COSMOS/M

A COMPLETE FINITE ELEMENT ANALYSIS SYSTEM

2.7

Command Reference

STRUCTURAL RESEARCH & ANALYSIS CORP.

First Edition
COSMOS/M 2.7
December 2001

Copyright

Disclaimer

The authors have taken due care in preparing this manual and the examples presented herein. In no event shall SRAC assume any liability or responsibility to any person or company for direct or indirect damage resulting from the use of the information contained herein or any discrepancies between this information and the actual operation of the software.

Licenses & Trademarks





Contents










1 Introduction

All the material in this manual is contained in the online help system for GEOSTAR. The online help contains material that is not included in this book.

Introduction

GEOSTAR is an interactive full three dimensional graphic geometric modeler, mesh generator, and finite element pre- and postprocessor. The geometric modeling capabilities of GEOSTAR are based on mixed boundary representation (B-rep) and parametric cubic equations.

The primary application of GEOSTAR is to function as a pre- and postprocessor to the COSMOS/M finite element analysis system. The user can create the model, supply all related analysis information, invoke the analysis, and review the results, all from within GEOSTAR in an interactive, menu driven, graphic environment.

About the COSMOS/M Command Reference Guide

The COSMOS/M Command Reference provides a comprehensive description of all commands related to pre- and postprocessing as well as to those related to the various analyses modules. For every command, the reference gives a general description, the syntax, the valid input for every argument, notes and examples. The cryptic name as well as the path to access the command are provided for each command. Graphical illustrations are also given for some commands for complete presentation.

The reference consists of an introduction and ten chapters that correspond to GEOSTAR's main menus. Each chapter is divided into sections such that each section covers a submenu. Menus, and submenus are shown in the order they appear in GEOSTAR from left to right. Commands in each submenu are ordered from top to bottom. The Command Reference Guide is intended to serve as a reference for use with the COSMOS/M User Guide. *All the information in this reference are is available in the online help of GEOSTAR.*

Command names used in this reference are the cryptic names that appear by default in GEOSTAR screen. Command names are intuitive, but the MENUTYPE command (Control menu) may be used to show non-cryptic extended names instead (long form).

Chapter 2 contains commands to create new files, open and save databases, edit text files, load sessions or gfm files, as well as saving images and printing.

Chapter 3 contains commands to identify, compress, plot, list, delete, undelete, and erase geometric entities, and nodes and elements. The Reset command resets default settings.

Chapter 4 contains commands related to the creation and processing of geometric entities, grid and coordinate systems.

Chapter 5 contains commands related to the creation of nodes and elements by meshing geometric entities or by direct generation.

Chapter 6 contains commands related to specification of element types, materials, and other related properties.

Chapter 7 contains commands related to specifying various types of loads and boundary conditions. Loads and boundary conditions may be applied directly to nodes or elements, or through their association with geometric entities.

Chapter 8 contains general utility commands, commands to control active sets, and selection sets, in addition to commands related to interface with CAD systems, and other FEA packages. The parametric submenu contains commands related to parametric variables, functions, arrays, and macros and is described in detail in the COSMOS/M User Guide. The miscellaneous submenu provides a number of miscellaneous commands.

Chapter 9 contains commands related to controlling the view, display, multiple windows, and xy-plotting of pre- and postprocessing graphs as well as user-defined graphs.

Chapter 10 contains commands related to specifying and running the various analysis options of the COSMOS/M program.

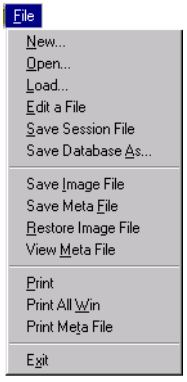
Chapter 11 contains commands related to postprocessing of the results for various types of analyses in both text and graphical formats.




2 File Menu

FILE Menu

Figure 2-1: File Menu



This menu contains commands to create, open, save, or print files, edit text files, load session (.ses) or neutral (.gfm) files, and save and restore images files.

NEWPROB

FILE, New or **File, Open**

File, New creates a new problem. If you specify an existing problem name, GEOSTAR prompts you on whether you want to replace it. If you open an existing problem as new, GEOSTAR saves the old session (*.ses) to (*.bck) and deletes all other database files.

File, Open opens an existing problem.

File name
Browse (or type in a name) to specify the file name.

Replace Problem?
This question is only prompted when you select File, New and specify an existing problem.

Yes	create a new database. The old session file is saved with bck extension.
No	do not replace the existing problem and return to the dialog box to specify another file.

Procedures:

To start GEOSTAR and open a new problem:

1. Start GEOSTAR. The **Open Problem Files** dialog box opens.
2. Browse to the folder in which you want to create the new problem.
3. In the **File Name** field, type in the name of the new problem.
4. Click **Open**.
5. Start issuing commands interactively or use **File, Load** to load files with valid GEOSTAR commands.

To start GEOSTAR and open an existing problem:

1. Start GEOSTAR. The **Open Problem Files** dialog box opens.
2. Browse to the folder which includes the problem.
3. In the **Files of Type** field, select **.GEN** or **.SES**.
4. Double-click the desired file or select the file and click **Open**.

To open an existing problem while in GEOSTAR:

1. Click **File, Open**. The **Open Problem Files** dialog box opens.
2. Browse to the folder which includes the problem.
3. In the **Files of Type** field, select **.GEN** or **.SES**.
4. Select the desired problem, its name appears in the **File Name** field.
5. Double-click the desired file or select the file and click **Open**. GEOSTAR will close the current problem and switch to the selected problem.

To start a new problem while in GEOSTAR:

1. Click **File, New**. The **New Problem Files** dialog box opens.
2. Browse to the folder in which you want to create the new problem.
3. In the **File Name** field, type in the name of the new problem.
4. Click **Save**. GEOSTAR will close the current problem and start the new one.
5. Start issuing commands interactively or use **File, Load** to load files with valid GEOSTAR commands. If a database exists for the new name, you will be given the choice to open it as a new or old problem. If you open an existing problem as new, the corresponding session file (.ses) will be saved in a backup file (.bck). All other database files are deleted.

FILE

FILE, Load

The FILE command executes a sequence of commands in the specified input file.

Input File Name
Name of the input file (with extension), in which the commands to be processed are listed.

Log Option Flag

A flag to enable or disable the writing of commands into Session file.
1: On write commands to the Session file (*.ses)
0: Off do not write commands to the Session file (*.ses).
(default is on)

Display Option Flag
A flag to activate or deactivate plotting on the screen.
1: On display plots
0: Off do not display plots
(default is on)

Echo Option Flag
A flag to enable or disable the echoing of commands read from the command input file in the message window.
0: Off do not echo commands
1: On echo commands
(default is on)

Message Option Flag
A flag to activate or deactivate the writing of GEOSTAR message in response to the given commands.
0: Off do not write messages
1: On write messages
(default is on)

Notes

1. Any number of files can be loaded sequentially.
2. Deactivating the loading flags (the display flag in particular) accelerates the loading process.

Procedure

To load a file to GEOSTAR:

1. Create a new problem with any name, or open an existing one.
2. Click **File, Load**. The **File** dialog box opens.
3. In the **Input File Name** field, type in the name of the file (in the default folder) to be loaded. Or click the Find button to select the desired folder and specify the name.
4. Select the desired options in the other fields.
5. Click **OK**. GEOSTAR will execute the commands in the specified file.

Example: `FILE,TEST.GEO,,0,,,`

This command reads in and executes all commands in the input file TEST.GEO. The display flag is deactivated to accelerate loading the file. After the file is processed, you can interactively issue more commands or load in other files.

EDIT

File, Edit a File

The EDIT command opens a text file.

File Name
Name of the text file to be edited.

Editor
Name of the editor to be used. Default is Notepad.

SAVE

FILE, Save Session File or **Save Database As**

Commands are temporarily saved in a buffer as they are issued. This buffer takes up to five commands and is dumped to the session file whenever filled. The **SAVE, Session File** command updates the session file even if the buffer has one command in it.

The **SAVE, Database As** command copies all database files to a new name. Use this command to save your work at a particular stage for later retrieval.

Save Option
Saving option. Used only if you type in the command in the GeoStar Console window where 0 saves the session file only and 1 copies all files to the new name.

Problem name without extension
New database name. Use the Find button to browse to the desired folder.

METAFILE

File, Save Meta File

The METAFILE command generates a meta file for the image in the active window. A meta file is an ASCII file that can be plotted using the PLT_METAFIL command.

File name
Name of file. Use the Find button to browse to the desired folder
(default is problem-name.M??, where?? is a number that is incremented each time a file is generated)

(Over-write)
Overwrite flag. Prompted only if the specified file exists.
0 do not overwrite and quit
1 overwrite file
(default is 0)

IMAGESAV

File, Save Image File

The IMAGESAV command saves the selected portion of an image on the screen into a file.

File name
Name of the file. Select DIB or BMP.
(default name is problem name .DIB)

To save a rectangular area:

1. Click **File, Save Image File**.
2. Browse to the folder in which you want to create the image file.
3. In the **File Name** field, type in the name of the file.
4. In the **Save As Type** field, select **DIB** or **BMP**.
5. Click **Save**. The cross hairs appear on the screen.
6. Move the cross hairs to one of the corners of the rectangular area and click the mouse.
7. Move the cross hairs to the opposite corner of the rectangular area and click the mouse.

IMAGERES

FILE, Restore Image File

The IMAGERES command restores a previously saved image.

File name
Name of the file. Choose

File Type
Choose DIB or BMP files.
(The default file name is "problem-name.DIB")

To restore a DIB or BMP file:

1. Click **File, Restore Image File**.
2. In the **Files of Type** field, select **DIB** or **BMP**.
3. Browse to the folder which contains the image file.
4. Select the desired file.
5. Click **Open**.




VIEW_META

FILE, View Meta File

The VIEW_META command displays a GEOSTAR meta file (generated by the METAFILE command) on the screen.

File name
Name of file to be viewed.

Note
A hardcopy of the viewed file cannot be obtained by using commands like LASERJET, PAINTJET, PLOTTER and similar commands that require the existence of database files. In addition to regular image capturing in windows, you can use the SCREENPLOT command to dump the viewed file to a hardcopy device. The PLT_METAFIL command can also be used to generate a hardcopy of the meta file.

PRINT

FILE, Print

The PRINT command opens the Print dialog box to print the active window.

PRINT ALL WIN

File, Print All Win

The PRINT ALL WIN command opens the Print dialog box to print all windows.

PRINT META FILE

File, Print Meta File

The PRINT META FILE command generates a hard copy from an existing META file (refer to the METAFILE command) on the system default printer or other printer connected to the system.

Meta filename
Name of metafile to be printed.
(default is problem_name.mij, where i and j are digits from 0 to 9)

EXIT

FILE, Exit

The EXIT command exits GEOSTAR. The model is already saved.

3 Edit Menu

EDIT Menu

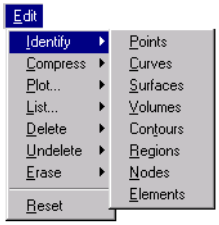
The Edit menu contains commands to identify, compress, plot, list, delete, undelete, and erase geometric entities, and nodes and elements. The Reset command resets default settings.

Figure 3-1 Edit Menu

▶ IDENTIFY Menu

Edit > IDENTIFY

Figure 3-2. Identify Menu



PTIDENT

Edit > IDENTIFY > Points

The PTIDENT identifies a keypoint by highlighting its label and displaying its label and coordinates in the dialogue area.

Notes

1. Only entities plotted on the screen can be selected by the mouse.
2. Pressing the right button of the mouse keeps selecting keypoints in the neighborhood.

CRIDENT

Edit > IDENTIFY > Curves

The CRIDENT identifies a curve by highlighting it and displaying its label in dialogue area.

(Refer to the PTIDENT command for syntax and details.)

SFIDENT

Edit > IDENTIFY > Surfaces

The SFIDENT identifies a surface by highlighting its label and also displaying its label in dialogue area.

(Refer to the PTIDENT command for syntax and details.)

VLIDENT

Edit > IDENTIFY > Volumes

The VLIDENT command identifies a volume by highlighting it and displaying its label in the dialogue area.

(Refer to the PTIDENT command for syntax and details.)

CTIDENT

Edit > IDENTIFY > Contours

The CTIDENT command identifies a contour by highlighting its member curves and displaying its label in the dialogue area.

(Refer to the PTIDENT command for syntax and details.)

RGIDENT

Edit > IDENTIFY > Regions

The RGIDENT command identifies a region by highlighting its boundaries and displaying its label in the dialogue area.

(Refer to the PTIDENT command for syntax and details.)

NIDENT

Edit > IDENTIFY > Nodes

The NIDENT command identifies the label of the node, pointed to by the mouse, and displays its label and coordinates in the dialogue area.

Note

Point the mouse to the entity of interest, and then press the left button of the mouse to display information about the selected entity. If necessary, press the right button of the mouse repeatedly to display information about the entities in the neighborhood of the selected one.

Example: `NIDENT`

Let the mouse be pointed to node number 5. If the NIDENT is issued, node number and coordinates are displayed in the dialogue area.

EIDENT

Edit > IDENTIFY > Elements

The EIDENT command identifies the label of the element, pointed to by the mouse, and displays it in the dialogue area.

Note

Point the mouse to the entity of interest, and then press the left button of the mouse to display information about the selected entity. If necessary, press the right button of the mouse repeatedly to display information about the entities in the neighborhood of the selected one.



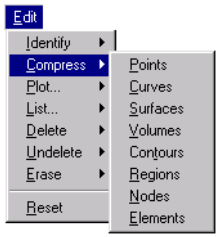

Example: `EIDENT`

Let element number 5 be picked by the mouse. If the EIDENT command is issued, the element is highlighted and its label is displayed in the dialog area.

▶ COMPRESS Menu

Edit > COMPRESS

Figure 3-3. Compress Menu



PTCOMPRESS

Edit > COMPRESS > Points

The PTCOMPRESS command renumbers the keypoints in the specified pattern by removing all numbering gaps.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.
(default is 1)

Ending keypoint
Ending keypoint in the pattern.
(default is the highest keypoint number defined)

Example: `PTCOMPRESS, 1, 100`

This command renumbers all keypoints numbered 1 to 100, so that no gaps exist. If defined keypoints were numbered 11 to 50 and 76 to 100, this command renumbers keypoints so that keypoint 11 is renumbered as 1, and keypoint 100 is renumbered as 65.

CRCOMPRESS

Edit > COMPRESS > Curves

The CRCOMPRESS command renumbers the curves in the pattern by removing all numbering gaps.

(Refer to the PTCOMPRESS command for syntax and details.)

SFCOMPRESS

Edit > COMPRESS > Surfaces

The SFCOMPRESS command renumbers the surfaces in the pattern by removing all numbering gaps.

(Refer to the PTCOMPRESS command for syntax and details.)

VLCOMPRESS

Edit > COMPRESS > Volumes

The VLCOMPRESS command renumbers the volumes in a pattern by removing all numbering gaps.

(Refer to the PTCOMPRESS command for syntax and details.)

CTCOMPRESS

Edit > COMPRESS > Contours

The CTCOMPRESS command renumbers the contours in the pattern by removing all numbering gaps.

(Refer to the PTCOMPRESS command for syntax and details.)

RGCOMPRESS

Edit > COMPRESS > Regions

The RGCOMPRESS command renumbers all regions in the pattern by removing all numbering gaps.

(Refer to the PTCOMPRESS command for syntax and details.)

NCOMPRESS

Edit > COMPRESS > Nodes

The NCOMPRESS command renumbers the nodes in the specified range continuously without any gaps.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is the highest node number defined)

Note
Nodes can be selected by the mouse.

Example: `NCOMPRESS, 1, 64`




Let there be node numbers 1 to 21 and 30 to 40 and 51 to 64 in the database. This command renumbers these sequences as 1 to 46 without leaving any gaps in between.

ECOMPRESS

Edit > COMPRESS > Elements

The ECOMPRESS command renumbers the elements in the specified range by removing all numbering gaps.

Entry & Option Description

Beginning element
Beginning element of the pattern.
(default is 1)

Ending element
Ending element of the pattern.
(default is the maximum of elements defined)

Example: `ECOMPRESS, 1, 64`

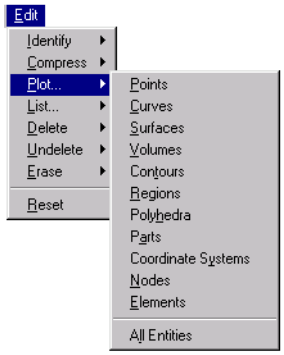
Let there be element numbers 1 to 21, 30 to 40 and 51 to 64 in the database. This command renumbers these sequences as 1 to 46 without leaving any gaps in between.




