometry checks. The default AUTO setting will use either a value of 10,000 or the number of lines in the Model Input File, whichever is larger.	Integer ≥ 0 AUTO	AUTO
MODLSTAB	When set to ON will force a linear static solution with one subcase and orthogonal acceleration loading, will add model stabilization to all DOF, and set the element STATUS results measure to 1 if a large relative displacement is detected. This model parameter is useful in locating disconnected parts.	ON/OFF	OFF
M6ROT	Specifies the inertia to be added to the normal rotation for CQUAD4 and CTRIA3 elements. The default AUTO setting will use a value of 1.0E-10 if K6ROT is also set to AUTO. This parameter is ignored for CQUADR and CTRIAR elements. See K6ROT above.	Real AUTO	0.0
NBEAMINTNODE	The number of beam internal nodes used when tapered material properties are specified. A higher value will produce more accurate results for tapered sections, but may result in slower performance and increased disk space requirements.	$1 \leq$ Integer ≤ 8	2

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
NSLDPLYINTPOINT	The number of layered solid element ply integration points in the 3-direction (thickness direction) of the ply. A higher value will produce more accurate results, but may result in slightly slower performance.	1, 3, or 5	3
PARTGEOMOUT	Individual part geometry statistics output option. When set to ON, additional part statistical information will be output including: • Material • Property type • Bounding box dimensions • Mass • Volume • Number of grid points • Number of elements	ON/OFF	OFF
PARTMASSOUT	Individual part mass properties output option. When set to ON, additional part mass properties information will be output including: • Material • Property type • Mass • Location of center of gravity • Mass moment of inertia	ON/OFF	OFF
PENTARTOL	Pent element aspect ratio tolerance. A warning message will be given if a pent element has an aspect ratio greater than or equal to the specified value.	Real	100.0
PENTFACEMAXIATOL	Pent element face maximum interior angle tolerance. A warning message will be given if a pent element has a face interior angle greater than or equal to the specified value.	Real	165.0
PENTFACEMINIATOL	Pent element face minimum interior angle tolerance. A warning message will be given if a pent element has a face interior angle less than or equal to the specified value.	Real	25.0
PENTFACESKEWTOL	Pent element face skew angle tolerance. A warning message will be given if a pent element has a face skew angle greater than or equal to the specified value.	Real	65.0
PENTFACETAPERTOL	Pent element face taper ratio tolerance. A warning message will be given if a pent element has a face taper ratio greater than or equal to the specified value.	Real	0.75
PENTFACEWARPTOL	Pent element face warping angle tolerance. A warning message will be given if a pent element has a quadrilateral face warping angle greater than or equal to the specified value.	Real	45.0
PENTMAXEPADTOL	Pent element maximum edge-point angular deviation tolerance. A warning message will be given if a pent element has an edge-point angular deviation greater than or equal to the specified value.	Real	30.0
PENTMINEPLRTOL	Pent element minimum edge-point length ratio tolerance. A warning message will be given if a pent element has an edge-point length ratio less than or equal to the specified value.	Real	0.5
PENTREDORD	Pent element reduced order integration option. When set to ON, pent elements will produce more accurate results by minimizing shear and Poisson's ratio locking. When set to OFF, the element may be too stiff and under predict results.	ON/OFF	ON

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
PYRARTOL	Pyr element aspect ratio tolerance. A warning message will be given if a pyr element has an aspect ratio greater than or equal to the specified value.	Real	100.0
PYRFACEMAXIATOL	Pyr element face maximum interior angle tolerance. A warning message will be given if a pyr element has a face interior angle greater than or equal to the specified value.	Real	170.0
PYRFACEMINIATOL	Pyr element face minimum interior angle tolerance. A warning message will be given if a pyr element has a face interior angle less than or equal to the specified value.	Real	5.0
PYRFACESKEWTOL	Pyr element face skew angle tolerance. A warning message will be given if a pyr element has a face skew angle greater than or equal to the specified value.	Real	80.0
PYRFACETAPERTOL	Pyr element face taper ratio tolerance. A warning message will be given if a pyr element has a face taper ratio greater than or equal to the specified value.	Real	0.75
PYRFACEWARPTOL	Pyr element face warping angle tolerance. A warning message will be given if a pyr element has a quadrilateral face warping angle greater than or equal to the specified value.	Real	45.0
PYRMAXEPADTOL	Pyr element maximum edge-point angular deviation tolerance. A warning message will be given if a pyr element has an edge-point angular deviation greater than or equal to the specified value.	Real	30.0
PYRMINEPLRTOL	Pyr element minimum edge-point length ratio tolerance. A warning message will be given if a pyr element has an edge-point length ratio less than or equal to the specified value.	Real	0.5
PYRREDORD	Pyr element reduced order integration option. When set to ON, pyr elements will produce more accurate results by minimizing shear and Poisson’s ratio locking. When set to OFF, the element will be too stiff and under predict results.	ON/OFF	ON
QUADARTOL	Quad element aspect ratio tolerance. A warning message will be given if a quad element has an aspect ratio greater than or equal to the specified value.	Real	100.0
QUADBNDREDORD	Quad element bending reduced order integration option. When set to ON, quad elements will produce more accurate results by minimizing transverse shear locking. When set to OFF, the element may be too stiff in bending and under predict results.	ON/OFF	ON

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
QUADELEMTYPE	Quad element bending formulation option. SRI – Selective Reduced-Order Integration. DKQ – Discrete Kirchhoff-Mindlin Quadrilateral. DKT – Discrete Kirchhoff-Mindlin Triangle (either two overlapping or four dissecting DKT elements depending on the setting for QUADINODE). The DKT and DKQ elements may be slightly more accurate than the SRI in very coarse meshes; however, the SRI element performs better in nonlinear and buckling solutions. All three element types handle finite transverse shear stiffness. The SRI and DKQ element types are supported in all solutions. The DKT element type is supported in linear solutions only. If QUADINODE is set to ON and the DKT element type is selected, the bending element will be comprised of four DKT subelements and a center node. If QUADINODE is set to OFF and the DKT element type is selected, the bending element will be comprised of two overlapping DKT sub elements.	SRI/DKQ/ DKT	SRI
QUADINODE	Quad element internal node option. When set to ON, quad elements will produce more accurate results with a small performance degradation. The AUTO setting (default) will use the ON setting for stiffness matrix and stress calculations for models less than DECOMPAUTOSIZE, models with composite shell elements, or nonlinear solutions. For models greater than DECOMPAUTOSIZE and AUTO, only the stiffness matrix assembly phase will use the ON setting. The AUTO setting provides optimal performance with accuracy.	ON/OFF AUTO	AUTO
QUADMAXEPADTOL	Quad element maximum edge-point angular deviation tolerance. A warning message will be given if a quad element has an edge-point angular deviation greater than or equal to the specified value.	Real	30.0
QUADMAXIATOL	Quad element maximum interior angle tolerance. A warning message will be given if a quad element has an interior angle greater than or equal to the specified value.	Real	165.0
QUADMEMREDORD	Quad element membrane reduced order integration option. When set to ON, quad elements will produce more accurate results by minimizing shear and Poisson's ratio locking. When set to OFF, the element may be too stiff in extension and under predict results.	ON/OFF	ON
QUADMINEPLRTOL	Quad element minimum edge-point length ratio tolerance. A warning message will be given if a quad element has an edge-point length ratio less than or equal to the specified value.	Real	0.5

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
QUADMINIATOL	Quad element minimum interior angle tolerance. A warning message will be given if a quad element has an interior angle less than or equal to the specified value.	Real	25.0
QUADREDORD	Quad element membrane and bending reduced order integration option. When set to ON, quad elements will produce more accurate results by minimizing shear and Poisson's ratio locking. When set to OFF, the element may be too stiff in extension and under predict results.	ON/OFF	ON
QUADRNODE	Quad element drill degree of freedom option. When set to ON, CQUAD4 entries will be converted to CQUADR entries.	ON/OFF	OFF
QUADSKEWTOL	Quad element skew angle tolerance. A warning message will be given if a quad element has a skew angle greater than or equal to the specified value.	Real	65.0
QUADTAPERTOL	Quad element taper ratio tolerance. A warning message will be given if a quad element has a taper ratio greater than or equal to the specified value.	Real	0.75
QUADWARPLIMIT	Quad element warping correction option. The value specified is the maximum element warping angle allowed using the standard quad element formulation. Quad elements with warping angles greater than this value will use the alternate formulation which has no limit for warping but is less accurate for coarse mesh densities.	Real	45.0
QUADWARPTOL	Quad element warping angle tolerance. A warning message will be given if a quad element has a warping angle greater than or equal to the specified value.	Real	45.0
RADMATRIX	Radiation matrix formulation option. When set to ON, requests the generation of radiation boundary condition matrix off diagonal terms.	ON/OFF	ON
RBCHECKLEVEL	Stiffness matrix equilibrium checks option. Equilibrium checks verify whether an unrestrained model can undergo simple rigid body motion without generating internal forces. There are six options: 0 – Do not perform any checks. 1 – Perform checks after stiffness matrix assembly before multipoint constraints are applied. 2 – Perform checks after multipoint constraints are applied before single point constraints are applied. 3 – Perform checks after single point constraints are applied before static condensation. 4 – Perform checks after static condensation before decomposition. 5 – Perform checks 1 – 4 above.	$0 \leq$ Integer ≤ 5	0
RBCHECKMODES	Specifies the number of modes to solve for in an automated modal rigid body check. When set to a value greater than zero will perform an eigenvalue extraction analysis requesting that number of specified modes on the unconstrained model. Displacements and strain energy are output. Multipoint constraints requested in the first subcase of the model will be included.	Integer ≥ 0	0

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
RESEQGRID	Grid point resequence option. When set to ON, the model grid point identification numbers will be resequenced internally to minimized model size and optimize performance. This action is completely transparent to the user so model results will still reference the original grid point identification numbers. If the resequenced model is bigger than the original model, the original is retained.	ON/OFF	ON
RESEQSTARTGRID	Grid point resequence start grid point identification number. The default is the model grid point with the lowest connectivity. This will usually result in the smallest resequenced model size. Selecting a different grid point in some cases may produce a smaller model size.	Integer $>$ 0	Lowest Connectivity Grid Point
ROTINERTIA	Diagonal element mass matrix rotary inertia option. When set to ON, rotary inertia terms (if significant) are added to the element mass matrix. The AUTO setting will use the OFF setting when the EXTRACTMETHOD directive is set to LANCZOS or set to AUTO and the LANCZOS eigensolver is selected.	ON/OFF AUTO	AUTO
SHEARELEMTYPE	Shear element formulation option. NASTRAN – Standard NASTRAN Garvey shear panel element. NORAN – V8.1 and below shear panel element. AUTO – Selects NASTRAN if the material is isotropic and NORAN if it is orthotropic or anisotropic.	NASTRAN/ NORAN AUTO	AUTO
SHELLRNODE	Shell element drill degree of freedom option. When set to ON, CQUAD4 and CTRIA3 entries will be converted to CQUADR and CTRIAR entries, respectively.	ON/OFF	OFF
SHELLTVSMATTYPE	Orthotropic shell element transverse shear stiffness type. Specifies the default type of transverse shear on MAT8 Bulk Data entries when the G1Z and G2Z fields are blank or zero. When set to RIGID, a rigid approach is used where the G1Z and G2Z are penalty values which provide a nearly rigid transverse shear stiffness. When set to FLEXIBLE, the G12 value is used. If a non-zero value is supplied for either G1Z or G2Z it will be used.	RIGID/ FLEXIBLE	FLEXIBLE
TEMPDEPCOMP	Option to enable temperature-dependent composite materials. When set to ON, ply material temperature dependence will be enabled for stiffness matrix and load vector assembly and element results calculations based on individual element ply temperature. Properties will be updated as temperatures change in nonlinear solutions. The OFF setting will use the reference temperature defined on the PCOMP entry.	ON/OFF	ON
TETARTOL	Tet element aspect ratio tolerance. A warning message will be given if a tet element has an aspect ratio greater than or equal to the specified value.	Real	100.0
TETFACEMAXIATOL	Tet element face maximum interior angle tolerance. A warning message will be given if a tet element has a face interior angle greater than or equal to the specified value.	Real	170.0
TETFACEMINIATOL	Tet element face minimum interior angle tolerance. A warning message will be given if a tet element has a face interior angle less than or equal to the specified value.	Real	5.0

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
TETFACESKEWTOL	Tet element face skew angle tolerance. A warning message will be given if a tet element has a face skew angle greater than or equal to the specified value.	Real	80.0
TETINODE	Tet element internal node option. When set to ON, parabolic tet elements will produce slightly more accurate results with a small performance degradation. The AUTO setting (default) will use the ON setting for stiffness matrix and stress calculations for models less than DECOMPAUTOSIZE or nonlinear solutions. For models greater than DECOMPAUTOSIZE and AUTO, only the stiffness matrix assembly phase will use the ON setting.	ON/OFF AUTO	OFF
TETMAXEPADTOL	Tet element maximum edge-point angular deviation tolerance. A warning message will be given if a tet element has an edge-point angular deviation greater than or equal to the specified value.	Real	30.0
TETMINEPLRTOL	Tet element minimum edge-point length ratio tolerance. A warning message will be given if a tet element has an edge-point length ratio less than or equal to the specified value.	Real	0.5
TETREDORD	Tet element reduced order integration option. When set to ON, tet elements will produce more accurate results by minimizing shear and Poisson's ratio locking. When set to OFF, the element will be too stiff and under predict results.	ON/OFF	ON
TRIARTOL	Tri element aspect ratio tolerance. A warning message will be given if a tri element has an aspect ratio greater than or equal to the specified value.	Real	100.0
TRIBNDREDORD	Tri element bending reduced order integration option. When set to ON, tri elements will produce more accurate results by minimizing transverse shear locking. When set to OFF, the element may be too stiff in bending and under predict results.	ON/OFF	ON
TRIELEMTYPE	Tri element bending formulation option. DKT – Discrete Kirchhoff-Mindlin Triangle. SRI – Selective Reduced-Order Integration. The DKT element is typically more accurate than the SRI in coarse meshes and like the SRI element, works well for both thick and thin plates. Both element types handle finite transverse shear stiffness and are supported in all solutions.	DKT/SRI	DKT
TRIINODE	Tri element internal node option. When set to ON, tri elements will produce more accurate results with a small performance degradation. The AUTO setting (default) will use the ON setting for stiffness matrix and stress calculations for models less than DECOMPAUTOSIZE, models with composite shell elements, or nonlinear solutions. For models greater than DECOMPAUTOSIZE and AUTO, only the stiffness matrix assembly phase will use the ON setting. The AUTO setting provides optimal performance with accuracy.	ON/OFF AUTO	AUTO
TRIMAXEPADTOL	Tri element maximum edge-point angular deviation tolerance. A warning message will be given if a tri element has an edge-point angular deviation greater than or equal to the specified value.	Real	30.0
TRIMAXIATOL	Tri element maximum interior angle tolerance. A warning message will be given if a tri element has an interior angle greater than or equal to the specified value.	Real	170.0

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
TRIMEMREDORD	Tri element membrane reduced order integration option. When set to ON, tri elements will produce more accurate results by minimizing shear and Poisson's ratio locking. When set to OFF, the element may be too stiff in extension and under predict results.	ON/OFF	ON
TRIMINEPLRTOL	Tri element minimum edge-point length ratio tolerance. A warning message will be given if a tri element has an edge-point length ratio less than or equal to the specified value.	Real	0.5
TRIMINIATOL	Tri element minimum interior angle tolerance. A warning message will be given if a tri element has an interior angle less than or equal to the specified value.	Real	10.0
TRIREDORD	Tri element membrane and bending reduced order integration option. When set to ON, tri elements will produce more accurate results by minimizing shear and Poisson's ratio locking. When set to OFF, the element may be too stiff in extension and under predict results.	ON/OFF	ON
TRIRNODE	Tri element drill degree of freedom option. When set to ON, CTRIA3 entries will be converted to CTRIAR entries.	ON/OFF	ON
TRISKEWTOL	Tri element skew angle tolerance. A warning message will be given if a tri element has a skew angle greater than or equal to the specified value.	Real	65.0
UNRESEQGRID	Unresequence model database option. When set to ON, the model database grid point identification numbers will be reset to original input values. This option is used primarily to generate a resequenced bulk data file by translating a resequenced database. See RESEQGRID.	ON/OFF	ON
WTMASS	Global mass matrix scaling factor. The terms of the global mass matrix are multiplied by the value of WTMASS when they are generated. This parameter is used when material density is input in weight instead of mass units. It does not affect loads generated by GRAV or RFORCE Bulk Data entries or mass properties calculated by the Grid Point Weight Generator. The value of WTMASS is calculated using the relation: $\rho_m = \left(\frac{1}{g}\right)\rho_w$ where ρ_m is mass or mass density, g is acceleration of gravity, ρ_w is weight or weight density	Real	1.0
VFMADDMETHOD	Specifies when in the solution sequence virtual fluid mass is added to the global mass matrix. There are two options: after mass matrix ASSEMBLY and after mass matrix REDUCTION.	ASSEMBLY/ REDUCTION	ASSEMBLY

Geometry Processor Parameters (Continued):

Parameter	Description	Type	Default
VFMINTERACTTOL	Tolerance for removing negligible off-diagonal fluid interaction terms from the assembled fluid mass matrix. A larger VFMINTERACTTOL value will result in a more sparse virtual fluid mass matrix (using less memory) but with a corresponding reduction in accuracy. Enclosed fluid volumes will have a dense virtual fluid mass matrix due to fluid interaction between adjacent and distant wet surfaces. Distant surfaces relative to a single point will have a negligible contribution but can still result in a dense virtual fluid mass matrix requiring large amounts of memory. A larger VFMINTERACTTOL value may be useful for reducing memory requirements and increasing performance for these of models.	Real	1.0E-10
VFMNORMTOL	Angular tolerance for excluding adjacent grid point surfaces in the fluid mass matrix. An average element surface normal is calculated for all wet surface elements connected at a grid point. If the angular difference between the average element surface normal and an adjacent individual element normal is greater than VFMNORNTOL, its fluid mass is excluded.	Real	30.0
VMOPT	See VFMADDMETHOD above.		
VOXELMESH	When ON, if MCID of first encountered hex element uses the basic coordinate system (MCID=0) and does not have temperature dependent material properties, then the element stiffness will be applied for every other hex element in the model.	ON/OFF	OFF
ZERONPDELEMMASS	Zero non-positive definite element mass matrix option. When set to ON, an eigensolution is performed for each point mass element (CONMi) mass matrix. If a negative principal mass or inertia is detected, the mass matrix for that element is set to zero.	ON/OFF	OFF

Solution Processor Parameters:

Parameter	Description	Type	Default
ADAPTLNCONTACT	Linear contact adaptive stiffness update method. When set to ON, each contact segment will adjust stiffness on each iteration to maintain a fixed penetration of 1 percent of the contact segment reference length dimension. When set to OFF, stiffness is not adjusted individually. The AUTO setting will use ON for contact segments with initial gap openings that are 10 percent of the contact segment reference length dimension.	ON/OFF AUTO	AUTO
AUTOFIXMODLSING	When set to ON will automatically constrain singular degrees of freedom when detected during linear static PSS factorization	ON/OFF	OFF
AUTOSPC	Automatic single point constraint option. AUTOSPC specifies the action to take when singularities exist in the stiffness matrix ($[K_{ff}]$). Setting AUTOSPC to ON means that singularities will be constrained automatically. Setting AUTOSPC to OFF means that singularities will not be constrained. If AUTOSPC is ON, identified singularities with a ratio smaller than STIFFRATIOTOL (default = 1.0E-8) will be automatically constrained with single-point constraints. See STIFFRATIOTOL and PRGPST.	ON/OFF	ON
BAREQVLOAD	Bar and beam element equivalent load vector formulation option. When set to ON, the bar and beam element load vector will be calculated using a work equivalent approach. When set to OFF, the bar and beam element load vector will include forces only.	ON/OFF	ON
DELTASTRAINEGOUT	Delta strain energy output option. When set to ON, the residual strain energy vector is output. The residual strain energy vector is calculated using: $$\delta E = (Ku - P)\,u$$ where u is the global displacement vector P is the global load vector K is the global stiffness matrix The solution error measure, epsilon, is calculated using: $$\varepsilon = \frac{\sum_{i=1}^{NDOF} \delta E}{u^T P}$$	ON/OFF	OFF
EPSILONFLOAT	Floating point precision constant for stiffness matrix factorization.	Real	1.0E-15
EPZERO	See STIFFRATIOTOL.	Real	1.0E-8
FACTDIAG	See SOLUTIONERROR.	Real	1.0E-10
FACTRATIOTOL	Stiffness matrix factor diagonal tolerance. 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 FACTRATIOTOL, 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 FACTRATIOTOL and less than zero and their associated external grid point identities will be output. If the matrix is non-positive definite or a singularity is detected, the program will then take appropriate action as directed by the model parameter SOLUTIONERROR.	Real	1.0E+5

Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
GRIDTEMPASGN	Option to assign element temperatures to adjacent grid points. GRIDTEMPASGN set to ON will assign element temperatures defined on TEMPP1 and TEMPRB entries to the associated element grid points. Surface and line elements that reference TEMPP1 and TEMPRB entries, respectively, will use temperatures defined on the entry. Adjacent elements with no element temperature definition will use grid point temperatures from element temperatures when PARAM, GRIDTEMPASGN is set to ON and from TEMP and TEMPD entries when it is set to OFF.	ON/OFF	OFF
GRIDTEMPAVE	Element grid point temperature averaging option. When set to ON, element grid point temperatures are averaged to determine the extensional contribution to the element thermal equivalent load vector.	ON/OFF	OFF
INERTIALRELIEF	Controls the calculation of inertial relief or enforced acceleration in STATIC solutions. INERTIALRELIEF set to ON or –1 requests that inertial relief be performed using the fixed point method. A SUPORT entry is required to be defined for a single grid point. The model must be fully constrained against rigid body motion about that point. Loads due to unit body accelerations at the point referenced by PARAM, GPWEIGHT or PARAM, GRDPNT are calculated and then appended to the global load vector. If a SUPORT is not specified, one will be generated automatically for all six degrees of freedom at the grid point specified by PARAM, GPWEIGHT or PARAM, GRDPNT. The AUTO setting requests that inertial relief be performed using Automated Inertial Relief Analysis (AIRA). AIRA does not require any model constraints or SUPORT entry or PARAM, GRDPNT settings. The model center of mass is automatically located and selected as the frame of reference. The model is stabilized using internally generated bush elements with a stiffness that is based on model characteristics.	Integer ON/OFF AUTO	0 OFF
INREL	See INERTIALRELIEF above.	Integer ON/OFF	0 OFF
LINEARCONTACT	Option to control surface contact in linear static solutions. When set to ON, an iterative contact procedure is performed by checking the status of contact surfaces and adjusting the contact stiffness. Iteration convergence is defined by LNCONTACTITERTOL with a maximum number of iterations permitted defined by MAXLNCONTACTITER. Convergence is typically achieved in two to three iterations. When set to OFF or in other linear solutions, surface contact will default to welded behavior.	ON/OFF	ON
LNCONTACTITERTOL	Linear contact analysis iteration convergence tolerance. See LINEARCONTACT.	Real	1.0E-2

Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
MAXLNCONTACTITER	Linear contact analysis maximum number of convergence iterations permitted. The linear contact procedure will iterate until the convergence factor set by LNCONTACTITERTOL is reached or MAXLNCONTACTITER iterations have been performed. A zero setting will result in iteration until convergence is reached. See LINEARCONTACT.	Integer ≥ 0	30
MAXRATIO	See FACTRATIOTOL.	Real	1.0E5
MAXSPARSEITER	Iterative solver maximum number of iterations permitted. The iterative solver will iterate until MINSPARSEITER iterations have been performed regardless of convergence and then continue until the convergence factor set by SPARSEITERTOL is reached or MAXSPARSEITER iterations have been performed. The AUTO setting will set MAXSPARSEITER to the number of degrees of freedom of the model. See the Model Initialization directive, DECOMPMETHOD in Section 2, *Initialization,* for more information.	Integer > 0 AUTO	AUTO
MINSPARSEITER	Iterative solver minimum number of iterations required. The iterative solver will iterate, regardless of convergence, until the minimum MINSPARSEITER iterations have been performed.	Integer > 0	50
NITERLCUPDATE	Applicable to topology optimization stress constraints. If element stress is less than the the stress constraint limit scaled by the topology optimization activation threshold, then element adjacency calculations are skipped. AUTO adjusts the threshold value dynamically throughout the model.	1 ≤ Integer≤ 10	10
PRGPST	Controls the printout of singularities. When set to ON, all degrees of freedom automatically constrained (PARAM, AUTOSPC, ON) will be written out to the Grid Point Singularity Table in the Model Results Output File. When set to OFF, only non-zero degrees of freedom are listed. See AUTOSPC.	ON/OFF	ON
RESEQGRIDMETHOD	Matrix profile minimization method. Solution time is proportional to matrix profile. The VSS and PSS solvers minimize profile by reordering matrix rows and columns. For the VSS solver 10 matrix profile minimization methods are available: VRM1-VRM10. Each method can be selected individually (other methods not used) or the three best methods (VRM1, VRM7, and VRM10) considered with the best reordering method used automatically (AUTO). For the PSS solver two matrix profile minimization methods are available: VRM1 and VRM7. Each method can be selected individually or the best reordering method used automatically (AUTO).	VRM1-VRM10/ AUTO	AUTO
QUADEQVLOAD	Quad element equivalent load vector formulation option. When set to ON, the quad element load vector will be calculated using a work equivalent approach. When set to OFF, the quad element load vector will include forces only.	ON/OFF	OFF
SHELLEQVLOAD	Shell element equivalent load vector formulation option. When set to ON, the quad and tri element load vectors will be calculated using a work equivalent approach. When set to OFF, the element load vector will include forces only.	ON/OFF	OFF

Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
SIGMA	Stefan-Boltzmann constant. The radiant heat flux is proportional to SIGMA $*$ (T + TABS)^4, where SIGMA is the Stefan-Boltzmann constant, T is the temperature at a grid point and TABS is the scale factor for absolute temperature 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 is 5.67E-8 W/m^2-°K^4 or 3.97E-14 BTU/sec.-in.^{2}-°R^4. The default value causes radiant heat effects to be discarded.	Real	0.0
SPARSEITERMETHOD	Iterative solver preconditioner method: ITERATIVE – Selects the iterative solver. This method uses less memory and may be faster for solid models. If a modal solution is being performed and field 6 on the EIGRL entry is blank, the iterative solver will be used during Lanczos extraction. DIRECT – Selects the direct sparse solver. This method may be faster if the model contains large numbers of RBEi elements or MPC equations and/or has elements that are irregularly shaped. If a modal solution is being performed and field 6 on the EIGRL entry is blank, the direct solver will be used during Lanczos extraction. PRIMAL – Selects the primal solver. This solver is similar to the ITERATIVE solver but may require less iterations for models that contain elements with high initial distortion. AUTO – Selects the fastest method based on available memory and element type. This parameter is only applicable to the PCGLSS iterative solver.	ITERATIVE/ DIRECT/ PRIMAL/ AUTO	AUTO
SPARSEITERMODE	Iterative solver implicit matrix-vector multiply option for reducing memory requirements for models with parabolic tet elements. There are three options: 0 – Implicit matrix-vector multiply is disabled. The full tet element stiffness matrix is used by the solver and additional memory is required. 1 – Implicit matrix-vector multiply is enabled. A reduced tet element stiffness matrix is generated and used by the solver reducing memory usage and increasing performance. 2 – Same as option 1 but requires less memory with a possible degradation in performance. 3 – Same as option 2 but uses the least amount of memory by skipping the assembly of the global mass and stiffness matrixes. The following limitations exists with this setting: • The AUTOSPC function will use only diagonal stiffness and is therefore less robust (see AUTOSPC in this section). • Forces of multipoint constraint are not available. • The reported epsilon (solution error measure) is the value given by the PCGLSS solver and not the value determined independently (see DELTASTRAINEGOUT in this section). This parameter is only applicable to the PCGLSS iterative solver.	$0 \le$ Integer ≤ 3 AUTO	AUTO

Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
SPARSEITERTOL	Iterative solver convergence factor. The iterative solver will iterate until the convergence factor set by SPARSEITERTOL is reached or MAXSPARSEITER iterations have been performed and at least MINSPARSEITER iterations have been performed. The AUTO setting uses a convergence factor of 1.0E-09 when Automated Inertial Relief (AIR) is selected or spring elements with high stiffness values are specified and a convergence factor of 1.0E-06 otherwise. See the Model Initialization directive, DECOMPMETHOD in Section 2, *Initialization*, for more information.	$0.0 < \text{Real} \leq 1.0$ AUTO	AUTO
SPARSEMETHOD	Specifies the VSS sparse direct solver matrix reordering method: HEAT – Used for one degree of freedom per node models such as in heat transfer solutions. SHELL – Used for six degree of freedom per node models such as in structural models with shell and line element types. SOLID – Used for three degree of freedom per node models such as in structural models with only solid elements. SOLVER – Directs the solver to determine the best reordering method based on the input stiffness matrix. AUTO – The program picks the best method based on the element types and solution selected in the model. Additional reordering options can be selected using the RESEQGRIDMETHOD directive.	HEAT/ SHELL/ SOLID/ SOLVER/ AUTO	AUTO
SPARSEOUTOFCORE	Parallel sparse direct solver out-of-core option. When set to ON, the PSS solver will operate in out-of-core mode which will handle larger models but is slower due to I/O usage and single CPU operation. When set to OFF, the PSS solver will operate completely in memory, in parallel CPU mode. The AUTO setting initially attempts to run completely in memory and only reverts to out-of-core mode if an insufficient memory error occurs. See the Model Initialization directive, DECOMPMETHOD in Section 2, *Initialization,* for more information.	ON/OFF AUTO	AUTO
SOLUTIONERROR	When set to ON, it directs the program to substitute the value of FACTDIAG (default = 1.0E-10) for the factored diagonal term when a singularity or non-positive definite is detected. If FACTDIAG is set to zero, non-positive definites are ignored, while a singularity will result in program termination. SOLUTIONERROR and FACTDIAG are ignored in eigenvalue solutions and when the sparse iterative solvers (PCGLSS or VIS) are used. While this option is useful for modeling checkout, it may lead to solutions of poor quality or fatal messages later in the run. It is recommended that SOLUTIONERROR be set to OFF for production runs.	ON/OFF	OFF
SPCGEN	Grid point singularity translation option for Bulk Data Output File generation. When set to ON, identified singularities listed in the Grid Point Singularity Table (PARAM, AUTOSPC, ON) will be translated out as SPC1 Bulk Data entries. See the Model Initialization directive, TRSLSPCDATA in Section 2, *Initialization,* and AUTOSPC for more information.	ON/OFF	OFF
STIFFRATIOTOL	Specifies the minimum global stiffness matrix diagonal ratio for automatic singularity detection. Values below STIFFRATIOTOL are considered singular. See AUTOSPC.	Real	1.0E-8
STIFFZEROTOL	Specifies the minimum value for an off-diagonal term to be considered nonzero in the global stiffness or mass matrix. If the ratio of the off-diagonal term to the corresponding diagonal term is less than STIFFZEROTOL, the off-diagonal term will be considered zero and removed from the matrix.	Real	1.0E-15

Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
TABS	Scale factor for absolute temperature. TABS is used to convert units of temperature input (°F or °C) to the absolute temperature (°R or °K) when radiant heat effects are included. Specify PARAM, TABS, 273.16 when Celsius is used and PARAM, TABS, 459.69 when Fahrenheit is used. See SIGMA.	Real	0.0
TRIEQVLOAD	Tri-element equivalent load vector formulation option. When set to ON, the tri-element load vector will be calculated using a work equivalent approach. When set to OFF, the tri element load vector will include forces only.	ON/OFF	OFF
WTMASSMETHOD	Specifies how the WTMASS model parameter is used to determine gravity loads. The NORAN option (default) uses WTMASS to convert GRAV loads into units of gravity. The NASTRAN option does not use WTMASS and thus requires models in mass density units to specify gravity loads in weight units versus G's which is consistent with other versions of Nastran.	NASTRAN/ NORAN	NASTRAN

Eigenvalue Processor Parameters:

Parameter	Description	Type	Default
AUTOBPD	Automatic global mass matrix singularity and non-positive definite correction option. When set to ON, the global mass matrix is checked for zero or negative diagonal terms. A zero or negative diagonal term will result in the corresponding row and column being zeroed and the diagonal term replaced with BPDEFDIAG. If BPDEFDIAG is not specified (recommended), it will be calculated automatically.	ON/OFF	OFF
BPDEFDIAG	Mass diagonal coefficient to be used for correcting singular and non-positive definite matrixes. When AUTOBPD is set to ON, the global mass matrix is checked for zero or negative diagonal terms. A zero or negative diagonal term will result in the corresponding row and column being zeroed and the diagonal term replaced with BPDEFDIAG. If BPDEFDIAG is not specified (recommended), it will be calculated automatically.	Real	Model Dependent
CLOSE	See SCRSPEC.	Real	1.0
DDAMPHASE	DDAM multiphase analysis option. Divides a DDAM analysis sequence into four phases: 0 – Complete single phase analysis. 1 – Phase 1 DDAM operations consisting of an eigenvalue extraction analysis and a modal database store (*filename.MDB* is generated). 2 – Phase 2 DDAM operations consisting of a modal database fetch, the response/shock spectrum generation using the DDAMDAT Bulk Data entry input, and a DDAM database store (*filename.DDB* is generated). 3 – Phase 3 DDAM operations consisting of a DDAM database fetch and grid point and element results processing.	$0 \leq$ Integer ≤ 3	0
DMILABEL	Specifies the base label for exported matrix data (NAME field on the DMIG Bulk Data entry). The user specified label is concatenated with the matrix type where the exported boundary stiffness matrix label becomes Kcccccc, the mass Mcccccc, the damping Bcccccc, and the load Pcccccc and where cccccc is the user specified label (maximum 6 characters).	Character	Subcase or super element number
EIGENFLEXFREQ	Specifies the threshold frequency in cycles per unit time for defining the first flexible mode in a normal modes or modal response analysis. Eigenvalues with a frequency greater than this value will be considered as flexible modes.	Real	0.1
EIGENSHIFTSFACT	Specifies the shift scale multiplier used to increase the shift scale for an eigensolver restart. See MAXEIGENRESTART below.	Real	1.0E+4
EIGENSOLACCEL	Subspace eigensolver acceleration option. When set to OFF, no acceleration algorithms will be used and solution times may increase. This option is typically used when the eigensolver selects a shift scale that results in an unstable or inaccurate solution.	ON/OFF	ON

Eigenvalue Processor Parameters (Continued):

Parameter	Description	Type	Default
EXTOUT	Model and matrix data output: MODEL – Requests model data translation to the Bulk Data Output File. DMIGOUT – Requests global matrix output to the Model Results Output File. DMIGBDF – Requests global matrix export in DMIG format to the Bulk Data Output File. DMIGOP2 – Requests global matrix export to a NASTRAN Output 2 formatted results file. OFF – No output is requested. If matrix reduction is requested only the reduced matrix will be exported. For the global matrix output options mass, stiffness, and damping matrixes will be exported. To select specific matrixes to export use the EXTSEOUT Case Control command (see EXTSEOUT in Section 3, *Case Control,* for more information).	MODEL/ DMIGOUT/ DMIGBDF/ DMIGOP2/ OFF	OFF
LANCZOSVECT	Initial starting vector formulation to be used by the Subspace eigensolver. When set to ON, eigensolver starting iteration vectors will be formulated using the Lanczos method. This method may increase solution time, but can be useful when the standard formulation does not converge to an acceptable solution or is very slow to converge.	ON/OFF	OFF
MAXEIGENRESTART	Defines the permitted number of eigensolver restarts when an invalid shift scale is either externally defined or internally estimated. See also EIGENSHIFTSFACT.	Integer $>$ 0	5
MODALDATABASE	Controls the storage and retrieval of modal data such as eigenvalues and eigenvectors used in dynamic response analysis. The default value DELETE will purge all modal data when the program terminates normally. When set to STORE, the modal database is stored in a single file with the same base name as the Model Results Output File and a *.MDB* file extension. When set to FETCH, the database specified by the MODALDATFILE directive is retrieved and the eigenvalue extraction phase is skipped. When set to UPDATE, the modal database will be retrieved and stored.	DELETE/ FETCH/ STORE/ UPDATE	DELETE
MODEFSPCSTORE	Controls the storage and calculation of single point constraint forces in the modal database. When set to ON, single point constraint forces will be stored in the modal database file for modal restarts. When set to OFF and a modal database restart is performed, single point constraint forces will be calculated, if requested, using the first subcase SPCFORCES and SPC set requests.	ON/OFF	ON
MODEPFACTOR	Controls the calculation and output of modal participation factors and modal effective mass.	ON/OFF	ON
NCBMODE	Defines the number of component modes for superelement analysis. A Craig-Bampton reduction will be performed using NCBMODE modes.	Integer $>$ 0	1
OPTION	Defines the summation method used to combine modal results in response spectrum analysis. See SCRSPEC for more information.	ABS/SRSS/ NRL/CQC	ABS

Eigenvalue Processor Parameters (Continued):

Parameter	Description	Type	Default
RESVEC	Residual vector generation option. The default AUTO value will set RESVEC to ON for modal transient and frequency response solutions when direct enforced motion via the SPCD entry is specified. When set to ON, will enable generation of residual vectors based on applied, inertial relief, and RVDOFi loads. If no RVDOFi Bulk Data entries are defined, residual vectors will be based on applied and inertial loads only. The use of residual vectors improves the accuracy of modal dynamic response solutions by partially correcting mode truncation effects.	ON/OFF AUTO	AUTO
RESVPGF	Residual vector zero tolerance. RESVPGF is used to eliminate duplicate input load vectors and null residual vectors.	Real	1.0E-6
RIGIDBODYMODE	Subspace eigensolver option to specify how rigid body motion is detected and handled. The default AUTO value will automatically detect any rigid body motion and extract rigid body mode shapes. When set to FORCED, directions specified on the SUPORT entry corresponding to the first six modes will be replaced with exact zero eigenvalues and rigid eigenvectors. All unconstrained directions should be specified on the SUPORT entry when this option is used. When set to OFF, the structure is assumed properly constrained and free of any rigid body motion.	FORCED/ OFF AUTO	AUTO
SCRSPEC	Setting SCRSPEC to ON or 0 requests that structural response be calculated for response spectra input in a normal modes analysis. The responses are summed with the ABS, SRSS, NRL, or CQC convention, depending on the value of PARAM, OPTION. If the SRSS, NRL, or CQC options are used, close natural frequencies will be summed by the ABS convention, where close natural frequencies are defined as meeting the inequality. $f_{i+1} < CLOSE * f_i$	Integer ON/OFF	-1 OFF
SORTMODEMASS	Modal data sorting option. When set to ON, modes will be summed in order of increasing modal mass (DDAM solutions only).	ON/OFF	ON
ZONADATAOUT	Zona aeroelastic solver output option. When set to ON, addition data is calculated and output to the Model Results Output File which is required for subsequent analysis using Zona's ZAERO software.	ON/OFF	OFF

Transient Response Processor Parameters:

Parameter	Description	Type	Default
ADAPTTIMESTEP	Option for adaptive time stepping in linear direct transient response. When ADAPTTIMESTEP is set to ON, the default time step skip factor specified on the TSTEP Bulk Data entry is set to 5 enabling adaptive time stepping. When set to OFF, the default time step skip factor is set to 0 disabling adaptive time stepping. The additional parameters for adaptive time stepping are specified in fields 6 through 9 on the TSTEP entry. ADAPTTIMESTEP is overridden if a non-blank value is specified in field 6.	ON/OFF	OFF
ALPHA	Rayleigh damping stiffness matrix scale factor. See W3, W4.	Real	0.0
BETA	Rayleigh damping mass matrix scale factor. See W3, W4.	Real	0.0
DYNLMDIRECTDIF	Controls the type of differentiation used in the large mass enforced motion method when this option is requested on a TLOAD2 Bulk Data entry. When set to ON, enforced displacements and velocities requested on TLOAD2 entries will be computed using direct differentiation. When set to OFF, numerical differentiation will be used.	ON/OFF	OFF
DYNRESPEIGVOUT	Controls the output of normal modes results in modal response solutions.	ON/OFF	OFF
DYNSOLACCEL	Modal response solution acceleration option. When set to OFF, reduces memory requirements for modal transient and frequency response analyses by storing eigenvectors on disk. Disk storage is automatic if eigenvector memory cannot be allocated.	ON/OFF	ON
DYNSOLDIRECTINT	Controls the type of integration used in solving the dynamic differential equations of motion used in transient response analysis. When set to ON, the equations are integrated directly. When set to OFF, integration will be performed numerically using the Newmark-Beta method.	ON/OFF	ON
DYNSOLRELGRID	Specifies the reference point for enforced motion in linear transient and frequency response solutions when relative motion output is requested via the REL option on the DISPLACEMENT, VELOCITY, and ACCELERATION Case Control commands. The AUTO setting selects the direct enforced motion input point for direct enforced motion (SPCD) and the point with the largest mass for large mass enforced motion.	Integer > 0 AUTO	AUTO
G	Specifies the uniform structural damping coefficient in the formulation of global damping matrix in direct transient solutions. To obtain the value for the model parameter G, multiply the critical damping ratio, C/C_0, by 2.0. Note that PARAM, W3 must be greater than zero or PARAM, G will be ignored.	Real	0.0
HFREQ	The parameters LFREQ and HFREQ specify the frequency range in cycles per unit time (LFREQ is the lower limit and HFREQ is the upper limit) of the modes to be used in normal modes and dynamic response analysis. Note that the default for HREQ will usually include all modes computed. See also LMODES below.	Real	1.0E+30
LFREQ	See HFREQ.	Real	0.0
LMODES	Specifies the number of lowest modes to use in normal modes and dynamic response analysis. If LMODES is set equal to zero, the retained modes are determined by the model parameters LFREQ and HFREQ.	Integer > 0	0

Transient Response Processor Parameters (Continued):

Parameter	Description	Type	Default
MAXIMPACTSTEP	Specifies the maximum number of output steps in Automated Impact Analysis. If MAXIMPACTSTEP is set equal to zero, no limit is placed on the number of output steps.	Integer > 0	0
MODEVAROUT	Controls the output of modal variables in modal response solutions.	ON/OFF	OFF
NDAMP	Numerical damping option for direct transient solutions. Numerical damping may be specified to achieve numerical stability. A value of zero requests no numerical damping. The default AUTO setting selects the optimum value based on the solution specified. For nonlinear transient heat transfer solutions a value of 0.3 is used. For nonlinear transient response solutions a value of 0.01 is used. Larger values may improve solution stability and convergence especially when contact is present.	Real AUTO	AUTO
RSPECTRA	Setting RSPECTRA to ON or 0 requests that response spectra be generated in a transient response analysis.	Integer ON/OFF	-1 OFF
USAWETSURFACE	Underwater Shock Analysis (USA) interface option. A value greater than zero enables a special direct transient response solution sequence which generates input files to the USA program. Once the USA program run has completed Autodesk Nastran is restarted and will use USA output files to complete the analysis. USAWETSURFACE should be set to an existing load set id in the model consisting of pressure loads on the wet surface.	Integer	0
W3, W4	Frequency of interest for structural damping. The damping matrix for transient analysis is assembled from the equation: $$[B_{GLB}] = \text{CB1} * [B_1] + \text{CB2} * [B_2] + \text{ALPHA} * [K_{GLB}] + \text{BETA} * [M_{GLB}]$$ $$[B_1] = [B_{DAMP}] + \frac{\text{G}}{\text{W3}}[K_{GLB}] + \frac{1}{\text{W4}}\sum G_{ELEM} K_{ELEM}$$ In the second equation above, the first term contains terms from viscous damping elements (CDAMP). The second term is structural damping based on the global stiffness matrix multiplied by the overall structural damping coefficient, specified by PARAM, G. The third term is the structural damping matrix created when GE is specified on the MATi entries. The default values of 0.0 for W3 and W4 cause the second and third terms to be ignored regardless of the presence of PARAM, G. The units of W3 and W4 are radians per unit time. See also CB1, CB2.	Real	0.0
XDAMP	Controls the use of structural damping in modal response solutions. When set to OFF, only modal damping will be used regardless if structural damping is specified.	ON/OFF	ON

Frequency Response Processor Parameters:

Parameter	Description	Type	Default
ACBINTERACTTOL	Specifies the tolerance for removing negligible off-diagonal interaction terms from the acoustic coefficient matrix.	Real	1.0E-10
ACBPRESSET	Specifies the remote acoustic output set by reference to an output set command. The grid points in the specified output set define points not on the acoustic boundary where acoustic pressure is to be calculated and output. See the Case Control command, SET in Section 3, *Case Control, f*or more information.	Integer $>$ 0	0
ACBREFPRES	Specifies the acoustic reference pressure used to convert sound pressure into decibels for boundary acoustic analysis.	Real	0.0
ACBVC	Defines the speed of sound in the fluid medium for boundary acoustic analysis.	Real	0.0
ADDPSDAFREQ	Option for automatically adding analysis frequencies to random response solutions. When set to ON will add frequencies from TABRND1 Bulk Data entries referenced in the Case Control of a random response solution.	ON/OFF	OFF
DFREQ	Specifies the threshold for the elimination of duplicate frequencies. Duplicate frequencies will be ignored if, $$\lvert f_i - f_{i-1} \rvert < \text{DFREQ} * \lvert f_{MAX} - f_{MIN} \rvert$$ where f_{MAX} and f_{MIN} are the maximum and minimum solution frequencies of the combined FREQi Bulk Data entries.	Real	1.0E-5
FREQRESPRSLTINCR	Defines the precision used in calculating real results values from complex ones in frequency response solutions using a sinusoidal sweep. Larger values will provide more accurate invariant and composite results measures at the cost of performance. The default value of 10 provides a compromise between these and will result in a sweep every 18 degrees from zero to 180 degrees.	Integer $>$ 0	10
FREQRESPRSLTOUT	Controls neutral file output during random response solutions. When set to OFF, disables frequency response output to the results neutral file. The OFF setting may reduce file size dramatically for large models with a large number of solution frequencies.	ON/OFF	ON
KDAMP	Option for specifying viscous modal damping as structural damping. When KDAMP is set to -1 or OFF viscous modal damping is entered into the complex stiffness matrix as structural damping.	Integer ON/OFF	1 ON
RANDRESPINVLEVEL	Controls invariant stress output in frequency and random response solutions. When set to 1 will output von Mises stress or strain. When set to 2 will also include principal and max shear stress or strain and biaxiality ratio.	$0 \le$ Integer ≤ 2	1
RANDRESPRSLTOUT	Controls neutral file output during random response solutions. When set to OFF, disables power spectral density output to the results neutral file. The OFF setting may reduce file size dramatically for large models with a large number of solution frequencies.	ON/OFF	ON

Frequency Response Processor Parameters (Continued):

Parameter	Description	Type	Default
VFM2ACB	Option to perform boundary acoustic analysis when a virtual fluid mass boundary is specified. The acoustic boundary is defined using MFLUID and ELIST Bulk Data entries in the same manner as virtual fluid mass. PARAM, ACBVC is used to specify the speed of sound in the fluid medium. PARAM, ACBPRESSET defines pressure output points in the fluid via SET Case Control commands. PARAM, ACBREFPRES is used to convert sound pressure to decibels. PARAM, ACBINTERACTTOL is used to specify the tolerance for removing negligible off-diagonal acoustic coefficient interaction terms from the assembled acoustic coefficient matrix to reduce memory requirements and improve performance.	ON/OFF	OFF

Nonlinear Solution Processor Parameters:

Parameter	Description	Type	Default
ADDNLTOQUADLOAD	When set to ON will add extensional loads in tension-only quad and shear panel elements to adjacent line elements.	ON/OFF	OFF
ADPCON	See SLINEKSFACT below.	Real	1.0
AUTOFIXNLMAT	Stress-strain curve correction option for nonlinear material plasticity. Plastic materials require all slopes in the stress-strain curve to be less than Young's modulus (except for the first 2 data points). The ON setting will remove any data points that violate the requirement.	ON/OFF	ON
BARDKMETHOD	Specifies how differential stiffness is applied to rod, bar, and beam elements. There are four options: TENSION – Differential stiffness is only added when the element is in tension. COMPRESSION – Differential stiffness is only added when the element is in compression. COUPLED – Differential stiffness is added regardless of loading and includes coupled torsional terms. BOTH – Differential stiffness is added regardless of loading and does not include coupled torsional terms.	TENSION/ COMPRESSION/ COUPLED/ BOTH	BOTH
BISECT	Controls how a nonlinear solution will proceed when the load bisection limit is reached. When set to ON, the solution will terminate with a fatal error. When set to OFF, the solution will bisect until the load bisection limit is reached but will continue to the next full or subincrement of load if the reason for the bisection was due to a lack of convergence.	ON/OFF	ON
COMPE1RSF	Specifies the default nonlinear composite progressive ply failure E1 reduction scale factor when not explicitly defined on a MATi Bulk Data entry.	$0.0 \le$ Real ≤ 1.0 DISABLE	DISABLE
COMPE1RSFTID	Specifies the default nonlinear composite progressive ply failure E1 stress-strain table identification number when not explicitly defined on a MATi Bulk Data entry.	Integer > 0	DISABLE
COMPE2RSF	Specifies the default nonlinear composite progressive ply failure E2 reduction scale factor when not explicitly defined on a MATi Bulk Data entry.	$0.0 \le$ Real ≤ 1.0 DISABLE	DISABLE
COMPE2RSFTID	Specifies the default nonlinear composite progressive ply failure E2 stress-strain table identification number when not explicitly defined on a MATi Bulk Data entry.	Integer > 0	DISABLE
COMPE3RSF	Specifies the default nonlinear composite progressive ply failure E3 reduction scale factor when not explicitly defined on a MATi Bulk Data entry.	$0.0 \le$ Real ≤ 1.0 DISABLE	DISABLE
COMPE3RSFTID	Specifies the default nonlinear composite progressive ply failure E3 stress-strain table identification number when not explicitly defined on a MATi Bulk Data entry.	Integer > 0	DISABLE
COMPG12RSF	Specifies the default nonlinear composite progressive ply failure G12 reduction scale factor when not explicitly defined on a MATi Bulk Data entry.	$0.0 \le$ Real ≤ 1.0 DISABLE	DISABLE
COMPG12RSFTID	Specifies the default nonlinear composite progressive ply failure G12 stress-strain table identification number when not explicitly defined on a MATi Bulk Data entry.	Integer > 0	DISABLE
COMPG1ZRSF	Specifies the default nonlinear composite progressive ply failure G1Z reduction scale factor when not explicitly defined on a MATi Bulk Data entry.	$0.0 \le$ Real ≤ 1.0 DISABLE	DISABLE

Nonlinear Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
COMPG1ZRSFTID	Specifies the default nonlinear composite progressive ply failure G1Z stress-strain table identification number when not explicitly defined on a MATi Bulk Data entry.	Integer > 0	DISABLE
COMPG23RSF	Specifies the default nonlinear composite progressive ply failure G23 reduction scale factor when not explicitly defined on a MATi Bulk Data entry.	0.0 ≤ Real ≤ 1.0 DISABLE	DISABLE
COMPG23RSFTID	Specifies the default nonlinear composite progressive ply failure G23 stress-strain table identification number when not explicitly defined on a MATi Bulk Data entry.	Integer > 0	DISABLE
COMPG2ZRSF	Specifies the default nonlinear composite progressive ply failure G2Z reduction scale factor when not explicitly defined on a MATi Bulk Data entry.	0.0 ≤ Real ≤ 1.0 DISABLE	DISABLE
COMPG2ZRSFTID	Specifies the default nonlinear composite progressive ply failure G2Z stress-strain table identification number when not explicitly defined on a MATi Bulk Data entry.	Integer > 0	DISABLE
COMPG31RSF	Specifies the default nonlinear composite progressive ply failure G31 reduction scale factor when not explicitly defined on a MATi Bulk Data entry.	0.0 ≤ Real ≤ 1.0 DISABLE	DISABLE
COMPG31RSFTID	Specifies the default nonlinear composite progressive ply failure G31 stress-strain table identification number when not explicitly defined on a MATi Bulk Data entry.	Integer > 0	DISABLE
CONTACTGEN	Automated Surface Contact Generation (ASCG). A value between 0 and 5 defines the type of contact generated. The program automatically finds solid and shell element faces in or near contact and generates the appropriate contact element type between them. There are six options: 0 – Automated surface contact generation is disabled. 1 – Symmetric general contact is enabled. 2 – Symmetric welded contact is enabled. 3 – Symmetric bi-directional sliding contact is enabled. 4 – Symmetric rough contact is enabled. 5 – Symmetric offset welded contact is enabled. The character variables: DISABLE, GENERAL, WELDED, SLIDE, ROUGH, and OFFSET may be used in place of the numerical options 0 through 5. See also CONTACTTOL.	0 ≤ Integer ≤ 5 DISABLE/ GENERAL/ WELDED/ SLIDE/ ROUGH/ OFFSET	0
CONTACTSTAB	Surface contact solution stabilization option. When set to ON, will generate stabilization spring stiffness via the model parameters NLKDIAGSET, NLKDIAGAFACT, and NLKDIAGMINAFACT on the contact boundary. The default AUTO setting will automatically detect and stabilize all surface contact in the model with a significant initial gap (i.e., model reference dimension multiplied by 1.0E-04). The stabilization stiffness used can be controlled by specifying a scale factor which is a multiplier to the stabilization stiffness calculated automatically.	Real AUTO	AUTO

Nonlinear Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
CONTACTTOL	Specifies the contact tolerance used in Automated Surface Contact Generation (ASCG). The value set defines the maximum normal activation distance. A recommended value is a distance approximately 10% larger than the largest gap you want to be recognized as contact. The default AUTO setting is based on the model reference dimension multiplied by 1.0E-04. Note that specified values are actual distances and are not normalized. For some models (i.e., very large, very small, or with large gaps) the default CONTACTTOL value may not be well suited, therefore it is recommended the analyst define this explicitly.	Real AUTO	AUTO
EMODES	Specifies the number of modes to be extracted during the initialization phase of Automated Impact Analysis. A normal modes analysis is performed to determine the damping frequency of interest and the time step size.	Integer > 0	30
FIXNLTOQUAD	Option to control the reversion of tension-only shell elements. Setting FIXNLTOQUAD to ON prevents elements that have reverted to tension-only from changing back to standard shell elements if the element load state changes from compression to tension. The ON setting is recommended for better convergence and solution stability.	ON/OFF	ON
HPNLMATREDORD	Hyperelastic element volumetric reduced order integration option. When set to ON, volumetric hyperelastic terms will use a one point integration allowing larger volumetric material constants and better simulation of incompressible materials. The default AUTO setting will use hyperelastic material reduced order integration for hex and pent elements and full integration for tet elements.	ON/OFF AUTO	AUTO
HPNLMATSFACT	Specifies the scale factor applied to the material nonlinear portion of the hyperelastic element material stiffness matrix [E]. The default AUTO setting will use a value which minimizes solution divergence.	$0.0 <$ Real ≤ 1.0 AUTO	AUTO
INITSTRAINSFACT	Specifies the scale factor applied to initial strain values defined on STRAIN Bulk Data entries.	Real	1.0
LANGLE	Specifies the method for processing large rotations in nonlinear analysis. Two methods are available, the gimbal angle method (default) and the rotation vector method. If LANGLE is set to 1, the gimbal angle method is selected. If LANGLE is set to 2, the rotation vector method is selected. Both methods give comparable results.	Integer 1 or 2	1

Nonlinear Solution Processor Parameters (Continued):

<table>
<tr><th>Parameter</th><th>Description</th><th>Type</th><th>Default</th></tr>
<tr><td>LGDISP</td><td>Controls the use of large displacement and follower force effects and differential stiffness in nonlinear analysis. If LGDISP is set to 1, or ON, large displacement and follower force effects and differential stiffness will be included. If LGDISP is set to 0, -1, or OFF, large displacement and follower force effects and differential stiffness will not be included. There are six options:
<table>
<tr><th></th><th colspan="6">LGDISP Setting</th></tr>
<tr><th>Nonlinear Effect</th><th>0</th><th>1</th><th>2</th><th>3</th><th>4</th><th>5</th></tr>
<tr><td>Large Displacement</td><td></td><td>✓</td><td>✓</td><td></td><td>✓</td><td>✓</td></tr>
<tr><td>Differential Stiffness</td><td></td><td>✓</td><td>✓</td><td>✓</td><td></td><td></td></tr>
<tr><td>Follower Force</td><td></td><td>✓</td><td></td><td>✓</td><td>✓</td><td></td></tr>
</table>
In Automated Impact Analysis (AIA), if LGDISP is set to 0, a value of 1 will be forced.</td><td>$-1 \le$ Integer ≤ 5
ON/OFF</td><td>0
OFF</td></tr>
<tr><td>MAXBISECTRESTART</td><td>Nonlinear solver restart option after maximum bisection error. When set to ON, permits restarting a nonlinear static solution which has terminated due to an E5076 fatal error (maximum number of bisections permitted reached.</td><td>ON/OFF</td><td>OFF</td></tr>
<tr><td>MAXINCREFSTRAINP</td><td>Specifies the maximum effective plastic strain permitted at an element integration point for a single nonlinear iteration. The default AUTO setting will use a starting value of 1.0E-4 if contact exists in the model and 1.0E-2 if it does not. The tolerance is then increased by the square of the increment number. The tighter tolerance when contact is present prevents erroneous plastic strain from accumulating while contact is being initially established.</td><td>Real
AUTO</td><td>AUTO</td></tr>
<tr><td>NCONTACTGEOMITER</td><td>Specifies the number of iterations for repositioning surface contact element slave nodes with initial penetration and/or protrusion. See SLINEPENTOL and SLINEPROTOL in this section.</td><td>Integer ≥ 0</td><td>1</td></tr>
<tr><td>NITERCUPDATE</td><td>Nonlinear solver contact stiffness update option. Controls the nonlinear contact stiffness update strategy. The value set is the number of iterations before the contact stiffness is updated. The AUTO setting varies the value automatically during nonlinear iteration. A zero setting will result in a stiffness update if any contact element or segment has a status change during the nonlinear iteration sequence.</td><td>Integer ≥ 0
AUTO</td><td>AUTO</td></tr>
<tr><td>NITERPFUPDATE</td><td>Nonlinear composite ply failure and stiffness update option. Controls the composite ply failure and stiffness update strategy used in Progressive Ply Failure Analysis (PPFA).</td><td>Integer ≥ 0</td><td>1</td></tr>
</table>

Nonlinear Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
NITERKSUPDATE	Nonlinear differential stiffness update option. Controls the nonlinear differential stiffness update strategy when a non-positive definite error is detected in a nonlinear static solution. The value set is the number of iterations following a non-positive definite error before the differential stiffness is again added to the tangent stiffness. NITERKSUPDATE is only applicable when LGDISP is set to ON, 1, or 2.	Integer ≥ 0	3
NITERMUPDATE	Nonlinear solver material stiffness update option. Controls the nonlinear material stiffness update strategy. The value set is the number of iterations before the material stiffness is updated.	Integer ≥ 0	3
NITERSUPDATE	Nonlinear solver surface contact stiffness update option. The value set is the number of iterations to freeze slide line and surface contact status when two successive solution divergences occur. See SLINESTABOPTION in this section.	Integer ≥ 0	8
NLAYERS	Specifies the number of nonlinear material layers in quad and tri elements. A larger value of NLAYERS will give greater accuracy at the cost of computing time and storage requirements.	Integer > 1	10
NLCOMPPLYFAIL	Nonlinear composite Progressive Ply Failure Analysis (PPFA) option. When set to ON, composite plies that fail the user specified failure theory (FT field on the PCOMP Bulk Data entry) will be reduced in material stiffness based on reduction scale factors specified on MAT1 and MAT8 Bulk Data entries. PPFA is supported in nonlinear static and transient solution sequences only.	ON/OFF	OFF
NLINDATABASE	Controls the storage and retrieval of nonlinear data such as loads, displacements, stress, and strain used in nonlinear static analysis. The default value DELETE will purge all nonlinear data when the program terminates normally. When set to STORE, the nonlinear database is stored in a single file with the same base name as the Model Results Output File, plus an increment and a load scale factor designator, and a .TDB file extension. When set to FETCH, the nonlinear database specified by the NLINDATFILE directive is retrieved and the nonlinear solution (static or transient) starts at the database configuration and load scale factor. An integer value may be specified to designate a SET command which identifies which load increments (load scale factors) are to be stored. When set to UPDATE, the nonlinear database will be retrieved and stored.	Integer > 0 DELETE/ FETCH/ STORE/ UPDATE	DELETE
NLINDATALOADSF	Specifies the initial load scale factor to be used when performing a nonlinear database restart (PARAM, NLINDATABASE, FETCH). The default AUTO setting will use the load scale factor stored in the nonlinear database file specified using the NLINDATFILE directive.	Real AUTO	AUTO

Nonlinear Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
NLINSOLACCEL	Nonlinear solver iteration acceleration option. Controls nonlinear iteration acceleration, damping, and line search algorithms. There are five options: 0 – No acceleration, damping, or line search controls (OFF). 1 – Damping only. 2 – Line search only. 3 – Acceleration, damping, and line search controls. 4 – Acceleration and damping only (ON). See the NLPARM Bulk Data entry in Section 4, *Bulk Data*, for additional line search parameters.	$0 \le$ Integer ≤ 4 ON/OFF	4
NLINSOLTOL	See NLTOL below.		
NLKDIAGAFACT	Specifies the stiffness to be added to diagonal terms of the global stiffness matrix. Specifying a small positive value is useful in stabilizing a solution and preventing a non-positive definite or singularity error. In nonlinear static solutions the added stiffness is decreased at the completion of each increment so to reach the value defined by NLKDIAGMINAFACT at the completion of the last increment. See also NLKDIAGCOMP and NLKDIAGMINAFACT.	Real	0.0
NLKDIAGCOMP	Specifies component numbers that NLKDIAGAFACT will augment.	$1 \le$ Integers ≤ 6	123456
NLKDIAGMINAFACT	Specifies the minimum NLKDIAGAFACT value used in nonlinear static solutions where the NLKDIAGAFACT value is decreased at the completion of each increment so to reach NLKDIAGMINAFACT at the completion of the last increment.	Real	0.0
NLKDIAGSET	Specifies which grid points NLKDIAGAFACT will be applied to by reference to an output set command. The default zero setting will apply NLKDIAGAFACT to all grid points. See the Case Control command, SET in Section 3, *Case Control,* for more information.	Integer ≥ 0	0
NLLSSTRAINTYPE	Specifies the type of large strain strain output as either log strain (LOG) or Green strain (GREEN).	LOG/GREEN	LOG
NLLSSTRESSTYPE	Specifies the type of large strain stress output as either as either Cauchy stress (CAUCHY) or 2nd Piola-Kirchhoff stress (2NDPK).	CAUCHY/ 2NDPK	CAUCHY
NLMATSFACT	Specifies the scale factor applied to the material nonlinear portion of the element material stiffness matrix [E]. The default AUTO setting will use a value which minimizes solution divergence.	$0.0 <$ Real ≤ 1.0 AUTO	AUTO
NLMATTABLGEN	When set to a value greater than zero, will convert all bi-linear materials defined on MATS1 entries to stress-strain tables with an elastic-plastic transition controlled by the value set for NLMATTABLGEN.	$0.0 <$ Real ≤ 1.0	0.0
NLNPDKRESET	When set to ON, will use the last converged tangent stiffness when a non-positive definite is detected. If large displacement effects with differential stiffness are enabled, the differential stiffness is removed first.	ON/OFF	OFF

Nonlinear Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
NLSUBCREINIT	When set to ON will reinitialize the nonlinear database for each subcase thereby restarting the simulation from zero. The default setting of OFF carries results and loading over from the previous subcase. This parameter is only applicable for nonlinear static solution sequences.	ON/OFF	OFF
NLTOQUAD	When set to OFF will disable tension-only quad element support regardless of PSHELL Bulk Data entry settings and solution type.	ON/OFF	ON
NLTOL	Nonlinear solver default convergence tolerance option. Sets defaults for the EPSU, EPSP and EPSW fields of the NLPARM and TSTEPNL Bulk Data entries. There are four options for the level of accuracy: 0 – Very High 1 – High 2 – Engineering 3 – Preliminary Design See the NLPARM and TSTEPNL Bulk Data entries in Section 4, *Bulk Data,* for additional information.	$0 \leq$ Integer ≤ 3	2
NLTRUESTRESS	When set to ON, will output true stress and strain in large displacement nonlinear solutions. True stress and strain accounts for changes in element shape due to deformation.	ON/OFF	OFF
NSLINEPSURFDIV	The number of surface divisions on a triangular contact face. Can be set to 1, 4 or 6. This setting is only applicable for triangular contact faces with midside nodes and ENHCCONTACTRSLT=PARABOLIC	1, 4, or 6	4
NSUBINCRBISECT	Specifies the maximum number of sub-incremental plastic increments before bisection is activated. The default AUTO setting will use a value of 100 if contact exists in the model and 200 if it does not. The tighter tolerance when contact is present prevents erroneous plastic strain from accumulating while contact is being initially established.	Integer > 0 AUTO	AUTO
QUADSECT	Specifies how a load or time increment will be divided when a bisection condition exists in a nonlinear solution. When set to ON and a bisection condition is reached, the current load or time increment is quadsected.	ON/OFF	OFF
SLINE2RIGIDELEM	When set to ON will convert surface contact elements to RBE3 elements on Nastran Bulk Data file export	ON/OFF	OFF
SLINEEDGENORMTOL	Specifies the automated surface contact generation element edge normal tolerance in degrees. An edge to face contact element will not be generated if the edge normal and face normal differ by a value greater than this tolerance.	$0.0 \leq$ Real ≤ 90.0	60.0
SLINEFACENORMTOL	Specifies the automated surface contact generation element face normal tolerance in degrees. A face to face contact element will not be generated if the face normals differ by a value greater than this tolerance.	$0.0 \leq$ Real ≤ 90.0	30.0
SLINEFSLIPK	Controls components of nonlinear stiffness matrix for contact elements. DIAGONAL includes Normal, Shear X and Shear Y components. COUPLED, also includes off diagonal shear terms and Shear Slip X and Shear Slip Y components DISABLE calculates stiffness using original method.	COUPLED/ DIAGONAL/ DISABLE	DIAGONAL
SLINEKAVG	When set to ON, will use an average of the adjacent component stiffnesses used in determining surface contact penalty values. The default OFF setting uses only the normal stiffness component which may be too small or large for some element thicknesses and/or materials.	ON/OFF	OFF

Nonlinear Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
SLINEKSFACT	Specifies the initial penalty values used in slide line and surface contact analysis. Initial penalty values are calculated using: $$k * SFACT * \lvert SLINEKSFACT \rvert$$ where k is a value selected for each slave node based on the diagonal stiffness matrix coefficients. SFACT is specified in the SFACT field of the BCONP and BSCONP Bulk Data entries. The SLINEKSFACT value applies to all contact regions in the model. The default AUTO setting will automatically adjust model penalty values when convergence problems occur.	Real AUTO	AUTO
SLINEKSFACT2TC	When set to ON will treat the SFACT field specified on the BSCONP and BCONP Bulk Data entries and CONTACTGEN Case Control commands as thermal contact conductance in heat transfer solutions and force a value of unity in structural solutions.	ON/OFF	OFF
SLINEMAXACTCORD	Specifies the surface contact activation coordinate system corresponding to SLINEMAXACTDIR. See also SLINEMAXACTDIR and SLINEMAXACTWIDTH.	Integer > 0	0
SLINEMAXACTDIR	Specifies the direction of surface contact movement when significant sliding is specified reducing unnecessary contact surface generation and memory requirements. See also SLINEMAXACTWIDTH and SLINEMAXACTCORD.	XYZ/X/Y/Z	XYZ
SLINEMAXACTDIST	Specifies the maximum slide line and surface contact element activation distance. The primary purpose of this parameter is to prevent unnecessary generation of contact segments when little or no movement is expected. For general and rough contact penetration types, the default value is AUTO in linear solutions and 1.0E+30 in nonlinear solutions. For all other penetration types the default is AUTO. The AUTO setting will restrict contact generation to adjacent elements while the 1.0E+30 setting will generate contact to allow unlimited movement. The AUTO setting is recommended for optimal performance when little or no movement is expected such as with bolted connections. Note that a zero value should only be used if all master and slave nodes are collocated.	Real > 0.0 AUTO	1.0E+30 AUTO
SLINEMAXACTRATIO	Specifies the maximum surface contact element activation ratio. When set to a value greater than zero, specifies the ratio of activation distance to contact surface maximum edge length. This parameter may be useful in reducing solution time for nonlinear surface contact models with SLINEMAXACTDIST set to a value greater than zero by deactivating contact segments far from area of active contact.	Real ≥ 0.0	0.0
SLINEMAXACTWIDTH	Defines the total width of the surface contact activation vector. See also SLINEMAXACTDIR and SLINEMAXACTCORD.	Real > 0.0 AUTO	AUTO
SLINEMAXDISPTOL	Specifies the normalized maximum allowable contact surface penetration defined as $$SLINEMAXDISPTOL = TMAX / \sqrt{A_{contact}}$$ where TMAX is the maximum allowable contact surface penetration. $A_{contact}$ is the contact surface area. The recommended range for SLINEMAXDISPTOL 1.0E-02 to 1.0E-05. Larger values may provide better nonlinear convergence with a possible increase in contact surface penetration.	Real > 0.0	1.0E-4

Nonlinear Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
SLINEMAXPENDIST	Specifies the maximum slide line and surface contact element penetration distance. The primary purpose of this parameter is to prevent contact segments from unintentionally becoming active when the geometry is complex and large changes in configuration take place. The default AUTO setting uses the maximum contact surface or slide line reference length.	Real $>$ 0.0 AUTO	AUTO
SLINEOFFSETTOL	Specifies the tolerance for automatically converting surface weld elements to offset weld elements. Welded contact with an initial separation less than SLINEOFFSETTOL will be converted to offset welded contact. The default AUTO setting is based on the model reference dimension multiplied by 1.0E-03. Note that specified values are actual distances and are not normalized. For some models (i.e., very large, very small, or with large gaps) the default SLINEOFFSETTOL value may not be well suited, therefore it is recommended the analyst define this explicitly. Note that for Automated Surface Contact Generation (ASCG) when a CONTACTGENERATE Case Control command is specified with the MAXAD field, SLINEOFFSETTOL will be set to MAXAD.	Real AUTO	AUTO
SLINEOPENKSFACT	Specifies the open gap penalty value used in slide line and surface contact analysis.	Real	1.0E-10
SLINEPENTOL	Specifies tolerances for adjusting initial penetration errors on contact surfaces. The actual tolerance used varies for each contact segment and is equal to the product of the contact segment reference dimension (average segment edge length) and SLINEPENTOL. Any initial penetration past the normalized SLINEPENTOL value will result in a check normal warning message. Any penetration between than this value and zero will result in repositioning of the contact segment slave node to the contact surface.	Real	0.2
SLINEPLANEZDIR	Alternate slide line plane normal definition. Specifies which coordinate component direction should be used to define the normal for all slide line planes.	X/Y/Z/R/T	Z
SLINEPNODEOPTION	This option can limit the amount of contact generated for parabolic tri face contact. CORNER will only consider the corner nodes for contact. EDGE will only consider the midside nodes for contact. BOTH will consider all the nodes on the triangular face. Only applicable with ENHCCONTACTRSLT=PARABOLIC	BOTH/ CORNER/ EDGE/ AUTO	BOTH
SLINEPOSTOL	Used to control contact surface segment overlap. The actual tolerance used varies for each contact segment and is equal to the product of the contact segment reference dimension (average segment edge length) and SLINEPOSTOL. A slave node is considered off the contact surface when past the segment boundary plus this value.	Real	1.0E-2
SLINEPROTOL	Specifies tolerances for adjusting initial protrusion errors on contact surfaces. The actual tolerance used varies for each contact segment and is equal to the product of the contact segment reference dimension (average segment edge length) and SLINEPROTOL. Any protrusion between this value and zero will result in a reset of the contact zero datum to the actual protrusion. The AUTO setting determines an optimum tolerance to improve accuracy based on contact surface curvature and initial gap distance.	Real AUTO	AUTO

Nonlinear Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
SLINESLIDETYPE	Contact penalty stiffness update method. When SLINESLIDETYPE is set to DYNAMIC, the proximity stiffness based update method is selected. When SLINESLIDETYPE is set to STATIC, the displacement based stiffness update method is selected. For either setting the normalized SLINEMAXDISPTOL parameter defines the default TMAX value (maximum allowable penetration). See SLINEMAXDISPTOL in this section. When SLINESLIDETYPE is set to AUTO and SLINEMAXACTDIST is also set to AUTO or zero and the MAXAD field on all BSCONP Bulk Data entries are set to AUTO, blank, or zero, SLINESLIDETYPE will be set to STATIC otherwise DYNAMIC is selected.	DYNAMIC/ STATIC/ AUTO/ DISABLE	DYNAMIC
SLINESTABKSFACT	Used to stabilize surface contact in nonlinear static solutions. When set to a value greater than zero, will add a normal and in-plane stabilization stiffness between contact surfaces. The default zero value disables this feature. A value of 1.0 will add a stiffness approximately equal to the closed gap stiffness value. The stabilization stiffness is decreased with each full increment in each subcase using $$K_s^i = \frac{\overline{K}_s}{2^{i-1}}$$ where $\overline{K}_s$ is the initial stabilization stiffness base on the specified SLINESTABKSFACT value. K_s^i is the stabilization stiffness for the current increment i.	Real	0.0
SLINESTABOPTION	Surface contact solution stabilization option. Specifies the type of solution stabilization to be used when a model contains slide line or surface contact elements and the nonlinear solution diverges. Options are only active for stabilization iterations defined by NITRERSUPDATE. There are four options: 0 – Stabilization disabled. 1 – Contact status is frozen. 2 – Contact open gap stiffness (SLINEOPENKSFACT) is increased by 1.0E+7. 3 – Contact unload tolerance (SLINEUNLOADTOL) is increased by 1.0E+7. 4 – Options 1 – 3 are used simultaneously. See SLINEUNLOADTOL, SLINEOPENKSFACT, NITERSUPDATE, and in this section.	0 ≤ Integer ≤ 4	0
SLINESTRESSLOC	Specifies the location where surface contact nodal stresses are calculated: SLAVE surface, MASTER surface, or BOTH surfaces.	SLAVE/ MASTER/ BOTH	MASTER

Nonlinear Solution Processor Parameters (Continued):

Parameter	Description	Type	Default
SLINEUNLOADTOL	Tolerance for determining a contact surface unload condition. The actual tolerance used varies for each contact segment and is equal to the product of the contact segment reference dimension (average segment edge length) and SLINEUNLOADTOL. An unload condition occurs when the contact surface normal displacement is greater than the unload tolerance. This parameter is not applicable in nonlinear transient solutions.	Real	1.0E-10

Results Processor Parameters:

Parameter	Description	Type	Default
ADDPRESTRESS	Option for adding prestress subcase results to subsequent subcases. This parameter will only function in PRESTRESS STATIC or PRESTRESS MODAL solutions.	ON/OFF	ON
ALTFAILINDEXFORM	Alternate failure index formulation for the LaRC02 failure theory. When set to ON will output the square of the ply fiber failure indexes providing a more consistent basis with the matrix failure indexes.	ON/OFF	OFF
AUTOCORDROTATE	Option for automatically rotating a projected coordinate system axis that is normal to an element plane, when an in-plane component is required.	ON/OFF	ON
BOLTPRELOADTOL	Tolerance for warning when a bolt has lost preload.	Real	0.0
COMPILSMETHOD	Option for defining how composite bond material failure indexes and strength ratios are calculated. When set to COMPONENT, the maximum of a separate material x-direction and y-direction failure index is used. When set to RESULTANT, a resultant transverse shear stress is calculated from the component values and used. The RESULTANT method is always used when the MCT composite failure theory is requested.	COMPONENT/ RESULTANT	COMPONENT
COMPK1	Foam core composite sandwich stability allowable coefficient. The face sheet wrinkling allowable for a foam core sandwich is given by: $$\sigma_{wr} = k_1(E_f E_c G_c)^{1/3}$$ where k_1 is given by COMPK1 and is defaulted to 0.76 for thick cores and 0.63 for thin cores. E_f is Young's Modulus for the facesheet E_c is Young's Modulus for the core G_c is the transverse shear modulus for the core See the Autodesk Nastran User's Manual, Section 21.4, *Composites,* for more information.	Real	AUTO
COMPK2	Honeycomb core composite sandwich stability allowable coefficient. The face sheet wrinkling allowable for a honeycomb core sandwich is given by: $$\sigma_{wr} = k_2 E_f \sqrt{\frac{E_c t_f}{E_f t_c}}$$ where k_2 is given by COMPK2 and is defaulted to 0.82 regardless of core thickness. E_f is Young's Modulus for the facesheet E_c is Young's Modulus for the core G_c is the transverse shear modulus for the core t_f is facesheet thickness t_c is core thickness See the Autodesk Nastran User's Manual, Section 21.4, *Composites,* for more information.	Real	AUTO

Results Processor Parameters (Continued):