▶ PLOT Menu

Edit > PLOT

Figure 3-4. Plot Menu



PTPLOT

Edit > PLOT > Points

The PTPLOT command plots a pattern of keypoints on the screen.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.
(default is 1)

Ending keypoint
Ending keypoint in the pattern.
(default is the highest keypoint defined)

Increment
Increment between keypoints in the pattern.
(default is 1)

Notes

1. Using command STATUS1, flag for displaying keypoint labels can be toggled ON or OFF.
2. Keypoints can be picked by the mouse.

Example: `PTPLOT, 1, 20, 1`
The above command plots keypoints 1 through 20 on the screen.

CRPLOT

Edit > PLOT > Curves

The CRPLOT command plots a pattern of curves on the screen.

(Refer to the PTPLOT command for syntax and details.)

SFPLOT

Edit > PLOT > Surfaces

The SFPLOT command plots a pattern of surfaces on the screen.

(Refer to the PTPLOT command for syntax and details.)

VLPLOT

Edit > PLOT > Volumes

The VLPLOT command plots a pattern of volumes on the screen.

(Refer to the PTPLOT command for syntax and details.)

CTPLOT

Edit > PLOT > Contours

The CTPLOT command plots a pattern of contours on the screen.

(Refer to the PTPLOT command for syntax and details.)

RGPLOT

Edit > PLOT > Regions

The RGPLOT command plots a pattern of regions on the screen.

(Refer to the PTPLOT command for syntax and details.)

PHPLOT

Edit > PLOT > Polyhedra

The PHPLOT command plots a pattern of polyhedrons on the screen.

Entry and Option Description

Beginning polyhedron
Beginning polyhedron in the pattern.
(default is 1)

Ending polyhedron
Ending polyhedron in the pattern.
(default is highest polyhedron number defined)

Increment
Increment between polyhedrons in the pattern.
(default is 1)

Example: `PHPLOT, 1, 3, 1`
This command plots polyhedrons 1, 2 and 3 on the screen.

PARTPLOT

Edit > PLOT > Parts

The PARTPLOT command plots a pattern of parts on the screen.

Entry and Option Description

Beginning part
Beginning part in the pattern.
(default is 1)

Ending part
Ending part in the pattern.
(default is highest part number defined)

Increment
Increment between parts in the pattern.
(default is 1)

Example: `PARTPLOT, 1, 3, 1`
This command plots parts 1, 2 and 3 on the screen.

CSPLOT

Edit > PLOT > Coord Sys

The CSPLOT command plots a pattern of coordinate systems on the screen.

Entry & Option Description

Beginning coordinate system
Beginning coordinate system label in the pattern.

Ending coordinate system
Ending coordinate system label in the pattern.
(default is the maximum coordinate system number defined)

Increment
Increment between coordinate systems in the pattern.
(default is 1)

Note
Using the STATUS1 command, flag for displaying coordinate system labels can be toggled to ON or OFF.

Example: `CSPLOT, 3, 6, 1`
The above command plots coordinate systems 3 to 6 on the screen.




NPLOT

Edit > PLOT > Nodes

The NPLOT command plots the nodes specified in the pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node of the pattern.
(default is the maximum of nodes defined)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `NPLOT, 1, 5, 1`
This command plots nodes 1 through 5.

EPLOT

Edit > PLOT > Elements

The EPLOT command plots the elements specified in the pattern.

Entry & Option Description

Beginning element
Beginning element of the pattern.
(default is 1)

Ending element
Ending element of the pattern.
(default is the maximum of elements defined)

Increment
Increment between the elements in the pattern.
(default is 1)

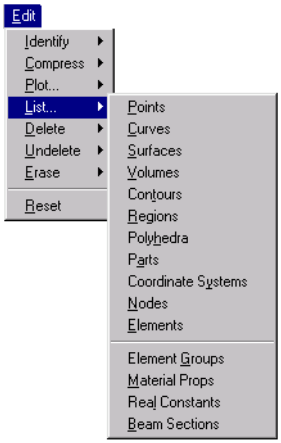
Example: `EPLOT, 1, 5, 1`
This command plots elements 1 through 5.




▶ LIST Menu

Edit > LIST

Figure 3-5. List Menu



PTLIST

Edit > LIST > Points

The PTLIST command lists labels and coordinates of all the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.
(default is 1)

Ending keypoint
Ending keypoint in the pattern.
(default is the highest keypoint defined)

Increment
Increment between keypoints in the pattern.
(default is 1)

Coordinate System
Coordinate system label to be used in listing.
(default is 0)

Example: `PTLIST, 10, 20, 2, 0`
This command lists labels and coordinates in the global coordinate system for keypoints 10, 12, 14, 16, 18 and 20.

CRLIST

Edit > LIST > Curves

The CRLIST command lists the keypoints of all the curves in the specified pattern.

(Refer to the PTLIST command for syntax and details.)

SFLIST

Edit > LIST > Surfaces

The SFLIST command lists the keypoints and boundary curves associated with the surfaces in the pattern.

(Refer to the PTLIST command for syntax and details.)

VLLIST

Edit > LIST > Volumes

The VLLIST command lists the keypoints, boundary curves and surfaces associated with the volumes in the pattern.

(Refer to the PTLIST command for syntax and details.)

CTLIST

Edit > LIST > Contours

The CTLIST command lists a pattern of contours by displaying their numbers, number of curves, element size and number of elements on curves associated with them.

(Refer to the PTLIST command for syntax and details.)

RGLIST

Edit > LIST > Regions

The RGLIST command lists a pattern of regions by displaying their number, number of contours and a list of the contours associated with them.

(Refer to the PTLIST command for syntax and details.)

PHLIST

Edit > LIST > Polyhedra

The PHLIST command lists a pattern of polyhedrons by displaying their labels, associated surfaces, regions and element size.

Entry and Option Description

Beginning polyhedron
Beginning polyhedron in the pattern.
(default is 1)

Ending polyhedron
Ending polyhedron in the pattern.
(default is the highest polyhedron number defined)

Increment
Increment between polyhedrons in the pattern.
(default is 1)

Example: `PHLIST, 1, 3, 1`
This command lists polyhedrons 1, 2 and 3 on the screen.




PARTLIST

Geo Panel: Edit > LIST > Parts

The PARTLIST command lists a pattern of parts by displaying their labels, associated polyhedrons and element size.

Entry and Option Description

Beginning part
Beginning part in the pattern.
(default is 1)

Ending part
Ending part in the pattern.
(default is the highest part number defined)

Increment
Increment between parts in the pattern.
(default is 1)

Example: `PARTLIST, 1, 3, 1`
This command lists parts 1, 2 and 3 on the screen.

CSLIST

Edit > LIST > Coordinate Systems

The CSLIST command lists a pattern of coordinate systems.

Entry & Option Description

Beginning coordinate system
Beginning coordinate system label in the pattern.

Ending coordinate system
Ending coordinate system label in the pattern.
(default is the maximum coordinate system number defined)

Increment
Increment between coordinate systems in the pattern.
(default is 1)

Example: `CSLIST, 3, 5, 1`
This command lists coordinate systems 3 to 5.

NLIST

Edit > LIST > Nodes

The NLIST command lists the nodal coordinates of the nodes specified in the pattern.




Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is the highest node number defined)

Increment
Increment between the nodes in the pattern.
(default is 1)

Coordinate system
Coordinate system to be used in listing.
(default is 0)

Example: `NLIST, 1, 5, 1, 0`

This command lists the coordinates of nodes 1 through 5 in the global coordinate system.

ELIST

Edit > LIST > Elements

The ELIST command lists the nodal connectivity and attributes of the elements in the specified pattern.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is the highest element label defined)

Increment
Increment between the elements in the pattern.
(default is 1)

Example: `ELIST, 1, 5, 1`

This command lists elements 1 through 5.

EGLIST

Edit > LIST > Element Groups

The EGLIST command lists a pattern of element groups.

Entry & Option Description

Beginning element group
Beginning element group to be listed.
(default is 1)

Ending element group
Ending element group to be listed.
(default is the highest element group number defined)

Increment
Increment between element groups.
(default is 1)

Example: `EGLIST, 1, 4, 1`
The above command lists element groups 1 through 4.

MPLIST

Edit > LIST > Material Props

The MPLIST command lists a pattern of material property sets.

Entry & Option Description

Beginning material property set
Beginning set label in the pattern.
(default is 1)

Ending material property set
Ending set label in the pattern.
(default is the highest material set defined)

Increment
Set label increment in the pattern.
(default is 1)

Example: `MPLIST, 1, 4, 1`
The above command lists material property sets 1 through 4.

RCLIST

Edit > LIST > Real Constants

The RCLIST command lists a pattern of real constant sets.

Entry & Option Description

Beginning real constant set
Beginning real constant set in the pattern.
(default is 1)

Ending real constant set
Ending real constant set in the pattern.
(default is the highest real constant set defined)

Increment
Increment between real constant sets in the pattern.
(default is 1)

Example: `RCLIST, 1, 4, 1`
The above command lists real constant sets 1 through 4.

BMSECLIST

Edit > LIST > Beam Sections

The BMSECLIST command lists a pattern of beam sections defined previously by the BMSECDEF command.

Entry & Option Description

Beginning real constant set
Beginning real constant set in the pattern.
(default is 1)

Ending real constant set
Ending real constant set in the pattern.
(default is the highest real constant set defined)

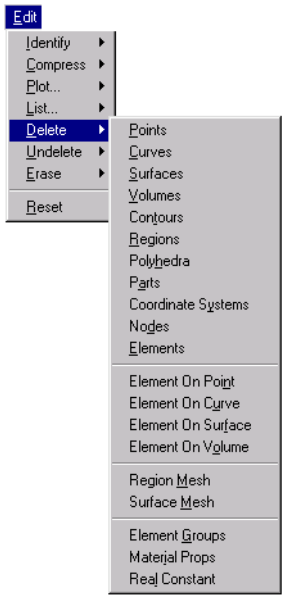
Increment
Increment between real constant sets in the pattern.
(default is 1)




▶ DELETE Menu

Edit > DELETE

Figure 3-6. Delete Menu



PTDEL

Edit > DELETE > Points

The PTDEL command deletes all the keypoints in the specified pattern from the database. The deleted keypoints can be undeleted by using the PTUNDEL command.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Note

This command deletes keypoint addresses temporarily from the database so that it is possible to undelete them using PTUNDEL command.

Example: `PTDEL, 10, 20, 2`

This command deletes keypoints 10, 12, 14, 16, 18 and 20 from the database.

CRDEL

Edit > DELETE > Curves

The CRDEL command deletes a pattern of curves from the database.

(Refer to the PTDEL command for syntax and details.)

SFDEL

Edit > DELETE > Surfaces

The SFDEL command deletes a pattern of surfaces from the database.

(Refer to the PTDEL command for syntax and details.)

VLDEL

Edit > DELETE > Volumes

The VLDEL command deletes a pattern of volumes from the database.

(Refer to the PTDEL command for syntax and details.)

CTDEL

Edit > DELETE > Contours

The CTDEL command deletes a pattern of contours from the database.

(Refer to the PTDEL command for syntax and details.)

RGDEL

Edit > DELETE > Regions

The RGDEL command deletes a pattern of regions from the database.

(Refer to the PTDEL command for syntax and details.)

PHDEL

Edit > DELETE > Polyhedra

The PHDEL command deletes a pattern of polyhedrons from the database. Associated lower entities are not deleted.

Entry and Option Description

Beginning polyhedron
Beginning polyhedron in the pattern.

Ending polyhedron
Ending polyhedron in the pattern.
(default is beginning polyhedra)

Increment
Increment between polyhedrons in the pattern.
(default is 1)

Example: `PHDEL, 1, 3, 1`
This command deletes polyhedrons 1, 2 and 3 from the database.

PARTDEL

Edit > DELETE > Parts

The PARTDEL command deletes a pattern of parts from the database. Associated lower entities are not deleted.

Entry and Option Description

Beginning part
Beginning part in the pattern.

Ending part
Ending part in the pattern.
(default is beginning part)

Increment
Increment between parts in the pattern.
(default is 1)

Example: `PARTDEL, 1, 3, 1`
This command deletes parts 1, 2 and 3 from the database.

CSDEL

Edit > DELETE > Coordinate Systems

The CSDEL command deletes a pattern of coordinate systems from the database.

Entry & Option Description

Beginning coordinate system
Beginning coordinate system label in the pattern.

Ending coordinate system
Ending coordinate system label in the pattern.
(default is beginning coordinate system)

Increment
Increment between coordinate systems in the pattern.
(default is 1)

Example: `CSDEL, 10, 20, 2`
This command deletes coordinate systems 10, 12, 14, 16, 18 and 20.

NDELETE

Edit > DELETE > Nodes

The NDELETE command deletes all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `NDELETE, 1, 5, 1`
This command deletes nodes 1 through 5 from the database.

EDELETE

Edit > DELETE > Elements

The EDELETE command deletes all elements in the specified pattern.

Entry & Option Description

Beginning element
Beginning element of the pattern.

Ending element
Ending element of the pattern.
(default is beginning element)

Increment
Increment between the elements in the pattern.
(default is 1)

Example: `EDELETE, 1, 5, 1`
This command deletes elements 1 through 5 from the database.

MPTDEL

Edit > DELETE > Element on Point

The MPTDEL command deletes elements and the corresponding nodes associated with any of the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.
(default is 1)

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between the keypoints in the pattern.
(default is 1)

Example: `MPTDEL, 1, 5, 1`

This command deletes nodes and associated 1-node elements at keypoints 1 through 5.

MCRDEL

Edit > DELETE > Element on Curve

The MCRDEL command deletes elements and the corresponding nodes associated with any of the curves in the specified pattern.

(Refer to the MPTDEL command for syntax and details.)

MSFDEL

Edit > DELETE > Element on Surface

The MSFDEL command deletes elements and the corresponding nodes associated with any of the surfaces in the specified pattern.

(Refer to the MPTDEL command for syntax and details.)

MVLDEL

Edit > DELETE > Element on Volume

The MVLDEL command deletes elements and the corresponding nodes associated with any of the volumes in the specified pattern.

(Refer to the MPTDEL command for syntax and details.)

MARGDEL

Edit > DELETE > Region Mesh

The MARGDEL command deletes the meshing of all regions in the specified pattern.

Entry & Option Description

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Example: `MARGDEL, 1, 5, 3`
This command deletes the finite element mesh in regions 1 and 4.

MASFDEL

Edit > DELETE > Surface Mesh

The MASFDEL command deletes the meshing of all surfaces in the specified pattern meshed by any of the MA_SF or the M_SF commands.

Entry & Option Description

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Note

The surfaces can be picked by the mouse.

Example: `MSFDEL, 1, 5, 3`
This command deletes the meshing of surfaces 1 and 4.

EGDEL

Edit > DELETE > Element Groups

The EGDEL command deletes a pattern of element group sets from the database.

Entry & Option Description

Beginning element group
Beginning element group set in the pattern.

Ending element group
Ending element group set in the pattern.
(default is beginning element group)

Increment
Increment between element group sets.
(default is 1)

Example: `EGDEL, 1, 5, 2`
This command deletes element groups 1, 3 and 5.

MPDEL

Edit > DELETE > Material Props

The MPDEL command deletes a pattern of material property sets.

Entry & Option Description

Beginning material property set
Beginning material property set in the pattern.

Ending material property set
Ending material property set in the pattern.
(default is beginning material property set)

Increment
Increment between material property sets.
(default is 1)

Example: `MPDEL, 1, 5, 2`
This command deletes material property sets number 1, 3 and 5.

RCDEL

Edit > DELETE > Real Constants

The RCDEL command deletes a pattern of real constant sets from the database.

Entry & Option Description

Beginning real constant set
Beginning real constant set in the pattern.

Ending real constant set
Ending real constant set in the pattern.
(default is beginning real constant set)

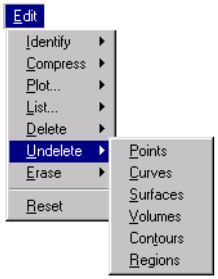
Increment
Increment between real constant sets.
(default is 1)

Example: `RCDEL, 1, 5, 2`
This command deletes real constant sets 1, 3 and 5.

▶ UNDELETE Menu

Edit > UNDELETE

Figure 3-7 Undelete Menu



PTUNDEL

Edit > UNDELETE > Points

The PTUNDEL command undeletes all keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Note
It is recommended that the PTUNDEL command be used immediately after an accidental deletion of keypoints and before the generation of any new geometric entities.

Example: `PTUNDEL, 10, 20, 2`
This command undeletes keypoints 10, 12, 14, 16, 18 and 20.

CRUNDEL

Edit > UNDELETE > Curves

The CRUNDEL command undeletes all curves in the specified pattern.

(Refer to the PTUNDEL command for syntax and details.)