Parameter	Description	Type	Default
COMPRSLTOUT	Controls the output of individual ply results to the element results neutral file for post processor results plotting. When set to ON, up to 200 individual ply results for each element are output in addition to laminate max/min results.	ON/OFF	ON
DATABASEACCEL	Model database acceleration option. When set to ON, the model database will be loaded into memory regardless of available RAM. When set to AUTO, RAM availability is checked for files that could use large memory blocks and only if sufficient RAM is available, will load into memory. When set to OFF, the model database will be stored on disk and memory requirements for internal data storage will be reduced, but performance may be degraded.	ON/OFF AUTO	AUTO
DIRSTRESSTYPE	Direct stress type option. Controls what stress type is output for bar, beam, and shell elements in the direct stress tensor results measure. There are three options: 0 – Direct stress tensor is output. 1 – Bending only stress tensor is output which excludes membrane/extensional stress components. 2 – Membrane/extensional only stress tensor is output which excludes bending stress components.	$0 \leq$ Integer ≤ 2	0
DISPGEOMSFACT	Specifies the scale factor applied to deformed geometry output. See the Model Initialization directive, TRSLDFGMDATA in Section 2, *Initialization,* for more information.	Real	1.0
ELEMRSLTCORD	Default coordinate system to be used for computing element results if a SURFACE and/or VOLUME Bulk Data entry is not specified. Note that grid point results will be output in the grid coordinate system.	Integer ELEMENT/ BASIC/ MATERIAL	MATERIAL
ELEMRSLTMAXTYPE	Element location where maximum/minimum stress/strain results are output. When AVGCENTER is selected the element centroid will be used (default in previous versions). When MAXCORNER is selected the maximum corner value will be used.	AVGCENTER/ MAXCORNER	MAXCORNER
ENHCBARRSLT	Option for enhanced CBAR and CBEAM element results. When set to ON, an improved method for calculating CBAR and CBEAM element stress results is used when a corresponding PBARL and PBEAML property type is specified. Maximum direct and invariant stresses are determined using an automatically generated internal cross-sectional mesh at each element end. A separate finite element solution is performed on each mesh with direct and invariant results calculated at each mesh point and the maximum and minimum values reported.	ON/OFF	OFF
ENHCCONTACTRSLT	Enhances the contact element formulation for parabolic tet elements. When activated, it will subdivide parabolic tet element primary surfaces into 4 separate sub-surfaces and avoid linearizing the element face. If the model does not have contact on parabolic tet elements, this parameter will have no effect on the solution.	ON/OFF/ SEGMENTED/ PARABOLIC/ AUTO	AUTO
ENHCQUADRSLT	Option for enhanced CQUADR element results. When set to ON, an improved method for calculating CQUADR element stress results is used which gives better accuracy in regions with stress concentrations.	ON/OFF	OFF

Results Processor Parameters (Continued):

Parameter	Description	Type	Default
EQVSTRESSTYPE	Equivalent stress type option. Controls what stress type is output in linear solutions for bar, beam, and shell elements in the equivalent stress results measure. There are three options: 0 – von Mises stress is output. 1 – Bending only von Mises stress is output which excludes membrane/extensional stress components. 2 – Membrane/extensional only von Mises stress is output which excludes bending stress components. This parameter is only applicable to linear solutions. Equivalent stress is always output in nonlinear solutions.	$0 \leq$ Integer ≤ 2	0
FLOATOUTZERO	Model results floating point zero tolerance. Real output data less than FLOATOUTZERO will be set to zero.	Real	1.0E-15
GPFORCEMETHOD	Specifies how grid point forces are calculated. The NORAN option only calculates element force contributions for elements which have an element FORCE request. This permits the calculation of internal loads along element point, edge, and face boundaries. The NASTRAN option considers all elements regardless of FORCE request.	NASTRAN/ NORAN	NASTRAN
GPRSLTAVEMETHOD	Specifies how shell element corner results are averaged to determine grid point values. When set to INVARIANT, all element corner result measures are calculated first and then averaged including invariant stress and strain. When set to DIRECT, only direct stress and strain is averaged and invariant results are determined from the averaged direct values.	INVARIANT/ DIRECT	INVARIANT
GPSTRESS	Grid point stress output option. When set to ON, grid point stresses for all subcases will be output unless the STRESS or STRAIN Case Control command is set to NONE for a specific subcase.	ON/OFF	OFF
LARC02TSAITOL	Option to revert failure theory used in composite laminate individual ply results from LaRC02 or Puck to Tsai-Wu if a non-unidirectional material is detected. The value set controls the tolerance that triggers reversion based on the ratio of E1/E2, XT/YT, and XC/YC.	Real	2.0
MATGEN	Homogenized material property translation for Bulk Data Output File generation. When set to ON, will translate calculated homogenized material property as a solid anisotropic material property (MAT9).	ON/OFF	OFF
MAXSRITER	Option to specify the maximum number of iterations used in determining composite LaRC02 strength ratios.	Integer ≥ 0	100
MECHSTRAIN	Controls the type of strain output. When thermal strains are generated, if MECHSTRAIN is set to ON, then mechanical strain (total minus thermal) is output. If MECHSTRAIN is set to OFF, then total strain (thermal plus mechanical) is output.	ON/OFF	OFF
NOCOMPS	Controls the computation and output of composite element ply results. If NOCOMPS is set to 1 or OFF, composite element ply results will be output while the equivalent homogeneous element results will be suppressed. If NOCOMPS is set to -1, 0 or ON, composite element ply results will be suppressed while the equivalent homogeneous element results will be output. When NOCOMPS is set to AUTO, NOCOMPS will be set to OFF (ply results are calculated) when either element force, stress or strain is requested or a nonlinear solution is performed and NLCOMPPLYFAIL is set to ON, and to ON (ply results are not calculated) otherwise reducing calculation time.	Integer ON/OFF AUTO	AUTO
OGEOM	Controls the output of geometry data blocks to the Nastran Binary Results File.	ON/OFF	ON

Results Processor Parameters (Continued):

Parameter	Description	Type	Default
OUTSETTOL	Tolerance for identifying real values in output set lists. A real value is considered as included if $$\left\|\frac{r_{SET} - r_{Input}}{r_{Input}}\right\| < \text{OUTSETTOL}$$ Where r_{SET} is the SET value; r_{Input} is the input value	Real	1.0E-5
POST	Controls the output of data blocks to the Nastran Binary Results File. See the Nastran Binary Results File Data Block Definition Table later in this section.	-7 ≤ Integer < 0	-1
RSLTDATABASE	Controls the storage and retrieval of results data such as loads, displacements, stress, and strain generated in linear and nonlinear structural solutions and subsequently used for restarts in fatigue and explicit dynamics. The default value DELETE will purge the results database when the program terminates normally. When set to STORE, the results database is stored in a single file with the same base name as the Model Results Output File and a .RDB file extension. When set to FETCH, the results database specified by the RSLTDATFILE directive is retrieved for use in multiaxial fatigue analysis. The EXCITEID on the TLOAD1 Bulk Data entry specifies the database results set to be used. If the results database was generated from a linear static analysis this would be the subcase sequence number (not identification number). If the results database was generated from a nonlinear static or transient analysis this would be the load or time step.	DELETE/ FETCH/ STORE/ EXPLICIT	DELETE
SOLIDGEN	Shell element to solid element generation option. When set to MODEL all shell elements in the model are converted to solid elements. When set to XSHL only Variable Shell Element Thickness (VSET) optimization design space elements will be converted.	DISABLE/ MODEL/ XSHL	DISABLE
SOLIDGENEXTDIR	Controls the SOLIDGEN thickness direction. The CENTER option will extrude the solid element in the positive and negative half thickness directions.	CENTER/ POSITIVE/ NEGATIVE	CENTER
SOLIDNODENORMTOL	Controls the SOLIDGEN normal tolerance in degrees. Adjacent shell elements will generate a connected solid element when their normals are within this tolerance.	0.0 ≤ Real ≤ 90.0	60.0

Results Processor Parameters (Continued):

Parameter	Description	Type	Default
SKINGEN	Automated Surface Skin Generation (ASSG). Generates non-structural surface skin elements used in stress and fatigue analysis. A value between 0 and 4 defines the method used to calculate element corner results on a solid element mesh surface. There are five options: 0 – Automated surface skin generation is disabled. 1 – Surface skin elements and results are generated on the solid element mesh surface. No changes are made to the connected solid element corner results. 2 – Surface skin elements and results are generated on the solid element mesh surface. Connected solid element corner stress and strain values are replaced with corresponding skin element values regardless of magnitude. 3 – Surface skin elements and results are generated on the solid element mesh surface. Connected solid element corner stress and strain values are replaced with corresponding skin element values if the magnitude of the skin element component is larger. 4 – Surface skin elements and results are generated on the solid element mesh surface. Connected solid element corner stress and strain values are averaged with corresponding skin element values. The character variables: DISABLE, SURFACE, HYBRIDX, HYBRIDM, and HYBRIDA may be used in place of the numerical options 0 through 4.	$0 \le$ Integer ≤ 4 DISABLE/ SURFACE/ HYBRIDX/ HYBRIDM/ HYBRIDA	0
STRENGTHRATIO	Controls the output of Tsai Strength Ratio, which is provided in place of Failure Index for composite element ply results output. When set to OFF, the standard NASTRAN Failure Index is output. When set to ON, the Tsai Strength Ratio is calculated. Strength Ratio is considered more useful than Failure Index because it indicates exactly how to change applied loading to achieve optimal ply performance (strength ratio equal to 1.0).	ON/OFF	OFF
STRESSERROR	Controls the output of normalized grid point stress error (mesh convergence error). When set to ON, stress error at each grid point is calculated using $$e_i = \left[\frac{1}{N}\sum_{n=1}^{N}\left(\sigma_i^n - \overline{\sigma}_i\right)^2\right]^{\frac{1}{2}}$$ where N is the number of shell or solid elements attached to the node. σ_i is the von Mises stress predicted by element n at grid point i. $\overline{\sigma}_i$ is the mean von Mises stress at grid point i. The normalized error output is generated using e_i and a relative stress error based on element volume.	ON/OFF	ON
TSAI2LARC02	When set to ON, will use the LaRC02 failure theory (LARC02) when the Tsai-Wu (TSAI) failure theory is specified in the FT field of the PCOMP Bulk Data entry.	ON/OFF	OFF

Results Processor Parameters (Continued):

Parameter	Description	Type	Default
TSAI2MCT	When set to ON, will use the MCT failure theory (MCT) when the Tsai-Wu (TSAI) failure theory is specified in the FT field of the PCOMP Bulk Data entry. Also the ON setting will automatically convert MAT8 Bulk Data entries to MATL8 by analyzing the MAT8 material properties and comparing to known values for carbon, glass, and Kevlar fibers in an epoxy matrix. Additionally MATL12 Bulk Data entries are converted by analyzing the MAT12 material properties. Unidirectional lamina with fiber volume fractions of approximately 0.6 and 0.52 respectively and plain weave fabrics with bundle volume fractions of approximately 0.373 are supported. Other fiber and bundle volume fractions may be specified using TSAI2MCTFVF and TSAI2MCTBVF. See TSAI2MCTFVF and TSAI2MCTBVF below and the MATL8 and MATL12 Bulk Data entries in Section 4, *Bulk Data,* for additional information.	ON/OFF CARBON/ GLASS/ KEVLAR	OFF
TSAI2MCTBVF	Bundle volume fraction for plain weave lamina used to automatically convert MAT8 Bulk Data entries to MATL8 when TSAI2MCT is set to ON. The AUTO setting will use 0.373 for carbon, glass, and Kevlar fibers.	$0.2 \leq$ Real ≤ 0.38 AUTO	AUTO
TSAI2MCTFVF	Fiber volume fraction for unidirectional lamina used to automatically convert MAT8 Bulk Data entries to MATL8 when TSAI2MCT is set to ON. The AUTO setting will use 0.6 for carbon and Kevlar fibers and 0.52 for glass.	$0.3 \leq$ Real ≤ 0.9 AUTO	AUTO
UNITS	Defines the model units system for output labeling and report generation. The format is D-F-H-T where D is the distance specifier, F is the force specifier, H is the heat specifier, and T is the time specifier. The following options are permitted: Distance: MM, CM, M, IN, or FT Force: KGF, TONF, N, KN, LBF, or KIPS Heat: CAL, KCAL, J, BTU, or KJ Time: SEC, MIN, or HR	D-F-H-T	Undefined

Topology Design Optimization Processor Parameters:

Parameter	Description	Type	Default
MAXITERRESTART	Topology design optimization control of initial iterative solver tolerance and maximum number tolerance changes. The iterative solver tolerance starts at MAXITERRESTART orders of magnitude larger than SPARSEITERTOL. When EPSILON is larger than 1.0E-5, an E5131 warning message will be issued and the iterative solver tolerance decrease by one order of magnitude . Adjustments to the tolerance will stop when the tolerance equals SPARSEITERTOL.	1 ≤ Integer ≤ 10	5
MAXTOPTITER	Topology design optimization maximum number of convergence iterations permitted. The solver will iterate until the convergence factor set by TOPTITERTOL is reached or MAXOPTITER iterations have been performed. A zero setting will result in iteration until convergence is reached.	Integer ≥ 0	200
NTOPTLPSLEVEL	Number of topology optimization smoothing levels. A value between 0 and 5 is recommended.	0 < Real ≤ 10	0
NTOPTSTRESSDIV	Topology design optimization number stress divisions. Applicable when a stress constraint is specified. A larger value will produce a more accurate result with a reduction in performance. A value between 5 and 10 is recommended.	1 ≤ Integer ≤ 100	200
TOPTACITERSOLTOL	Topology optimization sparse iterative solver convergence factor for adjoint operations. When solving for adjoint displacements the iterative solver will iterate until the convergence factor set by TOPTACITERSOLTOL is reached or MAXSPARSEITER iterations have been performed and at least MINSPARSEITER iterations have been performed. The AUTO setting uses the SPARSEITERTOL setting.	0.0 < Real ≤ 1.0 AUTO	AUTO
TOPTACTHRESHOLD	Applicable to topology optimization stress constraints. If element stress is less than the the stress constraint limit scaled by the topology optimization activation threshold, then element adjacency calculations are skipped. AUTO adjusts the threshold value dynamically throughout the model.	0.0 < Real ≤ 1.0 AUTO	AUTO
TOPTALMMETHOD	Topology design optimization boundary threshold used to export a Nastran Bulk Data file of the final optimized design. Elements with densities below this value will not be exported along with their associated grid points. Additive manufacturing constraint method for topology design optimization. There are three options: 1 - The constraint is strictly imposed using max and min functions. The sensitivity is not smooth. The optimum design tends not to produce overhang supporting structures. 2 - The constraint is approximately imposed by an exponential function. The approximation approaches the true max and min functions with a large TOPTMAXALPHA. 3 - The constraint is approximately imposed by a projection method. The optimum design tends to produce more overhang structures.	1 ≤ Integer ≤ 3	1
TOPTALMSPSFACT	Topology design optimization additive manufacturing constraint support scale factor. Applicable when TOPTALMMETHOD is set to 3. Controls the amount of support material generated. A larger value will result in higher density supports and smaller value will result in lower density supports.	0 ≤ Real ≤ 1.0	0.5

Topology Design Optimization Processor Parameters (Continued):

Parameter	Description	Type	Default
TOPTBTHRESHOLD	Topology design optimization boundary threshold used to export a Nastran Bulk Data file of the final optimized design. Elements with densities below this value will not be exported along with their associated grid points.	Real	0.5
TOPTCOMPINDEX	Topology design optimization compliance index design constraint value. Applicable only when TOPTGEN is set to 2, 3, or 4.	Real > 1.0	1.0E+10
TOPTDATABASE	Controls the storage and retrieval of topology design optimization density data. The default value DELETE purges all element density data when the program terminates normally. When set to STORE, the converged optimized design is stored in a single file with the same base name as the Model Results Output File and a *.ODB* file extension. When set to FETCH, the optimized design specified by the TOPTDATFILE directive is retrieved and used as the starting point for the subsequent topology design optimization solution sequence. When set to UPDATE, the optimized design data will be retrieved and stored.	DELETE FETCH STORE UPDATE	DELETE
TOPTDEPXITER	Topology Design Optimization skip factor for interprocess communication dependent models. Dependent models defined with TOPTDEPXFILE will be run every TOPTDEPXITER topology optimization iterations of the independent model.	1 ≤ Integer ≤ 99999999 AUTO	AUTO
TOPTDESIGNCONSTR	Topology design optimization design constraint value based on the TOPTGEN setting. See TOPTGEN.	Real > 0.0	1.0E+10
TOPTDESIGNMODE	When SPEED is selected, settings are adjusted to give best possible performance at the cost of accuracy. Applicable only in static solutions when minimizing compliance with volume fraction constraints, or in modal solutions when maximizing frequency with volume fraction constraints.	ACCURACY/ SPEED	ACCURACY
TOPTDESIGNREGION	Topology design optimization design region property identification number.	Integer ≥ 0	1
TOPTDESIGNTOL	Topology design optimization MAM engine convergence factor used when the TOPTENGINE Model Initialization directive is set to MAM or AUTO and MAM is selected. For each design iteration the MAM engine will iterate until the convergence factor set by TOPTDESIGNTOL is reached or the internal iteration limit has been exceeded. Using a larger factor may improve performance but could also result in a less optimized design.	0 ≤ Real ≤ 1	AUTO
TOPTDTHRESHOLD	Topology design optimization density threshold used for removing elements from the optimization process. Elements with densities below this value will not be included in the solution for displacements, internal forces, or compliance calculations. This parameter is only applicable when TOPTSOLACCEL is set to ON.	0.0 ≤ Real ≤ 0.1	
TOPTELEMEXTTOL	Near tolerance used to identify elements which are aligned in the extrude direction specified on the TOPVAR Bulk Data entry. The actual tolerance is derived using TOPTELEMEXTTOL and an element reference dimension.	Real	1.0E-2
TOPTELEMSYMTOL	Near tolerance used to identify elements which are symmetric with respect to the specified TOPVAR Bulk Data entry mirror symmetry plane. The actual tolerance is derived using TOPTELEMSYMTOL and an element reference dimension.	Real	1.0E-2

Topology Design Optimization Processor Parameters (Continued):

Parameter	Description	Type	Default
TOPTGEN	Automated Topology Optimization Generation (ATOG). A value between 0 and 8 defines one of the predefined topology design optimization solutions generated. There are eight options: 0 - Automated topology optimization generation is disabled. 1 - The objective is to minimize compliance, the design constraint is volume fraction (mass fraction), and the solution is linear statics. 2 - The objective is to minimize volume fraction (mass fraction), the design constraints are stress and compliance index, and the solution is linear statics. 3 - The objective is to minimize volume fraction (mass fraction), the design constraint is maximum displacement and compliance index, and the solution is linear statics. 4 - The objective is to minimize volume fraction (mass fraction), the design constraint is maximum singe point constraint force and compliance index, and the solution is linear statics. 5 - The objective is to minimize volume fraction (mass fraction), the design constraint is compliance index, and the solution is linear statics. 6 - The objective is to minimize volume fraction (mass fraction), the design constraint is modal frequency, and the solution is normal modes. The character variables: DISABLE, COMPVF, VFSTRESS, VFDISP, VFSPCF, VFCOMP, AND VFFREQ may be used in place of the numerical options 0 through 6. All objectives are minimized and all constraints are upper limits, except for modal frequency which is a lower frequency limit. Multiple subcases are supported and applicable to all constraints. See also TOPTMANCONSTR, TOPTDESIGNCONSTR, and TOPTCOMPINDEX.	0 ≤ Integer ≤ 6 DISABLE/ COMPVF/ VFSTRESS/ VFDISP/ VFSPCF/ VFCOMP/ VFFREQ	0
TOPTGLBDMETHOD	Global displacement constraint method for topology design optimization. There are three options: 1 - The maximum displacement is approximated by P-norm of all displacements in a region. The global displacement value can be different from the maximum displacement. 2 - The maximum displacement is approximated by an exponential function of all displacements in a region. The global displacement value can be different from the maximum displacement. 3 - The maximum displacement is used as it is without approximation. The grid of maximum displacement might change during optimization iteration. The global displacement value is identical to the maximum displacement.	1 ≤ Integer ≤ 3	3
TOPTGLBSMETHOD	Global stress constraint method for topology design optimization. There are three options: 1 - The maximum von Mises stress is approximated by P-norm of all stresses in a region. The global stress value can be different from the maximum von Mises stress. 2 - The maximum von Mises stress is approximated by an exponential function of all stresses in a region. The global stress value can be different from the maximum von Mises stress. 3 - Same as option 1, but the global stress is scaled to match with the maximum von Mises stress. The global stress value is identical to the maximum von Mises stress.	1 ≤ Integer ≤ 3	3

Topology Design Optimization Processor Parameters (Continued):

Parameter	Description	Type	Default
TOPTITERSOLMODE	Topology design optimization control of iterative solver initial starting displacement vector and solver mode. The auto setting will use option 1. There are four options: 1 - Use block right-hand-side solver mode and a zero starting displacement vector. 2 - Use block right-hand-side solver mode and a non-zero starting displacement vector. 3 - Use sequential solver mode and a non-zero starting displacement vector. 4 - Use sequential solver mode and a zero starting displacement vector. **Note:** The block right-hand-side solver mode is disabled regardless of the TOPTITERSOLMODE setting if the number of adjoint load cases is less than two.	1 ≤ Integer ≤ 4 AUTO	AUTO
TOPTITERTOL	Topology design optimization Iterative solver convergence factor. The topology optimization solver will iterate until the convergence factor set by TOPTITERTOL is reached or MAXTOPTITER iterations have been performed	Real	5.0E-3
TOPTMANCONSTR	Defines the type of manufacturing constraint to be used in Automated Topology Optimization Generation (ATOG). There are three options: 0 - No manufacturing constraint is specified. 1 - Symmetry using either 1, 2, or 3 planes of symmetry is specified. 2 - Extrude design constraint is specified. 3 - Additive layer manufacturing design constraint is specified. 4 - 3-axis milling manufacturing design constraint is specified. The character variables: DISABLE, SYM, EXT, and ALM may be used in place of the numerical options 0 through 3. See also TOPTMANDIR and TOPTMANCORD.	0 ≤ Integer ≤ 3 DISABLE/ SYM/ EXT/ ALM/ MILL	0
TOPTMANCORD	Specifies the topology design optimization manufacturing constraint coordinate system corresponding to TOPTMANCONSTR. See also TOPTMANDIR.	Integer > 0	0
TOPTMANDIR	Specifies the topology design optimization manufacturing constraint symmetry plane(s), extrude direction, or print direction depending on the TOPTMANCONSTR value specified.	+X/+Y/+Z/ -X/-Y/-Z/ XY/YZ/ZX/ XYYZ/YZZX/ XYZX/ XYYZZX	XY
TOPTMAXACTDIST	Topology design optimization maximum distance for identifying adjacent elements. Elements within distance TOPTMAXACTDIST are used for sensitivity filtering. The default AUTO setting is recommended since large values may result in slower performance and undesired results.	Real AUTO	AUTO
TOPTMAXALPHA	The maximum value of the exponential quotient for additive manufacturing method (TOPTALMMETHOD = 2). The exponential quotient increases gradually during optimization iteration up to TOPTMAXALPHA. AUTO option sets TOPTMAXALPHA = 6.0. Values greater than 12.0 are not recommended and may result in that the exponential function returning an NaN error.	Real > 0.0 AUTO	AUTO

Topology Design Optimization Processor Parameters (Continued):

Parameter	Description	Type	Default
TOPTMAXBETA	Topology design optimization maximum minimum member size manufacturing constraint penalty value. A value between 1.0 and 16.0 is recommended. The AUTO setting will select the best value based on the TOPTENGINE used and the constraints defined in the model.	Real > 0.0 AUTO	AUTO
TOPTMAXDELTAOBJ	Topology design optimization maximum permitted delta objective. Applicable only when using an adjustable volume fraction and when the solution has trouble converging. The auto setting uses a value of 2.0. The delta objective is computed using: $$\delta_{Obj} = \frac{Obj_i - Obj_{min}}{Obj_{min}}$$ where Obj_i is the current objective Obj_{min} is the minimum objective, which is updated when the objective gradient changes sign	Real > 0.0 AUTO	AUTO
TOPTMAXGRAYSCALE	Topology optimization maximum permitted gray scale. Applicable only when using an adjustable volume fraction and when the solution has trouble converging. The auto setting uses a value of 2.0. The gray scale is computed using: $$\gamma = \frac{\eta_d}{\eta_g}$$ where η_d is the number of elements in the design space between 0.2 and TOPTBTHRESHOLD η_g is the number of elements in the design space greater than TOPTBTHRESHOLD	Real > 0.0 AUTO	AUTO
TOPTMAXPNORMEXP	Exponent of p-norm for global displacement and global temperature.	1.0 < Real ≤ 100.0	10.0
TOPTMINADJVF	Topology design optimization lower bound for allowed adjustable volume fraction.	0 ≤ Real ≤ 1.0	5.00E-02
TOPTSOLACCEL	Topology design optimization solution acceleration option. When set to ON, low density elements defined as less than TOPTDTHRESHOLD are removed and solution times will decrease. It is recommended to only use this option with the PCGLSS solver.	ON/OFF	OFF
TOPTSTRESSTOL	Decimal value representing the percent of high-density elements with the highest stress to be ignored when evaluating whether a stress constraint is within limits.	0 ≤ Real ≤ 1	4.00E-03
TOPTSTRESSTYPE	Topology Optimization stress type option. A value of 1 will set VonMises stress equal to the equivalent stress (square root of density x VM stress). A value of 0 will keep VM stress unchanged. The AUTO setting will use 1 when TOPTSOLACCEL is ON and 0 when it is OFF	0 ≤ Integer≤ 1 AUTO	1
TOPTTDMAXEPSILON	Affects size of members when maximum member size manufacturing constraints are used.	1.0E-3 ≤ Real ≤ 1.0	0.5

Nastran Binary Results File Geometry Data Block Definition Table:

POST							Geometry Data Block	Description
-1	-2	-4	-6	-7	-8	-9/10		
NO	YES	YES	YES	YES	YES	YES	CSTM	Coordinate System Transformation Matrixes
YES	NO	NO	NO	NO	YES	YES	GEOM1	Grid Point Definitions
YES	YES	YES	NO	NO	YES	YES	GEOM2	Element Definitions
YES	NO	NO	NO	YES	YES	YES	EPT	Element Properties
YES	NO	NO	NO	YES	YES	YES	MPT	Material Properties
NO	NO	NO	NO	YES	YES	YES	CASECC	Case Control information
NO	NO	NO	NO	YES	YES	YES	BGPDT	Basic Grid Point Definition Table
NO	YES	YES	YES	YES	YES	YES	GPL	Grid Point List
NO	YES	YES	YES	NO	YES	YES	GPDT	Grid Point Definitions
NO	NO	NO	NO	YES	YES	YES	GEOM2S	Element Definitions (superelements)

Nastran Binary Results File Results Data Block Definition Table:

POST							Results Data Block	Description
-1	-2	-4	-6	-7	-8	-9/10		
YES	YES	NO	YES	YES	YES	YES	OUGV1	Displacements
YES	YES	NO	YES	YES	YES	YES	OUPV1	Velocities and accelerations
YES	YES	NO	YES	YES	YES	YES	OPG1	Applied loads
YES	YES	NO	YES	YES	YES	YES	OQG1	Single constraint forces
YES	YES	NO	YES	NO	YES	YES	OQMG1	Multipoint constraint forces
YES	YES	NO	YES	YES	YES	YES	OES1	Element stresses
YES	YES	NO	YES	NO	YES	YES	OES1C	Composite element stresses
YES	YES	NO	YES	YES	YES	YES	ONRGY1	Element strain energy and energy densities
YES	YES	NO	YES	YES	YES	YES	OGPFB1	Grid point forces
YES	YES	NO	YES	YES	YES	YES	OSTR1	Element strains
YES	YES	NO	YES	NO	YES	YES	OSTR1C	Composite element strains
YES	YES	NO	YES	NO	YES	YES	OEFIT	Composite element failure indices
YES	YES	NO	YES	YES	YES	YES	OEF1X	Element forces and heat fluxes

Nastran Binary Results File Modeler Compatibility Table:

POST	Modeler
-1	MSC Patran
-2	UGS/Siemens I-DEAS
-4	LMS International Virtual Lab
-6	UGS/Siemens Unigraphics
-7	TMP Vision
-8	Anaglyph Laminate Tools

Model Parameter/Solution Applicability Matrix:

Parameter	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
ACBINTERACTTOL												✓	✓	✓	✓								
ACBPRESSET												✓	✓	✓	✓								
ACBREFPRES												✓	✓	✓	✓								
ACBVC												✓	✓	✓	✓								
ADAPTTIMESTEP																✓							
ADDNLTOQUADLOAD			✓						✓						✓				✓				
ADDPRESTRESS		✓				✓	✓											✓	✓				
ALIGNEDGENODE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
ALPHA					✓			✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
ALTFAILINDEXFORM	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
AUTOBPD				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓				
AUTOFIXELEMGEOM	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
AUTOFIXRIGIGELEM	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
AUTOFIXRIGIGSPC	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
AUTOSPC	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
BARDKMETHOD		✓	✓			✓	✓	✓	✓	✓	✓			✓	✓			✓	✓	✓			
BAREQVLOAD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓				
BETA					✓			✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
BISECT			✓				✓		✓		✓				✓				✓			✓	

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	Solution: 101 Linear Static	181 Prestress Static	106 Nonlinear Static	103 Modal	110 Modal Complex Eigenvalue	182 Linear Prestress Modal	185 Nonlinear Prestress Modal	188 Linear Prestress Complex Eigenvalue	189 Nonlinear Prestress Complex Eigenvalue	105 Linear Buckling	180 Nonlinear Buckling	108 Direct Frequency Response	111 Modal Frequency Response	183 Linear Prestress Frequency Response	186 Nonlinear Prestress Frequency Response	109 Direct Transient Response	112 Modal Transient Response	184 Linear Prestress Transient Response	187 Nonlinear Prestress Transient Response	129 Nonlinear Transient Response	101 Linear Steady State Heat Transfer	153 Nonlinear Steady State Heat Transfer	159 Nonlinear Transient Heat Transfer
CB1, CB2					✓			✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
CHECKRUN	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
CLOSE				✓																			
CK1, CK2	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
CM1, CM2	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
COMPE1RSF			✓				✓		✓		✓				✓				✓	✓			
COMPE1RSFTID			✓				✓		✓		✓				✓				✓	✓			
COMPE2RSF			✓				✓		✓		✓				✓				✓	✓			
COMPE2RSFTID			✓				✓		✓		✓				✓				✓	✓			
COMPE3RSF			✓				✓		✓		✓				✓				✓	✓			
COMPE3RSFTID			✓				✓		✓		✓				✓				✓	✓			
COMPG12RSF			✓				✓		✓		✓				✓				✓	✓			
COMPG12RSFTID			✓				✓		✓		✓				✓				✓	✓			
COMPG1ZRSF			✓				✓		✓		✓				✓				✓	✓			
COMPG1ZRSFTID			✓				✓		✓		✓				✓				✓	✓			
COMPG23RSF			✓				✓		✓		✓				✓				✓	✓			
COMPG23RSFTID			✓				✓		✓		✓				✓				✓	✓			
COMPG2ZRSF			✓				✓		✓		✓				✓				✓	✓			

Model Parameter/Solution Applicability Matrix (Continued):

	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
Parameter	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
COMPG2ZRSFTID			✓				✓		✓		✓				✓				✓	✓			
COMPG31RSF			✓				✓		✓		✓				✓				✓	✓			
COMPG31RSFTID			✓				✓		✓		✓				✓				✓	✓			
COMPK1, COMPK2	✓	✓	✓	✓		✓	✓			✓	✓						✓	✓	✓	✓			
COMPILSMETHOD	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
COMPRSLTOUT	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
CONTACTGEN	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
CONTACTSTAB	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
CONTACTTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
CONVMATRIX																					✓	✓	✓
COUPMASS				✓	✓	✓	✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
CP1, CP2	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
CYSYMGEN	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
CYSYMTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
DATABASEACCEL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
DDAMPHASE				✓																			
DFREQ												✓	✓	✓	✓								
DIRSTRESSTYPE	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
DISPGEOMSFACT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
DMILABEL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
DMIPDIAG	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
DYNLMDIRECTDIF																✓	✓	✓	✓				
DYNRESPEIGVOUT													✓	✓	✓		✓	✓	✓				
DYNSOLACCEL													✓	✓	✓		✓	✓	✓				
DYNSOLDIRECTINT																✓	✓	✓	✓	✓			
DYNSOLRELGRID												✓	✓	✓	✓	✓	✓	✓	✓				
EIGENFLEXFREQ				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
EIGENSHIFTSFACT				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
EIGENSOLACCEL				✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓		✓	✓	✓				
EDGENODETOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
ELEMGEOMCHECKS	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
ELEMGEOMFATAL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
ELEMGEOMOUT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
ELEMRSLTCORD	✓	✓	✓	✓		✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
ELEMRSLTMAXTYPE	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
EMODES																				✓			
ENHCBARRSLT	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
ENHCQUADRSLT	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
EPSILONFLOAT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
EPZERO	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
EQVSTRESSTYPE	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
FACTDIAG	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
FACTRATIOTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
FIXNLTOQUAD			✓				✓		✓		✓									✓			
FLOATINZERO	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
FLOATOUTZERO	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
FREQRESPRSLTOUT												✓	✓	✓	✓								
FREQRESPRSLTINCR												✓	✓	✓	✓								
G					✓			✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
GPFORCEMETHOD	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
GPSTRESS	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
GRDPNT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
GRIDCOLTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
GRIDTEMPASGN	✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
GRIDTEMPAVE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HEXARTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	101 Linear Static	181 Prestress Static	106 Nonlinear Static	103 Modal	110 Modal Complex Eigenvalue	182 Linear Prestress Modal	185 Nonlinear Prestress Modal	188 Linear Prestress Complex Eigenvalue	189 Nonlinear Prestress Complex Eigenvalue	105 Linear Buckling	180 Nonlinear Buckling	108 Direct Frequency Response	111 Modal Frequency Response	183 Linear Prestress Frequency Response	186 Nonlinear Prestress Frequency Response	109 Direct Transient Response	112 Modal Transient Response	184 Linear Prestress Transient Response	187 Nonlinear Prestress Transient Response	129 Nonlinear Transient Response	101 Linear Steady State Heat Transfer	153 Nonlinear Steady State Heat Transfer	159 Nonlinear Transient Heat Transfer
HEXENODE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HEXFACEMAXIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HEXFACEMINIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HEXFACESKEWTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HEXFACEWARPTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HEXINODE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
HEXMAXEPADTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HEXMINEPLRTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HEXREDORD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HFREQ				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
HPNLMATREDORD			✓				✓				✓				✓				✓	✓			
HPNLMATSFACT			✓				✓				✓				✓				✓	✓			
INITSTRNSFACT			✓				✓		✓		✓				✓				✓	✓			
INREL	✓																						
KRIGIDELEM	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
J4ROT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
HEXARTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
HEXENODE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
K6ROT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			

Model Parameter/Solution Applicability Matrix (Continued):

	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
Parameter	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
KDAMP					✓			✓	✓			✓											
LANCZOSVECT				✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓		✓	✓	✓				
LANGLE			✓				✓		✓		✓				✓				✓	✓			
LARC02TSAITOL	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
LFREQ				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
LGDISP			✓				✓				✓				✓				✓	✓			
LINEARCONTACT	✓																						
LMODES				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
LNCONTACTITERTOL	✓																						
M6ROT				✓	✓	✓	✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
MAXBISECTRESTART			✓				✓		✓		✓				✓				✓				
MAXEIGENRESTART				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
MAXELEMGEOMMSG	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
MAXIMPACTSTEP																				✓			
MAXLNCONTACTITER	✓																						
MAXTOPTITER	✓																						
MAXRATIO	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
MAXSPARSEITER	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
MAXSRITER	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
MECHSTRAIN	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
MINSPARSEITER	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	
MODALDATABASE				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
MODEPFACTOR				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
MODFSPCSTORE				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
MODEVAROUT													✓	✓	✓		✓	✓	✓				
NCBMODE	✓			✓									✓				✓						
NCONTACTGEOMITER	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
NITERCUPDATE			✓				✓		✓		✓				✓				✓	✓			
NITERKSUPDATE			✓				✓		✓		✓				✓				✓				
NITERMUPDATE			✓				✓		✓		✓				✓				✓	✓			
NITERPFUPDATE			✓				✓		✓		✓				✓				✓	✓			
NLAYERS			✓				✓		✓		✓				✓				✓	✓			
NLINDATABASE							✓		✓		✓				✓				✓				
NLINSOLACCEL			✓				✓		✓		✓				✓				✓			✓	✓
NLKDIAGAFACT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
NLKDIAGCOMP	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
NLKDIAGMINAFACT			✓	✓			✓		✓		✓				✓								
NLKDIAGSET	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
NLLSSTRAINTYPE			✓				✓				✓				✓				✓	✓			

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
NLLSSTRESSTYPE			✓				✓				✓				✓				✓	✓			
NLMATSFACT			✓				✓				✓				✓				✓	✓			
NLMATTABLGEN			✓				✓				✓				✓				✓	✓			
NLTOL			✓				✓		✓		✓				✓				✓	✓			
NLTRUESTRESS			✓				✓		✓		✓				✓				✓	✓			
NLSUBCREINIT			✓				✓				✓				✓				✓			✓	
NOCOMPS	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
NSLDPLYINTPOINT	✓	✓		✓		✓				✓						✓	✓	✓					
OGEOM	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
OPTION				✓																			
OUTSETTOL			✓		✓		✓	✓	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
OUTZEROVECT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PARTGEOMOUT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PARTMASSOUT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PENTARTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PENTFACEMAXIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PENTFACEMINIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PENTFACESKEWTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PENTFACEWARPTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
PENTMAXEPADTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PENTMINEPLRTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PENTREDORD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
POST	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PRGPST	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PYRTARTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PYRFACEMAXIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PYRFACEMINIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PYRFACESKEWTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PYRFACEWARPTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PYRMAXEPADTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PYRMINEPLRTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
PYRREDORD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
QUADARTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
QUADELEMTYPE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
QUADEQVLOAD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
QUADINODE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
QUADMAXEPADTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
QUADMAXIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

Model Parameter/Solution Applicability Matrix (Continued):

	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
Parameter	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
QUADMINEPLRTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
QUADMINIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
QUADREDORD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
QUADRNODE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
QUADSKEWTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
QUADWARPLIMIT	✓	✓		✓	✓	✓		✓		✓		✓	✓	✓		✓	✓	✓					
QUADWARPTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
RADMATRIX																					✓	✓	✓
RANDRESPINVLEVEL												✓	✓	✓	✓								
RBCHECKLEVEL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
RBCHECKMODES	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
RESEQGRID	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
RESEQGRIDMETHOD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
RESVEC				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
RESVPGF				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
RIGIDBODYMODE				✓	✓	✓	✓	✓	✓				✓	✓	✓		✓	✓	✓				
RIGIDELEM2ELAS	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
ROTINERTIA	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
RSLTDATABASE	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
SLINEKAVG	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
SHEARELEMTYPE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
SHELLRNODE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
SHELLTVSMATTYPE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
SIGMA																					✓	✓	✓
SLINEKAVG	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEKSFACT	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEKSFACT2TC																					✓	✓	✓
SLINEMAXACTCORD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEMAXACTDIR	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEMAXACTDIST	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEMAXACTRATIO			✓				✓				✓				✓				✓	✓			
SLINEMAXACTWIDTH	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEMAXDISPTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEMAXPENDIST	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEOFFSETTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEPENTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEPLANEZDIR	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEPOSTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

Model Parameter/Solution Applicability Matrix (Continued):

	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
Parameter	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
SLINEPROTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINESLIDETYPE			✓				✓				✓				✓				✓	✓			
SLINESTABKSFACT			✓				✓				✓				✓				✓	✓			
SLINESTRESSLOC	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SLINEUNLOADTOL			✓				✓				✓				✓				✓	✓			
SOLUTIONERROR	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SORTMODEMASS				✓		✓	✓																
SPARSEITERMETHOD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓		✓	✓	✓		✓	✓	
SPARSEITERMODE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓		✓	✓	✓		✓	✓	✓		✓	✓	
SPARSEITERTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SPARSEMETHOD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
SPCGEN	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
STIFFZEROTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
STRENGTHRATIO	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
STRESSERROR	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
TABS																					✓	✓	✓
TETARTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TETFACEMAXIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TETFACEMINIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
TETFACESKEWTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TETMAXEPADTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TETMINEPLRTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TETREDORD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TOPTBTHRESHOLD	✓																						
TOPTELEMSYMTOL	✓																						
TOPTITERTOL	✓																						
TOPTMAXACTDIST	✓																						
TRIARTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TRIELEMTYPE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TRIEQVLOAD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
TRIMAXEPADTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TRIMAXIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TRIMINEPLRTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TRIMINIATOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TRIREDORD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
TRIRNODE	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓			
TRISKEWTOL	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓

Model Parameter/Solution Applicability Matrix (Continued):

Parameter	Solution																						
	101	181	106	103	110	182	185	188	189	105	180	108	111	183	186	109	112	184	187	129	101	153	159
	Linear Static	Prestress Static	Nonlinear Static	Modal	Modal Complex Eigenvalue	Linear Prestress Modal	Nonlinear Prestress Modal	Linear Prestress Complex Eigenvalue	Nonlinear Prestress Complex Eigenvalue	Linear Buckling	Nonlinear Buckling	Direct Frequency Response	Modal Frequency Response	Linear Prestress Frequency Response	Nonlinear Prestress Frequency Response	Direct Transient Response	Modal Transient Response	Linear Prestress Transient Response	Nonlinear Prestress Transient Response	Nonlinear Transient Response	Linear Steady State Heat Transfer	Nonlinear Steady State Heat Transfer	Nonlinear Transient Heat Transfer
TSAI2LARC02	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
TSAI2MCT	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
TSAI2MCTBVF	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
TSAI2MCTFVF	✓	✓	✓	✓		✓	✓			✓	✓					✓	✓	✓	✓	✓			
UNITS	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
UNRESEQGRID	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
USAWETSURFACE																✓							
VMOPT				✓	✓	✓	✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
WARNING	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
VFM2ACB												✓	✓	✓	✓								
VFMADDMETHOD				✓	✓	✓	✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
VFMINTERACTTOL				✓	✓	✓	✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
VFMNORMTOL				✓	✓	✓	✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
WTMASS				✓	✓	✓	✓	✓	✓			✓	✓	✓	✓	✓	✓	✓	✓	✓			
XDAMP													✓	✓	✓		✓	✓	✓				
ZONADATAOUT				✓		✓	✓																

RESULTS NEUTRAL FILE FORMAT

Results Neutral Files

The result neutral file system is the primary interface for graphical processing of model results data. The file system is also used for:

- Source of expanded model results output.
- Input file for results limits search via the RESULTLIMITS Case Control command.
- Input file for automated SET entry generation via the SETGENERATE Case Control command.