SFUNDEL

Edit > UNDELETE > Surfaces

The SFUNDEL command undeletes a pattern of surfaces.

(Refer to the PTUNDEL command for syntax and details.)

VLUNDEL

Edit > UNDELETE > Volumes

The VLUNDEL command undeletes a pattern of volumes.

(Refer to the PTUNDEL command for syntax and details.)

CTUNDEL

Edit > UNDELETE > Contours

The CTUNDEL command undeletes a pattern of contours.

(Refer to the PTUNDEL command for syntax and details.)

RGUNDEL

Edit > UNDELETE > Regions

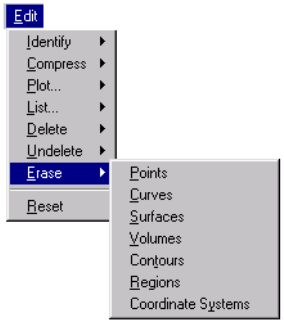
The RGUNDEL command undeletes a pattern of regions.

(Refer to the PTUNDEL command for syntax and details.)

▶ ERASE Menu

Edit > ERASE

Figure 3-8 Erase Menu



PTERASE

Edit > ERASE > Points

The PTERASE command erases keypoints from the screen. The erased keypoints may be replotted using the PTPLOT command.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `PTERASE, 10, 20, 2`

This command erases keypoints 10, 12, 14, 16, 18 and 20 from the screen.

CRERASE

Edit > ERASE > Curves

The CRERASE command erases a pattern of curves from the screen. The erased curves may be replotted using the CRPLOT command.

(Refer to the PTERASE command for syntax and details.)

SFERASE

Edit > ERASE > Surfaces

The SFERASE command erases surfaces from the screen. The erased surfaces may be replotted using the SFPLOT command.

(Refer to the PTERASE command for syntax and details.)

VLERASE

Edit > ERASE > Volumes

The VLERASE command erases volumes from the screen. They may be replotted using the VLPLOT command.

(Refer to the PTERASE command for syntax and details.)

CTERASE

Edit > ERASE > Contours

The CTERASE command erases a pattern of contours from the screen. The erased contours may be replotted by using the CTPLOT command.

(Refer to the PTERASE command for syntax and details.)

RGERASE

Edit > ERASE > Regions

The RGERASE command erases regions from the screen. The erased regions may be replotted using the RGPLOT command.

(Refer to the PTERASE command for syntax and details.)

CSERASE

Edit > ERASE > Coordinate Systems

The CSERASE command erases coordinate system plots from the screen. They may be replotted using the CSPLOT command.

Entry & Option Description

Beginning coordinate system
Beginning coordinate system label in the pattern.

Ending coordinate system
Ending coordinate system label in the pattern.
(default is beginning coordinate system)

Increment
Increment between coordinate systems in the pattern
(default is 1)

Example: `CSERASE, 10, 14, 2`
This command erases coordinate systems 10, 12, and 14 from the screen.

RESET

EDIT > Reset

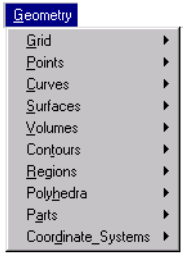
The RESET command resets the view, zoom, translation, shrink, scale, boundary and rotation parameters back to the default values.




4 Geometry Menu

GEOMETRY Menu

Figure 4-1 Geometry Menu



GEOMETRY

This main menu includes commands to generate and process geometric entities. Geometric entities provide a convenient and powerful vehicle for generating the finite element mesh and specifying loading and boundary conditions.

The geometric entities and the limit on their numbers in GEOSTAR are given below.

Entity	Symbol	Limit
Keypoints	PT	24000
Curves	CR	24000
Surfaces	SF	8000
Volumes	VL	5000
Contours	CT	5000
Regions	RG	5000
Polyhedra	PH	100
Parts	PART	100
Coordinate Systems	CS	5000

The Grid and Coordinate Systems menus are included under the GEOMETRY menu for convenience since they are important tools for creating the geometry. The hierarchies of these entities are (from high to low): VL, SF, CR, and PT or PART, PH, SF or RG, CT, CR, and PT. A higher entity must have member(s) of all lower

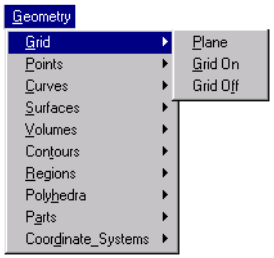
entities associated with it. Whenever an entity is generated, all associated lower entities are automatically generated by GEOSTAR.

▶ GRID Menu

Geometry > GRID

This menu contains commands to define a plane and control the grid. A grid can be created in any plane and can be conveniently used to create geometric entities. The grid is particularly useful with sketching commands.

Figure 4-2 Grid Menu



PLANE

Geometry > GRID > Plane

The PLANE command defines a plane parallel to one of the planes of the active coordinate system (must be Cartesian) by specifying the normal axis to the plane and an offset. You need to issue this command before defining a grid. To create a grid on the plane, choose **Geometry, Grid, Grid On**.

Axis Normal to Plane
The axis normal to the plane. Choose X, Y or Z of the active coordinate system..
(default is Z)

Offset on axis
The magnitude of the offset on the axis to locate the position of the plane on the axis.
(default is 0)

Grid line style
The style of grid lines.

= 0: Blank	blank, do not plot the grid
= 1: Solid	solid grid lines
= 2: Broken	broken grid lines
	(default is solid)

To define a plane:

1. Activate (or create) the desirecd coordinate system if needed.
2. Click **Geometry, Grid, Plane**.
3. From the **Axis Normal to Plane** drop down menu, select the desired axis.
4. In the **Offset on Axis**, type in the desired value. Use (-) to define the plane on the negative side of the axis.
5. From the **Grid Line Style** drop down menu, select the desired option.
6. Click **OK**.

GRIDON

Geometry > GRID > Grid on

The GRIDON command plots a grid on the active plane defined by the Geometry, Grid, PLANE command. The style of the grid lines is based on options specified by the (**Geometry, Grid, PLANE**) command. Axisymmetric models must be created in the global X-Y plane with Y as the axis of symmetry.

Coordinate of origin of 1st axis
The coordinate value for the origin the first axis.
(default is 0)

Coordinate of origin of 2nd axis
The coordinate value for the origin of the second axis.
(default is 0)

Increment along the 1st axis
Spacing between the grid lines along the first axis.
(default is 5.0)

Increment along the 2nd axis
Spacing between the grid lines along the second axis.
(default is 5.0)

Number of increments along the 1st axis
Number of grid line spacings along the first axis.
(default is 20)

Number of increments along the 2nd axis
Number of grid line spacings along the second axis.
(default is 20)

Grid line color index
Color for the grid lines.
(default is 2)

Note

In the course of developing your model, you define as many planes and grids as needed.

To create a grid:

1. Click **Geometry, Grid, Plane** to define a plane.
2. Click **Geometry, Grid, Grid On**.
3. Specify the desired value to define the origin, spacing, and range of the grid.

Example

```
GRIDON,0.,0.,6.0,5.0,20.0,20.0,3
```

This command creates a grid on the active plane. The grid extends from 0.0 to 120.0 along the first axis and 0.0 to 100.0 along the second

axis with the grid spacing of 6.0 and 5.0 along the first and second axes respectively. The grid line color is 3.

GRIDOFF

Geometry > GRID > Grid off

The GRIDOFF command erases and turns off the active grid.

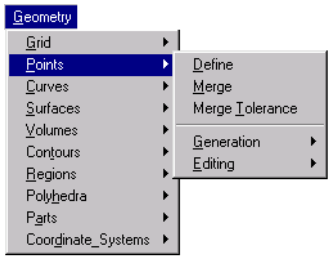
▶ POINTS Menu

Geometry > POINTS

A keypoints is the lowest entity type in geometry. This menu includescommands to generate and process keypoints.

The short names of the commands in this menu start with “PT”. The rest of the characters describe the function of the comand. Keypoints can be meshed to generate1-node elements like MASS and 1-node GAP and SPRING elements.

Figure 4-3 Points Menu



PT

Geometry > POINTS > Define

The PT command creates a keypoint by defining its coordinates in the active coordinate system, or conveniently by snapping onto a predefined grid using the mouse.

Keypoint
Keypoint number.
(default is the highest keypoint number defined + 1)

XYZ-Coordinate value
Keypoint coordinates in the currently active coordinate system.
(defaults are 0.0)

Note

Keypoints are defined in the active coordinate system which can be Cartesian, cylindrical, or spherical.

To snap keypoints on a grid:

1. Click **Geometry, Grid, Plane** to define a plane.
2. Click **Geometry, Grid, Grid On** to create a grid.
3. Click **Geometry, Points, Define** or type in **PT** in the GeoStar Console window.
4. In the **Keypoint** field, accept the default keypoint label or enter a new label.

5. Move the mouse pointer to the desired location on the grid and click the left button. Notice that the coordinates of the point are shown as you move the mouse pointer. You can create a keypoint at locations other than the intersection of the grid lines by typing in the desired coordinates.

Example

```
PT, 1, 1.0, 2.0, 3.0
```

This command defines keypoint 1 at the location defined by x-coordinate = 1.0; y-coordinate = 2.0; z-coordinate = 3.0 in the active coordinate system.

PTMERGE

Geometry > POINTS > Merge

The PTMERGE command merges keypoints. Two keypoints are merged if the differences between their corresponding X, Y, and Z-coordinates are equal to, or smaller than the tolerance specified by the PTTOL (**Geometry, Points, Merge Tolerance**) command. Merging can be performed with respect to all keypoints or with respect to the keypoints in the specified pattern only.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is the max. keypoint label defined)

Increment
Increment between keypoints in the pattern.
(default is 1)

Tolerance
Magnitude of tolerance. Keypoints are merged only if all of their three components are within the specified tolerance.
(default is .0001)

All/Among flag
Merging flag.
= 0: All merge with respect to all keypoints in the model
= 1: Amongmerge with respect to the keypoints specified in the pattern only
(default is all)

Echo flag
Message flag.
= 1: On GEOSTAR issues a message when two keypoints are merged
= 0: Off GEOSTAR does not issue messages
(default is on)

Low/High flag

Direction flag.
= 0: Low merge higher labels with lower labels of keypoints, such that if two keypoints are merged the keypoint with higher label is deleted
= 1: High merge lower labels with higher labels of keypoints, such that if two keypoints are merged the keypoint with lower label is deleted
(default is low)

Notes

1. Keypoints can be selected by the mouse only if they are plotted on the screen.
2. If merging with respect to the keypoints in the pattern is selected, then other keypoints (keypoints not specified in the pattern) are not checked for merging.

PTTOL

Geometry > POINTS > Merge Tolerance

The PTTOL command sets the tolerance for merging keypoints. Any two keypoints with the differences between their corresponding X, Y, and Z coordinates are equal to or less than the specified tolerance will be considered as one keypoint. This command lets you set the desired tolerance before importing CAD files. A tolerance of zero will result in keeping all keypoints even if they are concident.

Tolerance
Keypoint merging tolerance.
(default is 5.e-005)

Example

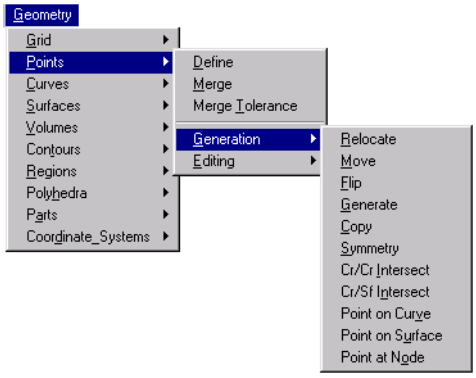
```
PTTOL, 0.0002
```

The above command specifies 0.0002 as the keypoint merging tolerance for all keypoints created after this command is issued.

▼GENERATION Menu

Geometry > POINTS > GENERATION

Figure 4-4 Generation Menu



PTRELOC

Geometry > POINTS > GENERATION > Relocate

The PTRELOC command relocates a pattern of existing keypoints by a specified translation and/or rotation in the currently active Cartesian coordinate system. A keypoint must

be free from assoiation with higher entities for a successful relocation. Relcoating higher entities (curves, surfaces, volumes, or regions) automatically relocates all associated keypoints. This command does not generate any new keypoints. The PTGEN (**Geometry, Points, Generation, Generate**) command can be used to generate new keypoints from existing ones.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

Notes

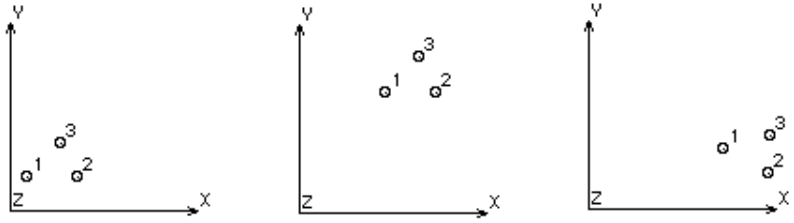
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. If a keypoint in the specified pattern is part of a higher entity, that keypoint is not relocated. The PTGEN command (**Geometry, Keypoints, Generation, Generate**) can be used instead to generate new keypoint from existing ones.

Example

Graphic Example: PTRELOC

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,5,10,0,
PT,2,20,10,0,
PT,3,15,20,0,
PTRELOC,1,3,1,0,20,25,0,
```

```
PTRELOC,1,3,1,1,0,0,-30,
```



Original | After the First PTRELOC Command | After the Second PTRELOC Command

PTMOVE

Geometry > POINTS > GENERATION > Move

The PTMOVE command moves a pattern of keypoints from the current coordinate system to the destination coordinate system. A keypoint must be free from assoiation with higher entities for a successful moving. Moving higher entities (curves, surfaces, volumes, or regions) automatically moves all associated keypoints. This command does not generate any new keypoints. The PTCOPY (**Geometry, Points, Generation, Copy**) command can be used to generate new keypoints from existing ones.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Destination coordinate
Destination coordinate system label.
(default is 0)

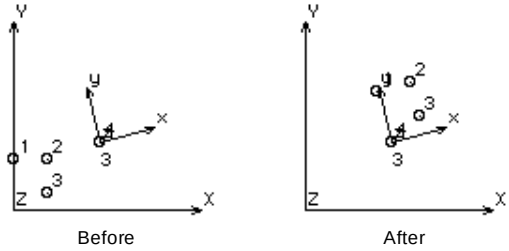
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. If a keypoint in the specified pattern is part of a higher entity, that keypoint is not moved.

Example

Graphic Example: PTMOVE

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,0,15,0,
PT,2,10,15,0,
PT,3,10,5,0,
PT,4,25,20,0,
CSANGL,3,0,25,20,0,0
  ,0,15,
ACTSET,CS,0,
PTMOVE,1,3,1,3,
```



PTFLIP

Geometry > POINTS > GENERATION > Flip

The PTFLIP command flips a pattern of keypoints about the specified plane in the current coordinate system. An offset can be specified with the flipping. A keypoint must be free from assoiation with higher entities for a successful flipping. Flipping higher entities (curves, surfaces, volumes, or regions) automatically flips all associated keypoints. This command does not generate any new keypoints. The PTSYM (**Geometry, Points, Generation, Symmetry**) command can be used to generate new keypoints from existing ones.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Rotation/Sweep axis
Axis representing normal to plane about which the pattern of keypoints is to be flipped.
= X y-z plane
= Y z-x plane
= Z x-y plane
(default is Z)

Offset
Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

Notes

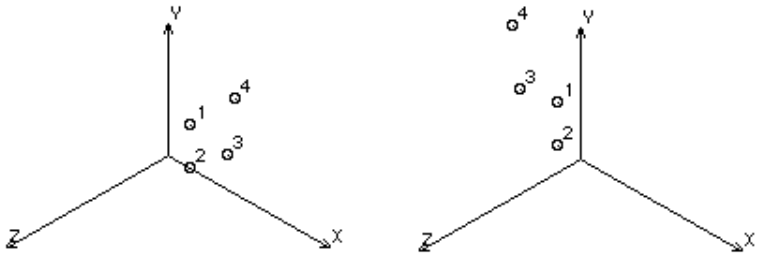
1. The Entities can be picked by the mouse only if they are plotted on the screen.

2. If a keypoint in the specified pattern is part of a higher entity, that keypoint is not flipped.
3. The specified offset is applied to all keypoints in the pattern.

Example

Graphic Example: PTFLIP

```
PLANE,Z,0,1,
PT,1,15,25,0,
PT,2,15,0,0,
PT,3,40,20,0,
PT,4,45,55,0,
PTFLIP,1,4,1,X,0,
```



PTGEN

Geometry > POINTS > GENERATION > Generate

The PTGEN command generates one or more patterns of keypoints from an existing pattern, by translating and/or rotating about the currently active Cartesian coordinate system. The PTRELOC (**Geometry, Points, Generation, Relocate**) command can be used to relocate existing keypoints without generating new ones.

Generation number
Number of patterns to be generated. (must be > 0 for generation to occur)
(default is 1)

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

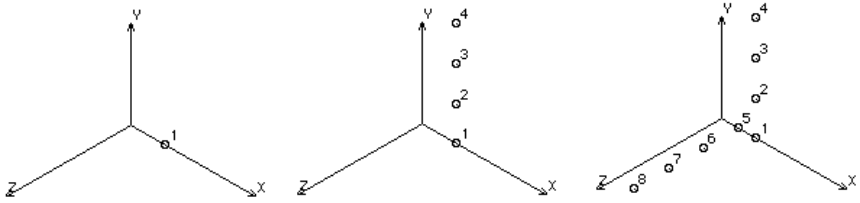
X, Y, Z-Displacement
Magnitudes of the translation in the currently active coordinate system.
(defaults are 0.0)

X, Y, Z-Rotation
Magnitudes of the rotations (degrees) in the currently active coordinate system.
(defaults are 0.0)

Example

Graphic Example: PTGEN

```
PT,1,30,0,0,
PTGEN,3,1,1,1,0,0,30.,0,
PTGEN,1,1,4,1,2,-15,0,0,90,0,0,
```



PTCOPY

Geometry > POINTS > GENERATION > Copy

The PTCOPY command copies a pattern of keypoints from the active coordinate system to the destination coordinate system. Use the PTMOVE (**Geomtery, Points, Generation, Move**) command to move existing keypoints without generating new ones.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

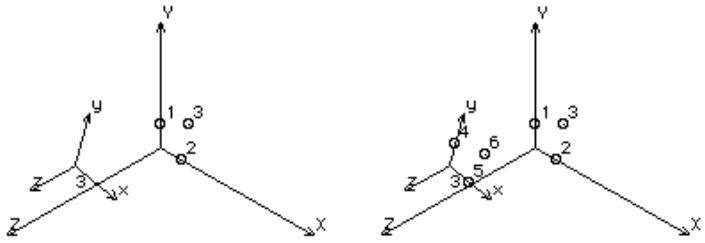
Increment
Increment between keypoints in the pattern.
(default is 1)

Destination coordinate
Destination coordinate system.
(default is 0)

Example

Graphic Example: PTCOPY

```
PLANE,Z,0,1,
PT,1,0,15,0,
PT,2,15,0,0,
PT,3,20,25,0,
CSANGL,3,0,-50,-30,10,0,0,-15,
ACTSET,CS,0,
PTCOPY,1,3,1,3,
```



Before After

PTSYM

Geometry > POINTS > ERATION MENU > Symmetry

The PTSYM command generates a pattern of keypoints by symmetry about the specified plane of the active coordinate system. An offset can also be specified. Use the PTFLIP (**Geomtery, Points, Generation, Flip**) command to flip existing keypoints without generating new ones.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Rotation/Sweep axis
Axis representing normal to the plane about which pattern of keypoints are to be generated symmetrically.
= X y-z plane

= Y x-z plane
= Z x-y plane
(default is Z)

Offset
Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

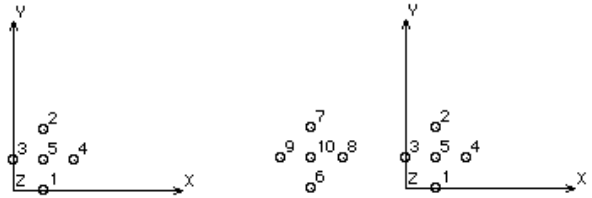
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A specified offset is applied to all keypoints of the specified pattern.

Example

Graphic Example: PTSYM

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,10,0,0,
PT,2,10,20,0,
PT,3,0,10,0,
PT,4,20,10,0,
PT,5,10,10,0,
PTSYM,1,5,1,X,-20,
```



Before After

PTINTCC

Geometry > POINTS > GENERATION > Cr/Cr Intersect

The PTINTCC command creates keypoints at the intersection of a primary curve with a pattern of specified intersecting curves. The keypoints are created on the intersecting curves if they are found within the specified tolerance from the primary curve.

Primary curve
Primary intersecting curve.

Beginning curve
Beginning curve in the pattern.

(default is 1)

Ending curve
Ending curve in the pattern.
(default is beginning curve)

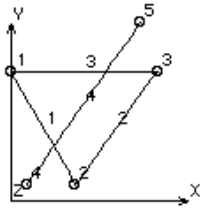
Increment
Increment between curves in the pattern.
(default is 1)

Tolerance
Convergence tolerance.
(default is 0.00005)

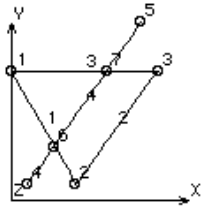
Example

Graphic Example: PTINTCC

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRPCORD,1,0,40,0,20,
  5,0,45,40,0,0,40,0,
CRPCORD,4,5,5,0,40,5
  5,0,40,55,0,
PTINTCC,4,1,3,1,
```



Before



After

PTINTCS

Geometry > POINTS > ERATION MENU > Cr/Sf Intersect

The PTINTCS command creates keypoints at the intersections of a curve with a pattern of specified surfaces.

Primary curve
Primary intersecting curve.

Beginning surface
Beginning surface in the pattern.
(default is 1)

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Tolerance
Convergence tolerance.
(default is 0.00005)

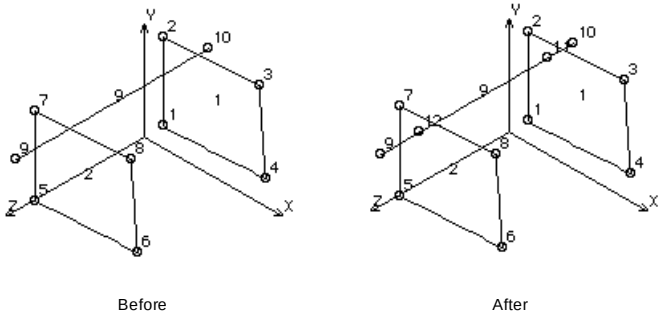
Notes

1. The curve and the beginning and ending surfaces can be picked by the mouse.
2. No keypoints are generated from the intersection of a curve and a surface lying in the same plane.

Example

Graphic Example: PTINTCS

```
PLANE,Z,0,1,
SF4CORD,1,15,15,0,15,75,0,90,80,0,95,20,0,
SFGEN,1,1,1,1,0,0,0,100,
PT,9,30,65,130,
CREXTRUDE,9,9,1,Z,-150,
PTINTCS,9,1,2,1,1
```



Before After

PTONCR

Geometry > POINTS > GENERATION > Point on Curve

The PTONCR command creates a keypoint on the specified curve. The location of the keypoint is defined in terms of the parametric coordinate of the curve.

Underlying curve
Curve on which the keypoint is to be created.

Parametric coordinate u
Parametric coordinate of keypoint on the curve (between 0.0 and 1.0).

Note

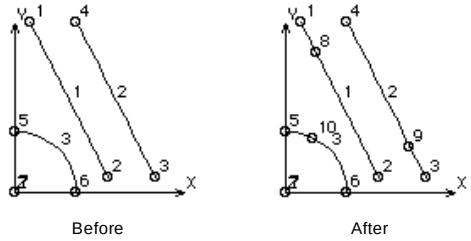
The curve can be picked by the mouse only if it is plotted on the screen.

Example

Graphic Example: PTONCR

```
PLANE,Z,0,1,
VIEW,0,0,1,
ACTMARK,CR,1
```

```
CRPCORD,1,5,55,0,30,5
  ,0,30,5,0,
CRPCORD,2,45,5,0,20,5
  5,0,20,55,0,
PT,5,0,20,0,
PT,6,20,0,0,
PT,7,0,0,0,
CRARC,3,6,5,7,20,
PTONCR,1,.2,
PTONCR,2,.2,
PTONCR,3,.7,
```



Note

The arrows define the parametric coordinates of the curves.

PTONSF

Geometry > POINTS > ERATION MENU > Point on Surface

The PTONSF command defines a keypoint on the specified surface. The location of the keypoint is defined in terms of the parametric coordinates of the surface.

Underlying surface
Surface on which the keypoint is to be defined.

Parametric coordinate u
First parametric coordinate of keypoint on the surface (between 0. and 1.).

Parametric coordinate v
Second parametric coordinate of keypoint on the surface (between 0. and 1.).

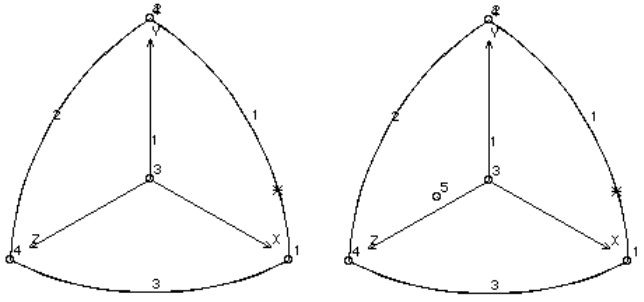
Note

The surface can be picked by the mouse only if it is plotted on the screen.

Example

Graphic Example: PTONSF

```
PT,1,60,0,0,
PT,2,0,60,0,
PT,3,0,0,0,
ACTMARK,SF,1
CRARC,1,1,2,
  3,60,
SFSWEEP,1,1,
  1,Y,-90,1,
PTONSF,1,.32
  ,.7,
```



Note

The asterisk identifies the parametric coordinates of the surfaces. The first parametric coordinate is along curve 1 starting at keypoint 1, the second is along curve 3 starting at keypoint 1.

PTND

Geometry > POINTS > ERATION MENU > Point at Node

The PTND command creates a keypoint at the location of the specified node. The keypoint is preserved.

Keypoint
Keypoint label.
(default is the highest keypoint number defined + 1)

Node
Node label.

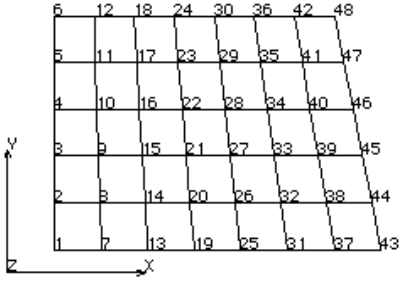
Notes

1. The node can be picked by the mouse only if it is plotted on the screen.
2. The NPTPUSH command can be used to push a node to the location of a keypoint, or alternatively use the command NMODIFY to modify the coordinates of a node. The NDPT command creates a node at the location of a keypoint.

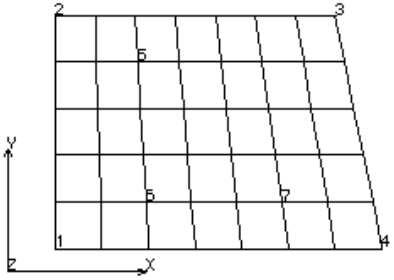
Example

Graphic Example: PTND

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,1,10,5,0,10,55,0,70,55,0,80,5,0,
M_SF,1,1,1,4,5,7,1,1,
PTND,5,17,
PTND,6,14,
PTND,7,32,
```



Before



After

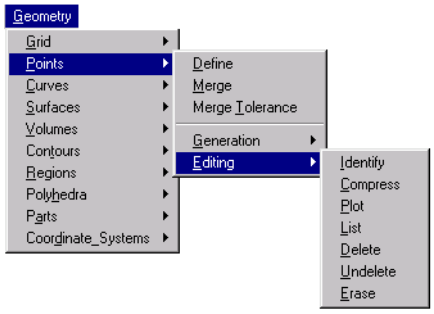
Note

Keypoints 5, 6, and 7 are created at location of node 17, 14, and 32 respectively.

▼EDITING Menu

Geometry > POINTS > EDITING

Figure 4-5 Editing Menu



PTIDENT

Geometry > POINTS > EDITING > Identify

The PTIDENT identifies a keypoint by highlighting it and displaying its label and coordinates in the dialogue area.

keypoint

Select keypoint .

Notes

1. Only entities plotted on the screen can be selected by the mouse.
2. Pressing the right button of the mouse keeps selecting keypoints in the neighborhood.

PTCOMPRESS

Geometry > POINTS > EDITING > Compress

The PTCOMPRESS command renumbers the keypoints in the specified pattern by removing all numbering gaps.

Beginning keypoint

Beginning keypoint in the pattern.
(default is 1)

Ending keypoint

Ending keypoint in the pattern.
(default is the highest keypoint number defined)

Example: `PTCOMPRESS, 1, 100`

This command renumbers all keypoints numbered 1 to 100 by removing any labeling gaps. If defined keypoints were numbered 11 to 50 and 76 to 100, this command renumbers keypoints so that keypoint 11 is renumbered as 1, keypoint 76 as 41, and keypoint 100 as 65.

PTPLOT

Geometry > POINTS > EDITING > Plot

The PTPLOT command plots a pattern of keypoints on the screen.

Beginning keypoint
Beginning keypoint in the pattern.
(default is 1)

Ending keypoint
Ending keypoint in the pattern.
(default is the highest keypoint defined)

Increment
Increment between keypoints in the pattern.
(default is 1)

Note
Use the STATUS1 table to control label plotting and color.

PTLIST

Geometry > POINTS > EDITING > List

The PTLIST command lists a pattern of keypoints.

Beginning keypoint
Beginning keypoint in the pattern.
(default is 1)

Ending keypoint
Ending keypoint in the pattern.
(default is the highest keypoint defined)

Increment
Increment between keypoints in the pattern.
(default is 1)

Coordinate System
Label of the coordinate system to be used in listing.
(default is 0)

Example: `PTLIST, 10, 20, 2`

This command lists labels and coordinates of keypoints 10, 12, 14, 16, 18 and 20 in the global coordinate system.




PTDEL

Geometry > POINTS > EDITING > Delete

The PTDEL command deletes a pattern of keypoints. The deleted keypoints can be undeleted by using the PTUNDEL (**Geometry, Undelete, Keypoints**) command. Keypoints associated with higher entities cannot be deleted.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Note
This command deletes keypoint addresses temporarily from the database so that it will be possible to undelete them using the PTUNDEL (**Edit, Delete, Points**) command.

Example: `PTDEL, 10, 20, 2`

This command deletes keypoints 10, 12, 14, 16, 18 and 20 from the database.

PTUNDEL

Geometry > POINTS > EDITING > Undelete

The PTUNDEL command undeletes all keypoints in the specified pattern.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Note
It is recommended that the PTUNDEL (**Edit, Undelete, Points**) command be used immediately after an accidental deletion of keypoints and before the generation of any new keypoints.

Example: `PTUNDEL, 10, 20, 2`

This command undeletes keypoints 10, 12, 14, 16, 18 and 20.

PTERASE

Geometry > POINTS > EDITING > Erase

The PTERASE command erases plotted keypoints. The erased keypoints can be replotted using the PTPLOT (**Edit, Plot, Points**) command.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `PTERASE, 10, 20, 2`

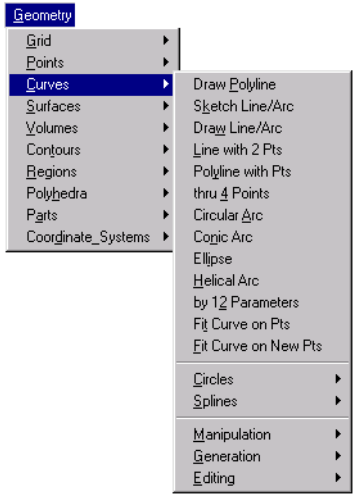
This command erases keypoints 10, 12, 14, 16, 18 and 20 from the screen.




▶ CURVES Menu

Geometry > CURVES

Figure 4-6 Curves Menu



This menu includes commands to generate, process and manipulate curves. Curves are 1D parametric entities. The cryptic names of the commands in this menu start with "CR", the rest of the characters describe the function of the command. Curves can be meshed to generate uniaxial (1D) elements like TRUSS3D, BEAM3D, PIPE and 2-node GAP elements.

CRPCORD

Geometry > CURVES > Draw Polyline

The CRPCORD command creates a series of keypoints and connects them with straight lines. Up to 20 keypoints and 19 curves (straight lines) can be created. The keypoints are specified by their coordinates, or by conveniently snapping to a pre-defined grid by the mouse. Created keypoints and curves are labeled by default. To sketch a combination of straight lines and circular curves, use the CRSPOLY command (Geometry, Curves, Draw Line/Arc) or the CRSKETCH command (Geometry, Curves, Draw Line/Arc).

Curve
Label of the first curve to be created.
(default is the highest curve number defined + 1)

Keypoint (i) XYZ-Coordinate value
X, Y, Z-coordinate of the ith keypoint created. (i=1,2,.....,20)

Note
GEOSTAR keeps prompting for more keypoints until a keypoint is selected twice in a raw, the first keypoint is selected again (closed polygon), or the limit of 20 keypoints is reached.

To Create a closed polygon:

1. Click **Geometry, Grid, Plane** to define a plane.
2. Click **Geometry, Grid, Grid On** to create the desired grid on the active plane.
3. Click **Geometry, Curves, Create Line/Arc**.
4. If desired, click in the **Curve** field and enter a label. Otherwise, click the left button of the mouse to accept the default label (recommended).
5. Move the mouse pointer to the desired location and notice that the X, Y, and Z fields of the first point display the coordinates of the mouse pointer.

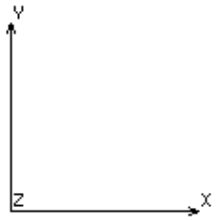
✍ Note that you can type in other values to specify locations other than the grid intersections.

6. Click the left button on the location of the first keypoint.
7. Move to the next location and click the left button.
8. Repeat steps 6 and 7 to define other sides of the polygon.
9. Move the mouse pointer to the first keypoint and click the left button to close the polygon.

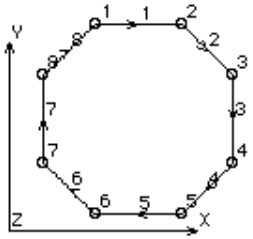
Example

Graphic Example: CRPCORD

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRCORD,1,25,60,0,50,
  60,0,65,45,0,65,20
  ,0,50,5,0,25,5,0,1
  0,20,0,10,45,0,25,
  60,0,
```



Before



After

CRSKETCH

Geometry > CURVES > Sketch Line/Arc

The CRSKETCH command sketches a series of curves in a predefined plane. After defining a reference keypoint, you can create continuous straight lines or circular arcs by specifying an exterior angle and length or radius for each curve. The polygon is created in a plane parallel to the active plane with an offset determined by the reference keypoint. After picking a grid point by the mouse, you can use the Back Space key to trace back and redefine the desired coordinates. Options to delete the last segment and to close the polygon with a straight line, normal, or tangential arc, are provided for convenience.