The results neutral file system consists of eight types of files, each generated by the Results Processor. A specific Model Initialization directive as shown below controls output of each type:

File Type	Model Initialization Directive	Default Neutral Filename
Grid Point Displacement Vector	DISPFILE = [d:] [path] filename[.ext]	*model output filename.DIS*
Grid Point Force Vector	FORCFILE = [d:] [path] filename[.ext]	*model output filename.GPF*
Element Internal Load Vector	LOADFILE = [d:] [path] filename[.ext]	*model output filename.ELF*
Element Results	ELEMFILE = [d:] [path] filename[.ext]	*model output filename.ELS*
Grid Point Results	GRIDFILE = [d:] [path] filename[.ext]	*model output filename.GPS*
Femap Results	Defined by DISPFILE	*model output filename.NEU* *model output filename.FNO*
NASTRAN Binary Results	Defined by DISPFILE	*model output filename.OP2*
NASTRAN XDB Results	Defined by DISPFILE	*model output filename.XDB*
NASTRAN ASCII Results	Defined by MODLOUTFILE	*model output filename.PCH*
MS Excel ASCII Results	Defined by MODLOUTFILE	*model output filename.CSV*

The DISPFILE, FORCFILE, LOADFILE, ELEMFILE, and GRIDFILE directives control the filenames and whether a file is to be generated. Setting a specific directive equal to the character variable NONE will disable output of that neutral file type.

Another useful Model Initialization directive is RSLTFILETYPE which controls file type and format. When RSLTFILETYPE is set to FEMAPASCII or FEMAPBINARY, a single Femap® compatible results neutral file of the entire results database is generated. When RSLTFILETYPE is set to PATRANASCII or PATRANBINARY, multiple PATRAN 2.5 compatible results neutral files are generated. PATRAN results neutral files have a two digit subcase number added to the base of the filename to facilitate multiple subcases. When RSLTFILETYPE is set to NASTRANBINARY, a single NASTRAN Output 2 compatible results file of the entire results database is generated.

When RSLTFILETYPE is set to FEMAPBINARY and the INRCRSLTOUT directive is set to ON, a separate Femap binary results neutral file will be generated for each load increment or time step. At the end of the analysis a single neutral file with all steps will be generated.

For a detailed description of each directive see Section 2, *Initialization*.

Results Neutral File Descriptions

<u>Grid Point Displacement Vector</u>

The Grid Point Displacement Vector Neutral File contains the calculated displacement vector at each grid point in the basic coordinate system. There are six columns where the first three are the x, y, and z components of translation and the last 3 are the x, y, and z components of rotation.

The ASCII formatted version has the following structure:

Record 1:	TITLE	(A80)
Record 2:	NGRID, MAXGID, MAXDISP, MAXDISPGID, NDISPVECTCOL	(2I8, E15.6, 2I8)
Record 3:	SUBTITLE	(A80)
Record 4:	LABEL	(A80)
Record 5 to NGRID+4 :	GRIDID, (DISPVECT(COL), COL=1, NDISPVECTCOL)	(I8, (5E13.7))

The binary unformatted version has the following structure:

Record 1:	TITLE, NGRID, MAXGID, MAXDISP, MAXDISPGID, NDISPVECTCOL
Record 2:	SUBTITLE
Record 3:	LABEL
Record 4 to NGRID+3 :	GRIDID, (DISPVECT(COL), COL=1, NDISPVECTCOL)

Where

TITLE	The set title	CHAR80
SUBTITLE	The set subtitle	CHAR80
LABEL	The set label	CHAR80
NGRID	Number of grid points	INT4
MAXGID	Largest grid point ID	INT4
MAXDISP	Maximum absolute displacement	REAL4
MAXDISPGID	Grid point ID where the maximum displacement occurs	INT4
NDISPVECTCOL	The number of displacement vector components or columns (6)	INT4
GRIDID	Grid point ID number	INT4
DISPVECT	Displacement vector component values at GRIDID	REAL4

Grid Point Force Vector

The Grid Point Force Vector Neutral File contains the calculated internal, applied and reacted force vector at each grid point in the basic coordinate system. The internal force vector is the resultant of all internal forces at the grid point. For transient response solutions, acceleration and velocity is also included in this file.

The ASCII formatted version has the following structure:

Record 1:	TITLE	(A80)
Record 2:	NGRID, MAXGID, MAXVECT, MAXVECTGID, NFORCVECTCOL	(2I8, E15.6, 2I8)
Record 3:	SUBTITLE	(A80)
Record 4:	LABEL	(A80)
Record 5 to NGRID+4 :	GRIDID, (FORCVECT(COL), COL=1, NFORCVECTCOL)	(I8, (5E13.7))

The binary unformatted version has the following structure:

Record 1:	TITLE, NGRID, MAXGID, MAXVECT, MAXVECTGID, NFORCVECTCOL
Record 2:	SUBTITLE
Record 3:	LABEL
Record 4 to NGRID+3 :	GRIDID, (FORCVECT(COL), COL=1, NFORCVECTCOL)

Where

TITLE	Set title	CHAR80
SUBTITLE	Set subtitle	CHAR80
LABEL	Set label	CHAR80

NGRID	Number of grid points	INT4
MAXGID	Largest grid point ID	INT4
MAXVECT	Maximum absolute vector value	REAL4
MAXVECTGID	Grid point ID where the maximum value occurs	INT4
NFORCVECTCOL	The number of force vector components or columns	INT4
GRIDID	Grid point ID number	INT4
FORCVECT	Force vector component values at GRIDID (internal force, applied force, SPC force, MPC force, velocity, and acceleration)	REAL4

Element Internal Load Vector

The Element Internal Load Vector Neutral File contains the calculated element internal forces at each node in the basic coordinate system.

The binary unformatted version has the following structure:

Record 1: TITLE, NLOADVECTCOL
Record 2: SUBTITLE
Record 3: LABEL
Record 4 to NELEM+3 : ELEMID, ELEMTYPE, (LOADVECT(COL), COL=1, NLOADVECTCOL)

Where

TITLE	Set title	CHAR80
SUBTITLE	Set subtitle	CHAR80
LABEL	Set label	CHAR80
NLOADVECTCOL	The number of load vector components or columns	INT4
ELEMID	Element ID number	INT4
ELEMTYPE	Element type code	INT4
LOADVECT	Load vector component values at ELEMID	REAL4

Element Results

The Element Results Neutral File contains various result types calculated at requested points on the element in a user-specified coordinate system. The coordinate system for shell element results is specified using the Case Control command SURFACE and solid element results using the Case Control command VOLUME (see SURFACE and VOLUME in Section 3, *Case Control*). Shell and solid elements that do not have a coordinate system defined via a SURFACE or VOLUME command will not be included. The default SURFACE is all shell elements in the element coordinate system. The default VOLUME is all solid elements in the element coordinate system. The default SURFACE/VOLUME coordinate system can be changed using the ELEMRSLTCORD parameter (see ELEMRSLTCORD in Section 5, *Parameters*). Composite shell element ply results will not be included in PARAM, NOCOMPS, -1 is included in the Model Input File (see NOCOMPS in Section 5, *Parameters*).
The ASCII formatted version has the following structure:

Record 1:	TITLE	(A80)
Record 2:	NELEMVECTCOL	(I8)
Record 3:	SUBTITLE	(A80)
Record 4:	LABEL	(A80)
Record 5 to NELEM+4 :	ELEMID, ELEMTYPE, (ELEMVECT(COL), COL=1, NELEMVECTCOL)	(2I8, /, (6E13.7))

The uncompressed binary unformatted version (RSLTFILECOMP directive set to OFF) has the following structure:

Record 1: TITLE, NELEMVECTCOL
Record 2: SUBTITLE
Record 3: LABEL
Record 4 to NELEM+3 : ELEMID, ELEMTYPE, (ELEMVECT(COL), COL=1, NELEMVECTCOL)

The compressed binary unformatted version (RSLTFILECOMP directive set to ON) has the following structure:

Record 1:	TITLE, NELEMVECTCOL
Record 2:	SUBTITLE
Record 3:	LABEL
Record 4 to NELEM+3 :	ELEMID, ELEMTYPE, NCOL, (ELEMVECTP(COL), COL=1, NCOL), (ELEMVECTC(COL), COL=1, NCOL)

Where

TITLE	Set title	CHAR80
SUBTITLE	Set subtitle	CHAR80
LABEL	Set label	CHAR80
NELEMVECTCOL	The number of load vector components or columns	INT4
ELEMID	Element ID number	INT4
ELEMTYPE	Element type code	INT4
ELEMVECT	Element vector component values at ELEMID	REAL4
NCOL	The number of non-zero element vector component values at ELEMID	INT4
ELEMVECTP	Non-zero element vector component value locations at ELEMID	INT4
ELEMVECTC	Non-zero element vector component values at ELEMID	REAL4

<u>Grid Point Results</u>

The Grid Point Results Neutral File contains various result types calculated at the grid points in a user-specified coordinate system. The coordinate system for shell element results is specified using the Case Control command SURFACE and solid element results using the Case Control command VOLUME (see SURFACE and VOLUME in Section 3, *Case Control*). Grid points connected to shell and solid elements that do not have a coordinate system defined via a SURFACE or VOLUME command will not be included.

The ASCII formatted version has the following structure:

Record 1:	TITLE	(A80)
Record 2:	NGRID, MAXGID, MAXVECT, MAXVECTGID, NGRIDVECTCOL	(2I8, E15.6, 2I8)
Record 3:	SUBTITLE	(A80)
Record 4:	LABEL	(A80)
Record 5 to NGRID+4 :	GRIDID, (GRIDVECT(COL), COL=1, NGRIDVECTCOL)	(I8, (5E13.7))

The binary unformatted version has the following structure:

Record 1:	TITLE, NGRID, MAXGID, MAXVECT, MAXVECTGID, NGRIDVECTCOL
Record 2:	SUBTITLE
Record 3:	LABEL
Record 4 to NGRID+3 :	GRIDID, (GRIDVECT(COL), COL=1, NGRIDVECTCOL)

Where

TITLE	Set title	CHAR80
SUBTITLE	Set subtitle	CHAR80
LABEL	Set label	CHAR80
NGRID	Number of grid points	INT4
MAXGID	Largest grid point ID	INT4
MAXVECT	Maximum absolute vector value	REAL4
MAXVECTGID	Grid point ID where the maximum value occurs	INT4
NGRIDVECTCOL	The number of force vector components or columns	INT4
GRIDID	Grid point ID number	INT4
GRIDVECT	Grid point vector component values at GRIDID	REAL4

Structural Solutions – Real

Element Results Neutral File Column Definition (*filename*.ELS):

Solid and Shell Elements		
Column Number	**Solid**	**Shell**
1	ENERGY	ENERGY
2	% TOTAL ENERGY	% TOTAL ENERGY
3	ENERGY DENSITY	ENERGY DENSITY
4	NORMAL -X	NORMAL -X1
5	NORMAL -Y	NORMAL -Y1
6	NORMAL -Z	SHEAR -XY1
7	SHEAR -XY	0-SHEAR ANGLE-1
8	SHEAR -YZ	MAX SHEAR-1
9	SHEAR -ZX	MAJOR PRINCIPAL-1
10	PRINCIPAL -A	MINOR-PRINCIPAL-1
11	PRINCIPAL -B	VON MISES-1
12	PRINCIPAL -C	FIBER DISTANCE-1
13	PRINCIPAL -A COS-X	NORMAL-X2
14	PRINCIPAL -B COS-X	NORMAL-Y2
15	PRINCIPAL -C COS X	SHEAR-XY2
16	PRINCIPAL -A COS-Y	0-SHEAR ANGLE-2
17	PRINCIPAL -B COS-Y	MAX SHEAR-2
18	PRINCIPAL -C COS-Y	MAJOR-PRINCIPAL-2
19	PRINCIPAL -A COS-Z	MINOR-PRINCIPAL-2
20	PRINCIPAL -B COS-Z	VON MISES-2
21	PRINCIPAL -C COS-Z	FIBER DISTANCE-2
22	VON MISES	MAX VON MISES-1/2
23	MAX SHEAR/TRESCA	MAX SHEAR-1/2
24	MAX PRINCIPAL	MAX PRINCIPAL-1/2
25	MIN PRINCIPAL	MIN PRINCIPAL-1/2
26	MEAN PRESSURE	STATUS
27	EQV STRESS	EQV STRESS-1
28	EFF STRAIN-P	EFF STRAIN-P1
29	EFF STRAIN-C	EFF STRAIN-C1
30	OCTAHEDRAL	EQV STRESS-2
31	STATUS	EFF STRAIN-P2
32	0	EFF STRAIN-C2
33	0	MEMBRANE FX
34	0	MEMBRANE FY
35	0	MEMBRANE FXY
36	0	MOMENT MX
37	0	MOMENT MY
38	0	MOMENT MXY
39	0	TRANSV. SHEAR QX
40	0	TRANSV. SHEAR QY

Note: When STRESS(CORNER) is specified in the Case Control Section of the model, columns 1-40 for solid and shell elements are repeated for each element node. The corresponding column number is equal to:

COLUMN NUMBER + (40 x NODE NUMBER).

Element Results Neutral File Column Definition (Continued):

Composite Shell Elements		
Column Number	**Composite Shell**	**Individual Ply**
1	0	0
2	0	0
3	0	0
4	MAX NORMAL-1	NORMAL-1
5	MAX NORMAL-2	NORMAL-2
6	MAX SHEAR-12	SHEAR-12
7	MAX SHEAR-XZ	SHEAR-XZ
8	MAX SHEAR-YZ	SHEAR-YZ
9	MAX PLY FAIL INDX	PLY FAIL INDX
10	MAX BOND FAIL INDX	BOND FAIL INDX
11	MAX STAB FAIL INDX	STAB FAIL INDX
12	MAX EQV STRESS	EQV STRESS
13	MIN NORMAL-1	STAB ALLW
14	MIN NORMAL-2	STAB ALLW FM
15	MIN SHEAR-12	STAB INDX WR
16	MIN SHEAR-XZ	STAB INDX DP
17	MIN SHEAR-YZ	STAB INDX CR
18	MIN PLY FAIL INDX	STAB ALLW WR
19	MIN BOND FAIL INDX	STAB ALLW DP
20	MIN STAB FAIL INDX	STAB ALLW CR
21	MAX EFF STRAIN	EFF STRAIN
22	MAX VON MISES	VON MISES
23	MAX MAX SHEAR	MAX SHEAR
24	MAX PRINCIPAL	MAX PRINCIPAL
25	MIN PRINCIPAL	MIN PRINCIPAL
26	MAX FAIL INDX	FAILURE THEORY
27	MAX FAIL INDX PLY	PLY FAIL MT-T
28	STATUS	PLY FAIL MT-C
29	STAB CORE PLY	PLY FAIL FB-T
30	MIN STAB ALLW	PLY FAIL FB-C
31	MIN STAB ALLW PLY	FRACTURE ANGLE
32	0	0
33	MEMBRANE FX	0
34	MEMBRANE FY	0
35	MEMBRANE FXY	0
36	MOMENT MX	0
37	MOMENT MY	0
38	MOMENT MXY	0
39	TRANSV. SHEAR QX	0
40	TRANSV. SHEAR QY	0

Note: When PARAM, COMPRSLTOUTPUT is set to ON, columns 1-40 for composite shell elements are repeated for each ply. The corresponding column number is equal to:

COLUMN NUMBER + (40 x PLY NUMBER).

Note: When PARAM, STRENGTHRATIO is set to ON, Failure Indexes are replaced with Strength Ratios.

Element Results Neutral File Column Definition (Continued):

Shear Elements	
Column Number	**Shear**
1	ENERGY
2	% TOTAL ENERGY
3	ENERGY DENSITY
4	SHEAR-XY EDGE 1
5	SHEAR-XY EDGE 2
6	SHEAR-XY EDGE 3
7	SHEAR-XY EDGE 4
8	MAX SHEAR-XY
9	MIN SHEAR-XY
10	AVE SHEAR-XY
11	0
12	0
13	0
14	0
15	0
16	0
17	0
18	0
19	0
20	0
21	0
22	0
23	0
24	0
25	0
26	0
27	0
28	0
29	KICK NODE 1
30	KICK NODE 2
31	KICK NODE 3
32	KICK NODE 4
33	MAX KICK LOAD
34	MIN KICK LOAD
35	SHEAR FLOW EDGE 1
36	SHEAR FLOW EDGE 2
37	SHEAR FLOW EDGE 3
38	SHEAR FLOW EDGE 4
39	MAX SHEAR FLOW
40	MIN SHEAR FLOW

Element Results Neutral File Column Definition (Continued):

Axisymmetric Elements	
Column Number	**Axisymmetric**
1	ENERGY
2	% TOTAL ENERGY
3	ENERGY DENSITY
4	NORMAL-RADIAL
5	NORMAL-TANGENTIAL
6	NORMAL-AXIAL
7	0
8	0
9	SHEAR-RADIAL/AXIAL
10	0
11	0
12	0
13	0
14	0
15	0
16	0
17	0
18	0
19	0
20	0
21	0
22	VON MISES
23	MAX SHEAR/TRESCA
24	MAX PRINCIPAL
25	MIN PRINCIPAL
26	MEAN PRESSURE
27	EQV STRESS
28	EFF STRAIN-P
29	EFF STRAIN-C
30	OCTAHEDRAL
31	STATUS
32	0
33	0
34	0
35	0
36	0
37	0
38	0
39	0
40	0

Element Results Neutral File Column Definition (Continued):

Line Elements			
Column Number	**Bar/Beam**	**Rod**	**Spring**
1	ENERGY	ENERGY	ENERGY
2	% TOTAL ENERGY	% TOTAL ENERGY	% TOTAL ENERGY
3	ENERGY DENSITY	ENERGY DENSITY	ENERGY DENSITY
4	SA-AXIAL	S-AXIAL	STRESS
5	SA-C	S-TORSIONAL	0
6	SA-D	0	0
7	SA-E	0	0
8	SA-F	0	0
9	SB-AXIAL	0	0
10	SB-C	0	0
11	SB-D	0	0
12	SB-E	0	0
13	SB-F	0	0
14	SA-MIN	0	0
15	SB-MIN	0	0
16	SA-MAX	0	0
17	SB-MAX	0	0
18	S-MAX	0	0
19	S-MIN	0	0
20	EQV STRESS	EQV STRESS	EQV STRESS
21	EFF STRAIN-P	EFF STRAIN-P	0
22	EFF STRAIN-C	EFF STRAIN-C	0
23	LOCATION A	0	0
24	LOCATION B	0	0
25	LOCATION S-MAX	0	0
26	LOCATION S-MIN	0	0
27	STATUS	STATUS	STATUS
28	VON MISES	0	0
29	FORCE A-X	FORCE A-X	0
30	FORCE A-Y PLANE 1	0	0
31	FORCE A-Z PLANE 2	0	0
32	MOMENT A-X	MOMENT A-X	0
33	MOMENT A-Y PLANE 2	0	0
34	MOMENT A-Z PLANE 1	0	0
35	FORCE B-X	FORCE B-X	FORCE
36	FORCE B-Y	0	0
37	FORCE B-Z	0	0
38	MOMENT B-X	MOMENT B-X	0
39	MOMENT B-Y PLANE 2	0	0
40	MOMENT B-Z PLANE 1	0	0

Element Results Neutral File Column Definition (Continued):

Line Elements			
Column Number	**Pipe**	**Weld**	**Bush**
1	ENERGY	ENERGY	ENERGY
2	% TOTAL ENERGY	% TOTAL ENERGY	% TOTAL ENERGY
3	ENERGY DENSITY	ENERGY DENSITY	ENERGY DENSITY
4	SA-LONGITUDINAL	SA-LONGITUDINAL	S-TX
5	SA-HOOP	SA-TORSIONAL	S-TY
6	SA-TORSIONAL	SA-SHEAR	S-TZ
7	SA-SHEAR	SA-MAX PRINCIPAL	S-RX
8	SA-MAX PRINCIPAL	SA-MAX SHEAR	S-RY
9	SA-MAX SHEAR	SB-LONGITUDINAL	S-RZ
10	SA-OCTAHEDRAL	SB-TORSIONAL	S-T MAX
11	SB-LONGITUDINAL	SB-SHEAR	S-R MAX
12	SB-HOOP	SB-MAX PRINCIPAL	0
13	SB-TORSIONAL	SB-MAX SHEAR	0
14	SB-SHEAR	0	0
15	SB-MAX PRINCIPAL	0	0
16	SB-MAX SHEAR	0	0
17	SB-OCTAHEDRAL	0	0
18	S-MAX PRINCIPAL	0	0
19	S-OCTAHEDRAL	0	0
20	EQV STRESS	EQV STRESS	EQV STRESS
21	EFF STRAIN-P	EFF STRAIN-P	EFF STRAIN
22	EFF STRAIN-C	EFF STRAIN-C	0
23	LOCATION A	LOCATION A	FORCE-K
24	LOCATION B	LOCATION B	FORCE-B
25	0	0	FORCE-C
26	0	0	FORCE-T
27	STATUS	STATUS	STATUS
28	0	0	0
29	FORCE A-X	FORCE A-X	VISC DAMP FORCE-X
30	FORCE A-Y PLANE 1	FORCE A-Y PLANE 1	VISC DAMP FORCE-Y
31	FORCE A-Z PLANE 2	FORCE A-Z PLANE 2	VISC DAMP FORCE-Z
32	MOMENT A-X	MOMENT A-X	VISC DAMP MOMENT-X
33	MOMENT A-Y PLANE 2	MOMENT A-Y PLANE 2	VISC DAMP MOMENT-Y
34	MOMENT A-Z PLANE 1	MOMENT A-Z PLANE 1	VISC DAMP MOMENT-Z
35	FORCE B-X	FORCE B-X	FORCE-X
36	FORCE B-Y	FORCE B-Y	FORCE-Y
37	FORCE B-Z	FORCE B-Z	FORCE-Z
38	MOMENT B-X	MOMENT B-X	MOMENT-X
39	MOMENT B-Y PLANE 2	MOMENT B-Y PLANE 2	MOMENT-Y
40	MOMENT B-Z PLANE 1	MOMENT B-Z PLANE 1	MOMENT-Z

Element Results Neutral File Column Definition (Continued):

Contact Elements			
Column Number	**Gap**	**Slide Line**	**Contact Surface**
1	0	0	0
2	0	0	0
3	0	0	0
4	AXIAL FORCE	MAX NORMAL FORCE	MAX NORMAL FORCE
5	SHEAR FORCE-Y	MAX NORMAL STRESS	MAX NORMAL STRESS
6	SHEAR FORCE-Z	MAX NORMAL GAP	MAX NORMAL GAP
7	AXIAL DISPLACEMENT	MIN NORMAL FORCE	MIN NORMAL FORCE
8	TOTAL DISP-Y	MIN NORMAL STRESS	MIN NORMAL STRESS
9	TOTAL DISP-Z	MIN NORMAL GAP	MIN NORMAL GAP
10	SLIP DISP-Y	MAX SHEAR FORCE	MAX SHEAR FORCE-X
11	SLIP DISP-Z	MAX SHEAR STRESS	MAX SHEAR FORCE-Y
12	GAP STATUS	MAX SLIP DISP	MAX SHEAR STRESS-X
13	RSLT SHEAR FORCE	MIN SHEAR FORCE	MAX SHEAR STRESS-Y
14	RSLT INPLANE DISP	MIN SHEAR STRESS	MAX SLIP DISP-X
15	RSLT SLIP DISP	MIN SLIP DISP	MAX SLIP DISP-Y
16	0	CONTACT STATUS	MIN SHEAR FORCE-X
17	0		MIN SHEAR FORCE-Y
18	0	0	MIN SHEAR STRESS-X
19	0	0	MIN SHEAR STRESS-Y
20	0	0	MIN SLIP DISP-X
21	0	0	MIN SLIP DISP-Y
22	0	0	CONTACT STATUS
23	0	0	RSLT SHEAR FORCE
24	0	0	RSLT SHEAR STRESS
25	0	0	RSLT SLIP DISP
26	0	0	
27	0	0	
28	0	0	
29	0	0	
30	0	0	0
31	0	0	0
32	0	0	0
33	0	0	0
34	0	0	
35	0	0	
36	0	0	
37	0	0	0
38	0	0	0
39	0	0	0
40	0	0	0

Element Results Neutral File Column Definition (Continued):

Cable Elements	
Column Number	**Cable**
1	0
2	0
3	0
4	TENSION FORCE
5	TENSION STRESS
6	AXIAL DISPLACEMENT
7	SLIP
8	CABLE STATUS
9	0
10	0
11	0
12	0
13	0
14	0
15	0
16	0
17	0
18	0
19	0
20	EQV STRESS
21	EFF STRAIN
22	0
23	0
24	0
25	0
26	0
27	0
28	0
29	0
30	0
31	0
32	0
33	0
34	0
35	0
36	0
37	0
38	0
39	0
40	0

Grid Point Results Neutral File Column Definition (*filename*.GPS):

Solid and Shell Elements		
Column Number	**Solid**	**Shell**
1	0	0
2	0	0
3	0	0
4	NORMAL -X	NORMAL-X1
5	NORMAL -Y	NORMAL-Y1
6	NORMAL -Z	SHEAR-XY1
7	SHEAR -XY	0-SHEAR ANGLE-1
8	SHEAR -YZ	MAX SHEAR-1
9	SHEAR -ZX	MAJOR PRINCIPAL-1
10	PRINCIPAL -A	MINOR-PRINCIPAL-1
11	PRINCIPAL -B	VON MISES-1
12	PRINCIPAL -C	FIBER DISTANCE-1
13	PRINCIPAL -A COS-X	NORMAL-X2
14	PRINCIPAL -B COS-X	NORMAL-Y2
15	PRINCIPAL -C COS-X	SHEAR-XY2
16	PRINCIPAL -A COS-Y	0-SHEAR ANGLE-2
17	PRINCIPAL -B COS-Y	MAX SHEAR-2
18	PRINCIPAL -C COS-Y	MAJOR-PRINCIPAL-2
19	PRINCIPAL -A COS-Z	MINOR-PRINCIPAL-2
20	PRINCIPAL -B COS-Z	VON MISES-2
21	PRINCIPAL -C COS-Z	FIBER DISTANCE-2
22	VON MISES	MAX VON MISES-1/2
23	MAX SHEAR/TRESCA	MAX SHEAR-1/2
24	MAX PRINCIPAL	MAX PRINCIPAL-1/2
25	MIN PRINCIPAL	MIN PRINCIPAL-1/2
26	MEAN PRESSURE	EQV STRESS-1
27	EQV STRESS	EFF STRAIN-P1
28	EFF STRAIN-P	EFF STRAIN-C1
29	EFF STRAIN-C	EQV STRESS-2
30	OCTAHEDRAL	EFF STRAIN-P2
31	0	EFF STRAIN-C2
32	0	0
33	0	0
34	0	0
35	0	0
36	0	0
37	0	0
38	0	MESH ERROR-1
39	0	MESH ERROR-2
40	MESH ERROR	MAX MESH ERROR-1/2

Grid Point Results Neutral File Column Definition (Continued):

Contact Elements	
Column Number	**Contact Surface**
1	0
2	0
3	0
4	0
5	0
6	0
7	0
8	0
9	0
10	0
11	0
12	0
13	0
14	0
15	0
16	0
17	0
18	0
19	0
20	0
21	0
22	0
23	0
24	0
25	0
26	0
27	0
28	0
29	0
30	0
31	0
32	CONTACT PRESSURE
33	CONTACT TRACTION-X
34	CONTACT TRACTION-Y
35	BOND EQV STRESS
36	BOND EFF DISP
37	BOND DAMAGE
38	0
39	0
40	0

Element Internal Load Vector Neutral File Column Definition (*filename*.ELF):

Column Number	Component
1	T1 INTERNAL FORCE
2	T2 INTERNAL FORCE
3	T3 INTERNAL FORCE
4	R1 INTERNAL MOMENT
5	R2 INTERNAL MOMENT
6	R3 INTERNAL MOMENT

Note: Data for columns 1-6 repeat for each node of the element.

Grid Point Displacement Vector Neutral File Column Definition (*filename*.DIS):

Column Number	Component
1	TRANSLATION-1
2	TRANSLATION-2
3	TRANSLATION-3
4	ROTATION-1
5	ROTATION-2
6	ROTATION-3

Grid Point Force Vector Neutral File Column Definition (*filename*.GPF):

Column Number	Component
1	INTERNAL FORCE-1
2	INTERNAL FORCE-2
3	INTERNAL FORCE-3
4	INTERNAL MOMENT-1
5	INTERNAL MOMENT-2
6	INTERNAL MOMENT-3
7	APPLIED FORCE-1
8	APPLIED FORCE-2
9	APPLIED FORCE-3
10	APPLIED MOMENT-1
11	APPLIED MOMENT-2
12	APPLIED MOMENT-3
13	SPC FORCE-1
14	SPC FORCE-2
15	SPC FORCE-3
16	SPC MOMENT-1
17	SPC MOMENT-2
18	SPC MOMENT-3
19	MPC FORCE-1
20	MPC FORCE-2
21	MPC FORCE-3
22	MPC MOMENT-1
23	MPC MOMENT-2
24	MPC MOMENT-3
25	VELOCITY-1
26	VELOCITY-2
27	VELOCITY-3
28	ANGULAR VELOCITY-1
29	ANGULAR VELOCITY-2
30	ANGULAR VELOCITY-3
31	ACCELERATION-1
32	ACCELERATION-2
33	ACCELERATION-3
34	ANGULAR ACCELERATION-1
35	ANGULAR ACCELERATION-2
36	ANGULAR ACCELERATION-3
37	CONTACT FORCE-1
38	CONTACT FORCE-2
39	CONTACT FORCE-3

Structural Solutions – Complex

Element Results Neutral File Column Definition (*filename*.ELS):

Solid and Shell Elements		
Column Number	**Solid**	**Shell**
1	0	0
2	0	0
3	0	0
4	NORMAL -X	NORMAL -X1
5	NORMAL -Y	NORMAL -Y1
6	NORMAL -Z	SHEAR -XY1
7	SHEAR -XY	0
8	SHEAR -YZ	0
9	SHEAR -ZX	0
10	0	0
11	0	VON MISES-1
12	0	FIBER DISTANCE-1
13	0	NORMAL-X2
14	0	NORMAL-Y2
15	0	SHEAR-XY2
16	0	0
17	0	0
18	0	0
19	0	0
20	0	VON MISES-2
21	0	FIBER DISTANCE-2
22	VON MISES	MAX VON MISES-1/2
23	0	0
24	0	0
25	0	0
26	0	0
27	0	0
28	0	0
29	0	0
30	0	0
31	0	0
32	0	0
33	0	MEMBRANE FX
34	0	MEMBRANE FY
35	0	MEMBRANE FXY
36	0	MOMENT MX
37	0	MOMENT MY
38	0	MOMENT MXY
39	0	TRANSV. SHEAR QX
40	0	TRANSV. SHEAR QY

Note: Complex data is stored as columns 1-40 are real/magnitude and columns 41-80 are imaginary/phase. When STRESS(CORNER) is specified in the Case Control Section of the model, columns 1-80 for solid and shell elements are repeated for each element node. The corresponding column number is equal to:

COLUMN NUMBER + (80 x NODE NUMBER).

Element Results Neutral File Column Definition (Continued):

Line Elements			
Column Number	**Bar/Beam**	**Rod**	**Spring**
1	0	0	0
2	0	0	0
3	0	0	0
4	SA-AXIAL	S-AXIAL	STRESS
5	SA-C	S-TORSIONAL	0
6	SA-D	0	0
7	SA-E	0	0
8	SA-F	0	0
9	SB-AXIAL	0	0
10	SB-C	0	0
11	SB-D	0	0
12	SB-E	0	0
13	SB-F	0	0
14	SA-MIN	0	0
15	SB-MIN	0	0
16	SA-MAX	0	0
17	SB-MAX	0	0
18	S-MAX	0	0
19	S-MIN	0	0
20	0	0	0
21	0	0	0
22	0	0	0
23	LOCATION A	0	0
24	LOCATION B	0	0
25	LOCATION S-MAX	0	0
26	LOCATION S-MIN	0	0
27	0	0	0
28	0	0	0
29	FORCE A-X	FORCE A-X	FORCE
30	FORCE A-Y PLANE 1	0	0
31	FORCE A-Z PLANE 2	0	0
32	MOMENT A-X	MOMENT A-X	0
33	MOMENT A-Y PLANE 2	0	0
34	MOMENT A-Z PLANE 1	0	0
35	FORCE B-X	FORCE B-X	0
36	FORCE B-Y	0	0
37	FORCE B-Z	0	0
38	MOMENT B-X	MOMENT B-X	0
39	MOMENT B-Y PLANE 2	0	0
40	MOMENT B-Z PLANE 1	0	0

Note: Complex data is stored as columns 1-40 are real/magnitude and columns 41-80 are imaginary/phase.

Element Results Neutral File Column Definition (Continued):

Line Elements		
Column Number	**Weld**	**Bush**
1	0	0
2	0	0
3	0	0
4	SA-LONGITUDINAL	S-TX
5	SA-TORSIONAL	S-TY
6	SA-SHEAR	S-TZ
7	0	S-RX
8	0	S-RY
9	SB-LONGITUDINAL	S-RZ
10	SB-TORSIONAL	S-T MAX
11	SB-SHEAR	S-R MAX
12	0	0
13	0	0
14	S-MAX LONGITUDINAL	0
15	S-MAX TORSIONAL	0
16	S-MAX SHEAR	0
17	0	0
18	0	0
19	0	0
20	0	0
21	0	0
22	0	0
23	LOCATION A	0
24	LOCATION B	0
25	0	0
26	0	0
27	0	0
28	0	0
29	FORCE A-X	VISC DAMP FORCE-X
30	FORCE A-Y PLANE 1	VISC DAMP FORCE-Y
31	FORCE A-Z PLANE 2	VISC DAMP FORCE-Z
32	MOMENT A-X	VISC DAMP MOMENT-X
33	MOMENT A-Y PLANE 2	VISC DAMP MOMENT-Y
34	MOMENT A-Z PLANE 1	VISC DAMP MOMENT-Z
35	FORCE B-X	FORCE-X
36	FORCE B-Y	FORCE-Y
37	FORCE B-Z	FORCE-Z
38	MOMENT B-X	MOMENT-X
39	MOMENT B-Y PLANE 2	MOMENT-Y
40	MOMENT B-Z PLANE 1	MOMENT-Z

Grid Point Results Neutral File Column Definition (*filename*.GPS):

Solid and Shell Elements		
Column Number	**Solid**	**Shell**
1	0	0
2	0	0
3	0	0
4	NORMAL -X	NORMAL-X1
5	NORMAL -Y	NORMAL-Y1
6	NORMAL -Z	SHEAR-XY1
7	SHEAR -XY	0
8	SHEAR -YZ	0
9	SHEAR -ZX	0
10	0	0
11	0	VON MISES-1
12	0	FIBER DISTANCE-1
13	0	NORMAL-X2
14	0	NORMAL-Y2
15	0	SHEAR-XY2
16	0	0
17	0	0
18	0	0
19	0	0
20	0	VON MISES-2
21	0	FIBER DISTANCE-2
22	VON MISES	MAX VON MISES-1/2
23	0	0
24	0	0
25	0	0
26	0	0
27	0	0
28	0	0
29	0	0
30	0	0
31	0	0
32	0	0
33	0	0
34	0	0
35	0	0
36	0	0
37	0	0
38	0	0
39	0	0
40	0	0

Note: Complex data is stored as columns 1-40 are real/magnitude and columns 41-80 are imaginary/phase.

Grid Point Displacement Vector Neutral File Column Definition (*filename*.DIS):

Column Number	Component
1	TRANSLATION-1
2	TRANSLATION-2
3	TRANSLATION-3
4	ROTATION-1
5	ROTATION-2
6	ROTATION-3

Note: Complex data is stored as columns 1-6 are real/magnitude and columns 7-12 are imaginary/phase.

Grid Point Force Vector Neutral File Column Definition (*filename*.GPF):

Column Number	Component
1	INTERNAL FORCE-1
2	INTERNAL FORCE-2
3	INTERNAL FORCE-3
4	INTERNAL MOMENT-1
5	INTERNAL MOMENT-2
6	INTERNAL MOMENT-3
7	APPLIED FORCE-1
8	APPLIED FORCE-2
9	APPLIED FORCE-3
10	APPLIED MOMENT-1
11	APPLIED MOMENT-2
12	APPLIED MOMENT-3
13	SPC FORCE-1
14	SPC FORCE-2
15	SPC FORCE-3
16	SPC MOMENT-1
17	SPC MOMENT-2
18	SPC MOMENT-3
19	MPC FORCE-1
20	MPC FORCE-2
21	MPC FORCE-3
22	MPC MOMENT-1
23	MPC MOMENT-2
24	MPC MOMENT-3
25	VELOCITY-1
26	VELOCITY-2
27	VELOCITY-3
28	ANGULAR VELOCITY-1
29	ANGULAR VELOCITY-2
30	ANGULAR VELOCITY-3
31	ACCELERATION-1
32	ACCELERATION-2
33	ACCELERATION-3
34	ANGULAR ACCELERATION-1
35	ANGULAR ACCELERATION-2
36	ANGULAR ACCELERATION-3

Note: Complex data is stored as columns 1-6 are real/magnitude and columns 7-12 are imaginary/phase. The remaining result types follow this same pattern (i.e., columns 13-18 are real/magnitude and columns 19-24 are imaginary/phase)

Heat Transfer Solutions

Element Results Neutral File Column Definition (*filename*.ELS):

Solid, Shell, and Line Elements			
Column Number	**Solid**	**Shell**	**Line**
1	THERMAL GRADIENT-X	THERMAL GRADIENT-X	THERMAL GRADIENT-X
2	THERMAL GRADIENT-Y	THERMAL GRADIENT-Y	0
3	THERMAL GRADIENT-Z	0	0
4	THERMAL GRAD. RSLT	THERMAL GRAD. RSLT	THERMAL GRAD. RSLT
5	HEAT FLUX-X	HEAT FLUX-X	HEAT FLUX-X
6	HEAT FLUX-Y	HEAT FLUX-Y	0
7	HEAT FLUX-Z	0	0
8	HEAT FLUX RSLT	HEAT FLUX RSLT	HEAT FLUX RSLT
9	0	0	0
10	0	0	0
11	0	0	0
12	0	0	0
13	0	0	0
14	0	0	0
15	0	0	0
16	0	0	0
17	0	0	0
18	0	0	0
19	0	0	0
20	0	0	0
21	0	0	0
22	0	0	0
23	0	0	0
24	0	0	0
25	0	0	0
26	0	0	0
27	0	0	0
28	0	0	0
29	0	0	0
30	0	0	0
31	0	0	0
32	0	0	0
33	0	0	0
34	0	0	0
35	0	0	0
36	0	0	0
37	0	0	0
38	0	0	0
39	0	0	0
40	0	0	0

Note: When FLUX(CORNER) is specified in the Case Control Section of the model, columns 1-40 for solid and shell elements are repeated for each element node. The corresponding column number is equal to:

COLUMN NUMBER + (40 x NODE NUMBER).

Element Results Neutral File Column Definition (Continued):

HBDY Elements	
Column Number	**HBDY**
1	APPLIED LOAD
2	CONVECTION LOAD
3	RADIATION LOAD
4	TOTAL LOAD
5	0
6	0
7	0
8	0
9	0
10	0
11	0
12	0
13	0
14	0
15	0
16	0
17	0
18	0
19	0
20	0
21	0
22	0
23	0
24	0
25	0
26	0
27	0
28	0
29	0
30	0
31	0
32	0
33	0
34	0
35	0
36	0
37	0
38	0
39	0
40	0

Grid Point Results Neutral File Column Definition (*filename*.GPS):

Solid and Shell Elements		
Column Number	**Solid**	**Shell**
1	THERMAL GRADIENT-X	THERMAL GRADIENT-X
2	THERMAL GRADIENT-Y	THERMAL GRADIENT-Y
3	THERMAL GRADIENT-Z	THERMAL GRADIENT-Z
4	THERMAL GRAD. RSLT	THERMAL GRAD. RSLT
5	HEAT FLUX-X	HEAT FLUX-X
6	HEAT FLUX-Y	HEAT FLUX-Y
7	HEAT FLUX-Z	HEAT FLUX-Z
8	HEAT FLUX RSLT	HEAT FLUX RSLT
9	0	0
10	0	0
11	0	0
12	0	0
13	0	0
14	0	0
15	0	0
16	0	0
17	0	0
18	0	0
19	0	0
20	0	0
21	0	0
22	0	0
23	0	0
24	0	0
25	0	0
26	0	0
27	0	0
28	0	0
29	0	0
30	0	0
31	0	0
32	0	0
33	0	0
34	0	0
35	0	0
36	0	0
37	0	0
38	0	0
39	0	0
40	0	0

Grid Point Displacement Vector Neutral File Column Definition (*filename*.DIS):

Column Number	Component
1	TEMPERATURE
2	0
3	0
4	0
5	0
6	0

Grid Point Force Vector Neutral File Column Definition (*filename*.GPF):

Column Number	Component
1	INTERNAL HEAT FLUX
2	APPLIED HEAT FLUX
3	SPC HEAT FLUX
4	MPC HEAT FLUX
5	ENTHALPY
6	ENTHALPY RATE
7	0
8	0
9	0
10	0
11	0
12	0
13	0
14	0
15	0
16	0
17	0
18	0
19	0
20	0
21	0
22	0
23	0
24	0
25	0
26	0
27	0
28	0
29	0
30	0
31	0
32	0
33	0
34	0
35	0
36	0

Element Type Code Definition:

Element Type	ELEMTYPE
ELAS	1
ROD	2
BAR	3
BEAM	4
SHELL	5
COMPOSITE SHELL	6
SHEAR	7
SOLID	8
GAP CONTACT	9
SLIDE LINE CONTACT	10
QUAD SURFACE CONTACT	11
TRI SURFACE CONTACT	12
CABLE	13
PIPE	14
SHELL 4-NODE	15
SHELL 3-NODE	16
SOLID 8-NODE	18
SOLID 6-NODE	19
SOLID 4-NODE	20
SOLID 20-NODE	21
SOLID 15-NODE	22
SOLID 15-NODE	23
HBDY	24
BUSH	25
WELD	26
SURFACE SHELL	27
LAYERED SOLID	28
SOLID 5-NODE	29
SOLID 13-NODE	30
AXISYMMETRIC	31
AXISYMMETRIC 3-NODE	32
AXISYMMETRIC 4-NODE	33
AXISYMMETRIC 2-NODE	34

Element Type Label Definition:

Element Type Label	Element Type Definition
ELAS	ELAS
ROD	ROD
BAR	BAR
BEAM	BEAM
SHELL	SHELL
COMP	COMPOSITE SHELL
SHEAR	SHEAR
SOLID	SOLID
GAP	GAP CONTACT
SLINE	SLIDE LINE CONTACT
SQUAD	QUAD SURFACE CONTACT
STRI	TRI SURFACE CONTACT
CABLE	CABLE
PIPE	PIPE
HBDY	HBDY
BUSH	BUSH
WELD	WELD
AQUAD	AXISYMMETRIC QUAD
ATRI	AXISYMMETRIC TRI

Vector Id Offset Definition for Complex Results:

Offset	Definition
0	Magnitude
10000000	Phase
20000000	Real
30000000	Imaginary

Note: The above offset values are added to the vector ids listed in the following tables to define a complex result type used in frequency and random response and complex eigenvalue analysis.

Structural Neutral File Element Results Column Descriptions

Spring Element Results Column Descriptions:

Vector Id	Label	Description
3028	ELAS FORCE	Spring element force. Controlled by FORCE Case Control command.
3182	ELAS STRESS	Spring element stress. Controlled by STRESS Case Control command.
3285	ELAS EQUIVALENT STRESS	Spring element equivalent stress. Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
3481	ELAS STATUS	Solution and option dependent. In modal summation solutions (DDAM) STATUS, is the mode number with the maximum response in the NRL summation.

Bush Element Results Column Descriptions:

Vector Id	Label	Description
3028	BUSH FORCE-X	Bush element force in element x-direction. Controlled by FORCE Case Control command.
3030	BUSH FORCE-Y	Bush element force in element y-direction. Controlled by FORCE Case Control command.
3031	BUSH FORCE-Z	Bush element force in element z-direction. Controlled by FORCE Case Control command.
3032	BUSH MOMENT-X	Bush element moment in element x-direction. Controlled by FORCE Case Control command.
3033	BUSH MOMENT-Y	Bush element moment in element y-direction. Controlled by FORCE Case Control command.
3034	BUSH MOMENT-Z	Bush element moment in element z-direction. Controlled by FORCE Case Control command.
3285	BUSH EQUIVALENT STRESS	Bush element maximum translational stress. Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
3286	BUSH EFFECTIVE STRAIN	Bush element maximum translational strain. Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
3481	BUSH STATUS	Solution and option dependent. In modal summation solutions (DDAM), STATUS is the mode number with the maximum response in the NRL summation.
3490	BUSH STRESS TRANSLATIONAL-X	Bush element x-direction translational stress. Controlled by STRESS Case Control command.
3491	BUSH STRESS TRANSLATIONAL-Y	Bush element y-direction translational stress. Controlled by STRESS Case Control command.
3492	BUSH STRESS TRANSLATIONAL-Z	Bush element z-direction translational stress. Controlled by STRESS Case Control command.
3493	BUSH STRESS ROTATIONAL-X	Bush element x-direction rotational stress. Controlled by STRESS Case Control command.
3494	BUSH STRESS ROTATIONAL-Y	Bush element y-direction rotational stress. Controlled by STRESS Case Control command.
3495	BUSH STRESS ROTATIONAL-Z	Bush element z-direction rotational stress. Controlled by STRESS Case Control command.
3496	BUSH STRESS TRANSLATIONAL-MAX	Bush element maximum translational stress. Controlled by STRESS Case Control command.
3497	BUSH STRESS ROTATIONAL-MAX	Bush element maximum rotational stress. Controlled by STRESS Case Control command.
3501	BUSH VISCOUS DAMPING FORCE-X	Bush element force in element x-direction due to viscous damping. Controlled by FORCE Case Control command.

Bush Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3502	BUSH VISCOUS DAMPING FORCE-Y	Bush element force in element y-direction due to viscous damping. Controlled by FORCE Case Control command.
3503	BUSH VISCOUS DAMPING FORCE-Z	Bush element force in element z-direction due to viscous damping. Controlled by FORCE Case Control command.
3504	BUSH VISCOUS DAMPING MOMENT-X	Bush element moment in element x-direction due to viscous damping. Controlled by FORCE Case Control command.
3505	BUSH VISCOUS DAMPING MOMENT-Y	Bush element moment in element y-direction due to viscous damping. Controlled by FORCE Case Control command.
3506	BUSH VISCOUS DAMPING MOMENT-Z	Bush element moment in element z-direction due to viscous damping. Controlled by FORCE Case Control command.
3507	BUSH FORCE-STIFFNESS	Bush element axial force due to stiffness. Controlled by FORCE Case Control command.
3508	BUSH FORCE-DAMPING	Bush element axial force due to damping. Controlled by FORCE Case Control command.
3509	BUSH FORCE-COUPLING	Bush element axial force due to coupled stiffness-damping. Controlled by FORCE Case Control command.
3510	BUSH FORCE-TOTAL	Bush element axial force due to stiffness, damping, and coupled stiffness-damping. Controlled by FORCE Case Control command.
3990	BUSH STRAIN TRANSLATIONAL-X	Bush element x-direction translational strain. Controlled by STRAIN Case Control command.
3991	BUSH STRAIN TRANSLATIONAL-Y	Bush element y-direction translational strain. Controlled by STRAIN Case Control command.
3992	BUSH STRAIN TRANSLATIONAL-Z	Bush element z-direction translational strain. Controlled by STRAIN Case Control command.
3993	BUSH STRAIN ROTATIONAL-X	Bush element x-direction rotational strain. Controlled by STRAIN Case Control command.
3994	BUSH STRAIN ROTATIONAL-Y	Bush element y-direction rotational strain. Controlled by STRAIN Case Control command.
3995	BUSH STRAIN ROTATIONAL-Z	Bush element z-direction rotational strain. Controlled by STRAIN Case Control command.
3996	BUSH STRAIN TRANSLATIONAL-MAX	Bush element maximum translational strain. Controlled by STRAIN Case Control command.
3997	BUSH STRAIN ROTATIONAL-MAX	Bush element maximum rotational strain. Controlled by STRAIN Case Control command.

Rod Element Results Column Descriptions:

Vector Id	Label	Description
3012	ROD MOMENT END A-X	Rod element torque at end A about element x-direction. Controlled by FORCE Case Control command.
3013	ROD MOMENT END B-X	Rod element torque at end B about element x-direction. Controlled by FORCE Case Control command.
3036	ROD FORCE END A-X	Rod element axial force at end A in element x-direction. Controlled by FORCE Case Control command.
3037	ROD FORCE END B-X	Rod element axial force at end B in element x-direction. Controlled by FORCE Case Control command.
3183	ROD AXIAL STRESS	Rod element axial stress. Controlled by STRESS Case Control command.
3186	ROD TORSIONAL STRESS	Rod element torsional stress. Controlled by STRESS Case Control command.
3290	ROD EQUIVALENT STRESS	Rod element nonlinear equivalent axial stress (material nonlinear solutions) or axial stress (linear solutions). Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
3291	ROD EFFECTIVE STRAIN-ELASTIC	Rod element effective axial strain. Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
3291	ROD EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC	Rod element effective strain (nonlinear elastic material) or plastic strain (elastic-plastic material). Controlled by NLSTRESS Case Control command.
3292	ROD EFFECTIVE STRAIN-CREEP	Rod element effective creep strain. Controlled by NLSTRESS Case Control command.
3481	ROD STATUS	Solution and option dependent. In modal summation solutions (DDAM) STATUS, is the mode number with the maximum response in the NRL summation.
3683	ROD AXIAL STRAIN	Rod element axial strain. Controlled by STRAIN Case Control commands.
3686	ROD TORSIONAL STRAIN	Rod element torsional strain. Controlled by STRAIN Case Control commands.
3998	ROD DAMAGE	Rod element fatigue damage. Controlled by FATIGUE, STRESS, and STRAIN Case Control commands.
3999	ROD LIFE	Rod element fatigue life. Controlled by FATIGUE, STRESS, and STRAIN Case Control commands.

Bar Element Results Column Descriptions:

Vector Id	Label	Description
3000	BAR MOMENT END A-Z PLANE 1	Bar element bending moment at end A about element z-direction. Controlled by FORCE Case Control command.
3001	BAR MOMENT END A-Y PLANE 2	Bar element bending moment at end A about element y-direction. Controlled by FORCE Case Control command.
3002	BAR MOMENT END B-Z PLANE 1	Bar element bending moment at end B about element z-direction. Controlled by FORCE Case Control command.
3003	BAR MOMENT END B-Y PLANE 2	Bar element bending moment at end B about element y-direction. Controlled by FORCE Case Control command.
3004	BAR FORCE END A-Y PLANE 1	Bar element transverse shear force at end A in element y-direction. Controlled by FORCE Case Control command.
3005	BAR FORCE END A-Z PLANE 2	Bar element transverse shear force at end A in element z-direction. Controlled by FORCE Case Control command.
3006	BAR FORCE END B-Y PLANE 1	Bar element transverse shear force at end B in element y-direction. Controlled by FORCE Case Control command.
3007	BAR FORCE END B-Z PLANE 2	Bar element transverse shear force at end B in element z-direction. Controlled by FORCE Case Control command.
3008	BAR FORCE END A-X	Bar element axial force at end A in element x-direction. Controlled by FORCE Case Control command.
3009	BAR FORCE END B-X	Bar element axial force at end B in element x-direction. Controlled by FORCE Case Control command.
3010	BAR MOMENT END A-X	Bar element torque at end A about element x-direction. Controlled by FORCE Case Control command.
3011	BAR MOMENT END B-X	Bar element torque at end B about element x-direction. Controlled by FORCE Case Control command.
3075	BAR STRESS END A POINT C	Bar element stress at end A, stress recovery point C. For bar elements this stress will only include bending contributions. Controlled by STRESS Case Control command.
3076	BAR STRESS END A POINT D	Bar element stress at end A, stress recovery point D. For bar elements this stress will only include bending contributions. Controlled by STRESS Case Control command.
3077	BAR STRESS END A POINT E	Bar element stress at end A, stress recovery point E. For bar elements this stress will only include bending contributions. Controlled by STRESS Case Control command.
3078	BAR STRESS END A POINT F	Bar element stress at end A, stress recovery point F. For bar elements this stress will only include bending contributions. Controlled by STRESS Case Control command.
3083	BAR STRESS END B POINT C	Bar element stress at end B, stress recovery point C. For bar elements this stress will only include bending contributions. Controlled by STRESS Case Control command.
3084	BAR STRESS END B POINT D	Bar element stress at end B, stress recovery point D. For bar elements this stress will only include bending contributions. Controlled by STRESS Case Control command.

Bar Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3085	BAR STRESS END B POINT E	Bar element stress at end B, stress recovery point E. For bar elements this stress will only include bending contributions. Controlled by STRESS Case Control command.
3086	BAR STRESS END B POINT F	Bar element stress at end B, stress recovery point F. For bar elements this stress will only include bending contributions. Controlled by STRESS Case Control command.
3107	BAR STRESS END A-AXIAL	Bar element axial stress at end A. Controlled by STRESS Case Control command.
3108	BAR STRESS END B-AXIAL	Bar element axial stress at end B. Controlled by STRESS Case Control command.
3109	BAR STRESS END A-MAX	Bar element maximum stress (bending and axial) for all points at end A. Controlled by STRESS Case Control command.
3110	BAR STRESS END A-MIN	Bar element minimum stress (bending and axial) for all points at end A. Controlled by STRESS Case Control command.
3111	BAR STRESS END B-MAX	Bar element maximum stress (bending and axial) for all points at end B. Controlled by STRESS Case Control command.
3112	BAR STRESS END B-MIN	Bar element minimum stress (bending and axial) for all points at end B. Controlled by STRESS Case Control command.
3195	BAR VON MISES STRESS	Bar element von Mises stress. Controlled by STRESS Case Control command.
3293	BAR EQUIVALENT STRESS	Bar element nonlinear equivalent axial stress (material nonlinear solutions) or axial stress (linear solutions). Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
3293	BAR VON MISES STRESS-BENDING	Bar element von Mises stress computed without membrane stress contribution. Controlled by STRESS or STRAIN Case Control commands and PARAM, EQVSTRESSTYPE setting.
3293	BAR VON MISES STRESS-MEMBRANE	Bar element von Mises stress computed without bending stress contribution. Controlled by STRESS or STRAIN Case Control commands and PARAM, EQVSTRESSTYPE setting.
3294	BAR EFFECTIVE STRAIN-ELASTIC	Bar element effective axial strain. Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
3294	BAR EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC	Bar element effective strain (nonlinear elastic material) or plastic strain (elastic-plastic material). Controlled by NLSTRESS Case Control command.
3295	BAR EFFECTIVE STRAIN-CREEP	Bar element effective creep strain. Controlled by NLSTRESS Case Control command.
3440	BAR MAX STRESS	Bar element maximum stress (bending and axial) for all points at end A and B. Controlled by STRESS Case Control command.

Bar Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3441	BAR MIN STRESS	Bar element minimum stress (bending and axial) for all points at end A and B. Controlled by STRESS Case Control command.
3442	BAR LOCATION A	Bar element end A location.
3443	BAR LOCATION B	Bar element end B location.
3481	BAR STATUS	Solution and option dependent. In modal summation solutions (DDAM) STATUS, is the mode number with the maximum response in the NRL summation. In solutions where a factor of safety calculation method has been defined on a MAT1 entry, STATUS is the factor of safety.
3575	BAR STRAIN END A POINT C	Bar element strain at end A, strain recovery point C. For bar elements this stress will only include bending contributions. Controlled by STRAIN Case Control command.
3576	BAR STRAIN END A POINT D	Bar element strain at end A, strain recovery point D. For bar elements this strain will only include bending contributions. Controlled by STRAIN Case Control command.
3577	BAR STRAIN END A POINT E	Bar element strain at end A, strain recovery point E. For bar elements this strain will only include bending contributions. Controlled by STRAIN Case Control command.
3578	BAR STRAIN END A POINT F	Bar element strain at end A, strain recovery point F. For bar elements this strain will only include bending contributions. Controlled by STRAIN Case Control command.
3583	BAR STRAIN END B POINT C	Bar element strain at end B, strain recovery point C. For bar elements this strain will only include bending contributions. Controlled by STRAIN Case Control command.
3584	BAR STRAIN END B POINT D	Bar element strain at end B, strain recovery point D. For bar elements this strain will only include bending contributions. Controlled by STRAIN Case Control command.
3585	BAR STRAIN END B POINT E	Bar element strain at end B, strain recovery point E. For bar elements this strain will only include bending contributions. Controlled by STRAIN Case Control command.
3586	BAR STRAIN END B POINT F	Bar element strain at end B, strain recovery point F. For bar elements this strain will only include bending contributions. Controlled by STRAIN Case Control command.
3607	BAR STRAIN END A-AXIAL	Bar element axial strain at end A. Controlled by STRAIN Case Control command.
3608	BAR STRAIN END B-AXIAL	Bar element axial strain at end B. Controlled by STRAIN Case Control command.
3609	BAR STRAIN END A-MAX	Bar element maximum strain (bending and axial) for all points at end A. Controlled by STRAIN Case Control command.
3610	BAR STRAIN END A-MIN	Bar element minimum strain (bending and axial) for all points at end A. Controlled by STRAIN Case Control command.
3611	BAR STRAIN END B-MAX	Bar element maximum strain (bending and axial) for all points at end B. Controlled by STRAIN Case Control command.

Bar Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3612	BAR STRAIN END B-MIN	Bar element minimum strain (bending and axial) for all points at end B. Controlled by STRAIN Case Control command.
3695	BAR VON MISES STRAIN	Bar element von Mises strain. Controlled by STRAIN Case Control command.
3940	BAR MAX STRAIN	Bar element maximum strain (bending and axial) for all points at ends A and B. Controlled by STRAIN Case Control command.
3941	BAR MIN STRAIN	Bar element minimum strain (bending and axial) for all points at ends A and B. Controlled by STRAIN Case Control command.
3998	BAR DAMAGE	Bar element fatigue damage. Controlled by FATIGUE, STRESS, and STRAIN Case Control commands.
3999	BAR LIFE	Bar element fatigue life. Controlled by FATIGUE, STRESS, and STRAIN Case Control commands.

Beam Element Results Column Descriptions:

Vector Id	Label	Description
3014	BEAM MOMENT END A-Z PLANE 1	Beam element bending moment at end A about element z-direction. Controlled by FORCE Case Control command.
3015	BEAM MOMENT END A-Y PLANE 2	Beam element bending moment at end A about element y-direction. Controlled by FORCE Case Control command.
3016	BEAM MOMENT END B-Z PLANE 1	Beam element bending moment at end B about element z-direction. Controlled by FORCE Case Control command.
3017	BEAM MOMENT END B-Y PLANE 2	Beam element bending moment at end B about element y-direction. Controlled by FORCE Case Control command.
3018	BEAM FORCE END A-Y PLANE 1	Beam element transverse shear force at end A in element y-direction. Controlled by FORCE Case Control command.
3019	BEAM FORCE END A-Z PLANE 2	Beam element transverse shear force at end A in element z-direction. Controlled by FORCE Case Control command.
3020	BEAM FORCE END B-Y PLANE 1	Beam element transverse shear force at end B in element y-direction. Controlled by FORCE Case Control command.
3021	BEAM FORCE END B-Z PLANE 2	Beam element transverse shear force at end B in element z-direction. Controlled by FORCE Case Control command.
3022	BEAM FORCE END A-X	Beam element axial force at end A in element x-direction. Controlled by FORCE Case Control command.
3023	BEAM FORCE END B-X	Beam element axial force at end B in element x-direction. Controlled by FORCE Case Control command.
3024	BEAM MOMENT END A-X	Beam element torque at end A about element x-direction. Controlled by FORCE Case Control command.
3025	BEAM MOMENT END B-X	Beam element torque at end B about element x-direction. Controlled by FORCE Case Control command.
3139	BEAM STRESS END A POINT C	Beam element stress at end A, stress recovery point C. For beam elements this stress will include both bending and axial contributions. Controlled by STRESS Case Control commands.
3140	BEAM STRESS END A POINT D	Beam element stress at end A, stress recovery point D. For beam elements this stress will include both bending and axial contributions. Controlled by STRESS Case Control commands.
3141	BEAM STRESS END A POINT E	Beam element stress at end A, stress recovery point E. For beam elements this stress will include both bending and axial contributions. Controlled by STRESS Case Control commands.
3142	BEAM STRESS END A POINT F	Beam element stress at end A, stress recovery point F. For beam elements this stress will include both bending and axial contributions. Controlled by STRESS Case Control commands.
3151	BEAM STRESS END B POINT C	Beam element stress at end B, stress recovery point C. For beam elements this stress will include both bending and axial contributions. Controlled by STRESS Case Control commands.
3152	BEAM STRESS END B POINT D	Beam element stress at end B, stress recovery point D. For beam elements this stress will include both bending and axial contributions. Controlled by STRESS Case Control commands.

Beam Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3153	BEAM STRESS END B POINT E	Beam element stress at end B, stress recovery point E. For beam elements this stress will include both bending and axial contributions. Controlled by STRESS Case Control commands.
3154	BEAM STRESS END B POINT F	Beam element stress at end B, stress recovery point F. For beam elements this stress will include both bending and axial contributions. Controlled by STRESS Case Control commands.
3164	BEAM STRESS END A-MAX	Beam element maximum stress (bending and axial) for all points at end A. Controlled by STRESS Case Control commands.
3165	BEAM STRESS END A-MIN	Beam element minimum stress (bending and axial) for all points at end B. Controlled by STRESS Case Control commands.
3166	BEAM STRESS END B-MAX	Beam element maximum stress (bending and axial) for all points at end A. Controlled by STRESS Case Control commands.
3167	BEAM STRESS END B-MIN	Beam element minimum stress (bending and axial) for all points at end A. Controlled by STRESS Case Control commands.
3170	BEAM STRESS END A-AXIAL	Beam element axial stress at end A. Controlled by STRESS Case Control command.
3176	BEAM STRESS END B-AXIAL	Beam element axial stress at end B. Controlled by STRESS Case Control command.
3195	BEAM VON MISES STRESS	Beam element von Mises stress. Controlled by STRESS Case Control command.
3296	BEAM EQUIVALENT STRESS	Beam element nonlinear equivalent axial stress (material nonlinear solutions) or axial stress (linear solutions). Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
3296	BEAM VON MISES STRESS-BENDING	Beam element von Mises stress computed without membrane stress contribution. Controlled by STRESS or STRAIN Case Control commands and PARAM, EQVSTRESSTYPE setting.
3296	BEAM VON MISES STRESS-MEMBRANE	Beam element von Mises stress computed without bending stress contribution. Controlled by STRESS or STRAIN Case Control commands and PARAM, EQVSTRESSTYPE setting.
3297	BEAM EFFECTIVE STRAIN-ELASTIC	Beam element effective axial strain. Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
3297	BEAM EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC	Beam element effective strain (nonlinear elastic material) or plastic strain (elastic-plastic material). Controlled by NLSTRESS Case Control command.
3298	BEAM EFFECTIVE STRAIN-CREEP	Beam element effective creep strain. Controlled by NLSTRESS Case Control command.
3446	BEAM MAX STRESS	Beam element maximum stress (bending and axial) for all points at ends A and B. Controlled by STRESS Case Control command.

Beam Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3447	BEAM MIN STRESS	Beam element minimum stress (bending and axial) for all points at ends A and B. Controlled by STRESS Case Control command.
3448	BEAM LOCATION A	Beam element end A location.
3449	BEAM LOCATION B	Beam element end B location.
3481	BEAM STATUS	Solution and option dependent. In modal summation solutions (DDAM), STATUS is the mode number with the maximum response in the NRL summation. In solutions where a factor of safety calculation method has been defined on a MAT1 entry, STATUS is the factor of safety.
3639	BEAM STRAIN END A POINT C	Beam element stress at end A, stress recovery point C. For beam elements this strain will include both bending and axial contributions. Controlled by STRAIN Case Control commands.
3640	BEAM STRAIN END A POINT D	Beam element strain at end A, strain recovery point D. For beam elements this strain will include both bending and axial contributions. Controlled by STRAIN Case Control commands.
3641	BEAM STRAIN END A POINT E	Beam element STRAIN at end A, STRAIN recovery point E. For beam elements this strain will include both bending and axial contributions. Controlled by STRAIN Case Control commands.
3642	BEAM STRAIN END A POINT F	Beam element STRAIN at end A, STRAIN recovery point F. For beam elements this strain will include both bending and axial contributions. Controlled by STRAIN Case Control commands.
3651	BEAM STRAIN END B POINT C	Beam element STRAIN at end B, STRAIN recovery point C. For beam elements this strain will include both bending and axial contributions. Controlled by STRAIN Case Control commands.
3652	BEAM STRAIN END B POINT D	Beam element STRAIN at end B, STRAIN recovery point D. For beam elements this strain will include both bending and axial contributions. Controlled by STRAIN Case Control commands.
3653	BEAM STRAIN END B POINT E	Beam element stress at end B, stress recovery point E. For beam elements this strain will include both bending and axial contributions. Controlled by STRAIN Case Control commands.
3654	BEAM STRAIN END B POINT F	Beam element stress at end B, stress recovery point F. For beam elements this strain will include both bending and axial contributions. Controlled by STRAIN Case Control commands.
3664	BEAM STRAIN END A-MAX	Beam element maximum strain (bending and axial) for all points at end A. Controlled by STRAIN Case Control commands.
3665	BEAM STRAIN END A-MIN	Beam element minimum strain (bending and axial) for all points at end B. Controlled by STRAIN Case Control commands.
3666	BEAM STRAIN END B-MAX	Beam element maximum strain (bending and axial) for all points at end A. Controlled by STRAIN Case Control commands.
3667	BEAM STRAIN END B-MIN	Beam element minimum strain (bending and axial) for all points at end A. Controlled by STRAIN Case Control commands.
3670	BEAM STRAIN END A-AXIAL	Beam element axial strain at end A. Controlled by STRAIN Case Control command.

Beam Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3676	BEAM STRAIN END B-AXIAL	Beam element axial strain at end B. Controlled by STRAIN Case Control command.
3695	BEAM VON MISES STRAIN	Beam element von Mises strain. Controlled by STRAIN Case Control command.
3948	BEAM MAX STRAIN	Beam element maximum strain (bending and axial) for all points at ends A and B. Controlled by STRAIN Case Control command.
3949	BEAM MIN STRAIN	Beam element minimum strain (bending and axial) for all points at ends A and B. Controlled by STRAIN Case Control command.
3998	BEAM DAMAGE	Beam element fatigue damage. Controlled by FATIGUE, STRESS, and STRAIN Case Control commands.
3999	BEAM LIFE	Beam element fatigue life. Controlled by FATIGUE, STRESS, and STRAIN Case Control commands.

Pipe Element Results Column Descriptions:

Vector Id	Label	Description
3223	PIPE EFFECTIVE STRAIN-CREEP	Pipe element effective creep strain. Controlled by NLSTRESS Case Control command.
3222	PIPE EFFECTIVE STRAIN-ELASTIC	Pipe element effective axial strain. Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
3222	PIPE EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC	Pipe element effective strain (nonlinear elastic material) or plastic strain (elastic-plastic material). Controlled by NLSTRESS Case Control command.
3221	PIPE EQUIVALENT STRESS	Pipe element nonlinear equivalent axial stress (material nonlinear solutions) or axial stress (linear solutions). Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
3314	PIPE FORCE END A-X	Pipe element axial force at end A in element x-direction. Controlled by FORCE Case Control command.
3310	PIPE FORCE END A-Y PLANE 1	Pipe element transverse shear force at end A in element y-direction. Controlled by FORCE Case Control command.
3311	PIPE FORCE END A-Z PLANE 2	Pipe element transverse shear force at end A in element z-direction. Controlled by FORCE Case Control command.
3315	PIPE FORCE END B-X	Pipe element axial force at end B in element x-direction. Controlled by FORCE Case Control command.
3312	PIPE FORCE END B-Y PLANE 1	Pipe element transverse shear force at end B in element y-direction. Controlled by FORCE Case Control command.
3313	PIPE FORCE END B-Z PLANE 2	Pipe element transverse shear force at end B in element z-direction. Controlled by FORCE Case Control command.
3224	PIPE LOCATION A	Pipe element end A location.
3225	PIPE LOCATION B	Pipe element end B location.
3316	PIPE MOMENT END A-X	Pipe element torque at end A about element x-direction. Controlled by FORCE Case Control command.
3307	PIPE MOMENT END A-Y PLANE 2	Pipe element bending moment at end A about element y-direction. Controlled by FORCE Case Control command.
3306	PIPE MOMENT END A-Z PLANE 1	Pipe element bending moment at end A about element z-direction. Controlled by FORCE Case Control command.
3317	PIPE MOMENT END B-X	Pipe element torque at end B about element x-direction. Controlled by FORCE Case Control command.
3309	PIPE MOMENT END B-Y PLANE 2	Pipe element bending moment at end B about element y-direction. Controlled by FORCE Case Control command.
3308	PIPE MOMENT END B-Z PLANE 1	Pipe element bending moment at end B about element z-direction. Controlled by FORCE Case Control command.

Pipe Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3206	PIPE STRESS END A-HOOP	Pipe element hoop stress at end A. Controlled by STRESS Case Control command.
3205	PIPE STRESS END A-LONGITUDINAL	Pipe element longitudinal stress at end A. Controlled by STRESS Case Control command.
3209	PIPE STRESS END A-MAX PRINCIPAL	Pipe element maximum principal stress at end A. Controlled by STRESS Case Control command.
3210	PIPE STRESS END A-MAX SHEAR	Pipe element maximum shear stress at end A. Controlled by STRESS Case Control command.
3211	PIPE STRESS END A-OCTAHEDRAL	Pipe element maximum principal stress at end A. Controlled by STRESS Case Control command.
3208	PIPE STRESS END A-SHEAR	Pipe element shear stress at end A. Controlled by STRESS Case Control command.
3207	PIPE STRESS END A-TORSIONAL	Pipe element torsional stress at end A. Controlled by STRESS Case Control command.
3213	PIPE STRESS END B-HOOP	Pipe element hoop stress at end B. Controlled by STRESS Case Control command.
3212	PIPE STRESS END B-LONGITUDINAL	Pipe element longitudinal stress at end B. Controlled by STRESS Case Control command.
3216	PIPE STRESS END B-MAX PRINCIPAL	Pipe element maximum principal stress at end B. Controlled by STRESS Case Control command.
3217	PIPE STRESS END B-MAX SHEAR	Pipe element maximum shear stress at end B. Controlled by STRESS Case Control command.
3218	PIPE STRESS END B-OCTAHEDRAL	Pipe element octahedral stress at end B. Controlled by STRESS Case Control command.
3215	PIPE STRESS END B-SHEAR	Pipe element shear stress at end B. Controlled by STRESS Case Control command.
3214	PIPE STRESS END B-TORSIONAL	Pipe element torsional stress at end B. Controlled by STRESS Case Control command.
3219	PIPE STRESS-MAX PRINCIPAL	Pipe element maximum principal stress at end A and B. Controlled by STRESS Case Control command.
3220	PIPE STRESS-OCTAHEDRAL	Pipe element octahedral stress at end A and B. Controlled by STRESS Case Control command.
3481	PIPE STATUS	Solution and option dependent. In modal summation solutions (DDAM) STATUS, is the mode number with the maximum response in the NRL summation.

Weld Element Results Column Descriptions:

Vector Id	Label	Description
3301	WELD EFFECTIVE STRAIN-CREEP	Weld element effective creep strain. Controlled by NLSTRESS Case Control command.
3300	WELD EFFECTIVE STRAIN-ELASTIC	Weld element effective axial strain. Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
3300	WELD EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC	Weld element effective strain (nonlinear elastic material) or plastic strain (elastic-plastic material). Controlled by NLSTRESS Case Control command.
3299	WELD EQUIVALENT STRESS	Weld element nonlinear equivalent axial stress (material nonlinear solutions) or axial stress (linear solutions). Note that for prestress solutions, regardless of PARAM, ADDPRESTRESS setting, equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
3328	WELD FORCE END A-X	Weld element axial force at end A in element x-direction. Controlled by FORCE Case Control command.
3324	WELD FORCE END A-Y PLANE 1	Weld element transverse shear force at end A in element y-direction. Controlled by FORCE Case Control command.
3325	WELD FORCE END A-Z PLANE 2	Weld element transverse shear force at end A in element z-direction. Controlled by FORCE Case Control command.
3329	WELD FORCE END B-X	Weld element axial force at end B in element x-direction. Controlled by FORCE Case Control command.
3326	WELD FORCE END B-Y PLANE 1	Weld element transverse shear force at end B in element y-direction. Controlled by FORCE Case Control command.
3327	WELD FORCE END B-Z PLANE 2	Weld element transverse shear force at end B in element z-direction. Controlled by FORCE Case Control command.
3254	WELD LOCATION A	Weld element end A location.
3255	WELD LOCATION B	Weld element end B location.
3330	WELD MOMENT END A-X	Weld element torque at end A about element x-direction. Controlled by FORCE Case Control command.
3321	WELD MOMENT END A-Y PLANE 2	Weld element bending moment at end A about element y-direction. Controlled by FORCE Case Control command.
3320	WELD MOMENT END A-Z PLANE 1	Weld element bending moment at end A about element z-direction. Controlled by FORCE Case Control command.
3331	WELD MOMENT END B-X	Weld element torque at end B about element x-direction. Controlled by FORCE Case Control command.
3323	WELD MOMENT END B-Y PLANE 2	Weld element bending moment at end B about element y-direction. Controlled by FORCE Case Control command.
3322	WELD MOMENT END B-Z PLANE 1	Weld element bending moment at end B about element z-direction. Controlled by FORCE Case Control command.

Weld Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3240	WELD STRESS END A-LONGITUDINAL	Weld element longitudinal stress at end A. Controlled by STRESS Case Control command.
3243	WELD STRESS END A-MAX PRINCIPAL	Weld element maximum principal stress at end A. Controlled by STRESS Case Control command.
3244	WELD STRESS END A-MAX SHEAR	Weld element maximum shear stress at end A. Controlled by STRESS Case Control command.
3242	WELD STRESS END A-SHEAR	Weld element shear stress at end A. Controlled by STRESS Case Control command.
3241	WELD STRESS END A-TORSIONAL	Weld element torsional stress at end A. Controlled by STRESS Case Control command.
3245	WELD STRESS END B-LONGITUDINAL	Weld element longitudinal stress at end B. Controlled by STRESS Case Control command.
3248	WELD STRESS END B-MAX PRINCIPAL	Weld element maximum principal stress at end B. Controlled by STRESS Case Control command.
3249	WELD STRESS END B-MAX SHEAR	Weld element maximum shear stress at end B. Controlled by STRESS Case Control command.
3247	WELD STRESS END B-SHEAR	Weld element shear stress at end B. Controlled by STRESS Case Control command.
3246	WELD STRESS END B-TORSIONAL	Weld element torsional stress at end B. Controlled by STRESS Case Control command.
3250	WELD STRESS-MAX PRINCIPAL	Weld element maximum principal stress at end A and B. Controlled by STRESS Case Control command.
3481	WELD STATUS	Solution and option dependent. In modal summation solutions (DDAM) STATUS, is the mode number with the maximum response in the NRL summation.