Curve
Curve label.
(default is the highest curve number defined + 1)

XYZ-Coordinates
Coordinates of the reference keypoint.

Line type
Type of the next curve to be created.

= L: Straight Line	create a straight line
= A: Tangential Arc	create a tangential circular arc
= N: Normal Arc	create a normal circular arc
	(default is straight line)

Angle
Exterior angle (positive if counter clockwise) measured in the right hand coordi-

nate system defined in the active plane. For creating a straight line, in refers to the angle between the direction of the last line (or tangent) and the direction of the new line. For creating an arc, it refers to the angle between the previous line (or tangent) and the line defined by connecting the start and end of the arc to be created. In other words, the circular arc end is determined by the intersection of a circle with the specified radius and the line that makes the specified angle with the previous line (or arc tangent). The direction of the created curve points towards the loose end.

Length/Radius of segment
The length of the straight line, or the radius of circular arc.

Notes

1. The command can be terminated by specifying a zero radius or length, or will be automatically terminated if the path is closed or the limit of 32 segments is reached. The "Escape" key and the "Cancel" button abort the command without creating any curves.
2. The active plane can be defined in any Cartesian coordinate system.
3. The initial angle is measured with respect to the positive x direction for the x-y or x-z planes and positive y-direction for y-z planes.

To Create an octagon of side 20:

1. Click **Geometry, Grid, Plane** to define a plane.
2. Click **Geometry, Grid, Grid On** to create the desired grid on the active plane.
3. Click **Geometry, Curves, Create Line/Arc.**
4. If desired, click in the **Curve** field and enter a label. Otherwise, click the left button of the mouse to accept the default label (recommended).
5. Move the mouse pointer to the desired location and notice that the X, Y, and Z fields of the first point display the coordinates of the mouse pointer.

Note that you can type in other values to specify locations other than the grid intersections.

6. Click the left button on the location of the first keypoint.
7. From the **Line Type** drop-down menu, select **L:Line**.
8. In the **Angle for Segment 1** field, enter **0** to start with a line parallel to the X axis of the grid.
9. In the **Length/Radius of Segment 1**, entr **20**.
10. From the **Line Type** drop-down menu (for segment 2), select **L:Line**.
11. In the **Angle for Segment 2** field, enter **45**.
12. In the **Length/Radius of Segment 2**, entr **20**.
13. Repeat steps 10, 11, and 12 to define segments 3 through 8.
14. Click **OK**. The octagon is created.

CRSPOLY

Geometry > CURVES > Draw Line/Arc

The CRSPOLY command creates a series of continuous lines and arcs in a predefined plane with an active grid. Up to 25 curves can be created at a time. The keypoints are specified by their coordinates, or conveniently snapped to a predefined grid by the mouse.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint (i) XYZ-Coordinate value
X, Y, Z-coordinate of the ith keypoint created. (i=1,2,.....,25)

Type
Defines curve type.

= L: Straight line	straight line
= A: Tangential arc	tangential circular arc
= N: Normal arc	normal circular arc
	(default is straight line)

Note
The program prompts for more keypoints until a keypoint is snapped twice, the first keypoint is snapped again (closed polygon), or the limit of 25 keypoints is reached.

To create a sketch using this command:

1. Click **Geometry, Grid, Plane** to define a plane.
2. Click **Geometry, Grid, Grid On** to create the desired grid on the active plane.
3. Click **Geometry, Curves, Draw Line/Arc.**
4. If desired, click in the **Curve** field and enter a label. Otherwise, click the left button of the mouse to accept the default label (recommended).
5. Move the mouse pointer to the desired location and notice that the X, Y, and Z fields of the first point display the coordinates of the mouse pointer.

Note that you can type in other values to specify locations other than the grid intersections.

6. Click the left button on the location of the first keypoint.
7. To change the **Line Type**, click the right button of the mouse until the desired curve type is selected.
8. Move to the new location, the curve is displayed on the screen as you move the mouse pointer.
9. Click the left button to create the displayed curve.
10. Repeat steps 7, 8, and 9 as desired.
11. Click **OK**.

CRLINE

Geometry > CURVES > Line with 2 Pts

The CRLINE command creates a straight line connecting two existing keypoints.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint 1
Start keypoint.

Keypoint 2
End keypoint.

Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The direction of the line is determined by starting from the first keypoint going towards the second.

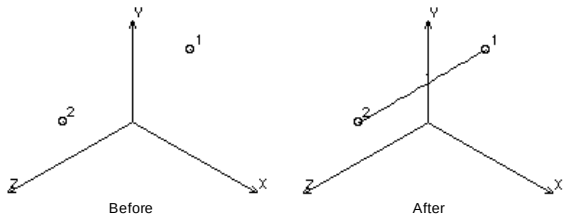
To create a line:

1. Click **Geometry, Curves, Line with 2 Pts**.
2. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default lable (recommended).
3. Select the first key point or type its label.
4. Select the second keypoint.
5. Click **OK**.

Example

Graphic Example:

```
CRLINE
PT,1,50,80,0,
PT,2,-10,20,50,
CRLINE,1,1,2,
```



CRPLINE

Geometry > CURVES > Polyline with Pts

The CRPLINE command connects a series of existing keypoints by straight lines. Up to 20 keypoints can be used to create 19 curves (straight lines). The keypoints are specified by their labels, or conveniently picked by the mouse.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint (i)

ith keypoint. (i=1,2,.....,20)

Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The program prompts for more keypoints until a keypoint is selected twice, the first keypoint is selected again (closed polygon), or the limit of 20 keypoints is r 9eached.

To use this command:

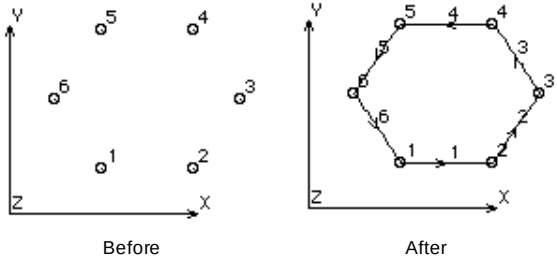
1. Click **Edit, Plot, Points** to plot the keypoints on the screen.
2. Click **Geometry, Curves, Polyline with Pts**.
3. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default lable (recommended).
4. Select the first keypoint or type in its label.
5. Continue to select or type in the labels of the other keypoints.
6. Click **OK**.

Example

Graphic Example:

CRPLINE

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,20,10,0,
PT,2,40,10,0,
PT,3,50,25,0,
PT,4,40,40,0,
PT,5,20,40,0,
PT,6,10,25,0,
CRPLINE,1,1,2,3,4,
  5,6,1,
```



CR4PT

Geometry > CURVES > thru 4 Points

The CR4PT command creates a curve by fitting a cubic polynomial through the four specified keypoints.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint 1
First keypoint.

Keypoint 2
Second keypoint.

Keypoint 3

Third keypoint.

Keypoint 4
Fourth keypoint.

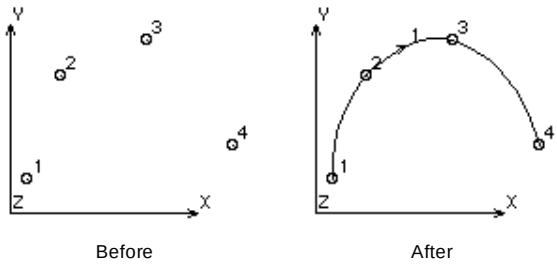
To use this command:

1. Click **Geometry, Curves, Thru 4 Pts**.
2. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default lable (recommended).
3. Select the first key point or type its label.
4. Select the other 3 keypoints.
5. Click **OK**.

Example

Graphic Example: CR4PT

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,5,10,0,
PT,2,15,40,0,
PT,3,40,50,0,
PT,4,65,20,0,
CR4PT,1,1,2,3,4,
```



CRARC

Geometry > CURVES > Circular Arc

The CRARC command creates a circular arc using three keypoints. The first two keypoints define the ends of the arc and the third keypoint is a reference point for fixing the direction of curvature. The third keypoint does not have to be located at the center but it must be defined towards the center of the arc.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint at one end
Start keypoint.

Keypoint at the other end
End keypoint.

Keypoint towards center
Keypoint towards center of curvature.

Radius
Radius of arc.
(default radius is computed based on a quarter circle between start point and end point)

Notes

1. Entities can be picked by the mouse only if the are plotted on the screen.
2. The direction of the arc is determined by starting from the first keypoint going towards the second.

To create an arc:

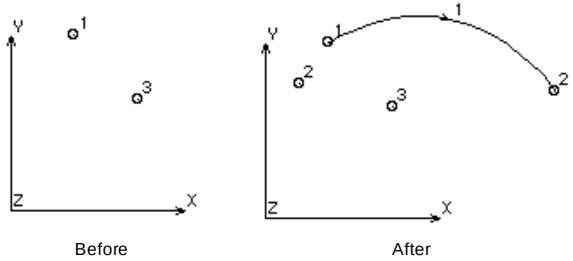
1. Click **Geometry, Curves, Circular Arc**.
2. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default lable (recommended).
3. Select a keypoint to define one end of the arc.
4. Select a keypoint to define the other end of the arc
5. Select any keypoint towards the center of the arc.
6. In the **Radius** field, enter the radius of the arc.
7. Click **OK**.

Example

Graphic Example:

CRARC

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,20,55,0,
PT,2,90,40,0,
PT,3,40,35,0,
CRARC,1,1,2,3,50.
  6211,
```



CRCONIC

Geometry > CURVES > Conic Arc

The CRCONIC command creates a conical arc. A conical arc may be parabolic, hyperbolic, or elliptic.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint at one end
Keypoint at the start of the curve.

Keypoint at the other end
Keypoint at the end of the curve.

Keypoint at intersect of tangents
Keypoint at the intersection of end tangents.

Rho
Parameter specifying type of arc. (must be between 0.0 and 1.0)
= 0.5 parabolic

< 0.5	elliptic
> 0.5	hyperbolic
	(default is 0.5)

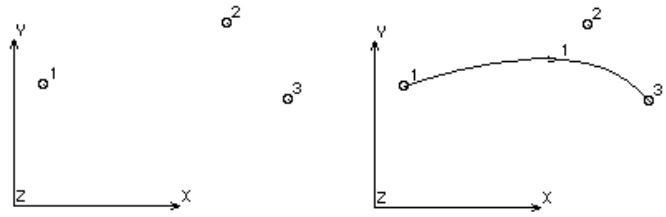
To create a parabolic curve:

1. Click **Geometry, Curves, Conic Arc**.
2. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default lable (recommended).
3. Select a keypoint to define one end of the arc.
4. Select a keypoint to define the other end of the arc
5. Select a keypoint to define the intersection of the tangents to the curve at the end points.
6. In the **RHO** field, enter **0.5**.
7. Click **OK**.

Example

Graphic Example: CRCONIC

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,10,40,0,
PT,2,70,60,0,
PT,3,90,35,0,
CRCONIC,1,1,3,2,0.5,
```



Before After

CRELLIPSE

Geometry > CURVES > Ellipse

The CRELLIPSE command creates an ellipse in space using a keypoint at one end of the major axis, a keypoint at one end of the minor axis and a keypoint at the center.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint at the end of major axis

Keypoint at one end of the major axis.

Keypoint at the end of minor axis
Keypoint at one end of the minor axis.

Keypoint at center
Keypoint at center.

Number of quadrants
Number of quadrants to be drawn. (must be between 1 and 4)
(default is 4)

To create an ellipse:

1. Click **Geometry, Curves, Ellipse**.
2. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default lable (recommended).
3. Select a keypoint to define the major axis.
4. Select a keypoint to define the minor axis
5. Select a keypoint to define the center.
6. In the **Number of Quadrants** field, enter **4**, to create a full ellipse made of 4 curves.
7. Click **OK**.

Notes

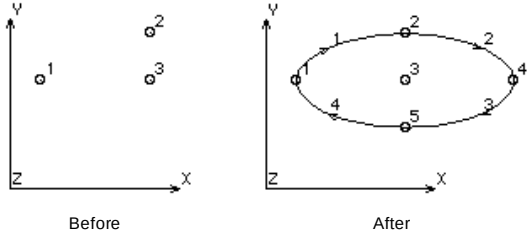
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A curve is generated for each quadrant.

Example

Graphic Example:

CRELLIPSE

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,10,35,0,
PT,2,45,50,0,
PT,3,45,35,0,
CRELLIPSE,1,1,2,3,4
  ,
```



CRHELIX

Geometry > CURVES > Helical Arc

The CRHELIX command creates a helical arc using 3 keypoints, a radius, a pitch and an angle.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint at center
Keypoint at the center.

Keypoint along axis
Keypoint to determine the axis of the helix (along with the center).

Keypoint on 0 degree line
Keypoint to determine the zero degree line of the helix.

Radius
Radius of the helix.

Pitch
Pitch of the helix, defined as the distance moved by the helix along its axis in a full turn (360 degrees).
(default is 1)

Angle of the helix
Angle of the arc in degrees.
(default is 360)

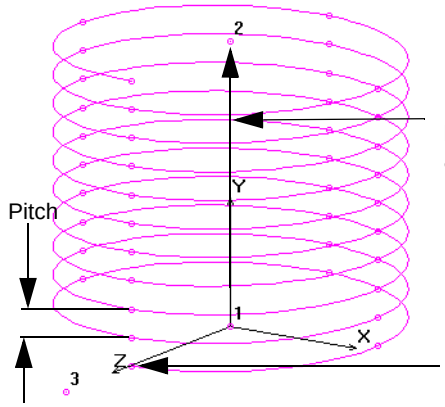
Number of segments
Number of curves. A number higher than the default can be specified. (between 1 and 360)
(default is 1 curve for each 90-degree segment)

Note
Entities can be picked by the mouse only if they are plotted on the screen.

Example

Graphic Example: CRHELIX