Gap Element Results Column Descriptions:

Vector Id	Label	Description
3226	GAP AXIAL FORCE	Gap element axial force (contact force). Controlled by FORCE or STRESS Case Control command.
3227	GAP RESULTANT SHEAR FORCE	Gap element shear force (due to friction) vector resultant. Controlled by FORCE or STRESS Case Control command.
3228	GAP SHEAR FORCE-Y	Gap element shear force (due to friction) in element y-direction. Controlled by FORCE or STRESS Case Control command.
3229	GAP SHEAR FORCE-Z	Gap element shear force (due to friction) in element z-direction. Controlled by FORCE or STRESS Case Control command.
3230	GAP AXIAL DISPLACEMENT	Gap element axial displacement. Controlled by FORCE or STRESS Case Control command.
3231	GAP TOTAL DISPLACEMENT-Y	Gap element total displacement in element y-direction. Controlled by FORCE or STRESS Case Control command.
3232	GAP TOTAL DISPLACEMENT-Z	Gap element total displacement in element z-direction. Controlled by FORCE or STRESS Case Control command.
3233	GAP SLIP DISPLACEMENT-Y	Gap element slip displacement in element y-direction. Controlled by FORCE or STRESS Case Control command.
3234	GAP SLIP DISPLACEMENT-Z	Gap element slip displacement in element z-direction. Controlled by FORCE or STRESS Case Control command.
3460	GAP STATUS	Gap element status (1=open, 2=slide – closed with no friction defined, 3=stick – closed with friction and holding, 4=slip – closed with friction and slipping). Controlled by FORCE or STRESS Case Control command.
3461	GAP RESULTANT INPLANE DISPLACEMENT	Gap element total displacement vector resultant. Controlled by FORCE or STRESS Case Control command.
3462	GAP RESULTANT SLIP DISPLACEMENT	Gap element slip displacement vector resultant. Controlled by FORCE or STRESS Case Control command.

Cable Element Results Column Descriptions:

Vector Id	Label	Description
3288	CABLE EFFECTIVE STRAIN	Cable element extensional strain. This value does not include slip. Controlled by FORCE, STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
3287	CABLE EQUIVALENT STRESS	Cable element extensional stress. Controlled by FORCE, STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
3463	CABLE FORCE	Cable element extensional force. Controlled by FORCE, STRESS, or STRAIN Case Control commands.
3466	CABLE SLIP DISPLACEMENT	Cable element slip displacement (slack). This value represents the amount of displacement before load is carried. Controlled by FORCE, STRESS, or STRAIN Case Control commands.
3467	CABLE STATUS	Solution and option dependent. In modal summation solutions (DDAM), STATUS is the mode number with the maximum response in the NRL summation. In nonlinear solutions STATUS is the cable status (1=loaded, 2=unloaded, 3=failed).
3464	CABLE STRESS	Cable element extensional stress. Controlled by FORCE, STRESS, or STRAIN Case Control commands.
3465	CABLE TOTAL DISPLACEMENT	Cable element total displacement, slack plus extension. Controlled by FORCE, STRESS, or STRAIN Case Control commands.

Shell Element Results Column Descriptions:

Vector Id	Label	Description
6036	SHELL MAX PRINCIPAL STRESS BOTTOM/TOP	Shell element maximum principal stress (of bottom and top). Controlled by STRESS Case Control command.
6037	SHELL MIN PRINCIPAL STRESS BOTTOM/TOP	Shell element minimum principal stress (of bottom and top). Controlled by STRESS Case Control command.
6038	SHELL MAX TRESCA STRESS BOTTOM/TOP	Shell element maximum Tresca stress (of bottom and top). Controlled by STRESS Case Control command.
6038	SHELL MAX MAX SHEAR STRESS BOTTOM/TOP	Shell element maximum maximum shear stress (of bottom and top). Controlled by STRESS Case Control command.
6039	SHELL MAX VON MISES STRESS BOTTOM/TOP	Shell element maximum von Mises stress. Controlled by STRESS Case Control command.
6043	SHELL FIBER DISTANCE TOP	Shell element stress/strain recovery distance (element z-direction) for top side (side 2).
6044	SHELL EFFECTIVE STRAIN-ELASTIC BOTTOM	Shell element bottom side (side 1) effective strain (von Mises). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
6044	SHELL FIBER DISTANCE BOTTOM	Shell element stress/strain recovery distance (element z-direction) for bottom side (side 1).
6046	SHELL STATUS	Solution and option dependent. In modal summation solutions (DDAM) STATUS is the mode number with the maximum response in the NRL summation. In nonlinear solutions with tension-only shell elements STATUS is the reversion status code (1=reverted to a shear panel or tension-only element, 0=has not reverted). In solutions where a factor of safety calculation method has been defined on a MAT1 entry, STATUS is the factor of safety. In topological optimization solutions STATUS is the element density.
6105	SHELL MAX PRINCIPAL STRAIN BOTTOM/TOP	Shell element maximum principal strain (of bottom and top). Controlled by STRAIN Case Control command.
6106	SHELL MIN PRINCIPAL STRAIN BOTTOM/TOP	Shell element minimum principal strain (of bottom and top). Controlled by STRAIN Case Control command.
6107	SHELL MAX TRESCA STRAIN BOTTOM/TOP	Shell element maximum Tresca strain (of bottom and top). Controlled by STRAIN Case Control command.
6107	SHELL MAX SHEAR STRAIN BOTTOM/TOP	Shell element maximum maximum shear strain (of bottom and top). Controlled by STRAIN Case Control command.
6108	SHELL MAX VON MISES STRAIN BOTTOM/TOP	Shell element maximum von Mises strain (of bottom and top). Controlled by STRAIN Case Control command.
6175	SHELL MAX DAMAGE BOTTOM/TOP	Shell element maximum fatigue damage (of bottom and top). Controlled by FATIGUE, VIBFATIGUE, and STRESS Case Control commands.
6176	SHELL MIN LIFE BOTTOM/TOP	Shell element minimum fatigue life (of bottom and top). Controlled by VIBFATIGUE and STRESS Case Control commands.
7020	SHELL NORMAL-X STRESS TOP	Shell element top side (side 2) normal stress in SURFACE x-direction. Controlled by STRESS Case Control command.

Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
7021	SHELL NORMAL-Y STRESS TOP	Shell element top side (side 2) normal stress in SURFACE y-direction. Controlled by STRESS Case Control command.
7023	SHELL SHEAR-XY STRESS TOP	Shell element top side (side 2) shear stress in SURFACE xy-direction (tensor x-face, y-direction). Controlled by STRESS Case Control command.
7026	SHELL MAJOR PRINCIPAL STRESS TOP	Shell element top side (side 2) major principal stress. Controlled by STRESS Case Control command.
7027	SHELL MINOR PRINCIPAL STRESS TOP	Shell element top side (side 2) minor principal stress. Controlled by STRESS Case Control command.
7029	SHELL ZERO SHEAR STRESS ANGLE TOP	Shell element top side (side 2) zero shear stress angle in degrees. Controlled by STRESS Case Control command.
7031	SHELL MAX SHEAR STRESS TOP	Shell element top side (side 2) maximum shear stress. Controlled by STRESS Case Control command.
7031	SHELL TRESCA STRESS TOP	Shell element top side (side 2) Tresca stress. Controlled by STRESS Case Control command.
7032	SHELL EQUIVALENT STRESS TOP	Shell element top side (side 2) nonlinear equivalent stress (material nonlinear solutions) or von Mises stress (linear solutions). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
7032	SHELL VON MISES STRESS-BENDING TOP	Shell element top side (side 2) von Mises stress computed without membrane stress contribution. Controlled by STRESS Case Control command and PARAM, EQVSTRESSTYPE setting.
7032	SHELL VON MISES STRESS-MEMBRANE TOP	Shell element top side (side 2) von Mises stress computed without bending stress contribution. Controlled by STRESS Case Control command and PARAM, EQVSTRESSTYPE setting.
7033	SHELL VON MISES STRESS TOP	Shell element top side (side 2) von Mises stress. Controlled by STRESS Case Control command.
7065	SHELL NORMAL-X STRAIN TOP	Shell element top side (side 2) normal strain in SURFACE x-direction. Controlled by STRAIN Case Control command.
7066	SHELL NORMAL-Y STRAIN TOP	Shell element top side (side 2) normal strain in SURFACE y-direction. Controlled by STRAIN Case Control command.
7068	SHELL SHEAR-XY STRAIN TOP	Shell element top side (side 2) shear strain in SURFACE xy-direction (tensor x-face, y-direction). Controlled by STRAIN Case Control command.
7071	SHELL MAJOR PRINCIPAL STRAIN TOP	Shell element top side (side 2) major principal strain. Controlled by STRAIN Case Control command.
7072	SHELL MINOR PRINCIPAL STRAIN TOP	Shell element top side (side 2) minor principal strain. Controlled by STRAIN Case Control command.
7074	SHELL ZERO SHEAR STRAIN ANGLE TOP	Shell element top side (side 2) zero shear strain angle in degrees. Controlled by STRAIN Case Control command.

Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
7076	SHELL MAX SHEAR STRAIN TOP	Shell element top side (side 2) maximum shear strain. Controlled by STRAIN Case Control command.
7076	SHELL TRESCA STRAIN TOP	Shell element top side (side 2) Tresca strain . Controlled by STRAIN Case Control command.
7077	SHELL VON MISES STRAIN TOP	Shell element top side (side 2) von Mises strain. Controlled by STRAIN Case Control command.
7088	SHELL EFFECTIVE STRAIN-ELASTIC TOP	Shell element top side (side 2) effective strain (von Mises). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
7088	SHELL EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC TOP	Shell element top side (side 2) effective (nonlinear elastic material) or plastic (elastic-plastic material) strain. Controlled by NLSTRESS Case Control command.
7089	SHELL EFFECTIVE STRAIN-CREEP TOP	Shell element top side (side 2) effective creep strain. Controlled by NLSTRESS Case Control command.
7122	SHELL BIAXIALITY RATIO BOTTOM	Shell element bottom side (side 1) stress biaxiality ratio. Controlled by STRESS Case Control command.
7123	SHELL DAMAGE BOTTOM	Shell element bottom side (side 1) fatigue damage. Controlled by FATIGUE, VIBFATIGUE, and STRESS Case Control commands.
7124	SHELL LIFE BOTTOM	Shell element bottom side (side 1) fatigue life. Controlled by FATIGUE, VIBFATIGUE, and STRESS Case Control commands.
7125	SHELL BIAXIALITY RATIO BOTTOM	Shell element bottom side (side 1) strain biaxiality ratio. Controlled by STRAIN Case Control command.
7206	SHELL MEMBRANE FORCE-FX	Shell element inplane normal force per unit length in SURFACE x-direction. Controlled by FORCE Case Control command.
7207	SHELL MEMBRANE FORCE-FY	Shell element inplane normal force per unit length in SURFACE y-direction. Controlled by FORCE Case Control command.
7208	SHELL MEMBRANE FORCE-FXY	Shell element inplane shear force per unit length in SURFACE xy-direction (tensor x-face, y-direction). Controlled by FORCE Case Control command.
7211	SHELL BENDING MOMENT-MX	Shell element bending moment per unit length in SURFACE x-direction. Controlled by FORCE Case Control command.
7212	SHELL BENDING MOMENT-MY	Shell element bending moment per unit length in SURFACE y-direction. Controlled by FORCE Case Control command.
7213	SHELL BENDING MOMENT-MXY	Shell element twisting moment per unit length in SURFACE xy-direction (tensor x-face, y-direction). Controlled by FORCE Case Control command.
7214	SHELL TRANSVERSE SHEAR FORCE-QX	Shell element transverse shear force per unit length in SURFACE xz-direction. Controlled by FORCE Case Control command.
7215	SHELL TRANSVERSE SHEAR FORCE-QY	Shell element transverse shear force per unit length in SURFACE yz-direction. Controlled by FORCE Case Control command.

Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
7420	SHELL NORMAL-X STRESS BOTTOM	Shell element bottom side (side 1) normal stress in SURFACE x-direction. Controlled by STRESS Case Control command.
7421	SHELL NORMAL-Y STRESS BOTTOM	Shell element bottom side (side 1) normal stress in SURFACE y-direction. Controlled by STRESS Case Control command.
7423	SHELL SHEAR-XY STRESS BOTTOM	Shell element bottom side (side 1) shear stress in SURFACE xy-direction (tensor x-face, y-direction). Controlled by STRESS Case Control command.
7426	SHELL MAJOR PRINCIPAL STRESS BOTTOM	Shell element bottom side (side 1) major principal stress. Controlled by STRESS Case Control command.
7427	SHELL MINOR PRINCIPAL STRESS BOTTOM	Shell element bottom side (side 1) minor principal stress. Controlled by STRESS Case Control command.
7429	SHELL ZERO SHEAR STRESS ANGLE BOTTOM	Shell element bottom side (side 1) zero shear stress angle in degrees. Controlled by STRESS Case Control command.
7431	SHELL MAX SHEAR STRESS BOTTOM	Shell element bottom side (side 1) maximum shear stress. Controlled by STRESS Case Control command.
7431	SHELL TRESCA STRESS BOTTOM	Shell element bottom side (side 1) Tresca stress. Controlled by STRESS Case Control command.
7432	SHELL EQUIVALENT STRESS BOTTOM	Shell element bottom side (side 1) nonlinear equivalent stress (material nonlinear solutions) or von Mises stress (linear solutions). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
7432	SHELL VON MISES STRESS-BENDING BOTTOM	Shell element bottom side (side 1) von Mises stress computed without membrane stress contribution. Controlled by STRESS Case Control command and PARAM, EQVSTRESSTYPE setting.
7432	SHELL VON MISES STRESS-MEMBRANE BOTTOM	Shell element bottom side (side 1) von Mises stress computed without bending stress contribution. Controlled by STRESS Case Control command and PARAM, EQVSTRESSTYPE setting.
7433	SHELL VON MISES STRESS BOTTOM	Shell element bottom side (side 1) von Mises stress. Controlled by STRESS Case Control command.
7465	SHELL NORMAL-X STRAIN BOTTOM	Shell element bottom side (side 1) normal strain in SURFACE x-direction. Controlled by STRAIN Case Control command.
7466	SHELL NORMAL-Y STRAIN BOTTOM	Shell element bottom side (side 1) normal strain in SURFACE y-direction. Controlled by STRAIN Case Control command.
7468	SHELL SHEAR-XY STRAIN BOTTOM	Shell element bottom side (side 1) shear strain in SURFACE xy-direction (tensor x-face, y-direction). Controlled by STRAIN Case Control command.
7471	SHELL MAJOR-PRINCIPAL STRAIN BOTTOM	Shell element bottom side (side 1) major principal strain. Controlled by STRAIN Case Control command.

Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
7472	SHELL MINOR PRINCIPAL STRAIN BOTTOM	Shell element bottom side (side 1) minor principal strain. Controlled by STRAIN Case Control command.
7474	SHELL ZERO SHEAR STRAIN ANGLE BOTTOM	Shell element bottom side (side 1) zero shear strain angle in degrees. Controlled by STRAIN Case Control command.
7476	SHELL MAX SHEAR STRAIN BOTTOM	Shell element bottom side (side 1) maximum shear strain. Controlled by STRAIN Case Control command.
7476	SHELL TRESCA STRAIN BOTTOM	Shell element bottom side (side 1) Tresca strain. Controlled by STRAIN Case Control command.
7477	SHELL VON MISES STRAIN BOTTOM	Shell element bottom side (side 1) von Mises strain. Controlled by STRAIN Case Control command.
7488	SHELL EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC BOTTOM	Shell element bottom side (side 1) effective (nonlinear elastic material) or plastic (elastic-plastic material) strain. Controlled by NLSTRESS Case Control command.
7489	SHELL EFFECTIVE STRAIN-CREEP BOTTOM	Shell element bottom side (side 1) effective creep strain. Controlled by NLSTRESS Case Control command.
7522	SHELL BIAXIALITY RATIO TOP	Shell element top side (side 2) stress biaxiality ratio. Controlled by STRESS Case Control command.
7523	SHELL DAMAGE TOP	Shell element top side (side 2) fatigue damage. Controlled by FATIGUE, VIBFATIGUE, and STRESS Case Control commands.
7524	SHELL LIFE TOP	Shell element top side (side 2) fatigue life. Controlled by FATIGUE, VIBFATIGUE, and STRESS Case Control commands.
7525	SHELL BIAXIALITY RATIO TOP	Shell element top side (side 2) strain biaxiality ratio. Controlled by STRAIN Case Control command.

Composite Shell Element Results Column Descriptions:

Vector Id	Label	Description
6054	COMP MAX EFFECTIVE STRAIN	2-Dimensional composite laminate element maximum effective strain (von Mises). Controlled by STRESS or STRAIN Case Control commands.
6055	COMP MAX EQUIVALENT STRESS	2-Dimensional composite laminate element maximum equivalent stress (von Mises). Controlled by STRESS or STRAIN Case Control commands.
6060	COMP MAX STABILITY FAILURE INDEX	2-Dimensional composite sandwich element maximum face sheet stability index. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
6061	COMP MIN STABILITY FAILURE INDEX	2-Dimensional composite sandwich element minimum face sheet stability index. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
6064	COMP MIN STABILITY ALLOWABLE	2-Dimensional composite sandwich element minimum stability allowable. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
6065	COMP MIN STABILITY ALLOWABLE PLY	2-Dimensional composite sandwich element minimum stability allowable ply. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
6066	COMP STABILITY CORE PLY	2-Dimensional composite sandwich element core ply selected by solver. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
6079	COMP MAX NORMAL-1 STRESS	2-Dimensional composite laminate element maximum ply normal stress in material 1-direction (of all plies). Controlled by STRESS Case Control command.
6080	COMP MAX NORMAL-2 STRESS	2-Dimensional composite laminate element maximum ply normal stress in material 2-direction (of all plies). Controlled by STRESS Case Control command.
6081	COMP MAX SHEAR-12 STRESS	2-Dimensional composite laminate element maximum ply shear stress in material 12-direction (of all plies). Controlled by STRESS Case Control command.
6082	COMP MAX SHEAR-XZ STRESS	2-Dimensional composite laminate element maximum interlaminar shear stress in material xz-direction (of all plies). Controlled by STRESS Case Control command.
6083	COMP MAX SHEAR-YZ STRESS	2-Dimensional composite laminate element maximum interlaminar shear stress in material yz-direction (of all plies). Controlled by STRESS Case Control command.
6084	COMP MIN NORMAL-1 STRESS	2-Dimensional composite laminate element minimum ply normal stress in material 1-direction (of all plies). Controlled by STRESS Case Control command.
6085	COMP MIN NORMAL-2 STRESS	2-Dimensional composite laminate element minimum ply normal stress in material 2-direction (of all plies). Controlled by STRESS Case Control command.
6086	COMP MIN SHEAR-12 STRESS	2-Dimensional composite laminate element minimum ply shear stress in material 12-direction (of all plies). Controlled by STRESS Case Control command.

Composite Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
6087	COMP MIN SHEAR-XZ STRESS	2-Dimensional composite laminate element minimum interlaminar shear stress in material xz-direction (of all plies). Controlled by STRESS Case Control command.
6088	COMP MIN SHEAR-YZ STRESS	2-Dimensional composite laminate element minimum interlaminar shear stress in material yz-direction (of all plies). Controlled by STRESS Case Control command.
6089	COMP MAX NORMAL-1 STRAIN	2-Dimensional composite laminate element maximum ply normal strain in material 1-direction (of all plies). Controlled by STRAIN Case Control command.
6090	COMP MAX NORMAL-2 STRAIN	2-Dimensional composite laminate element maximum ply normal strain in material 2-direction (of all plies). Controlled by STRAIN Case Control command.
6091	COMP MAX SHEAR-12 STRAIN	2-Dimensional composite laminate element maximum ply shear strain in material 12-direction (of all plies). Controlled by STRAIN Case Control command.
6092	COMP MAX SHEAR-XZ STRAIN	2-Dimensional composite laminate element maximum interlaminar shear strain in material xz-direction (of all plies). Controlled by STRAIN Case Control command.
6093	COMP MAX SHEAR-YZ STRAIN	2-Dimensional composite laminate element maximum interlaminar shear strain in material yz-direction (of all plies). Controlled by STRAIN Case Control command.
6094	COMP MIN NORMAL-1 STRAIN	2-Dimensional composite laminate element minimum ply normal strain in material 1-direction (of all plies). Controlled by STRAIN Case Control command.
6095	COMP MIN NORMAL-2 STRAIN	2-Dimensional composite laminate element minimum ply normal strain in material 2-direction (of all plies). Controlled by STRAIN Case Control command.
6096	COMP MIN SHEAR-12 STRAIN	2-Dimensional composite laminate element minimum ply shear strain in material 12-direction (of all plies). Controlled by STRAIN Case Control command.
6097	COMP MIN SHEAR-XZ STRAIN	2-Dimensional composite laminate element minimum interlaminar shear strain in material xz-direction (of all plies). Controlled by STRAIN Case Control command.
6098	COMP MIN SHEAR-YZ STRAIN	2-Dimensional composite laminate element minimum interlaminar shear strain in material yz-direction (of all plies). Controlled by STRAIN Case Control command.
6099	COMP MAX PLY FAILURE INDEX	2-Dimensional composite laminate element maximum ply failure index (of all plies). Controlled by STRESS or STRAIN Case Control commands.
6099	COMP MAX PLY STRENGTH RATIO	2-Dimensional composite laminate element maximum ply strength ratio (of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.

Composite Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
6100	COMP MAX BOND FAILURE INDEX	2-Dimensional composite laminate element maximum bond failure index (of all plies). Controlled by STRESS or STRAIN Case Control commands.
6100	COMP MAX BOND STRENGTH RATIO	2-Dimensional composite laminate element maximum bond strength ratio (of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
6101	COMP MIN PLY FAILURE INDEX	2-Dimensional composite laminate element minimum ply failure index (of all plies). Controlled by STRESS or STRAIN Case Control commands.
6101	COMP MIN PLY STRENGTH RATIO	2-Dimensional composite laminate element minimum ply strength ratio (of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
6102	COMP MIN BOND FAILURE INDEX	2-Dimensional composite laminate element minimum bond failure index (of all plies). Controlled by STRESS or STRAIN Case Control commands.
6102	COMP MIN BOND STRENGTH RATIO	2-Dimensional composite laminate element minimum bond strength ratio (of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
6103	COMP MAX FAILURE INDEX	2-Dimensional composite laminate element maximum failure index (both ply and bond of all plies). Controlled by STRESS or STRAIN Case Control commands.
6103	COMP MIN STRENGTH RATIO	2-Dimensional composite laminate element minimum strength ratio (both ply and bond of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
6104	COMP MAX FAILURE INDEX PLY	2-Dimensional composite laminate element maximum failure index ply (both ply and bond of all plies). Controlled by STRESS or STRAIN Case Control commands.
6104	COMP MIN STRENGTH RATIO PLY	2-Dimensional composite laminate element maximum failure index ply (both ply and bond of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
6109	COMP MAX PRINCIPAL STRESS	2-Dimensional composite laminate element maximum ply principal stress (of all plies). Controlled by STRESS Case Control command.
6110	COMP MIN PRINCIPAL STRESS	2-Dimensional composite laminate element minimum ply principal stress (of all plies). Controlled by STRESS Case Control command.
6111	COMP MAX MAX SHEAR STRESS	2-Dimensional composite laminate element maximum maximum shear stress (of all plies). Controlled by STRESS Case Control command.
6112	COMP MAX VON MISES STRESS	2-Dimensional composite laminate element maximum von Mises stress (of all plies). Controlled by STRESS Case Control command.
6113	COMP MAX PRINCIPAL STRAIN	2-Dimensional composite laminate element maximum ply principal strain (of all plies). Controlled by STRAIN Case Control command.
6114	COMP MIN PRINCIPAL STRAIN	2-Dimensional composite laminate element minimum ply principal strain (of all plies). Controlled by STRAIN Case Control command.

Composite Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
6115	COMP MAX MAX SHEAR STRAIN	2-Dimensional composite laminate element maximum maximum shear strain (of all plies). Controlled by STRAIN Case Control command.
6116	COMP MAX VON MISES STRAIN	2-Dimensional composite laminate element maximum von Mises strain (of all plies). Controlled by STRAIN Case Control command.
6117	COMP STATUS	2-Dimensional composite laminate element ply failure status in percent of total plies failed. Controlled by STRESS or STRAIN Case Control commands and PARAM, NLCOMPPLYFAIL.
7206	COMP MEMBRANE FORCE-FX	2-Dimensional composite laminate element inplane normal force per unit length in SURFACE x-direction. Controlled by FORCE Case Control command.
7207	COMP MEMBRANE FORCE-FY	2-Dimensional composite laminate element inplane normal force per unit length in SURFACE y-direction. Controlled by FORCE Case Control command.
7208	COMP MEMBRANE FORCE-FXY	2-Dimensional composite laminate element inplane shear force per unit length in SURFACE xy-direction (tensor x-face, y-direction). Controlled by FORCE Case Control command.
7211	COMP BENDING MOMENT-MX	2-Dimensional composite laminate element bending moment per unit length in SURFACE y-direction. Controlled by FORCE Case Control command.
7212	COMP BENDING MOMENT-MY	2-Dimensional composite laminate element bending moment per unit length in SURFACE x-direction. Controlled by FORCE Case Control command.
7213	COMP BENDING MOMENT-MXY	2-Dimensional composite laminate element twisting moment per unit length in SURFACE xy-direction (tensor x-face, y-direction). Controlled by FORCE Case Control command.
7214	COMP TRANSVERSE SHEAR FORCE-QX	2-Dimensional composite laminate element transverse shear force per unit length in SURFACE xz-direction. Controlled by FORCE Case Control command.
7215	COMP TRANSVERSE SHEAR FORCE-QY	2-Dimensional composite laminate element transverse shear force per unit length in SURFACE yz-direction. Controlled by FORCE Case Control command.
1000020 + 200(ply - 1)	COMP PLY NORMAL-1 STRESS	2-Dimensional composite laminate element ply normal stress in ply 1-direction (longitudinal). Controlled by STRESS Case Control command.
1000021 + 200(ply - 1)	COMP PLY NORMAL-2 STRESS	2-Dimensional composite laminate element ply normal stress in ply 2-direction (lateral). Controlled by STRESS Case Control command.
1000023 + 200(ply - 1)	COMP PLY SHEAR-12 STRESS	2-Dimensional composite laminate element ply normal stress in ply 12-direction. Controlled by STRESS Case Control command.
1000024 + 200(ply - 1)	COMP PLY SHEAR-XZ STRESS	2-Dimensional composite laminate element interlaminar shear stress in material xz-direction. Controlled by STRESS Case Control command.

Composite Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
1000025 + 200(ply - 1)	COMP PLY SHEAR-YZ STRESS	2-Dimensional composite laminate element interlaminar shear stress in material yz-direction. Controlled by STRESS Case Control command.
1000026 + 200(ply - 1)	COMP PLY MAX PRINCIPAL STRESS	2-Dimensional composite laminate element ply maximum principal stress. Controlled by STRESS Case Control command.
1000027 + 200(ply - 1)	COMP PLY MIN PRINCIPAL STRESS	2-Dimensional composite laminate element ply minimum principal stress. Controlled by STRESS Case Control command.
1000031 + 200(ply - 1)	COMP PLY MAX SHEAR STRESS	2-Dimensional composite laminate element ply maximum shear stress. Controlled by STRESS Case Control command.
1000032 + 200(ply - 1)	COMP PLY EQUIVALENT STRESS	2-Dimensional composite laminate element equivalent stress (von Mises). Controlled by STRESS or STRAIN Case Control commands.
1000033 + 200(ply - 1)	COMP PLY VON MISES STRESS	2-Dimensional composite laminate element von Mises stress. Controlled by STRESS Case Control command.
1000090 + 200(ply - 1)	COMP PLY PLY FAILURE INDEX	2-Dimensional composite laminate element ply ply failure index. Controlled by STRESS or STRAIN Case Control commands.
1000090 + 200(ply - 1)	COMP PLY PLY STRENGTH RATIO	2-Dimensional composite laminate element ply ply strength ratio. Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
1000091 + 200(ply - 1)	COMP PLY BOND FAILURE INDEX	2-Dimensional composite laminate element ply bond failure index. Controlled by STRESS or STRAIN Case Control commands.
1000091 + 200(ply - 1)	COMP PLY BOND STRENGTH RATIO	2-Dimensional composite laminate element ply bond strength ratio. Controlled by STRESS or STRAIN Case Control commands.
1000092 + 200(ply - 1)	COMP PLY STABILITY INDEX	2-Dimensional composite sandwich element face sheet stability index. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
1000093 + 200(ply - 1)	COMP PLY STABILITY ALLOWABLE	2-Dimensional composite sandwich element stability allowable (minimum of the wrinkling, dimpling, and crimping allowables). Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
1000094 + 200(ply - 1)	COMP PLY STABILITY ALLOWABLE FAILURE MODE	2-Dimensional composite sandwich element stability allowable failure mode (1=wrinkling, 2=dimpling, 3=crimping). Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
1000095 + 200(ply - 1)	COMP PLY STABILITY INDEX WRINKLING	2-Dimensional composite sandwich element face sheet wrinkling stability index. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
1000096 + 200(ply - 1)	COMP PLY STABILITY INDEX DIMPLING	2-Dimensional composite sandwich element face sheet dimpling stability index. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
1000097 + 200(ply - 1)	COMP PLY STABILITY INDEX CRIMPING	2-Dimensional composite sandwich element face sheet crimping stability index. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.

Composite Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
1000098 + 200(ply - 1)	COMP PLY STABILITY ALLOWABLE WRINKLING	2-Dimensional composite sandwich element wrinkling stability allowable. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
1000099 + 200(ply - 1)	COMP PLY STABILITY ALLOWABLE DIMPLING	2-Dimensional composite sandwich element dimpling stability allowable. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
1000000 + 200(ply - 1)	COMP PLY STABILITY ALLOWABLE CRIMPING	2-Dimensional composite sandwich element crimping stability allowable. Controlled by STRESS or STRAIN Case Control commands and the LAM field on the PCOMP entry.
1000101 + 200(ply – 1)	COMP PLY PLY FAILURE INDEX MATRIX-TENSION	2-Dimensional composite laminate element ply matrix-tension failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000101 + 200(ply – 1)	COMP PLY PLY FAILURE INDEX MATRIX-1	2-Dimensional composite laminate element ply matrix failure index (MCT failure theory). Fill-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000101 + 200(ply – 1)	COMP PLY PLY STRENGTH RATIO MATRIX-TENSION	2-Dimensional composite laminate element ply matrix-tension failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000101 + 200(ply – 1)	COMP PLY PLY STRENGTH RATIO MATRIX-1	2-Dimensional composite laminate element ply matrix strength ratio (MCT failure theory). Fill-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000102 + 200(ply - 1)	COMP PLY PLY FAILURE INDEX MATRIX-COMPRESSION	2-Dimensional composite laminate element ply matrix-compression failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000102 + 200(ply - 1)	COMP PLY PLY FAILURE INDEX MATRIX-2	2-Dimensional composite laminate element ply matrix failure index (MCT failure theory). Warp-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000102 + 200(ply - 1)	COMP PLY PLY STRENGTH RATIO MATRIX-COMPRESSION	2-Dimensional composite laminate element ply matrix-compression failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000102 + 200(ply - 1)	COMP PLY PLY STRENGTH RATIO MATRIX-2	2-Dimensional composite laminate element ply matrix strength ratio (MCT failure theory). Warp-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000103 + 200(ply - 1)	COMP PLY PLY FAILURE INDEX FIBER-TENSION	2-Dimensional composite laminate element ply fiber-tension failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.

Composite Shell Element Results Column Descriptions (Continued):

Vector Id	Label	Description
1000103 + 200(ply - 1)	COMP PLY PLY FAILURE INDEX FIBER-1	2-Dimensional composite laminate element ply fiber failure index (MCT failure theory). Fill-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000103 + 200(ply - 1)	COMP PLY PLY STRENGTH RATIO FIBER-TENSION	2-Dimensional composite laminate element ply fiber-tension failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000103 + 200(ply - 1)	COMP PLY PLY STRENGTH RATIO FIBER-1	2-Dimensional composite laminate element ply fiber strength ratio (MCT failure theory). Fill-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000104 + 200(ply - 1)	COMP PLY PLY FAILURE INDEX FIBER-COMPRESSION	2-Dimensional composite laminate element ply fiber-compression failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000104 + 200(ply - 1)	COMP PLY PLY FAILURE INDEX FIBER-2	2-Dimensional composite laminate element ply fiber failure index (MCT failure theory). Warp-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000104 + 200(ply - 1)	COMP PLY PLY STRENGTH RATIO FIBER-COMPRESSION	2-Dimensional composite laminate element ply fiber-compression failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000104 + 200(ply - 1)	COMP PLY PLY STRENGTH RATIO FIBER-2	2-Dimensional composite laminate element ply fiber failure index (MCT failure theory). Warp-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
1000105 + 200(ply - 1)	COMP PLY FAILURE THEORY	2-Dimensional composite laminate failure theory code [1=Hill, 2=Hoffman, 3=Tsai-Wu, 4=Max Strain (MSC), 5=Max Strain (Autodesk), 6=Max Stress, 7=LaRC02, 8=Puck, 9=MCT, 0=None]. Controlled by the FT field on the PCOMP Bulk Data entry.
1000106 + 200(ply - 1)	COMP PLY FRACTURE ANGLE	2-Dimensional composite laminate fracture plane angle (LaRC02 and Puck failure theories only). Controlled by STRESS or STRAIN Case Control commands.
1000107 + 200(ply - 1)	COMP PLY STRENGTH RATIO ERROR	2-Dimensional composite laminate element strength ratio error. Controlled by STRESS or STRAIN Case Control commands.
1000108 + 200(ply - 1)	COMP PLY STATUS	2-Dimensional composite laminate element ply failure status (1=ply has failed, 0=ply has not failed). Controlled by STRESS or STRAIN Case Control commands and PARAM, NLCOMPPLYFAIL.
1000109 + 200(ply - 1)	COMP PLY EFFECTIVE STRAIN	2-Dimensional composite laminate element effective strain (von Mises). Controlled by STRESS or STRAIN Case Control commands.

Shear Element Results Column Descriptions:

Vector Id	Label	Description
6007	SHEAR MAX KICK LOAD	Shear element maximum kick load. Controlled by FORCE Case Control command.
6008	SHEAR MIN KICK LOAD	Shear element minimum kick load. Controlled by FORCE Case Control command.
6009	SHEAR MAX SHEAR FLOW	Shear element maximum shear flow (all edges). Controlled by FORCE Case Control command.
6010	SHEAR MIN SHEAR FLOW	Shear element minimum shear flow (all edges). Controlled by FORCE Case Control command.
6011	SHEAR KICK LOAD NODE 1	Shear element node-1 kick load. Controlled by FORCE Case Control command.
6012	SHEAR KICK LOAD NODE 2	Shear element node-2 kick load. Controlled by FORCE Case Control command.
6013	SHEAR KICK LOAD NODE 3	Shear element node-3 kick load. Controlled by FORCE Case Control command.
6014	SHEAR KICK LOAD NODE 4	Shear element node-4 kick load. Controlled by FORCE Case Control command.
6015	SHEAR SHEAR FLOW STRESS EDGE 1	Shear element inplane shear force on element edge 1 (nodes 1-2). Controlled by STRESS Case Control command.
6016	SHEAR SHEAR FLOW STRESS EDGE 2	Shear element inplane shear force on element edge 2 (nodes 2-3). Controlled by STRESS Case Control command.
6017	SHEAR SHEAR FLOW STRESS EDGE 3	Shear element inplane shear force on element edge 3 (nodes 3-4). Controlled by STRESS Case Control command.
6018	SHEAR SHEAR FLOW STRESS EDGE 4	Shear element inplane shear force on element edge 4 (nodes 4-1). Controlled by STRESS Case Control command.
6020	SHEAR AVERAGE SHEAR FLOW	Shear element average shear flow (all edges). Controlled by FORCE Case Control command.
6024	SHEAR SHEAR-XY STRESS EDGE 1	Shear element inplane shear stress on element edge 1 (nodes 1-2). Controlled by STRESS Case Control command.
6025	SHEAR SHEAR-XY STRESS EDGE 2	Shear element inplane shear stress on element edge 2 (nodes 2-3). Controlled by STRESS Case Control command.
6026	SHEAR SHEAR-XY STRESS EDGE 3	Shear element inplane shear stress on element edge 3 (nodes 3-4). Controlled by STRESS Case Control command.
6027	SHEAR SHEAR-XY STRESS EDGE 4	Shear element inplane shear stress on element edge 4 (nodes 4-1). Controlled by STRESS Case Control command.
6028	SHEAR MAX SHEAR-XY STRESS	Shear element maximum shear stress (all edges). Controlled by STRESS Case Control command.
6029	SHEAR MIN SHEAR-XY STRESS	Shear element minimum shear stress (all edges). Controlled by STRESS Case Control command.
6030	SHEAR AVERAGE SHEAR-XY STRESS	Shear element average shear stress (all edges). Controlled by STRESS Case Control command.

Solid Element Results Column Descriptions:

Vector Id	Label	Description
60010	SOLID NORMAL-X STRESS	Solid element normal stress in VOLUME x-direction. Controlled by STRESS Case Control command.
60011	SOLID NORMAL-Y STRESS	Solid element normal stress in VOLUME y-direction. Controlled by STRESS Case Control command.
60012	SOLID NORMAL-Z STRESS	Solid element normal stress in VOLUME z-direction. Controlled by STRESS Case Control command.
60013	SOLID SHEAR-XY STRESS	Solid element shear stress in VOLUME xy-direction (tensor x-face, y-direction). Controlled by STRESS Case Control command.
60014	SOLID SHEAR-YZ STRESS	Solid element shear stress in VOLUME yz-direction (tensor y-face, z-direction). Controlled by STRESS Case Control command.
60015	SOLID SHEAR-ZX STRESS	Solid element shear stress in VOLUME zx-direction (tensor z-face, x-direction). Controlled by STRESS Case Control command.
60016	SOLID PRINCIPAL-A STRESS	Solid element maximum principal stress. Controlled by STRESS Case Control command.
60017	SOLID PRINICPAL-C STRESS	Solid element minimum principal stress. Controlled by STRESS Case Control command.
60018	SOLID PRINCIPAL-B STRESS	Solid element median principal stress. Controlled by STRESS Case Control command.
60019	SOLID PRINCIPAL-A COSINE-X	Solid element maximum principal stress x-direction cosine. Controlled by STRESS Case Control command.
60020	SOLID PRINCIPAL-B COSINE-X	Solid element median principal stress x-direction cosine. Controlled by STRESS Case Control command.
60021	SOLID PRINCIPAL-C COSINE-X	Solid element minimum principal stress x-direction cosine. Controlled by STRESS Case Control command.
60022	SOLID PRINCIPAL-A COSINE-Y	Solid element maximum principal stress y-direction cosine. Controlled by STRESS Case Control command.
60023	SOLID PRINCIPAL-B COSINE-Y	Solid element median principal stress y-direction cosine. Controlled by STRESS Case Control command.
60024	SOLID PRINCIPAL-C COSINE-Y	Solid element minimum principal stress y-direction cosine. Controlled by STRESS Case Control command.
60025	SOLID PRINCIPAL-A COSINE-Z	Solid element maximum principal stress z-direction cosine. Controlled by STRESS Case Control command.
60026	SOLID PRINCIPAL-B COSINE-Z	Solid element median principal stress z-direction cosine. Controlled by STRESS Case Control command.
60027	SOLID PRINCIPAL-C COSINE-Z	Solid element minimum principal stress z-direction cosine. Controlled by STRESS Case Control command.
60028	SOLID MAX SHEAR STRESS	Solid element maximum shear stress. Controlled by STRESS Case Control command.
60029	SOLID MEAN PRESSURE STRESS	Solid element mean pressure stress. Controlled by STRESS Case Control command.