```
PLANE,X,0,1,
PT,1,0,0,0,
PT,2,0,100,0,
PT,3,0,0,100,
CRHELIX,1,1,2,3,60.,10.,3600.,40,
```



Line from keypoint 1 to keypoint 2 determines the axis of the helix.

The angle 3600 degrees results in 10 loops.

The starting point of the helix lies on the line connecting point 1 to point 3.

CRGFORM

Geometry > CURVES > by 12 Parameters

The CRGFORM command creates a curve by specifying its geometric form parameters. A total of 12 constants are needed to uniquely define a curve in space in this form. The curve is defined by specifying the coordinates of its end keypoints and the components of the tangents at the end keypoints.

The command is not intended for on-line use. It is designed to minimize the time needed to recreate curves. The saving is achieved by storing 12 parameters to define each curve of the final geometry. The command is internally used by the GFORM_OUT command to write the final model in a compact geometric form (neutral form), that eliminates the need to recreate intermediate geometrical entities.

Curve
New curve label.
(default is the highest curve number defined + 1)

Curve equation p(1,1)
x-coordinate of one end keypoint.

Curve equation p(1,2)
y-coordinate of one end keypoint.

Curve equation p(1,3)
z-coordinate of one end keypoint.

Curve equation p(2,1)
x-coordinate of the other end keypoint.

Curve equation p(2,2)

y-coordinate of the other end keypoint.

Curve equation p(2,3)
z-coordinate of the other end keypoint.

Curve equation p(3,1)
x-component of the tangent at the first keypoint.

Curve equation p(3,2)
y-component of the tangent at the first keypoint.

Curve equation p(3,3)
z-component of the tangent at the first keypoint.

Curve equation p(4,1)
x-component of the tangent at the second keypoint.

Curve equation p(4,2)
y-component of the tangent at the second keypoint.

Curve equation p(4,3)
z-component of the tangent at the second keypoint.

Notes

1. The GFORM_OUT command internally implements this command to store the 12 parameters needed to regenerate each curve of the final model.
2. Use of the GFORM_OUT command results in the creation of a neutral-type file containing all existing geometry in a compact ASCII form. Refer to the GFORM_OUT command for more information.

CRFIT

Geometry > CURVES > Fit Curve on Pts

The CRFIT command creates a curve by fitting a cubic polynomial to a series of existing keypoints. Up to 25 keypoints may be specified. Use the CRFITCORD (**Geometry, Curves, Fit Curve on New Pts**) command to create a series of keypoints and fit a curve through them.

Curve
Curve label.
(default is the highest curve number defined + 1)

Tolerance
Allowable tolerance to define the curve. The maximum distance allowed between the curve to be created and any keypoint.
(default is 0.001)

Keypoint (i)
ith keypoint to define the curve. (i=1,2,.....,25)

To fit a curve through a series of keypoints:

1. Click **Geometry, Curves, Fit Curve on Pts**.

2. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default lable (recommended).
3. In the **Tolerance** field, accept the default or enter the desired value.
4. Select the desired keypoints one after the other.
5. Click **OK**.

CRFITCORD

Geometry > CURVES > Fit Curve on New Pts

The CRFITCORD command creates a a series of keypoints and a curve. The curve is created by fitting a cubic polynomial through the specified locations. Up to 25 keypoints may be specified by their coordinates, or conveniently snapped to a pre-defined grid by the mouse. The keypoints must be specified in one direction along the curve. Use the CRFIT command to fit a curve through existing keypoints.

Curve
Curve label.
(default is the highest curve number defined + 1)

Tolerance
Allowable tolerance to define the curve. The maximum distance allowed between the curve to be created and any of the given keypoints.

XYZ-Coordinate of Keypoint (i)
X, Y, Z Coordinates of the ith keypoint created. (i=1,2,.....,25)

To create a series of keypoints and fit a curve through them:

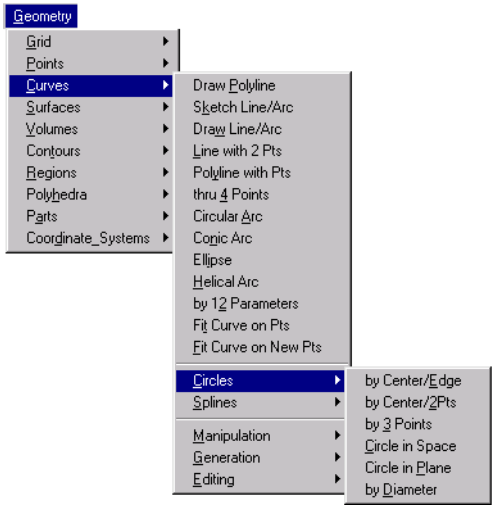
1. Click **Geometry, Grid, Plane** to define a plane.
1. Click **Geometry, Grid, Grid On** to create a grid on the defined plane.
1. Click **Geometry, Curves, Fit Curve on New Pts**.
2. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default lable (recommended).
3. In the **Tolerance** field, accept the default or enter the desired value.
4. Snap the desired locations one after the other.
5. Click **OK**.

▼CIRCLES Menu

Geometry > CURVES > CIRCLES

This section contains commands to create circular curves by various ways.

Figure 4-7 Circles Menu



CRSCIRCLE

Geometry > CURVES > CIRCLES > by Center/ Edge

The CRSCIRCLE command defines a circle in a prescribed plane by snapping the center and a point on the circumference. An active grid must exist to enable snapping, otherwise the 3 coordinate values must be specified for each point.

Curve
Curve label.
(default is the highest curve number defined + 1)

XYZ-Coordinate value of center of circle
X, Y and Z coordinates of center point.

XYZ-Coordinate value of a point on the circle
X, Y and Z coordinates of point on the circumference.

Number of segments
Number of segments of circle.
(default is 4 segments)

To create a circle by defining a centre and a point on the circumference:

1. Click **Geometry, Grid, Plane** to define a plane.
2. Click **Geometry, Grid, Grid On** to create a grid on the defined plane.
3. Click **Geometry, Curves, Circles, By Center/Edge**.
4. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default label (recommended).
5. In the **Tolerance** field, accept the default or enter the desired value.
6. Move the mouse pointer to the location of the center and click the left button.
7. Move the mouse pointer to a new location to determine the radius of the circle and click the left button. Notice the circle as you move the mouse pointer around.

8. In the **Number of Segments**, accept 4 or enter a larger number.
9. Click **OK**.

CRARCCPT

Geometry > CURVES > CIRCLES > by Center/2Pts

The CRARCCPT command creates a circular arc using two edge points and a center. The radius is defined by the distance between the center and the first keypoint. The angle of the arc is equal to the angle between the two straight lines connecting the edge points to the center. A new keypoint, lying on the straight line connecting the center to the second edge point is generated, unless the two edge points are equidistant from the center.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint at one end
First edge keypoint.

Keypoint at the other end
Second edge keypoint.

Keypoint at center
Keypoint at the center of the arc.

Notes

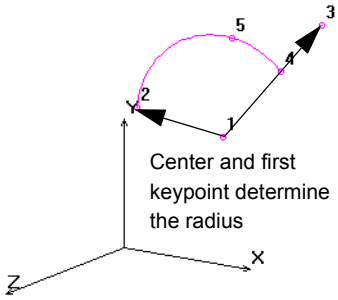
1. The Entities can be picked by the mouse only if they are plotted on the screen.
2. The circular arc will not pass through the second edge point unless the two edge points are equidistant from the center.

To create a circular arc by defining a centre and 2 other keypoints:

1. Click **Geometry, Curves, Circles, By Center/2 Pts**.
2. In the **Curve** field, enter the desired label or click the left button of the mouse to accept the default label (recommended).
3. Select the starting keypoint on the arc.
4. Select another keypoint to determone the other end of the arc. This keypoint does not have to lie on the arc.
5. Select a keypoint to be the center of the arc.



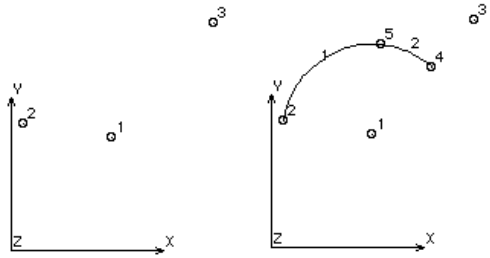

6. Click **OK**. One or more curves will be created.



Example

Graphic Example: CRARCCPT

```
PLANE,Z,0,1,
PT,1,40,45,0,
PT,2,5,50,0,
PT,3,80,90,0,
CRARCCPT,1,2,3,1,
```



CRARC3PT

Geometry > CURVES > CIRCLES > by 3 Points

The CRARC3PT command fits a circular arc through three noncolinear keypoints.

Curve
Curve label.
(default is the highest curve number defined +1)

Keypoint at one end
Keypoint at one end.

Keypoint at the other end
Keypoint at the other end.

Keypoint on the curve
Keypoint on the arc.

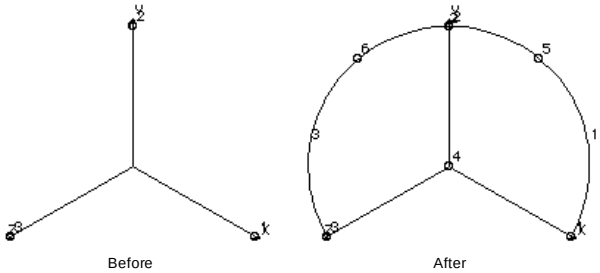
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. If the selected keypoints are colinear, a message is given and no arc is created.
3. The third keypoint lies on the arc and determines its direction.

Example

Graphic Example: CRARC3PT

```
PT,1,50,0,0,
PT,2,0,50,0,
PT,3,0,0,50.,
CRARC3PT,1,1,3,2,
```



CRCIRCLE

Geometry > CURVES > CIRCLES > Circle in Space

The CRCIRCLE command defines a circular arc in space by a defining a center, 2 other keypoints, a radius, and an angle.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint at center
Keypoint at the center of the circle.

Keypoint along axis
Keypoint along axis of the circle.

Keypoint on 0 degree line
Keypoint to the define zero degree line.

Radius
Radius of the circle.

Angle of the arc
Angle (between 0.0 and 360.0).
(default is 360 degrees)

Number of segments
Number of segments forming the circle.
(default is 1 segment for each 90 degrees, maximum is 360)

Notes

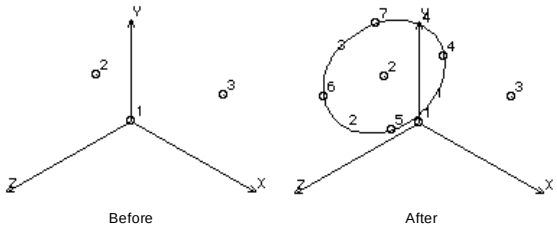
1. Entities can be picked by the mouse only if they are plotted on the screen.

2. A curve is generated for each segment.
3. New curves are numbered by default after the first curve number was entered.
4. The plane of the circle is normal to the line joining the keypoint at the center and the keypoint along the axis. The plane passes through the center and if the keypoint to define the zero degree line is not in the same plane, its projection is used to determine the actual zero degree line.

Graphic Example:

CRCIRCLE

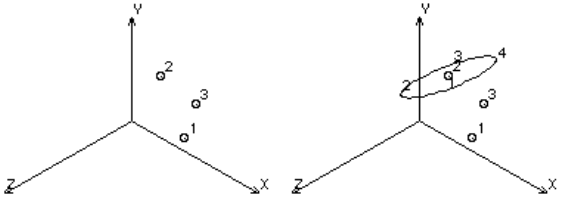
```
PT,1,0,0,0,
PT,2,0,50,30,
PT,3,40,20,-40,
CRCIRCLE,1,2,1,3,4
  0.,360,4,
```



Graphic Example 2:

CRCIRCLE

```
PLANE,Z,0,1,
VIEW,1,1,1,
PT,1,45,10,0,
PT,2,25,45,0,
PT,3,55,40,0,
CRCIRCLE,1,2,1,3,
  30.4138,360,4,
```



CRPCIRCLE

Geometry > CURVES > CIRCLES > Circle in Plane

The CRPCIRCLE command defines a circle in a prescribed plane by a center, a keypoint to define one end, .

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint at center
Keypoint at the center of the circle.

Keypoint on 0 degree line
Keypoint to define zero degree line on the predefined plane.

Radius
Radius of the circle.

Angle of the arc
Angle (between 0.0 and 360.0).
(default is 360 degrees)

Number of segments
Number of segments of circle (between default and 360).

(default is 1 segment for each 90 degrees, maximum is 360)

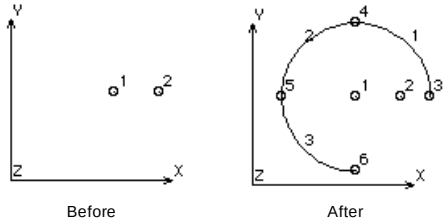
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A curve is generated for each segment specified.
3. New curves are numbered by default after the first curve number was entered.

Example

Graphic Example: CRPCIRCLE

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,35,30,0,
PT,2,50,30,0,
CRPCIRCLE,1,1,2,25,270,3,
```



CRPCIRDIA

Geometry > CURVES > CIRCLES > by Diameter

The CRPCIRDIA command creates a circular arc in a predefined plane using 2 keypoints to define the diameter. The radius is calculated as half the distance between the 2 keypoints.

Curve
Curve label.
(default is the highest curve number defined +1)

Keypoint at one end of diameter
Keypoint at one end of the diameter.

Keypoint at the other end of diameter
Keypoint at the other end of the diameter.

Angle of the arc
Angle of the arc in degrees.
(default is 360.0 degrees)

Number of segments
Number of segments in the arc to be generated (between default and 360).
(default is 1 segment for each 90 degrees, maximum is 360)

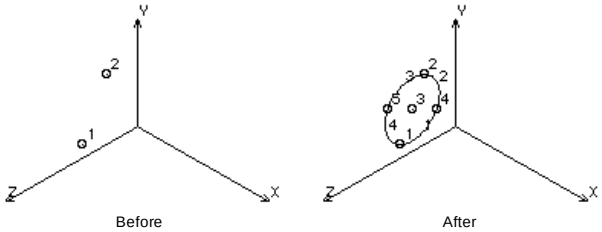
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A plane must be defined before this command can work.
3. The angle of the arc is measured starting from the first keypoint to the second, using the right hand rule convention to determine the direction.
4. Individual curve angles greater than 90 degrees are not allowed.

Example

Graphic Example: CRPCIRDIA

```
PLANE,X,30,1,
PT,1,30,40,75,
PT,2,30,80,55,
CRPCIRDIA,1,1,2,3
  60,4,
```

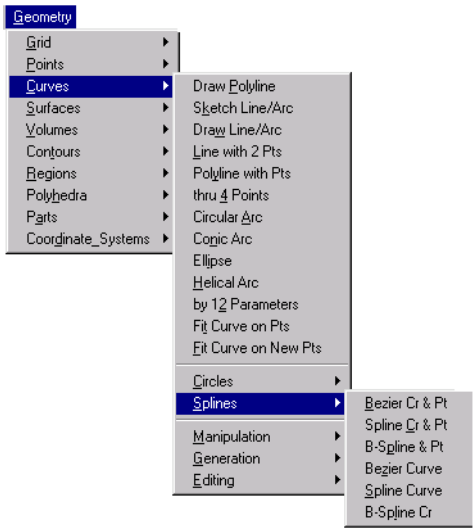


▼SPLINES Menu

Geometry > CURVES > SPLINES

This section contains commands to generate spline curves.

Figure 4-8 Splines Menu



CRBEZCORD

Geometry > CURVES > SPLINES > Bezier Cr & Pt

The CRBEZCORD command creates a Bezier curve by specifying the coordinates of 4 keypoints. The curve starts from the first keypoint and ends at the last. The keypoints are specified by their coordinates or conveniently snapped on a predefined grid by the mouse. The curve does not pass through the second and third keypoints which are used as control points.

Curve
Curve label.
(default is the highest curve number defined + 1)

Keypoint (i) XYZ-Coordinate value
X, Y, Z-coordinate of the ith keypoint. (i=1, 2, 3, 4)

Notes

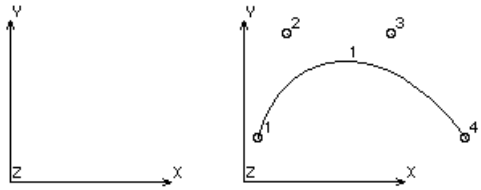
1. Keypoints can be snapped by the mouse on a predefined grid.
2. Bezier curves are extensively used in the automobile and aerospace industries to design profiles. The curve has a continuous derivative.

Examples

Graphic Example 1:

CRBEZCORD

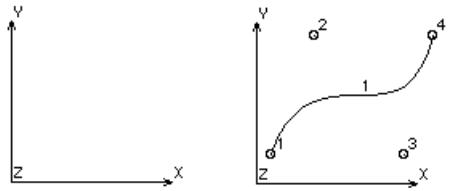
```
PLANE,Z,0,1,
VIEW,0,0,1,
CRBECORD,1,5,15,0,15,50
  ,0,50,50,0,75,15,0,
```



Graphic Example 2:

CRBEZCORD

```
PLANE,Z,0,1,
VIEW,0,0,1,
PLANE,Z,0,1,
CRBEZCORD,1,5,10,0,20,5
  0,0,50,10,0,60,50,0,
```



CRSPLCORD

Geometry > CURVES > SPLINES > Spline Cr & Pt

The CRSPLCORD command creates a series of keypoints and connects them with splines. Up to 20 keypoints and 19 curves (splines) can be created. The keypoints are specified by their coordinates, or conveniently snapped on a predefined grid by the mouse. The created keypoints and curves are numbered by default except the first curve, for which a label can be specified. To create a spline curve using existing keypoints, use the CRSSPL (**Geometry, Curves, Splines, Spline Cr**) command.