Solid Element Results Column Descriptions (Continued):

Vector Id	Label	Description
60030	SOLID EQUIVALENT STRESS	Solid element nonlinear equivalent stress (material nonlinear solutions) or von Mises stress (linear solutions). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
60031	SOLID VON MISES STRESS	Solid element von Mises stress. Controlled by STRESS Case Control command.
60032	SOLID OCTAHEDRAL STRESS	Solid element octahedral stress. Controlled by STRESS Case Control command.
60033	SOLID MAX PRINCIPAL STRESS	Solid element maximum principal stress. Controlled by STRESS Case Control command.
60034	SOLID MIN PRINCIPAL STRESS	Solid element minimum principal stress. Controlled by STRESS Case Control command.
60035	SOLID STATUS	Solution and option dependent. In modal summation solutions (DDAM) STATUS is the mode number with the maximum response in the NRL summation. In solutions where a factor of safety calculation method has been defined on a MAT1 entry, STATUS is the factor of safety. In topological optimization solutions STATUS is the element density.
60050	SOLID NORMAL-X STRAIN	Solid element normal strain in VOLUME x-direction. Controlled by STRAIN Case Control command.
60051	SOLID NORMAL-Y STRAIN	Solid element normal strain in VOLUME y-direction. Controlled by STRAIN Case Control command.
60052	SOLID NORMAL-Z STRAIN	Solid element normal strain in VOLUME z-direction. Controlled by STRAIN Case Control command.
60053	SOLID SHEAR-XY STRAIN	Solid element shear strain in VOLUME xy-direction (tensor x-face, y-direction). Controlled by STRAIN Case Control command.
60054	SOLID SHEAR-YZ STRAIN	Solid element shear strain in VOLUME yz-direction (tensor y-face, z-direction). Controlled by STRAIN Case Control command.
60055	SOLID SHEAR-ZX STRAIN	Solid element shear strain in VOLUME zx-direction (tensor z-face, x-direction). Controlled by STRAIN Case Control command.
60056	SOLID PRINCIPAL-A STRAIN	Solid element maximum principal strain. Controlled by STRAIN Case Control command.
60057	SOLID PRINICPAL-C STRAIN	Solid element minimum principal strain. Controlled by STRAIN Case Control command.
60058	SOLID PRINCIPAL-B STRAIN	Solid element median principal strain. Controlled by STRAIN Case Control command.
60059	SOLID MAX SHEAR STRAIN	Solid element maximum shear strain. Controlled by STRAIN Case Control command.
60060	SOLID MEAN PRESSURE STRAIN	Solid element mean pressure strain. Controlled by STRAIN Case Control command.
60061	SOLID VON MISES STRAIN	Solid element von Mises strain. Controlled by STRAIN Case Control command.

Solid Element Results Column Descriptions (Continued):

Vector Id	Label	Description
60062	SOLID PRINCIPAL-A COS-X	Solid element maximum principal strain x-direction cosine. Controlled by STRAIN Case Control command.
60063	SOLID PRINCIPAL-B COS-X	Solid element median principal strain x-direction cosine. Controlled by STRAIN Case Control command.
60064	SOLID PRINCIPAL-C COS-X	Solid element minimum principal strain x-direction cosine. Controlled by STRAIN Case Control command.
60065	SOLID PRINCIPAL-A COS-Y	Solid element maximum principal strain y-direction cosine. Controlled by STRAIN Case Control command.
60066	SOLID PRINICPAL-B COS-Y	Solid element median principal strain y-direction cosine. Controlled by STRAIN Case Control command.
60067	SOLID PRINCIPAL-C COS-Y	Solid element minimum principal strain y-direction cosine. Controlled by STRAIN Case Control command.
60068	SOLID PRINCIPAL-A COS-Z	Solid element maximum principal strain z-direction cosine. Controlled by STRAIN Case Control command.
60069	SOLID PRINCIPAL-B COS-Z	Solid element median principal strain z-direction cosine. Controlled by STRAIN Case Control command.
60070	SOLID PRINCIPAL-C COS-Z	Solid element minimum principal strain z-direction cosine. Controlled by STRAIN Case Control command.
60071	SOLID OCTAHEDRAL STRAIN	Solid element octahedral strain. Controlled by STRAIN Case Control command.
60072	SOLID EFFECTIVE STRAIN-ELASTIC	Solid element effective strain (von Mises). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
60072	SOLID EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC	Solid element effective (nonlinear elastic material) or plastic (elastic-plastic material) strain. Controlled by NLSTRESS Case Control command.
60073	SOLID EFFECTIVE STRAIN-CREEP	Solid element effective creep strain. Controlled by NLSTRESS Case Control command.
60073	SOLID VOLUMETRIC STRAIN	Solid element volumetric strain (large strain material). Controlled by NLSTRESS Case Control command.
60073	SOLID MARTENSITE VOLUME FRACTION	Solid element martensite volume fraction (Nitinol shape memory material). Controlled by NLSTRESS Case Control command.
60075	SOLID MAX PRINCIPAL STRAIN	Solid element maximum principal strain. Controlled by STRAIN Case Control command.
60076	SOLID MIN PRINCIPAL STRAIN	Solid element minimum principal strain. Controlled by STRAIN Case Control command.
60120	SOLID BIAXIALITY RATIO	Solid element stress biaxiality ratio. Controlled by STRESS Case Control command.
60121	SOLID DAMAGE	Solid element fatigue damage. Controlled by FATIGUE, VIBFATIGUE, STRESS, and STRAIN Case Control commands.
60122	SOLID LIFE	Solid element fatigue life. Controlled by FATIGUE, VIBFATIGUE, STRESS, and STRAIN Case Control commands.
60123	SOLID BIAXIALITY RATIO	Solid element strain biaxiality ratio. Controlled by STRAIN Case Control command.

Axisymmetric Solid Element Results Column Descriptions:

Vector Id	Label	Description
6175	AXSYM DAMAGE	Axisymmetric solid element fatigue damage. Controlled by FATIGUE, STRESS, and STRAIN Case Control commands.
6176	AXSYM LIFE	Axisymmetric solid element fatigue life. Controlled by FATIGUE, STRESS, and STRAIN Case Control commands.
6200	AXSYM NORMAL-RADIAL STRESS	Axisymmetric solid element normal stress in radial direction. Controlled by STRESS Case Control command.
6201	AXSYM NORMAL-TANGENTIAL STRESS	Axisymmetric solid element normal stress in tangential direction. Controlled by STRESS Case Control command.
6202	AXSYM NORMAL-AXIAL STRESS	Axisymmetric solid element normal stress in axial direction. Controlled by STRESS Case Control command.
6203	AXSYM SHEAR-RADIAL/AXIAL STRESS	Axisymmetric solid element shear stress in axial/radial direction. Controlled by STRESS Case Control command.
6204	AXSYM VON MISES STRESS	Axisymmetric solid element von Mises stress. Controlled by STRESS Case Control command.
6205	AXSYM MAX SHEAR/TRESCA STRESS	Axisymmetric solid element von Mises stress. Controlled by STRESS Case Control command.
6206	AXSYM MAX PRINCIPAL STRESS	Axisymmetric solid element maximum principal stress. Controlled by STRESS Case Control command.
6207	AXSYM MIN PRINCIPAL STRESS	Axisymmetric solid element minimum principal stress. Controlled by STRESS Case Control command.
6208	AXSYM MEAN PRESSURE STRESS	Axisymmetric solid element mean pressure stress. Controlled by STRESS Case Control command.
6209	AXSYM OCTAHEDRAL STRESS	Axisymmetric solid element octahedral stress. Controlled by STRESS Case Control command.
6210	AXSYM STATUS	In solutions where a factor of safety calculation method has been defined on a MAT1 entry, STATUS is the factor of safety.
6211	AXSYM EQUIVALENT STRESS	Axisymmetric solid element von Mises stress. Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
6212	AXSYM EFFECTIVE STRAIN-ELASTIC	Axisymmetric solid element von Mises strain. Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
6214	AXSYM NORMAL-RADIAL STRAIN	Axisymmetric solid element normal strain in radial direction. Controlled by STRAIN Case Control command.
6215	AXSYM NORMAL-TANGENTIAL STRAIN	Axisymmetric solid element normal strain in tangential direction. Controlled by STRAIN Case Control command.
6216	AXSYM NORMAL-AXIAL STRAIN	Axisymmetric solid element normal strain in axial direction. Controlled by STRAIN Case Control command.
6217	AXSYM SHEAR-RADIAL/AXIAL STRAIN	Axisymmetric solid element shear strain in axial/radial direction. Controlled by STRAIN Case Control command.

Axisymmetric Solid Element Results Column Descriptions (Continued):

Vector Id	Label	Description
6218	AXSYM VON MISES STRAIN	Axisymmetric solid element von Mises strain. Controlled by STRAIN Case Control command.
6219	AXSYM MAX SHEAR/TRESCA STRAIN	Axisymmetric solid element von Mises strain. Controlled by STRAIN Case Control command.
6220	AXSYM MAX PRINCIPAL STRAIN	Axisymmetric solid element maximum principal strain. Controlled by STRAIN Case Control command.
6221	AXSYM MIN PRINCIPAL STRAIN	Axisymmetric solid element minimum principal strain. Controlled by STRAIN Case Control command.
6222	AXSYM MEAN PRESSURE STRAIN	Axisymmetric solid element mean pressure strain. Controlled by STRAIN Case Control command.
6223	AXSYM OCTAHEDRAL STRAIN	Axisymmetric solid element octahedral strain. Controlled by STRAIN Case Control command.

Composite Solid Element Results Column Descriptions:

Vector Id	Label	Description
60036	LSLD MAX EFFECTIVE STRAIN	3-Dimensional composite laminate element maximum ply effective strain (von Mises, of all plies). Controlled by STRESS or STRAIN Case Control commands.
60037	LSLD MAX EQUIVALENT STRESS	3-Dimensional composite laminate element maximum ply equivalent stress (von Mises, of all plies). Controlled by STRESS or STRAIN Case Control commands.
60196	LSLD MAX NORMAL-1 STRESS	3-Dimensional composite laminate element maximum ply normal stress in ply 1-direction (longitudinal). Controlled by STRESS Case Control command.
60197	LSLD MAX NORMAL-2 STRESS	3-Dimensional composite laminate element maximum ply normal stress in ply 2-direction (lateral). Controlled by STRESS Case Control command.
60198	LSLD MAX NORMAL-3 STRESS	3-Dimensional composite laminate element maximum ply normal stress in ply 3-direction (thickness). Controlled by STRESS Case Control command.
60199	LSLD MAX SHEAR-12 STRESS	3-Dimensional composite laminate element maximum ply normal stress in ply 12-direction. Controlled by STRESS Case Control command.
60200	LSLD MAX SHEAR-YZ STRESS	3-Dimensional composite laminate element maximum ply interlaminar shear stress in material xz-direction. Controlled by STRESS Case Control command.
60201	LSLD MAX SHEAR-XZ STRESS	3-Dimensional composite laminate element maximum ply interlaminar shear stress in material yz-direction. Controlled by STRESS Case Control command.
60202	LSLD MIN NORMAL-1 STRESS	3-Dimensional composite laminate element minimum ply normal stress in ply 1-direction (longitudinal). Controlled by STRESS Case Control command.
60203	LSLD MIN NORMAL-2 STRESS	3-Dimensional composite laminate element minimum ply normal stress in ply 2-direction (lateral). Controlled by STRESS Case Control command.
60204	LSLD MIN NORMAL-3 STRESS	3-Dimensional composite laminate element minimum ply normal stress in ply 3-direction (thickness). Controlled by STRESS Case Control command.
60205	LSLD MIN SHEAR-12 STRESS	3-Dimensional composite laminate element minimum ply normal stress in ply 12-direction. Controlled by STRESS Case Control command.
60206	LSLD MIN SHEAR-YZ STRESS	3-Dimensional composite laminate element minimum ply interlaminar shear stress in material xz-direction. Controlled by STRESS Case Control command.
60207	LSLD MIN SHEAR-XZ STRESS	3-Dimensional composite laminate element minimum ply interlaminar shear stress in material yz-direction. Controlled by STRESS Case Control command.

Composite Solid Element Results Column Descriptions (Continued):

Vector Id	Label	Description
60208	LSLD MAX NORMAL-1 STRAIN	3-Dimensional composite laminate element maximum ply normal strain in ply 1-direction (longitudinal). Controlled by STRAIN Case Control command.
60209	LSLD MAX NORMAL-2 STRAIN	3-Dimensional composite laminate element maximum ply normal strain in ply 2-direction (lateral). Controlled by STRAIN Case Control command.
60210	LSLD MAX NORMAL-3 STRAIN	3-Dimensional composite laminate element maximum ply normal strain in ply 3-direction (thickness). Controlled by STRAIN Case Control command.
60211	LSLD MAX SHEAR-12 STRAIN	3-Dimensional composite laminate element maximum ply normal strain in ply 12-direction. Controlled by STRAIN Case Control command.
60212	LSLD MAX SHEAR-YZ STRAIN	3-Dimensional composite laminate element maximum ply interlaminar shear strain in material xz-direction. Controlled by STRAIN Case Control command.
60213	LSLD MAX SHEAR-XZ STRAIN	3-Dimensional composite laminate element maximum ply interlaminar shear strain in material yz-direction. Controlled by STRAIN Case Control command.
60214	LSLD MIN NORMAL-1 STRAIN	3-Dimensional composite laminate element minimum ply normal strain in ply 1-direction (longitudinal). Controlled by STRAIN Case Control command.
60215	LSLD MIN NORMAL-2 STRAIN	3-Dimensional composite laminate element minimum ply normal strain in ply 2-direction (lateral). Controlled by STRAIN Case Control command.
60216	LSLD MIN NORMAL-3 STRAIN	3-Dimensional composite laminate element minimum ply normal strain in ply 3-direction (thickness). Controlled by STRAIN Case Control command.
60217	LSLD MIN SHEAR-12 STRAIN	3-Dimensional composite laminate element minimum ply normal strain in ply 12-direction. Controlled by STRAIN Case Control command.
60218	LSLD MIN SHEAR-YZ STRAIN	3-Dimensional composite laminate element minimum ply interlaminar shear strain in material xz-direction. Controlled by STRAIN Case Control command.
60219	LSLD MIN SHEAR-XZ STRAIN	3-Dimensional composite laminate element minimum ply interlaminar shear strain in material yz-direction. Controlled by STRAIN Case Control command.
60220	LSLD MAX PLY FAILURE INDEX	3-Dimensional composite laminate element maximum ply failure index (of all plies). Controlled by STRESS or STRAIN Case Control commands.
60220	LSLD MAX PLY STRENGTH RATIO	3-Dimensional composite laminate element maximum ply strength ratio (of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
60221	LSLD MAX BOND FAILURE INDEX	3-Dimensional composite laminate element maximum bond failure index (of all plies). Controlled by STRESS or STRAIN Case Control commands.

Composite Solid Element Results Column Descriptions (Continued):

Vector Id	Label	Description
60221	LSLD MAX BOND STRENGTH RATIO	3-Dimensional composite laminate element maximum bond strength ratio (of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
60222	LSLD MIN PLY FAILURE INDEX	3-Dimensional composite laminate element minimum ply failure index (of all plies). Controlled by STRESS or STRAIN Case Control commands.
60222	LSLD MIN PLY STRENGTH RATIO	3-Dimensional composite laminate element minimum ply strength ratio (of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
60223	LSLD MIN BOND FAILURE INDEX	3-Dimensional composite laminate element minimum bond failure index (of all plies). Controlled by STRESS or STRAIN Case Control commands.
60223	LSLD MIN BOND STRENGTH RATIO	3-Dimensional composite laminate element minimum bond strength ratio (of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
60224	LSLD MAX FAILURE INDEX	3-Dimensional composite laminate element maximum failure index (both ply and bond of all plies). Controlled by STRESS or STRAIN Case Control commands.
60224	LSLD MIN STRENGTH RATIO	3-Dimensional composite laminate element minimum strength ratio (both ply and bond of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
60225	LSLD MAX FAILURE INDEX PLY	3-Dimensional composite laminate element maximum failure index ply (both ply and bond of all plies). Controlled by STRESS or STRAIN Case Control commands.
60225	LSLD MIN STRENGTH RATIO PLY	3-Dimensional composite laminate element maximum failure index ply (both ply and bond of all plies). Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.
60230	LSLD MAX PRINCIPAL STRESS	3-Dimensional composite laminate element maximum ply principal stress (of all plies). Controlled by STRESS Case Control command.
60231	LSLD MIN PRINCIPAL STRESS	3-Dimensional composite laminate element minimum ply principal stress (of all plies). Controlled by STRESS Case Control command.
60232	LSLD MAX MAX SHEAR STRESS	3-Dimensional composite laminate element maximum maximum shear stress (of all plies). Controlled by STRESS Case Control command.
60233	LSLD MAX VON MISES STRESS	3-Dimensional composite laminate element maximum von Mises stress (of all plies). Controlled by STRESS Case Control command.
60234	LSLD MAX PRINCIPAL STRAIN	3-Dimensional composite laminate element maximum ply principal strain (of all plies). Controlled by STRAIN Case Control command.
60235	LSLD MIN PRINCIPAL STRAIN	3-Dimensional composite laminate element minimum ply principal strain (of all plies). Controlled by STRAIN Case Control command.
60236	LSLD MAX MAX SHEAR STRAIN	3-Dimensional composite laminate element maximum maximum shear strain (of all plies). Controlled by STRAIN Case Control command.

Composite Solid Element Results Column Descriptions (Continued):

Vector Id	Label	Description
60237	LSLD MAX VON MISES STRAIN	3-Dimensional composite laminate element maximum von Mises strain (of all plies). Controlled by STRAIN Case Control command.
60610 + 200(ply - 1)	LSLD PLY PLY NORMAL-1 STRESS	3-Dimensional composite laminate element ply normal stress in ply 1-direction (longitudinal). Controlled by STRESS Case Control command.
60611 + 200(ply - 1)	LSLD PLY PLY NORMAL-2 STRESS	3-Dimensional composite laminate element ply normal stress in ply 2-direction (lateral). Controlled by STRESS Case Control command.
60612 + 200(ply - 1)	LSLD PLY PLY NORMAL-3 STRESS	3-Dimensional composite laminate element ply normal stress in ply 3-direction (thickness). Controlled by STRESS Case Control command.
60613 + 200(ply - 1)	LSLD PLY PLY SHEAR-12 STRESS	3-Dimensional composite laminate element ply normal stress in ply 12-direction. Controlled by STRESS Case Control command.
60614 + 200(ply - 1)	LSLD PLY PLY SHEAR-YZ STRESS	3-Dimensional composite laminate element interlaminar shear stress in material xz-direction. Controlled by STRESS Case Control command.
60615 + 200(ply - 1)	LSLD PLY PLY SHEAR-XZ STRESS	3-Dimensional composite laminate element interlaminar shear stress in material yz-direction. Controlled by STRESS Case Control command.
60616 + 200(ply - 1)	LSLD PLY PLY PRINCIPAL-A STRESS	3-Dimensional composite laminate element ply maximum principal stress. Controlled by STRESS Case Control command.
60617 + 200(ply - 1)	LSLD PLY PLY PRINCIPAL-B STRESS	3-Dimensional composite laminate element ply minimum principal stress. Controlled by STRESS Case Control command.
60618 + 200(ply - 1)	LSLD PLY PLY PRINCIPAL-C STRESS	3-Dimensional composite laminate element ply median principal stress. Controlled by STRESS Case Control command.
60628 + 200(ply - 1)	LSLD PLY PLY MAX SHEAR STRESS	3-Dimensional composite laminate element ply maximum shear stress. Controlled by STRESS Case Control command.
60629 + 200(ply - 1)	LSLD PLY PLY MEAN PRESSURE STRESS	3-Dimensional composite laminate element ply mean pressure stress. Controlled by STRESS Case Control command.
60630 + 200(ply - 1)	LSLD PLY PLY EQUIVALENT STRESS	3-Dimensional composite laminate element ply equivalent stress (von Mises). Controlled by STRESS or STRAIN Case Control commands.
60631 + 200(ply - 1)	LSLD PLY PLY VON MISES STRESS	3-Dimensional composite laminate element ply von Mises stress. Controlled by STRESS Case Control command.
60632 + 200(ply - 1)	LSLD PLY PLY OCTAHEDRAL STRESS	3-Dimensional composite laminate element ply octahedral stress. Controlled by STRESS Case Control command.
60633 + 200(ply - 1)	LSLD PLY PLY MAX PRINCIPAL STRESS	3-Dimensional composite laminate element ply maximum principal stress. Controlled by STRESS Case Control command.
60634 + 200(ply - 1)	LSLD PLY PLY MIN PRINCIPAL STRESS	3-Dimensional composite laminate element ply minimum principal stress. Controlled by STRESS Case Control command.

Composite Solid Element Results Column Descriptions (Continued):

Vector Id	Label	Description
60650 + 200(ply - 1)	LSLD PLY PLY NORMAL-1 STRAIN	3-Dimensional composite laminate element ply normal strain in ply 1-direction (longitudinal). Controlled by STRAIN Case Control command.
60651 + 200(ply - 1)	LSLD PLY PLY NORMAL-2 STRAIN	3-Dimensional composite laminate element ply normal strain in ply 2-direction (lateral). Controlled by STRAIN Case Control command.
60652 + 200(ply - 1)	LSLD PLY PLY NORMAL-3 STRAIN	3-Dimensional composite laminate element ply normal strain in ply 3-direction (thickness). Controlled by STRAIN Case Control command.
60653 + 200(ply - 1)	LSLD PLY PLY SHEAR-12 STRAIN	3-Dimensional composite laminate element ply normal strain in ply 12-direction. Controlled by STRAIN Case Control command.
60654 + 200(ply - 1)	LSLD PLY PLY SHEAR-YZ STRAIN	3-Dimensional composite laminate element interlaminar shear strain in material xz-direction. Controlled by STRAIN Case Control command.
60655 + 200(ply - 1)	LSLD PLY PLY SHEAR-XZ STRAIN	3-Dimensional composite laminate element interlaminar shear strain in material yz-direction. Controlled by STRAIN Case Control command.
60656 + 200(ply - 1)	LSLD PLY PLY PRINCIPAL-A STRAIN	3-Dimensional composite laminate element ply maximum principal strain. Controlled by STRAIN Case Control command.
60657 + 200(ply - 1)	LSLD PLY PLY PRINCIPAL-B STRAIN	3-Dimensional composite laminate element ply minimum principal strain. Controlled by STRAIN Case Control command.
60658 + 200(ply - 1)	LSLD PLY PLY PRINCIPAL-C STRAIN	3-Dimensional composite laminate element ply median principal strain. Controlled by STRAIN Case Control command.
60659 + 200(ply - 1)	LSLD PLY PLY MAX SHEAR STRAIN	3-Dimensional composite laminate element ply maximum shear strain. Controlled by STRAIN Case Control command.
60660 + 200(ply - 1)	LSLD PLY PLY MEAN PRESSURE STRAIN	3-Dimensional composite laminate element ply mean pressure strain. Controlled by STRAIN Case Control command.
60661 + 200(ply - 1)	LSLD PLY PLY VON MISES STRAIN	3-Dimensional composite laminate element ply von Mises strain. Controlled by STRAIN Case Control command.
60671 + 200(ply - 1)	LSLD PLY PLY OCTAHEDRAL STRAIN	3-Dimensional composite laminate element ply octahedral strain. Controlled by STRAIN Case Control command.
60675 + 200(ply - 1)	LSLD PLY PLY MAX PRINCIPAL STRAIN	3-Dimensional composite laminate element ply maximum principal strain. Controlled by STRAIN Case Control command.
60676 + 200(ply - 1)	LSLD PLY PLY MIN PRINCIPAL STRAIN	3-Dimensional composite laminate element ply minimum principal strain. Controlled by STRAIN Case Control command.
60690 + 200(ply - 1)	LSLD PLY PLY FAILURE INDEX	3-Dimensional composite laminate element ply failure index. Controlled by STRESS or STRAIN Case Control commands.
60690 + 200(ply - 1)	LSLD PLY PLY STRENGTH RATIO	3-Dimensional composite laminate element ply strength ratio. Controlled by STRESS or STRAIN Case Control commands and PARAM, STRENGTHRATIO.

Composite Solid Element Results Column Descriptions (Continued):

Vector Id	Label	Description
60691 + 200(ply - 1)	LSLD PLY PLY BOND FAILURE INDEX	3-Dimensional composite laminate element ply bond failure index. Controlled by STRESS or STRAIN Case Control commands.
60691 + 200(ply - 1)	LSLD PLY PLY BOND STRENGTH RATIO	3-Dimensional composite laminate element ply bond strength ratio. Controlled by STRESS or STRAIN Case Control commands.
60692 + 200(ply – 1)	LSLD PLY PLY FAILURE INDEX MATRIX-TENSION	3-Dimensional composite laminate element ply matrix-tension failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60692 + 200(ply – 1)	LSLD PLY PLY FAILURE INDEX MATRIX-1	3-Dimensional composite laminate element ply matrix failure index (MCT failure theory). Fill-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60692 + 200(ply – 1)	LSLD PLY PLY STRENGTH RATIO MATRIX-TENSION	3-Dimensional composite laminate element ply matrix-tension failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60692 + 200(ply – 1)	LSLD PLY PLY STRENGTH RATIO MATRIX-1	3-Dimensional composite laminate element ply matrix strength ratio (MCT failure theory). Fill-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60693 + 200(ply – 1)	LSLD PLY PLY FAILURE INDEX MATRIX-COMPRESSION	3-Dimensional composite laminate element ply matrix-compression failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60693 + 200(ply – 1)	LSLD PLY PLY FAILURE INDEX MATRIX-2	3-Dimensional composite laminate element ply matrix failure index (MCT failure theory). Warp-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60693 + 200(ply – 1)	LSLD PLY PLY STRENGTH RATIO MATRIX-COMPRESSION	3-Dimensional composite laminate element ply matrix-compression failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60693 + 200(ply – 1)	LSLD PLY PLY STRENGTH RATIO MATRIX-2	3-Dimensional composite laminate element ply matrix strength ratio (MCT failure theory). Warp-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60694 + 200(ply – 1)	LSLD PLY PLY FAILURE INDEX FIBER-TENSION	3-Dimensional composite laminate element ply fiber-tension failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.

Composite Solid Element Results Column Descriptions (Continued):

Vector Id	Label	Description
60694 + 200(ply – 1)	LSLD PLY PLY FAILURE INDEX FIBER-1	3-Dimensional composite laminate element ply fiber failure index (MCT failure theory). Fill-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60694 + 200(ply – 1)	LSLD PLY PLY STRENGTH RATIO FIBER-TENSION	3-Dimensional composite laminate element ply fiber-tension failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60694 + 200(ply – 1)	LSLD PLY PLY STRENGTH RATIO FIBER-1	3-Dimensional composite laminate element ply fiber strength ratio (MCT failure theory). Fill-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60695 + 200(ply – 1)	LSLD PLY PLY FAILURE INDEX FIBER-COMPRESSION	3-Dimensional composite laminate element ply fiber-compression failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60695 + 200(ply – 1)	LSLD PLY PLY FAILURE INDEX FIBER-2	3-Dimensional composite laminate element ply fiber failure index (MCT failure theory). Warp-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60695 + 200(ply – 1)	LSLD PLY PLY STRENGTH RATIO FIBER-COMPRESSION	3-Dimensional composite laminate element ply fiber-compression failure index (LaRC02 or Puck failure theories). Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60695 + 200(ply – 1)	LSLD PLY PLY STRENGTH RATIO FIBER-2	3-Dimensional composite laminate element ply fiber failure index (MCT failure theory). Warp-direction for plain weave fabrics. Controlled by STRESS or STRAIN Case Control commands and the FT field on the PCOMP entry.
60696 + 200(ply – 1)	LSLD PLY PLY FAILURE THEORY	3-Dimensional composite laminate failure theory code [1=Hill, 2=Hoffman, 3=Tsai-Wu, 4=Max Strain (MSC), 5=Max Strain (Autodesk), 6=Max Stress, 7=LaRC02, 8=Puck, 9=MCT, 0=None]. Controlled by the FT field on the PCOMP Bulk Data entry.
60697 + 200(ply – 1)	LSLD PLY PLY FRACTURE ANGLE	3-Dimensional composite laminate fracture plane angle (LaRC02 and Puck failure theories only). Controlled by STRESS or STRAIN Case Control commands.
60698 + 200(ply – 1)	LSLD PLY PLY STRENGTH RATIO ERROR	3-Dimensional composite laminate element strength ratio error. Controlled by STRESS or STRAIN Case Control commands.
60699 + 200(ply – 1)	LSLD PLY PLY STATUS	3-Dimensional composite laminate element ply failure status (1=ply has failed, 0=ply has not failed). Controlled by STRESS or STRAIN Case Control commands and PARAM, NLCOMPPLYFAIL.
60700 + 200(ply – 1)	LSLD PLY PLY EFFECTIVE STRAIN	3-Dimensional composite laminate element ply effective strain (von Mises). Controlled by STRESS or STRAIN Case Control commands.

Quad Contact Surface Element Results Column Descriptions:

Vector Id	Label	Description
3468	SQUAD MAX NORMAL FORCE	Quad contact surface maximum contact segment normal force. Controlled by FORCE or STRESS Case Control command.
3469	SQUAD MAX CONTACT PRESSURE	Quad contact surface maximum contact segment contact pressure. Controlled by FORCE or STRESS Case Control command.
3470	SQUAD MAX NORMAL GAP	Quad contact surface maximum normal gap. Controlled by FORCE or STRESS Case Control command.
3471	SQUAD MIN NORMAL FORCE	Quad contact surface minimum contact segment normal force. Controlled by FORCE or STRESS Case Control command.
3472	SQUAD MIN CONTACT PRESSURE	Quad contact surface minimum contact segment contact pressure. Controlled by FORCE or STRESS Case Control command.
3473	SQUAD MIN NORMAL GAP	Quad contact surface minimum normal gap. Controlled by FORCE or STRESS Case Control command.
3474	SQUAD MAX SHEAR FORCE-X	Quad contact surface maximum contact segment shear force in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3475	SQUAD MAX SHEAR FORCE-Y	Quad contact surface maximum contact segment shear force in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3476	SQUAD MAX CONTACT TRACTION-X	Quad contact surface maximum contact segment contact traction in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3477	SQUAD MAX CONTACT TRACTION -Y	Quad contact surface maximum contact segment contact traction in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3478	SQUAD MAX SLIP DISPLACEMENT-X	Quad contact surface maximum contact segment slip displacement in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3479	SQUAD MAX SLIP DISPLACEMENT-Y	Quad contact surface maximum contact segment slip displacement in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3480	SQUAD MIN CONTACT TRACTION-X	Quad contact surface minimum contact segment contact traction in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3517	SQUAD MIN CONTACT TRACTION -Y	Quad contact surface minimum contact segment contact traction in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3518	SQUAD MIN SHEAR STRESS-X	Quad contact surface minimum contact segment shear stress in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3519	SQUAD MIN SHEAR STRESS-Y	Quad contact surface minimum contact segment shear stress in the element y-direction. Controlled by FORCE or STRESS Case Control command.

Quad Contact Surface Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3520	SQUAD MIN SLIP DISPLACEMENT-X	Quad contact surface minimum contact segment slip displacement in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3521	SQUAD MIN SLIP DISPLACEMENT-Y	Quad contact surface minimum contact segment slip displacement in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3522	SQUAD STATUS	Quad contact status (1=open, 2=slide – closed with no friction defined, 3=stick – closed with friction and holding, 4=slip – closed with friction and slipping, 5=weld). Controlled by FORCE or STRESS Case Control command.
3523	SQUAD RESULTANT SHEAR FORCE	Quad contact surface maximum resultant shear force. Controlled by FORCE or STRESS Case Control command.
3524	SQUAD RESULTANT CONTACT TRACTION	Quad contact surface maximum resultant contact traction. Controlled by FORCE or STRESS Case Control command.
3525	SQUAD RESULTANT SLIP DISPLACEMENT	Quad contact surface maximum resultant slip displacement. Controlled by FORCE or STRESS Case Control command.

Tri Contact Surface Element Results Column Descriptions:

Vector Id	Label	Description
3468	STRI MAX NORMAL FORCE	Tri contact surface maximum contact segment normal force. Controlled by FORCE or STRESS Case Control command.
3469	STRI MAX CONTACT PRESSURE	Tri contact surface maximum contact segment contact pressure. Controlled by FORCE or STRESS Case Control command.
3470	STRI MAX NORMAL GAP	Tri contact surface maximum normal gap. Controlled by FORCE or STRESS Case Control command.
3471	STRI MIN NORMAL FORCE	Tri contact surface minimum contact segment normal force. Controlled by FORCE or STRESS Case Control command.
3472	STRI MIN CONTACT PRESSURE	Tri contact surface minimum contact segment contact pressure. Controlled by FORCE or STRESS Case Control command.
3473	STRI MIN NORMAL GAP	Tri contact surface minimum normal gap. Controlled by FORCE or STRESS Case Control command.
3474	STRI MAX SHEAR FORCE-X	Tri contact surface maximum contact segment shear force in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3475	STRI MAX SHEAR FORCE-Y	Tri contact surface maximum contact segment shear force in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3476	STRI MAX CONTACT TRACTION -X	Tri contact surface maximum contact segment contact traction in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3477	STRI MAX CONTACT TRACTION -Y	Tri contact surface maximum contact segment contact traction in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3478	STRI MAX SLIP DISPLACEMENT-X	Tri contact surface maximum contact segment slip displacement in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3479	STRI MAX SLIP DISPLACEMENT-Y	Tri contact surface maximum contact segment slip displacement in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3480	STRI MIN SHEAR FORCE-X	Tri contact surface minimum contact segment shear force in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3517	STRI MIN SHEAR FORCE-Y	Tri contact surface minimum contact segment shear force in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3518	STRI MIN CONTACT TRACTION -X	Tri contact surface minimum contact segment contact traction in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3519	STRI MIN CONTACT TRACTION -Y	Tri contact surface minimum contact segment contact traction in the element y-direction. Controlled by FORCE or STRESS Case Control command.

Tri Contact Surface Element Results Column Descriptions (Continued):

Vector Id	Label	Description
3520	STRI MIN SLIP DISPLACEMENT-X	Tri contact surface minimum contact segment slip displacement in the element x-direction. Controlled by FORCE or STRESS Case Control command.
3521	STRI MIN SLIP DISPLACEMENT-Y	Tri contact surface minimum contact segment slip displacement in the element y-direction. Controlled by FORCE or STRESS Case Control command.
3522	STRI STATUS	Tri contact status (1=open, 2=slide – closed with no friction defined, 3=stick – closed with friction and holding, 4=slip – closed with friction and slipping, 5=weld). Controlled by FORCE or STRESS Case Control command.
3523	STRI RESULTANT SHEAR FORCE	Tri contact surface maximum resultant shear force. Controlled by FORCE or STRESS Case Control command.
3524	STRI RESULTANT CONTACT TRACTION	Tri contact surface maximum resultant contact traction. Controlled by FORCE or STRESS Case Control command.
3525	STRI RESULTANT SLIP DISPLACEMENT	Tri contact surface maximum resultant slip displacement. Controlled by FORCE or STRESS Case Control command.

Miscellaneous Element Results Column Descriptions:

Vector Id	Label	Description
80000	ENERGY	Element strain energy. Controlled by ESE Case Control command.
80001	PERCENT TOTAL ENERGY	Element percent of total strain energy. Controlled by ESE Case Control command.
80002	ENERGY DENSITY	Element strain energy density. Controlled by ESE Case Control command.

Structural Neutral File Element Grid Point Results Column Descriptions

Virtual Fluid Mass Element Grid Point Results Column Descriptions:

Vector Id	Label	Description
61	TOTAL FLUID PRESSURE	Virtual fluid mass element total fluid pressure.
62	T1 FLUID PRESSURE	Virtual fluid mass element fluid pressure in T1 direction. Controlled by MPRES Case Control command.
63	T2 FLUID PRESSURE	Virtual fluid mass element fluid pressure in T2 direction. Controlled by MPRES Case Control command.
64	T3 FLUID PRESSURE	Virtual fluid mass element fluid pressure in T3 direction. Controlled by MPRES Case Control command.

Shell Element Grid Point Results Column Descriptions:

Vector Id	Label	Description
71	SHELL NORMAL-X TOP STRESS	Shell element top side (side 2) normal stress in SURFACE x-direction. Controlled by STRESS Case Control command.
72	SHELL NORMAL-Y TOP STRESS	Shell element top side (side 2) normal stress in SURFACE y-direction. Controlled by STRESS Case Control command.
73	SHELL SHEAR-XY TOP STRESS	Shell element top side (side 2) normal stress in SURFACE xy-direction (tensor x-face, y-direction). Controlled by STRESS Case Control command.
74	SHELL MAJOR PRINCIPAL TOP STRESS	Shell element top side (side 2) major principal stress. Controlled by STRESS Case Control command.
75	SHELL MINOR PRINCIPAL TOP STRESS	Shell element top side (side 2) minor principal stress. Controlled by STRESS Case Control command.
76	SHELL ZERO SHEAR STRESS ANGLE TOP	Shell element top side (side 2) zero shear stress angle in degrees. Controlled by STRESS Case Control command.
77	SHELL MAX SHEAR TOP STRESS	Shell element top side (side 2) maximum shear stress. Controlled by STRESS Case Control command.
77	SHELL TRESCA TOP STRESS	Shell element top side (side 2) Tresca stress . Controlled by STRESS Case Control command.
78	SHELL VON MISES TOP STRESS	Shell element top side (side 2) von Mises stress. Controlled by STRESS Case Control command.
81	SHELL NORMAL-X BOTTOM STRESS	Shell element bottom side (side 1) normal stress in SURFACE x-direction. Controlled by STRESS Case Control command.
82	SHELL NORMAL-Y BOTTOM STRESS	Shell element bottom side (side 1) normal stress in SURFACE y-direction. Controlled by STRESS Case Control command.
83	SHELL SHEAR-XY BOTTOM STRESS	Shell element bottom side (side 1) normal stress in SURFACE xy-direction (tensor x-face, y-direction). Controlled by STRESS Case Control command.
84	SHELL MAJOR PRINCIPAL STRESS BOTTOM	Shell element bottom side (side 1) major principal stress. Controlled by STRESS Case Control command.
85	SHELL MINOR PRINCIPAL STRESS BOTTOM	Shell element bottom side (side 1) minor principal stress. Controlled by STRESS Case Control command.
87	SHELL MAX SHEAR STRESS BOTTOM	Shell element bottom side (side 1) maximum shear strain. Controlled by STRESS Case Control command.
87	SHELL TRESCA STRESS BOTTOM	Shell element bottom side (side 1) Tresca stress . Controlled by STRESS Case Control command.
88	SHELL VON MISES STRESS BOTTOM	Shell element bottom side (side 1) von Mises stress. Controlled by STRESS Case Control command.