Curve
Curve label.
(default is the highest curve number defined + 1)

End condition code for the spline
End condition code.

= 0: Natural	natural spline
= 1: Forced at first	forced spline at the first keypoint
= 2: Forced at last	forced spline at the last keypoint
= 3: Forced at both	forced spline at both first and last keypoints
	(default is natural spline)

Point (i) XYZ-Coordinate value
X, Y, Z-coordinates of the ith keypoint. (i=1,2,.....,20)

Notes

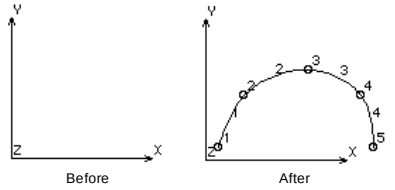
1. The program prompts for more keypoints until a keypoint is snapped twice consecutively, or the limit of 20 keypoints is reached.
2. Each spline is a cubic polynomial that passes through its two end points and satisfies the slope compatibility requirements with other splines.

3. A natural spline connects all corresponding keypoints. A forced spline at the first keypoint is similar to a natural spline except that the first segment, connecting the first keypoint to the second, is not drawn. Similarly, a spline is not drawn to connect the last keypoint if a forced spline is specified at the last keypoint.
4. Every spline connecting two consecutive keypoints is assigned a curve label.

Examples

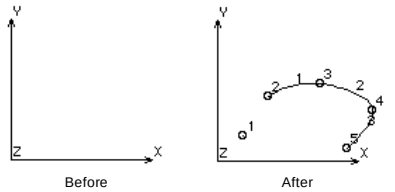
Graphic Example 1: CRSPLCORD

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRSPLCORD,1,0,5,5,0,15,25,0,
  40,35,0,60,25,0,65,5,0,65,
  5,0,
```



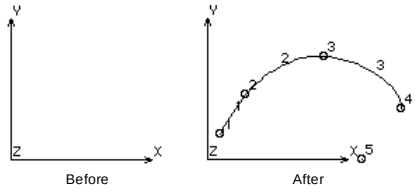
Graphic Example 2: CRSPLCORD

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRSPLCORD,1,1,10,10,0,20,25,
  0,40,30,0,60,20,0,50,5,0,5
  0,5,0,
```



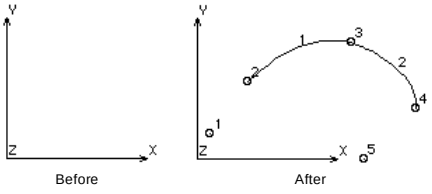
Graphic Example 3: CRSPLCORD

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRSPLCORD,1,2,5,10,0,15,25,0
  ,45,40,0,75,20,0,60,0,0,60
  ,0,0,
```



Graphic Example 4: CRSPLCORD

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRSPLCORD,1,3,5,10,0,20,30,0
  ,60,45,0,85,20,0,65,0,0,65
  ,0,0,
```



CRBSPLCORD

Geometry > CURVES > SPLINES > B-Spline & Pt

The CRBSPLCORD command creates a series of keypoints and an open or closed B-Splines. The keypoints are specified by their coordinates, or conveniently snapped on a predefined grid by the mouse. Up to 20 keypoints can be specified. The generated curves have first and second continuous derivatives. To create a B-Spline curve using existing keypoints, use the CRSBSPL (**Geometry, Curves, Splines, B-Spline Cr**) command.

Curve

Curve label.
(default is the highest curve number defined + 1)

Close flag
Flag to specify an open or closed loop.
= 0: Open open loop
= 1: Close closed loop
(default is open)

Number of control keypoints
Number of control keypoints (limit is 20).
(default is 6)

Keypoint (i) XYZ-Coordinate value
X, Y, Z-coordinates of the ith keypoint of the B-spline. (i= 1, 2, ..., 20)

Notes

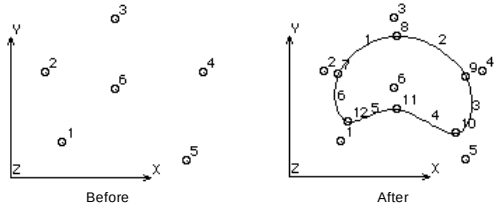
1. Keypoints can be snapped on a predefined grid.
2. The same control keypoint can be specified two or three times to give more weight to a keypoint.
3. The command uses 4 consecutive keypoints to generate a B-Spline near the second and the third keypoints. If a closed loop is specified, successive keypoints are used in a cyclic fashion until the loop is closed. Otherwise, the open loop starts at the second keypoint and ends at the keypoint before the last.

Examples

Graphic Example 1:

CRBSPLCORD

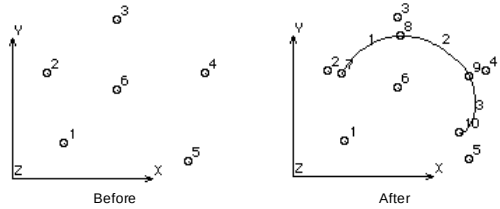
```
PLANE,Z,0,1,
VIEW,0,0,1,
CRBSPLCORD,1,0,6,10,1
  0,0,10,40,0,40,50,0,
  65,40,0,45,10,0,20,0
  ,0,
```



Graphic Example 2:

CRBSPLCORD

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRBSPLCORD,1,1,6,10,1
  0,0,10,40,0,40,50,0,
  65,40,0,45,10,0,20,0
  ,0,
```



CRBEZIER

Geometry > CURVES > SPLINES > Bezier Curve

The CRBEZIER command creates a Bezier curve using 4 keypoints. The curve starts from the first keypoint and ends at the last. The curve does not pass through the second and third keypoints.

Curve
Curve label.
(default is the highest curve number defined + 1)

Control keypoint (i)
ith keypoint of the Bezier curve. (i= 1, 2, 3, 4)

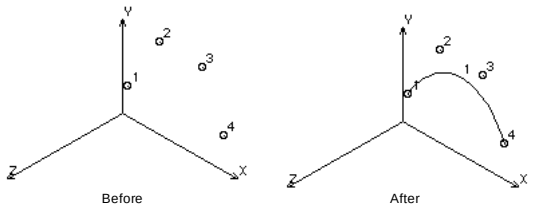
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. Bezier curves are extensively used in the automobile and aerospace industries to design profiles. The curve has a continuous first derivative.

Example

Graphic Example: CRBEZIER

```
PLANE,Z,0,1,
PT,1,5,25,0,
PT,2,35,75,0,
PT,3,75,75,0,
PT,4,95,30,0,
CRBEZIER,1,1,2,3,4,
```



CRSPLINE

Geometry > CURVES > SPLINES > Spline Curve

The CRSPLINE command connects a series of existing keypoints by natural or forced splines. Up to 20 keypoints can be selected. To create the keypoints and the curve in one command, use the CRSPLCORD (**Geometry, Curves, Splines, Spline Cr & pt**) command.

Curve
Curve label.
(default is the highest curve number defined + 1)

End condition code for the spline
End condition code.

= 0: Natural spline	natural spline
= 1: Forced at first	forced spline at the first keypoint
= 2: Forced at last	forced spline at the last keypoint
= 3: Forced at both	forced spline at both first and last keypoints

(default is natural spline)

Keypoint (i)
ith keypoint of the spline. (i= 1, 2, ..., 20)

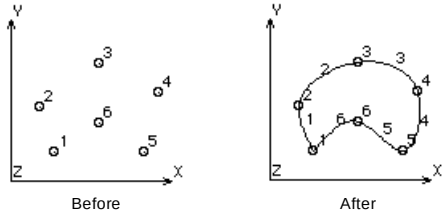
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The program prompts for more keypoints until a keypoint is selected twice, or the limit of 20 keypoints is reached.
3. Each spline is a cubic equation that passes through its two endpoints and satisfies the slope compatibility requirements with other splines.
4. A natural spline connects all corresponding keypoints. A forced spline at the first keypoint is similar to a natural spline except that the first segment, connecting the first keypoint to the second, is not drawn. Similarly, a spline is not drawn to connect the last keypoint if a forced spline is specified at the last keypoint.
5. Every spline connecting two consecutive keypoints is assigned a curve label.

Example

Graphic Example: CRSPLINE

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,15,10,0,
PT,2,10,25,0,
PT,3,30,40,0,
PT,4,50,30,0,
PT,5,45,10,0,
PT,6,30,20,0,
CRSPLINE,1,0,1,2,3,4,5,6
  ,1,1,
```



CRBSPLINE

Geometry > CURVES > SPLINES > B-Spline Cr

The CRBSPLINE command creates open or closed B-Splines using a series of specified control keypoints. Up to 20 keypoints can be specified. The generated curves have continuous first and second derivatives. To create the keypoints and the curve in one command, use the CRSBSPLCORD (**Geometry, Curves, Splines, B-Spline & pt**) command.

Curve
Curve label.
(default is the highest curve number defined + 1)

Close flag
Flag to specify an open or closed loop.
= 0: Open open
= 1: Close closed

(default is open)

Number of control keypoints
Number of control keypoints. Limited to 20.
(default is 6)

Keypoint (i)
ith keypoint of the B-spline. (i= 1, 2, ..., 20)

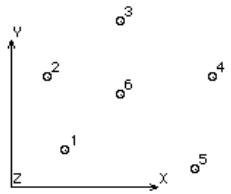
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The same control keypoint can be specified two or three times to give more weight to a keypoint.
3. The command uses 4 consecutive keypoints to generate a B-Spline curve between the second and the third keypoints. If a closed loop is specified, successive keypoints are used in a circular fashion until the loop is closed. Otherwise, the open loop starts at the second keypoint and ends at the keypoint before the last.

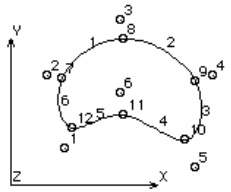
Examples

Graphic Example: CRBSPLINE

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,15,10,0,
PT,2,10,30,0,
PT,3,30,45,0,
PT,4,55,30,0,
PT,5,50,5,0,
PT,6,30,25,0,
CRBSPLINE,1,1,6,1,2,3
  ,4,5,6,
```



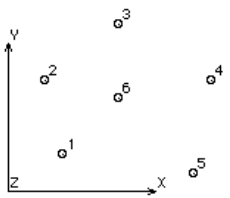
Before



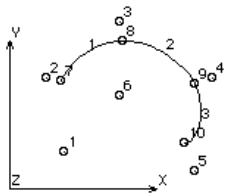
After

Graphic Example 2: CRBSPLINE

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,15,10,0,
PT,2,10,30,0,
PT,3,30,45,0,
PT,4,55,30,0,
PT,5,50,5,0,
PT,6,30,25,0,
CRBSPLINE,1,0,6,1,2,
  3,4,5,6,
```



Before



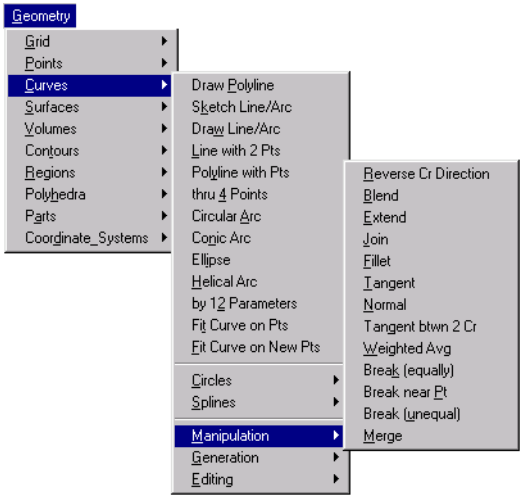
After

▼MANIPULATION Menu

Geometry > CURVES > MANIPULATION MENU

This menu contains commands to manipulate existing curves.

Figure 4-9 Manipulation Menu



CRREPAR

Geometry > CURVES > MANIPULATION MENU > Reverse Cr Direction

The CRREPAR command reverses the directions of curves in the specified pattern. Curve direction is used for non-uniform parametric meshing and creating a key-point on a curve (**Geometry, Points, Generation, Point On Curve**) command.

Beginning curve
Beginning curve of the pattern.

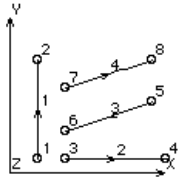
Ending curve
Ending curve of the pattern.
(default is beginning curve)

Increment
Increment between curves of the pattern.
(default is 1)

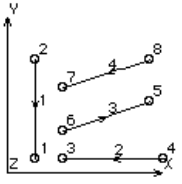
Example

Graphic Example: CRREPAR

```
PLANE,Z,0,1,
VIEW,0,0,1,
CR2CORD,1,10,5,0,10,40,0,
CR2CORD,2,20,5,0,55,5,0,
CR2CORD,3,50,25,0,20,15,0,
CR2CORD,4,20,30,0,50,40,0,
CRREPAR,1,4,1,
```



Before



After

CRBLEND

Geometry > CURVES > MANIPULATION MENU > Blend

The CRBLEND command blends two existing curves. The new blended curves have a common tangent at the common keypoint. The common tangent is calculated using the slopes of the original curves and the specified ratio and weight.

Curve 1
First curve to be blended.

Curve 2
Second curve to be blended.

Ratio
Ratio of change in the tangent of the first curve to that of the second curve in determining the new common tangent. (must be between 0.0 and 1.0)
(default is 0.5)

Weight
A factor that determines relative lengths of the straight line segments.
(default is 1.0)

Original curve keeping flag
Original curves keeping flag.
= 0: Do not keep do not keep original curves unless part of a higher entity
= 1: Keep original curve keep original curves
(default is keep original curve)

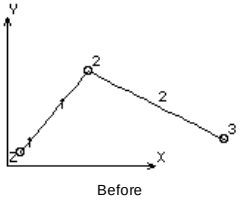
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A ratio of 1.0 forces no change in the tangent to the second curve.
3. The keep flag allows the user to keep the original curves even if they are not part of higher entities. The curves are always kept if part of higher entities.
4. The command is intended for the design of curve profiles and for approximate geometry.

Example

Graphic Example: CRBLEND

```
PLANE,Z,0,1,
VIEW,0,0,1,
CR2CORD,1,5,5,0,30,3
  5,0,
CR2CORD,2,30,35,0,80
  ,10,0,
CRBLEND,1,2,0.5,1,1,
```



Before After

CREXTEND

Geometry > CURVES > MANIPULATION MENU > Extend

The CREXTEND command extends a curve at a specified end by a specified length. The extension is a straight line with a new label, in the direction of the tangent to the curve at the specified end.

Curve
Label of curve to be generated.
(default is the highest curve number defined + 1)

Curve to be extended
Label of curve to be extended.

Keypoint of extension
Label of keypoint at the start of the extension.

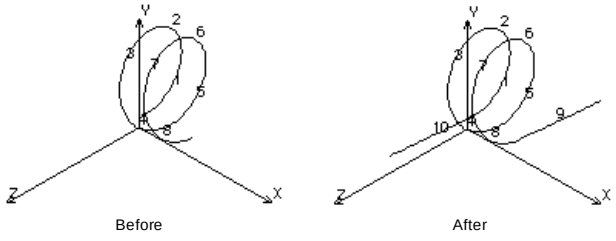
Length
Length of the extension.

Note
Labels can be picked by the mouse.

Example

Graphic Example: CREXTEND

```
PLANE,Z,0,1,
PT,1,0,40,0,
PT,2,100,50,0,
PT,3,0,0,0,
CRHELIX,1,1,2,
  3,30.,20,720.
  ,8,
CREXTEND,9,8,1
  2,60.,
CREXTEND,10,1,
  4,60.,
```



CRJOIN

Geometry > CURVES > MANIPULATION MENU > Join

The CRJOIN command joins two curves at the specified keypoints. The original curves to be joined are not affected by this command.

Curve
New curve label.
(default is the highest curve number defined + 1)

Curve 1
First curve to be joined.

Curve 2
Second curve to be joined.

Keypoint of curve 1
Keypoint of curve 1 at which the curve will be joined.

Keypoint of curve 2
Keypoint of curve 2 at which the curve will be joined.

Weight factor for curve 1
Weight factor (between 1 and 10) assigned to the slope of curve 1.
(default is 1)

Weight factor for curve 2
Weight factor (between 1 and 10) assigned to the slope of the curve 2.
(default is 1)

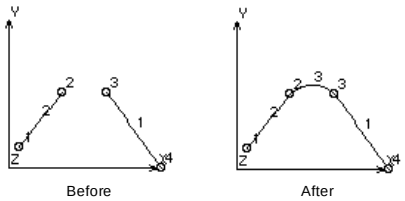
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The weights are used as scaling factors for the magnitudes of the tangents of the curves at reference points.
3. A higher relative weight for one of the base curves gives more weight to it in defining the new curve.
4. The command keeps the original curves unchanged to allow the user to make several trials before a satisfactory curve is obtained. Unwanted curves can be deleted by the CRDEL (**Edit, Delete, Curves**) command.
5. The command is intended for creating approximate geometry profiles.