Shell Element Grid Point Results Column Descriptions (Continued):

Vector Id	Label	Description
113	SHELL MAX VON MISES STRESS BOTTOM/TOP	Shell element maximum von Mises stress. Controlled by STRESS Case Control command.
114	SHELL MAX SHEAR STRESS BOTTOM/TOP	Shell element maximum shear stress (of bottom and top). Controlled by STRESS Case Control command.
114	SHELL TRESCA STRESS BOTTOM/TOP	Shell element maximum Tresca stress (of bottom and top). Controlled by STRESS Case Control command.
115	SHELL MAX PRINCIPAL STRESS BOTTOM/TOP	Shell element maximum principal stress (of bottom and top). Controlled by STRESS Case Control command.
116	SHELL MIN PRINCIPAL STRESS BOTTOM/TOP	Shell element minimum principal stress (of bottom and top). Controlled by STRESS Case Control command.
660	SHELL NORMAL-X STRAIN TOP	Shell element top side (side 2) normal strain in SURFACE x-direction. Controlled by STRAIN Case Control command.
661	SHELL NORMAL-Y STRAIN TOP	Shell element top side (side 2) normal strain in SURFACE y-direction. Controlled by STRAIN Case Control command.
662	SHELL SHEAR-XY STRAIN TOP	Shell element top side (side 2) normal strain in SURFACE xy-direction (tensor x-face, y-direction). Controlled by STRAIN Case Control command.
663	SHELL MAJOR PRINCIPAL STRAIN TOP	Shell element top side (side 2) major principal strain. Controlled by STRAIN Case Control command.
664	SHELL MINOR PRINCIPAL STRAIN TOP	Shell element top side (side 2) minor principal strain. Controlled by STRAIN Case Control command.
665	SHELL ZERO SHEAR STRAIN ANGLE TOP	Shell element top side (side 2) zero shear strain angle in degrees. Controlled by STRAIN Case Control command.
666	SHELL MAX SHEAR STRAIN TOP	Shell element top side (side 2) maximum shear strain. Controlled by STRAIN Case Control command.
666	SHELL TRESCA STRAIN TOP	Shell element top side (side 2) Tresca strain . Controlled by STRAIN Case Control command.
667	SHELL VON MISES STRAIN TOP	Shell element top side (side 2) von Mises strain. Controlled by STRAIN Case Control command.
680	SHELL NORMAL-X STRAIN BOTTOM	Shell element bottom side (side 1) normal strain in SURFACE x-direction. Controlled by STRAIN Case Control command.
681	SHELL NORMAL-Y STRAIN BOTTOM	Shell element bottom side (side 1) normal strain in SURFACE y-direction. Controlled by STRAIN Case Control command.
682	SHELL SHEAR-XY STRAIN BOTTOM	Shell element bottom side (side 1) normal strain in SURFACE xy-direction (tensor x-face, y-direction). Controlled by STRAIN Case Control command.
683	SHELL MAJOR-PRINCIPAL STRAIN BOTTOM	Shell element bottom side (side 1) major principal strain. Controlled by STRAIN Case Control command.
684	SHELL MINOR PRINCIPAL STRAIN BOTTOM	Shell element bottom side (side 1) minor principal strain. Controlled by STRAIN Case Control command.

Shell Element Grid Point Results Column Descriptions (Continued):

Vector Id	Label	Description
685	SHELL ZERO SHEAR STRAIN ANGLE BOTTOM	Shell element bottom side (side 1) zero shear strain angle in degrees. Controlled by STRAIN Case Control command.
686	SHELL MAX SHEAR STRAIN BOTTOM	Shell element bottom side (side 1) maximum shear strain. Controlled by STRAIN Case Control command.
686	SHELL TRESCA STRAIN BOTTOM	Shell element bottom side (side 1) Tresca strain. Controlled by STRAIN Case Control command.
687	SHELL VON MISES STRAIN BOTTOM	Shell element bottom side (side 1) von Mises strain. Controlled by STRAIN Case Control command.
712	SHELL MAX VON MISES STRAIN BOTTOM/TOP	Shell element maximum von Mises strain (of bottom and top). Controlled by STRAIN Case Control command.
713	SHELL MAX SHEAR STRAIN BOTTOM/TOP	Shell element maximum maximum shear strain (of bottom and top). Controlled by STRAIN Case Control command.
713	SHELL TRESCA STRAIN BOTTOM/TOP	Shell element maximum Tresca strain (of bottom and top). Controlled by STRAIN Case Control command.
714	SHELL MAX PRINCIPAL STRAIN BOTTOM/TOP	Shell element maximum principal strain (of bottom and top). Controlled by STRAIN Case Control command.
715	SHELL MIN PRINCIPAL STRAIN BOTTOM/TO	Shell element minimum principal strain (of bottom and top). Controlled by STRAIN Case Control command.
716	SHELL EQUIVALENT STRESS TOP	Shell element top side (side 2) nonlinear equivalent stress (material nonlinear solutions) or von Mises stress (linear solutions). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
717	SHELL EFFECTIVE STRAIN-ELASTIC TOP	Shell element top side (side 2) effective strain (von Mises). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
717	SHELL EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC TOP	Shell element top side (side 2) effective (nonlinear elastic material) or plastic (elastic-plastic material) strain. Controlled by NLSTRESS Case Control command.
718	SHELL EFFECTIVE STRAIN-CREEP TOP	Shell element top side (side 2) effective creep strain. Controlled by NLSTRESS Case Control command.
719	SHELL EQUIVALENT STRESS BOTTOM	Shell element bottom side (side 1) nonlinear equivalent stress (material nonlinear solutions) or von Mises stress (linear solutions). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting equivalent stress will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).

Shell Element Grid Point Results Column Descriptions (Continued):

Vector Id	Label	Description
720	SHELL EFFECTIVE STRAIN-ELASTIC BOTTOM	Shell element bottom side (side 1) effective strain (von Mises). Note that for prestress solutions regardless of PARAM, ADDPRESTRESS setting effective strain will not include prestress contribution. Controlled by STRESS or STRAIN Case Control commands.
720	SHELL EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC BOTTOM	Shell element bottom side (side 1) effective (nonlinear elastic material) or plastic (elastic-plastic material) strain. Controlled by NLSTRESS Case Control command.
721	SHELL EFFECTIVE STRAIN-CREEP BOTTOM	Shell element bottom side (side 1) effective creep strain. Controlled by NLSTRESS Case Control command.
727	SHELL FIBER DISTANCE TOP	Shell element stress/strain recovery distance (element z-direction) for top side (side 2).
728	SHELL FIBER DISTANCE BOTTOM	Shell element stress/strain recovery distance (element z-direction) for bottom side (side 1).

Solid Element Grid Point Results Column Descriptions:

Vector Id	Label	Description
91	SOLID NORMAL-X	Solid element grid point normal stress in VOLUME x-direction. Controlled by GPSTRESS Case Control command.
92	SOLID NORMAL-Y	Solid element grid point normal stress in VOLUME y-direction. Controlled by GPSTRESS Case Control command.
93	SOLID NORMAL-Z	Solid element grid point normal stress in VOLUME z-direction. Controlled by GPSTRESS Case Control command.
94	SOLID SHEAR-XY	Solid element grid point shear stress in VOLUME xy-direction (tensor x-face, y-direction). Controlled by GPSTRESS Case Control command.
95	SOLID SHEAR-YZ	Solid element grid point shear stress in VOLUME yz-direction (tensor y-face, z-direction). Controlled by GPSTRESS Case Control command.
96	SOLID SHEAR-ZX	Solid element grid point shear stress in VOLUME zx-direction (tensor z-face, x-direction). Controlled by GPSTRESS Case Control command.
97	SOLID PRINCIPAL-A	Solid element grid point maximum principal stress. Controlled by GPSTRESS Case Control command.
98	SOLID PRINICPAL-C	Solid element grid point minimum principal stress. Controlled by GPSTRESS Case Control command.
99	SOLID PRINCIPAL-B	Solid element grid point median principal stress. Controlled by GPSTRESS Case Control command.
100	SOLID MAX SHEAR	Solid element grid point maximum shear stress. Controlled by GPSTRESS Case Control command.
101	SOLID VON MISES	Solid element grid point von Mises stress. Controlled by GPSTRESS Case Control command.
102	SOLID MEAN PRESSURE	Solid element grid point mean pressure stress. Controlled by GPSTRESS Case Control command.
103	SOLID PRINCIPAL-A COSINE-X	Solid element grid point maximum principal stress x-direction cosine. Controlled by GPSTRESS Case Control command.
104	SOLID PRINCIPAL-B COSINE-X	Solid element grid point median principal stress x-direction cosine. Controlled by GPSTRESS Case Control command.
105	SOLID PRINCIPAL-C COSINE-X	Solid element grid point minimum principal stress x-direction cosine. Controlled by GPSTRESS Case Control command.
106	SOLID PRINCIPAL-A COSINE-Y	Solid element grid point maximum principal stress y-direction cosine. Controlled by GPSTRESS Case Control command.
107	SOLID PRINICPAL-B COSINE-Y	Solid element grid point median principal stress y-direction cosine. Controlled by GPSTRESS Case Control command.
108	SOLID PRINCIPAL-C COSINE-Y	Solid element grid point minimum principal stress y-direction cosine. Controlled by GPSTRESS Case Control command.

Solid Element Grid Point Results Column Descriptions (Continued):

Vector Id	Label	Description
109	SOLID PRINCIPAL-A COSINE-Z	Solid element grid point maximum principal stress z-direction cosine. Controlled by GPSTRESS Case Control command.
110	SOLID PRINCIPAL-B COSINE-Z	Solid element grid point median principal stress z-direction cosine. Controlled by GPSTRESS Case Control command.
110	SOLID PRINCIPAL-C COSINE-Z	Solid element grid point minimum principal stress z-direction cosine. Controlled by GPSTRESS Case Control command.
112	SOLID OCTAHEDRAL	Solid element grid point octahedral stress. Controlled by GPSTRESS Case Control command.
118	SOLID MAX PRINCIPAL	Solid element grid point maximum principal stress. Controlled by GPSTRESS Case Control command.
119	SOLID MIN PRINCIPAL	Solid element grid point minimum principal stress. Controlled by GPSTRESS Case Control command.
690	SOLID NORMAL-X	Solid element grid point normal strain in VOLUME x-direction. Controlled by GPSTRAIN Case Control command.
691	SOLID NORMAL-Y	Solid element grid point normal strain in VOLUME y-direction. Controlled by GPSTRAIN Case Control command.
692	SOLID NORMAL-Z	Solid element grid point normal strain in VOLUME z-direction. Controlled by GPSTRAIN Case Control command.
693	SOLID SHEAR-XY	Solid element grid point shear strain in VOLUME xy-direction (tensor x-face, y-direction). Controlled by GPSTRAIN Case Control command.
694	SOLID SHEAR-YZ	Solid element grid point shear strain in VOLUME yz-direction (tensor y-face, z-direction). Controlled by GPSTRAIN Case Control command.
695	SOLID SHEAR-ZX	Solid element grid point shear strain in VOLUME zx-direction (tensor z-face, x-direction). Controlled by GPSTRAIN Case Control command.
696	SOLID PRINCIPAL-A	Solid element grid point maximum principal strain. Controlled by GPSTRAIN Case Control command.
697	SOLID PRINICPAL-C	Solid element grid point minimum principal strain. Controlled by GPSTRAIN Case Control command.
698	SOLID PRINCIPAL-B	Solid element grid point median principal strain. Controlled by GPSTRAIN Case Control command.
699	SOLID MAX SHEAR	Solid element grid point maximum shear strain. Controlled by GPSTRAIN Case Control command.
700	SOLID VON MISES	Solid element grid point von Mises strain. Controlled by GPSTRAIN Case Control command.
701	SOLID MEAN PRESSURE	Solid element grid point mean pressure strain. Controlled by GPSTRAIN Case Control command.

Solid Element Grid Point Results Column Descriptions (Continued):

Vector Id	Label	Description
702	SOLID PRINCIPAL-A COS X	Solid element grid point maximum principal strain x-direction cosine. Controlled by GPSTRAIN Case Control command.
703	SOLID PRINCIPAL-B COS X	Solid element grid point median principal strain x-direction cosine. Controlled by GPSTRAIN Case Control command.
704	SOLID PRINCIPAL-C COS X	Solid element grid point minimum principal strain x-direction cosine. Controlled by GPSTRAIN Case Control command.
705	SOLID PRINCIPAL-A COS Y	Solid element grid point maximum principal strain y-direction cosine. Controlled by GPSTRAIN Case Control command.
706	SOLID PRINICPAL-B COS Y	Solid element grid point median principal strain y-direction cosine. Controlled by GPSTRAIN Case Control command.
707	SOLID PRINCIPAL-C COS Y	Solid element grid point minimum principal strain y-direction cosine. Controlled by GPSTRAIN Case Control command.
708	SOLID PRINCIPAL-A COS Z	Solid element grid point maximum principal strain z-direction cosine. Controlled by GPSTRAIN Case Control command.
709	SOLID PRINCIPAL-B COS Z	Solid element grid point median principal strain z-direction cosine. Controlled by GPSTRAIN Case Control command.
710	SOLID PRINCIPAL-C COS Z	Solid element grid point minimum principal strain z-direction cosine. Controlled by GPSTRAIN Case Control command.
711	SOLID OCTAHEDRAL	Solid element grid point octahedral strain. Controlled by GPSTRAIN Case Control command.
722	SOLID MAX PRINCIPAL	Solid element grid point maximum principal strain. Controlled by GPSTRAIN Case Control command.
723	SOLID MIN PRINCIPAL	Solid element grid point minimum principal strain. Controlled by GPSTRAIN Case Control command.
724	SOLID EQUIVALENT STRESS	Solid element grid point nonlinear equivalent stress (material nonlinear solutions) or von Mises stress (linear solutions). Note that for prestress solutions regardless of PARAM, ADDPREGPSTRESS setting equivalent stress will not include prestress contribution. Controlled by GPSTRESS or STRAIN Case Control commands (linear solutions) and NLSTRESS Case Control command (nonlinear solutions).
771	SOLID EFFECTIVE STRAIN-ELASTIC	Solid element grid point effective strain (von Mises). Note that for prestress solutions regardless of PARAM, ADDPREGPSTRESS setting effective strain will not include prestress contribution. Controlled by GPSTRESS or STRAIN Case Control commands.
771	SOLID EFFECTIVE STRAIN-PLASTIC/NONLINEAR ELASTIC	Solid element grid point effective (nonlinear elastic material) or plastic (elastic-plastic material) strain. Controlled by NLSTRESS Case Control command.
772	SOLID EFFECTIVE STRAIN-CREEP	Solid element grid point effective creep strain. Controlled by NLSTRESS Case Control command.

Contact Surface Element Grid Point Results Column Descriptions:

Vector Id	Label	Description
303	CONTACT STATUS	Quad and tri contact surface grid point contact status (1=open, 2=slide – closed with no friction defined, 3=stick – closed with friction and holding, 4=slip – closed with friction and slipping, 5=weld). Controlled by STRESS Case Control command.
332	SSHL CONTACT PRESSURE	Quad and tri contact surface grid point pressure. Positive indicates compression. Controlled by STRESS Case Control command.
333	SSHL CONTACT TRACTION-X	Quad and tri contact surface grid point traction in the x-direction. Controlled by STRESS Case Control command.
334	SSHL CONTACT TRACTION-Y	Quad and tri contact surface grid point traction in the y-direction. Controlled by STRESS Case Control command.
335	SSHL CONTACT EQUIVALENT STRESS	Quad and tri contact surface grid point equivalent stress used in weld bond failure analysis. Controlled by STRESS Case Control command.
336	SSHL BOND EFFECTIVE DISPLACEMENT	Quad and tri contact surface grid point bond effective displacement. Controlled by STRESS Case Control command.
337	SSHL BOND DAMAGE	Quad and tri contact surface grid point bond damage. Controlled by STRESS Case Control command.

Miscellaneous Element Grid Point Results Column Descriptions:

Vector Id	Label	Description
775	SHELL MESH CONVERGENCE ERROR BOTTOM	Shell element grid point bottom side (side 1) normalized mesh convergence error. Controlled by STRESS(CORNER) Case Control command and PARAM, STRESSERROR or GPDISCONT Case Control command.
776	SHELL MESH CONVERGENCE ERROR TOP	Shell element grid point top side (side 2) normalized mesh convergence error. Controlled by STRESS(CORNER) Case Control command and PARAM, STRESSERROR or GPDISCONT Case Control command.
777	SHELL MAX MESH CONVERGENCE ERROR BOTTOM/TOP	Shell element maximum normalized mesh convergence error (of bottom and top). Controlled by STRESS(CORNER) Case Control command and PARAM, STRESSERROR or GPDISCONT Case Control command.
778	SOLID MESH CONVERGENCE ERROR	Solid element grid point normalized mesh convergence error. Controlled by STRESS(CORNER) Case Control command and PARAM, STRESSERROR or GPDISCONT Case Control command.

Structural Neutral File Element Internal Load Vector Results Column Descriptions

Element Internal Load Vector Results Column Descriptions:

Vector Id	Label	Description
85000 + 6(node - 1)	NODE i T1 INTERNAL FORCE	Element nodal force at node i in direction T1 (translational).
85001 + 6(node - 1)	NODE i T2 INTERNAL FORCE	Element nodal force at node i in direction T2 (translational).
85002 + 6(node - 1)	NODE i T3 INTERNAL FORCE	Element nodal force at node i in direction T3 (translational).
85003 + 6(node - 1)	NODE i R1 INTERNAL MOMENT	Element nodal moment at node i in direction R1 (rotational).
85004 + 6(node - 1)	NODE i R2 INTERNAL MOMENT	Element nodal moment at node i in direction R2 (rotational).
85005 + 6(node - 1)	NODE i R3 INTERNAL MOMENT	Element nodal moment at node i in direction R3 (rotational).

Structural Neutral File Grid Point Vector Results Column Descriptions

Grid Point Displacement and Force Vector Results Column Descriptions:

Vector Id	Label	Description
1	TOTAL TRANSLATION	Grid point translational displacement vector resultant. Controlled by DISPLACEMENT Case Control command.
2	T1 TRANSLATION	Grid point displacement vector in T1 direction (translational). Controlled by DISPLACEMENT Case Control command.
3	T2 TRANSLATION	Grid point displacement vector in T2 direction (translational). Controlled by DISPLACEMENT Case Control command.
4	T3 TRANSLATION	Grid point displacement vector in T3 direction (translational). Controlled by DISPLACEMENT Case Control command.
5	TOTAL ROTATION	Grid point rotational displacement vector resultant. Controlled by DISPLACEMENT Case Control command.
6	R1 ROTATION	Grid point displacement vector in R1 direction (rotational). Controlled by DISPLACEMENT Case Control command.
7	R2 ROTATION	Grid point displacement vector in R2 direction (rotational). Controlled by DISPLACEMENT Case Control command.
8	R3 ROTATION	Grid point displacement vector in R3 direction (rotational). Controlled by DISPLACEMENT Case Control command.
11	TOTAL VELOCITY	Grid point translational velocity vector resultant. Controlled by VELOCITY Case Control command.
12	T1 VELOCITY	Grid point velocity vector in T1 direction (translational). Controlled by VELOCITY Case Control command.
13	T2 VELOCITY	Grid point velocity vector in T2 direction (translational). Controlled by VELOCITY Case Control command.
14	T3 VELOCITY	Grid point velocity vector in T3 direction (translational). Controlled by VELOCITY Case Control command.
15	TOTAL ANGULAR VELOCITY	Grid point angular velocity vector resultant. Controlled by VELOCITY Case Control command.
16	R1 ANGULAR VELOCITY	Grid point velocity vector in R1 direction (rotational). Controlled by OLOAD Case Control command.
17	R2 ANGULAR VELOCITY	Grid point velocity vector in R2 direction (rotational). Controlled by VELOCITY Case Control command.
18	R3 ANGULAR VELOCITY	Grid point velocity vector in R3 direction (rotational). Controlled by VELOCITY Case Control command.
21	TOTAL ACCELERATION	Grid point translational acceleration vector resultant. Controlled by ACCELERATION Case Control command.
22	T1 ACCELERATION	Grid point acceleration vector in T1 direction (translational). Controlled by ACCELERATION Case Control command.
23	T2 ACCELERATION	Grid point acceleration vector in T2 direction (translational). Controlled by ACCELERATION Case Control command.
24	T3 ACCELERATION	Grid point acceleration vector in T3 direction (translational). Controlled by ACCELERATION Case Control command.

Grid Point Displacement and Force Vector Results Column Descriptions (Continued):

Vector Id	Label	Description
25	TOTAL ANGULAR ACCELERATION	Grid point angular acceleration vector resultant. Controlled by ACCELERATION Case Control command.
26	R1 ACCELERATION	Grid point acceleration vector in R1 direction (rotational). Controlled by ACCELERATION Case Control command.
27	R2 ACCELERATION	Grid point acceleration vector in R2 direction (rotational). Controlled by ACCELERATION Case Control command.
28	R3 ACCELERATION	Grid point acceleration vector in R3 direction (rotational). Controlled by ACCELERATION Case Control command.
41	TOTAL APPLIED FORCE	Grid point applied force vector resultant. Controlled by OLOAD Case Control command.
42	T1 APPLIED FORCE	Grid point applied force vector in T1 direction (translational). Controlled by OLOAD Case Control command.
43	T2 APPLIED FORCE	Grid point applied force vector in T2 direction (translational). Controlled by OLOAD Case Control command.
44	T3 APPLIED FORCE	Grid point applied force vector in T3 direction (translational). Controlled by OLOAD Case Control command.
45	TOTAL APPLIED MOMENT	Grid point applied moment vector rotational resultant. Controlled by OLOAD Case Control command.
46	R1 APPLIED MOMENT	Grid point applied moment vector in R1 direction (rotational). Controlled by OLOAD Case Control command.
47	R2 APPLIED MOMENT	Grid point applied moment vector in R2 direction (rotational). Controlled by OLOAD Case Control command.
48	R3 APPLIED MOMENT	Grid point applied moment vector in R3 direction (rotational). Controlled by OLOAD Case Control command.
51	TOTAL SPC FORCE	Grid point single point constraint force vector resultant. Controlled by SPCFORCES Case Control command.
52	T1 SPC FORCE	Grid point single point constraint force vector in T1 direction (translational). Controlled by SPCFORCES Case Control command.
53	T2 SPC FORCE	Grid point single point constraint force vector in T2 direction (translational). Controlled by SPCFORCES Case Control command.
54	T3 SPC FORCE	Grid point single point constraint force vector in T3 direction (translational). Controlled by SPCFORCES Case Control command.
55	TOTAL SPC MOMENT	Grid point single point constraint moment vector resultant. Controlled by SPCFORCES Case Control command.
56	R1 SPC MOMENT	Grid point single point constraint moment vector in R1 direction (rotational). Controlled by SPCFORCES Case Control command.
57	R2 SPC MOMENT	Grid point single point constraint moment vector in R2 direction (rotational). Controlled by SPCFORCES Case Control command.
58	R3 SPC MOMENT	Grid point single point constraint moment vector in R3 direction (rotational). Controlled by SPCFORCES Case Control command.

Grid Point Displacement and Force Vector Results Column Descriptions (Continued):

Vector Id	Label	Description
61	TOTAL INTERNAL FORCE	Grid point internal force vector resultant. Controlled by GPFORCE Case Control command.
62	T1 INTERNAL FORCE	Grid point internal force vector in T1 direction (translational). Controlled by GPFORCE Case Control command.
63	T2 INTERNAL FORCE	Grid point internal force vector in T2 direction (translational). Controlled by GPFORCE Case Control command.
64	T3 INTERNAL FORCE	Grid point internal force vector in T3 direction (translational). Controlled by GPFORCE Case Control command.
65	TOTAL INTERNAL MOMENT	Grid point internal moment vector rotational resultant. Controlled by GPFORCE Case Control command.
66	R1 INTERNAL MOMENT	Grid point internal moment vector in R1 direction (rotational). Controlled by GPFORCE Case Control command.
67	R2 INTERNAL MOMENT	Grid point internal moment vector in R2 direction (rotational). Controlled by GPFORCE Case Control command.
68	R3 INTERNAL MOMENT	Grid point internal moment vector in R3 direction (rotational). Controlled by GPFORCE Case Control command.
151	TOTAL MPC FORCE	Grid point multipoint constraint force vector resultant. Controlled by MPCFORCES Case Control command.
152	T1 MPC FORCE	Grid point multipoint constraint force vector in T1 direction (translational). Controlled by MPCFORCES Case Control command.
153	T2 MPC FORCE	Grid point multipoint constraint force vector in T2 direction (translational). Controlled by MPCFORCES Case Control command.
154	T3 MPC FORCE	Grid point multipoint constraint force vector in T3 direction (translational). Controlled by MPCFORCES Case Control command.
155	TOTAL MPC MOMENT	Grid point multipoint constraint moment vector rotational resultant. Controlled by MPCFORCES Case Control command.
156	R1 MPC FORCE	Grid point multipoint constraint moment vector in R1 direction (rotational). Controlled by MPCFORCES Case Control command.
157	R2 MPC FORCE	Grid point multipoint constraint moment vector in R2 direction (rotational). Controlled by MPCFORCES Case Control command.
158	R3 MPC FORCE	Grid point multipoint constraint moment vector in R3 direction (rotational). Controlled by MPCFORCES Case Control command.

Heat Transfer Neutral File Element Results Column Descriptions

Rod Element Results Column Descriptions:

Vector Id	Label	Description
3101	ROD THERMAL GRADIENT	Rod element thermal gradient in element x-direction. Controlled by FLUX Case Control command.
3104	ROD THERMAL GRADIENT RESULTANT	Rod element thermal gradient vector resultant. Controlled by FLUX Case Control command.
3105	ROD HEAT FLUX	Rod element heat flux in element x-direction. Controlled by FLUX Case Control command.
3108	ROD HEAT FLUX RESULTANT	Rod element heat flux vector resultant. Controlled by FLUX Case Control command.

Bar Element Results Column Descriptions:

Vector Id	Label	Description
3201	BAR THERMAL GRADIENT	Bar element thermal gradient in element x-direction. Controlled by FLUX Case Control command.
3204	BAR THERMAL GRADIENT RESULTANT	Bar element thermal gradient vector resultant. Controlled by FLUX Case Control command.
3205	BAR HEAT FLUX	Bar element heat flux in element x-direction. Controlled by FLUX Case Control command.
3208	BAR HEAT FLUX RESULTANT	Bar element heat flux vector resultant. Controlled by FLUX Case Control command.

Beam Element Results Column Descriptions:

Vector Id	Label	Description
3301	BEAM THERMAL GRADIENT	Beam element thermal gradient in element x-direction. Controlled by FLUX Case Control command.
3304	BEAM THERMAL GRADIENT RESULTANT	Beam element thermal gradient vector resultant. Controlled by FLUX Case Control command.
3305	BEAM HEAT FLUX	Beam element heat flux in element x-direction. Controlled by FLUX Case Control command.
3308	BEAM HEAT FLUX RESULTANT	Beam element heat flux vector resultant. Controlled by FLUX Case Control command.

Cable Element Results Column Descriptions:

Vector Id	Label	Description
3801	CABLE THERMAL GRADIENT	Cable element thermal gradient in element x-direction. Controlled by FLUX Case Control command.
3804	CABLE THERMAL GRADIENT RESULTANT	Cable element thermal gradient vector resultant. Controlled by FLUX Case Control command.
3805	CABLE HEAT FLUX	Cable element heat flux in element x-direction. Controlled by FLUX Case Control command.
3808	CABLE HEAT FLUX RESULTANT	Cable element heat flux vector resultant. Controlled by FLUX Case Control command.

Pipe Element Results Column Descriptions:

Vector Id	Label	Description
3901	PIPE THERMAL GRADIENT	Pipe element thermal gradient in element x-direction. Controlled by FLUX Case Control command.
3904	PIPE THERMAL GRADIENT RESULTANT	Pipe element thermal gradient vector resultant. Controlled by FLUX Case Control command.
3905	PIPE HEAT FLUX	Pipe element heat flux in element x-direction. Controlled by FLUX Case Control command.
3908	PIPE HEAT FLUX RESULTANT	Pipe element heat flux vector resultant. Controlled by FLUX Case Control command.

Weld Element Results Column Descriptions:

Vector Id	Label	Description
4001	WELD THERMAL GRADIENT	Weld element thermal gradient in element x-direction. Controlled by FLUX Case Control command.
4004	WELD THERMAL GRADIENT RESULTANT	Weld element thermal gradient vector resultant. Controlled by FLUX Case Control command.
4005	WELD HEAT FLUX	Weld element heat flux in element x-direction. Controlled by FLUX Case Control command.
4008	WELD HEAT FLUX RESULTANT	Weld element heat flux vector resultant. Controlled by FLUX Case Control command.

Bush Element Results Column Descriptions:

Vector Id	Label	Description
4101	BUSH THERMAL GRADIENT	Bush element thermal gradient in element x-direction. Controlled by FLUX Case Control command.
4104	BUSH THERMAL GRADIENT RESULTANT	Bush element thermal gradient vector resultant. Controlled by FLUX Case Control command.
4105	BUSH HEAT FLUX	Bush element heat flux in element x-direction. Controlled by FLUX Case Control command.
4108	BUSH HEAT FLUX RESULTANT	Bush element heat flux vector resultant. Controlled by FLUX Case Control command.

HBDY Element Results Column Descriptions:

Vector Id	Label	Description
4201	HBDY APPLIED LOAD	HBDY element applied load. Controlled by FLUX Case Control command.
4202	HBDY CONVECTION LOAD	HBDY element convection load. Controlled by FLUX Case Control command.
4203	HBDY RADIATION LOAD	HBDY element radiation load. Controlled by FLUX Case Control command.
4204	HBDY TOTAL LOAD	Total of HBDY element applied, convection, and radiation loads. Controlled by FLUX Case Control command.

Shell Element Results Column Descriptions:

Vector Id	Label	Description
6001	SHELL THERMAL GRADIENT-X	Shell element thermal gradient in SURFACE x-direction. Controlled by FLUX Case Control command.
6002	SHELL THERMAL GRADIENT-Y	Shell element thermal gradient in SURFACE y-direction. Controlled by FLUX Case Control command.
6004	SHELL THERMAL GRADIENT RESULTANT	Shell element thermal gradient vector resultant. Controlled by FLUX Case Control command.
6005	SHELL HEAT FLUX-X	Shell element heat flux in SURFACE x-direction. Controlled by FLUX Case Control command.
6006	SHELL HEAT FLUX-Y	Shell element heat flux in SURFACE y-direction. Controlled by FLUX Case Control command.
6008	SHELL HEAT FLUX RESULTANT	Shell element heat flux vector resultant. Controlled by FLUX Case Control command.

Solid Element Results Column Descriptions:

Vector Id	Label	Description
60001	SOLID THERMAL GRADIENT-X	Solid element thermal gradient in VOLUME x-direction. Controlled by FLUX Case Control command.
60002	SOLID THERMAL GRADIENT-Y	Solid element thermal gradient in VOLUME y-direction. Controlled by FLUX Case Control command.
60003	SOLID THERMAL GRADIENT-Z	Solid element thermal gradient in VOLUME z-direction. Controlled by FLUX Case Control command.
60004	SOLID THERMAL GRADIENT RESULTANT	Solid element thermal gradient vector resultant. Controlled by FLUX Case Control command.
60005	SOLID HEAT FLUX-X	Solid element heat flux in VOLUME x-direction. Controlled by FLUX Case Control command.
60006	SOLID HEAT FLUX-Y	Solid element heat flux in VOLUME y-direction. Controlled by FLUX Case Control command.
60007	SOLID HEAT FLUX-Z	Solid element heat flux in VOLUME z-direction. Controlled by FLUX Case Control command.
60008	SOLID HEAT FLUX RESULTANT	Solid element heat flux vector resultant. Controlled by FLUX Case Control command.

Heat Transfer Neutral File Vector Results Column Descriptions

Grid Point Temperature and Heat Flow Vector Results Column Descriptions:

Vector Id	Label	Description
1	TEMPERATURE	Grid point temperature. Controlled by THERMAL Case Control command.
11	ENTHALPY	Grid point enthalpy. Controlled by ENTHALPY Case Control command.
21	ENTHALPY RATE	Grid point enthalpy rate of change. Controlled by HDOT Case Control command.
41	APPLIED HEAT FLOW	Grid point applied heat flow. Controlled by OLOAD Case Control command.
51	SPC HEAT FLOW	Grid point single point constraint heat flow. Controlled by SPCFORCES Case Control command.
151	MPC HEAT FLOW	Grid point multipoint constraint heat flow. Controlled by MPCFORCES Case Control command.

MODEL INPUT FILE COMMAND AND ENTRY SUMMARY

Model Input File Case Control Command Summary:

Case Control Commands				
Subcase Control	**Output Control**	**Model Modification**	**Model Generation**	**Miscellaneous**
ANALYSIS	ACCELERATION	ELEMDELETE*	CONTACTGENERATE*	INCLUDE
BEGIN BULK	CORELLATE*	GRIDSCALEFACTOR*	CYSYMGENERATE*	MODESET
B2GG	DISPLACEMENT	GRIDOFFSET*	DISPINTERPOLATE*	PARAM
CMETHOD	ECHO		FATIGUE*	RESVEC
CONTACTSET*	ELFORCE		IMPACTGENERATE*	SKIPOFF
DDAM*	ELSTRAIN*		LOADINTERPOLATE*	SKIPON
DEFORM	ELSTRESS		SELEMGENERATE*	
DMIGADD*	ENTHALPY		SETGENERATE*	
DLOAD	ESE		TEMPINTERPOLATE*	
ELEMSET*	EXTSEOUT		TEMPGENERATE*	
FREQUENCY	FLUX		TEMPSCALEFACTOR*	
IC	FORCE		VIBFATIGUE*	
INITIALSTRAIN	GEOMCHECK		WELDGENERATE*	
K2GG	GLBMATRIX*		XSETGENERATE*	
LOAD	GPDISCONT*			
LOADSET	GPFLUX*			
M2GG	GPFORCE			
METHOD	GPSTRAIN*			
MPC	GPSTRESS			
NONLINEAR	GROUNDCHECK			
NLPARM	HDOT			
P2G	LABEL			
RANDOM	LINE			
SDAMPING	MODES			
SOLUTION	MPCFORCES			
SPC	NLSTRESS			
SUBCASE	OFREQUENCY			
SUBCOM	OLOAD			
SUBSEQ	OTIME			
TEMPERATURE	RESULTSLIMITS*			
TSTEP	SET			
TSTEPNL	SPCFORCES			
	STRAIN			
	STRESS			
	SUBTITLE			
	SURFACE			
	THERMAL			
	TITLE			
	VECTOR			
	VELOCITY			
	VOLUME			
	XYDATA*			

Model Input File Case Control Command Summary (Continued):

Case Control Commands				
Subcase Control	**Output Control**	**Model Modification**	**Model Generation**	**Miscellaneous**
	XYDATAGENERATE*			
	XYPLOT			
	XYPRINT			

* Denotes Autodesk Nastran extension

Model Input File Bulk Data Entry Summary:

Bulk Data Entries						
Element	**Property**	**Material**	**Load**	**Displacement**	**Coordinate**	**Miscellaneous**
BCONP	PBAR	CONCRETE*	DAREA	MPC	CORD1C	ASET
BFRIC	PBEAM	ENDATA*	DEFORM	MPCADD	CORD1R	ASET1
BLSEG	PBUSH	MAT1	DELAY	SPC	CORD1S	BAROR
BOUTPUT	PBUSH1D	MAT2	DLOAD	SPC1	CORD2C	BEAMOR
BSCONP	PCABLE*	MAT4	DPHASE	SPCADD	CORD2R	BSET
BSSEG	PCOMP	MAT5	DTI, SPECSEL	SPCD	CORD2S	BSET1
BWIDTH	PCONV	MAT8	DTI, SPSEL	TEMPBC		CBARAO
CBAR	PDAMP	MAT9	FORCE			CSET
CBEAM	PDAMPT	MAT12*	FORCE1			CSET1
CBUSH	PELAS	MATHP	FREQ			DDAMDATA
CBUSH1D	PELAST	MATHP1*	FREQ1			DMIG
CCABLE*	PGAP	MATL8*	FREQ2			EIGRL
CDAMP1	PHBDY	MATS1	FREQ3			EIGC
CDAMP2	PMASS	MATST1*	FREQ4			EIGR
CDAMP3	PMOUNT*	MATT1	GRAV			ESET*
CDAMP4	PPIPE	MATT2	LOAD			ESET1*
CELAS1	PROD	MATT4	LSEQ			ENDDATA
CELAS2	PSHEAR	MATT5	MOMENT			EPOINT
CELAS3	PSHELL	MATT8*	MOMENT1			FATIGUE*
CELAS4	PTUBE	MATT9	NOLIN1			GRDSET
CGAP	PVISC	MATT12*	NOLIN2			GRID
CHBDYG	PWELD	MATVE	NOLIN3			INCLUDE
CHBDYP		NITINOL*	NOLIN4			NLPARM
CHEXA		RADM	PLOAD			NLPCI
CMASS1		RADMT	PLOAD1			OMIT
CMASS2		SNDATA*	PLOAD2			OMIT1
CMASS3		TABLEM1	PLOAD4			PARAM
CMASS4		TABLEM2	PLOADG			QSET
CONM1		TABLEM3	PLOADX1			QSET1
CONM2		TABLEM4	QBDY1			SEELT
CONROD		TABLES1	QBDY2			SELABEL
CONV		TABLEST	QBDYG*			SESET
CPENTA		TABVE	QHBDY			SNDATA*
CPIPE			QVOL			SPOINT
CQUAD4			RADBC			SUPORT
CQUAD8			RADSET			TABDMP1
CQUADR			RANDPS			TOPVAR
CROD			RANDT1			TSTEPNL
CSHEAR			RFORCE			VIEW
CTETRA			RLOAD1			VIEW3D
CTRIA3			RLOAD2			VFATIGUE
CTRIA6			SLOAD			XSET*
			STRAIN			XSET1*

(Continued)

Model Input File Bulk Data Entry Summary (Continued):

Bulk Data Entries						
Element	**Property**	**Material**	**Load**	**Displacement**	**Coordinate**	**Miscellaneous**
CTRIAR			TABFV			
CTRIAX6			TABLED1			
CTUBE			TABLED2			
CVISC			TABLED3			
CWELD			TABLED4			
GENEL			TABLEVF			
RBAR			TABRND1			
RBE1			TEMP			
RBE2			TEMPD			
RBE3			TEMPP1			
RROD			TEMPRB			
RSPLINE			TIC			
RTRPLT			TLOAD1			
			TLOAD2			
			TSTEP			

* Denotes Autodesk Nastran extension

(Continued)