Example

Graphic Example: CRJOIN

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,5,10,0,
PT,2,25,35,0,
CR2CORD,1,45,35,0,70,0,0,
CRLINE,2,1,2,
SCALE,.8,
CRJOIN,3,1,2,3,2,1.0,1.0,
```



CRFILLET

Geometry > CURVES > MANIPULATION MENU > Fillet

The CRFILLET command creates a circular fillet of a specified radius between 2 curves. It provides an option to trim the curves up to the start of the fillet.

Curve
Label of curve to be generated.
(default is the highest curve number defined + 1)

Curve 1

First curve label.

Curve 2
Second curve label.

Radius of fillet
Radius of fillet to be created.

Trim flag
Flag to trim the filleted curves.
= 0: Do not trim — do not trim original curves
= 1: Trim — trim original curves
(default is trim)

Original curve keeping flag
Original curves keeping flag.
= 0: Do not keep — do not keep original curves unless part of a higher entity
= 1: Keep original — always keep original curves
(default is do not keep original)

Tolerance
Tolerance to define the fillet.
(default depends on dimensions)

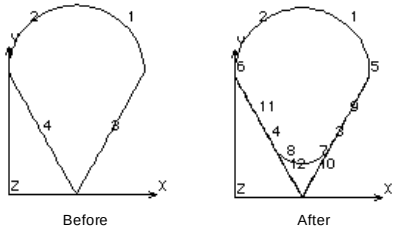
Notes

1. New keypoints at the center of the fillet and at its ends are created. Another keypoint is created on the fillet if its angle is more than 90 degrees.
2. The created fillet is labeled by default. If the fillet angle is more than 90 degrees, the fillet is broken into two curves.
3. Each original curve is broken into two curves at the fillet point.
4. The *keep* flag lets you keep the original curves even if they are not part of higher entities. The curves are always kept if they are part of higher entities.

Example

Graphic Example: CRFILLET

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,0,45,0,
PT,2,50,45,0,
CRPCIRDIA,1,2,1,180,2,
PT,5,25,0,0,
CRLINE,3,5,2,
CRLINE,4,1,5,
CRFILLET,5,1,3,10,1,0,
CRFILLET,6,2,4,10,1,0,
CRFILLET,7,3,4,12,0,
```



CRTANPT

Geometry > CURVES > MANIPULATION MENU > Tangent

The CRTANPT command creates a tangent from a keypoint to a base curve.

Curve
Curve label of the line to be generated.
(default is the highest curve number defined + 1)

Curve for tangency
Label of the base (original) curve.

Keypoint of tangency
Label of the keypoint from which tangent is drawn.

Break flag
Flag to break the base curve at the tangent point.

= 0: Do not break	do not break original curve
= 1: Break original	break the original curve to two segments at the tangent point *(default is do not break)*

Original curve keeping flag
Original curve keeping flag.

= 0: Do not keep	delete the original curve
= 1: Keep original	keep the original curve *(default is do not keep original)*

Notes

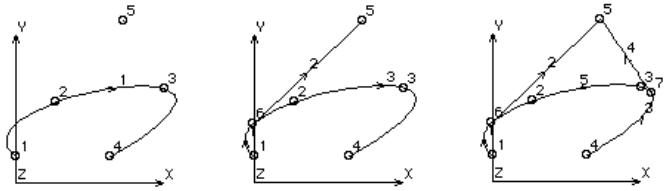
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The command searches the curve from its start, looking for a point to draw the tangent to. Once a point is found, the search is terminated. If two tangents exist and are needed, the direction of the curve can be reversed by the CRREPAR (**Geometry, Curves, Manipulate, Reverse Cr Direction**) command to start the search from the other end.
3. If more than two tangents exist and are needed, the original curve must be broken using (**Geometry, Curves, Manipulate, Break (equally) - Break Near Pt, or Break (unequally)** commands.

Example

Graphic Example: CRTANPT

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,0,10,0,
PT,2,15,30,0,
PT,3,55,35,0,
PT,4,35,10,0,
PT,5,40,60,0,
CR4PT,1,1,2,3,4,
```

```
CRTANPT,2,1,5,1,0,
CRREPAR,3,3,1,
CRTANPT,4,3,5,1,0,
```



CRNORMPT

Geometry > CURVES > MANIPULATION MENU > Normal

The CRNORMPT command projects a normal from a keypoint to a base curve.

Curve
Curve label of the new line to be generated.
(default is the highest curve number defined + 1)

Curve to construct normal
Label of the base (original) curve to which the normal is to be projected.

Keypoint of normal
Label of the keypoint from which the normal is projected.

Break flag
Flag to break the original curve at the point of intersection with the normal.

= 0: Do not break	do not break the original curve unless part of a higher entity
= 1: Break original	break the original curve into two segments at the point of intersection *(default is do not break original)*

Original curve keeping flag
Original curve keeping flag.

= 0: Do not keep	delete the original curve
= 1: Keep original	keep the original curve *(default is do not keep original)*

Notes

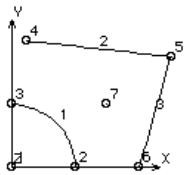
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The command sweeps the curve from its start, searching for a point of projection. Once a point is located, the sweeping is terminated. If two normals exist and are needed, the direction of the curve can be reversed by the CRREPAR (**Geometry, Curves, Manipulate, Reverse Cr Direction**) command to start the search from the other end.

3. If more than two normals exist and are needed, the original curve must be broken using the (**Geometry, Curves, Manipulate, Break (equally) - Break Near Pt, or Break (unequally)** commands.
4. The 'original curve keeping flag' lets you keep the original curve even if it is not part of a higher entity. The curve is always kept if it is a part of a higher entity regardless of the 'original curve keeping flag'.

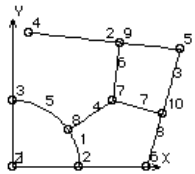
Example

Graphic Example: CRNORMPT

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,0,0,0,
PT,2,20,0,0,
PT,3,0,20,0,
CRARC,1,2,3,1,20,
CRPCORD,2,5,40,0,50,35,0,4
  0,0,0,40,0,0,
PT,7,30,20,0,
SCALE,.8,
CRNORMPT,4,1,7,1,0,
CRNORMPT,6,2,7,1,0,
CRNORMPT,7,3,7,1,0,
```



Before



After

CRTANLIN

Geometry > CURVES > MANIPULATION MENU > Tangent btwn 2 Cr

The CRTANLIN command creates a line that is tangential to two base curves.

Curve
Curve label of the line to be generated.
(default is the highest curve number defined +1)

Curve 1
Label of first base (original) curve.

Curve 2
Label of second base (original) curve.

Break flag
Flag to break both curves at tangent points.

= 0: Do not break	do not original base curves
= 1: Break original	break each original curve into 2 segments
	(default is do not break original)

Original curve keeping flag
Original curve keeping flag.

= 0: Do not keep	delete the original curves
= 1: Keep original	keep the original curves
	(default is do not keep original)

Notes

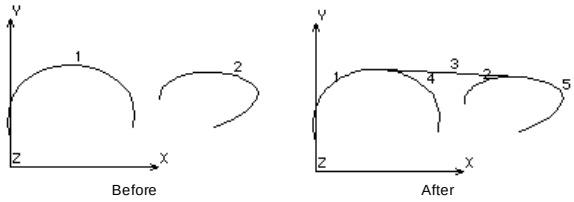
1. Entities can be picked bt the mouse only if they are plotted on the screen.
2. The 'original curve keeping flag' allows the user to preserve the original curves even if they are not part of higher entities. The curves are always preserved if they are part of higher entities regardless of the 'original curve keeping flag'.

Example

Graphic Example:

CRTANLIN

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,0,10,0,
PT,2,10,35,0,
PT,3,35,35,0,
PT,4,45,15,0,
PT,5,55,25,0,
PT,6,70,35,0,
PT,7,90,30,0,
PT,8,75,15,0,
CR4PT,1,1,2,3,4,
CR4PT,2,5,6,7,8,
CRTANLIN,3,1,2,1,0,
```



CRWAVRG

Geometry > CURVES > MANIPULATION MENU > Weighted Avg

The CRWAVRG command creates a curve by the weighted averaging of two base curves.

Curve
New curve label.
(default is the highest curve number defined + 1)

Curve 1
First base curve label.

Curve 2
Second base curve label.

Ratio
Ratio of weight for the first curve to that of the second curve (must be between 0.0 and 1.0).
(default is 0.5)

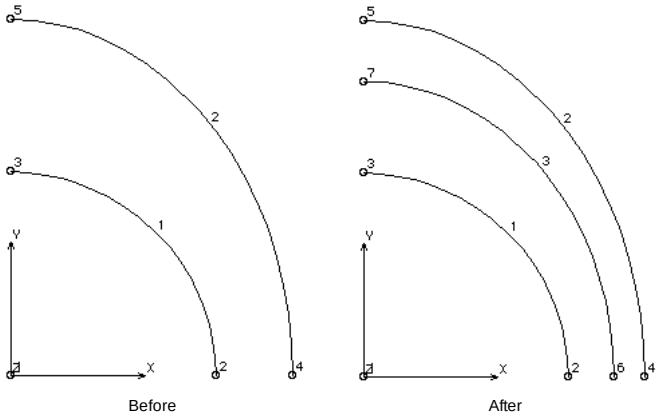
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The command is intended to mix existing curves to get a desired profile.

Example

Graphic Example: CRWAVRG

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,0,0,0,
PT,2,40,0,0,
PT,3,0,40,0,
PT,4,55,0,0,
PT,5,0,70,0,
CRARC,1,2,3,1,40,
CRELLIPSE,2,4,5,1,1,
CRWAVRG,3,1,2,.4,
```



CRBRK

Geometry > CURVES > MANIPULATION MENU > Break (equally)

The CRBRK command generates additional curves from a pattern of curves by breaking each curve into a prescribed number of uniform segments.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Number of segments
Number of segments.
(default is 2)

Original curve keeping flag

Original curves keeping flag.
= 0: Do not keep do not keep the original curves unless they are part of higher entities
= 1: Keep original always keep the original curves
(default is not to keep original curves)

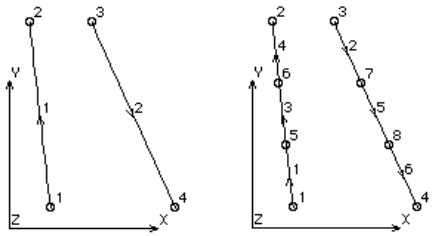
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The 'original curve keeping flag' lets you preserve the original curves even if they are not part of higher entities. A curve is always preserved if it is associated with a higher entity.

Example

Graphic Example: CRBRK

```
PLANE,Z,0,1,
VIEW,0,0,1,
CR2CORD,1,10,5,0,5,50,0,
CR2CORD,2,20,50,0,40,5,0,
CRBRK,1,2,1,3,0,
```



CRPTBRK

Geometry > CURVES > MANIPULATION MENU > Break Near Pt

The CRPTBRK command breaks a curve into two segments by projecting a normal from a reference keypoint.

Curve to be broken
Label of the curve to be broken into two segments.

Reference keypoint
Label of the reference keypoint.

Original curve keeping flag
Original curve keeping flag.
= 0: Do not keep delete the original curve unless they are part of a higher entity
= 1: Keep original always keep the original curve
(default is do not keep original)

Notes

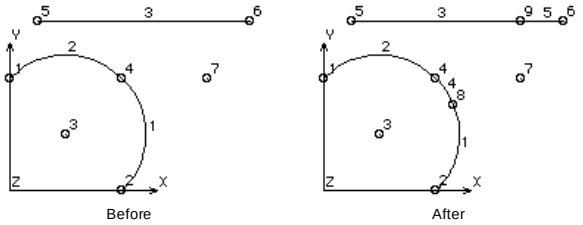
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The 'original curve keeping flag' allows the user to keep the original curve even if it is not part of a higher entity. The curve is always preserved if it is a part of a higher entity regardless of the 'original curve keeping flag'.

Example

Graphic Example: CRPTBRK

```
PLANE,Z,0,1,
```

```
VIEW,0,0,1,
PT,1,0,40,0,
PT,2,40,0,0,
CRPCIRDIA,1,2,1,
  180,2,
CR2CORD,3,10,60,
  0,85,60,0,
PT,7,70,40,0,
SCALE,.7,
CRPTBRK,1,7,0,
CRPTBRK,3,7,0,
```



CRNUBRK

Geometry > CURVES > MANIPULATION MENU > Break (unequally)

The CRNUBRK command breaks each curve in a pattern of curves into a specified number of segments. The segmentation is specified by parametric coordinates. Check the **Mark** column for curves in the STATUS1 table to display the direction of curves before issuing this command.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Number of segments
Number of segments.
(default is 2)

Parameter value
Parametric coordinate for ith point. You will be required to input (*Number of segments - 1*) parametric coordinate in ascending order. Each value must lie between 0.0 and 1.0.

Original curve keeping flag
Original curves keeping flag.
= 0: Do not keep do not keep original curves unless they are part of higher entities
= 1: Keep original always keep the original curves
(default is do not keep original)

Notes

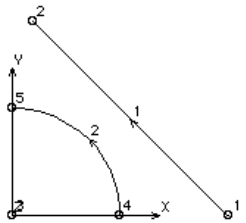
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The 'original curve keeping flag' lets you keep the original curves even if they are not part of higher entities. The curves are always preserved if part of

higher entities.

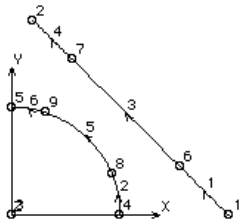
Example

Graphic Example: CRNUBRK

```
PLANE,Z,0,1,
VIEW,0,0,1,
CR2CORD,1,50,0,0,5,4
  5,0,
PT,3,0,0,0,
PT,4,25,0,0,
PT,5,0,25,0,
CRARC,2,4,5,3,25,
CRNUBRK,1,2,1,3,.25,
  .8,0,
```



Before



After

CRMERGE

Geometry > CURVES > MANIPULATION MENU > Merge

The CRMERGE command merges curves. Two curves are merged if they coincident (within the specified tolerance). The end keypoints as well as two intermediate points are checked. You must be careful since merged curves will be lost unless they are part of higher entities. We suggest that you use the **File, Save Database As** command before issuing this command. The command need not be used in most cases but may be useful in creating polyhedra and parts for some complicated models or imported 3-D solid geometry.

Beginning curve

Beginning curve in the pattern.

Ending curve

Ending curve in the pattern.
(default is the max. curve label defined)

Increment

Increment between curves in the pattern.
(default is 1)

Tolerance

Magnitude of tolerance.
(default is .0001)

All/Among flag

Merging flag.

= 0: All	merge with respect to all curves in the model
= 1: Among	merge with respect to the curves specified in the pattern only *(default is all)*

Echo flag

Message flag.

= 1: On — GEOSTAR issues a message when two curves are merged
= 0: Off — GEOSTAR will not issue any messages
(default is on)

Low/High flag
Direction flag.
= 0: Low — when two curves are merged, keep the curve with the lower label
= 1: High — when two curves are merged, keep the curve with the higher label
(default is low)

Notes

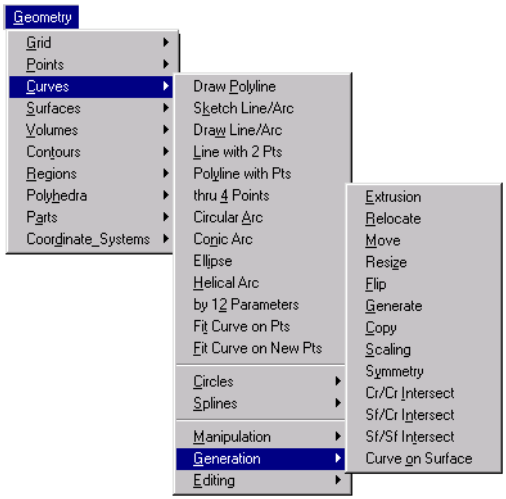
1. Entitis can be selected by the mouse only if they are plotted on the screen.
2. If merging with respect to the curves in the pattern is specified, then other curves (curves not specified in the pattern) are not checked for merging.
3. If a curve is merged, its keypoints will also be deleted, unless they are associated with other entities.

▼GENERATION Menu

Geometry > CURVES > GENERATION

This menu contains commands to generate curves by various operations using existing geometric entities.

Figure 4-10 Generation Menu



CREXTR

Geometry > CURVES > GENERATION > Extrusion

The CREXTR command defines a set of lines by extruding keypoints along a specified coordinate axis of the active Cartesian coordinate system.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.

Increment
Increment between keypoints in the pattern.
(default is 1)

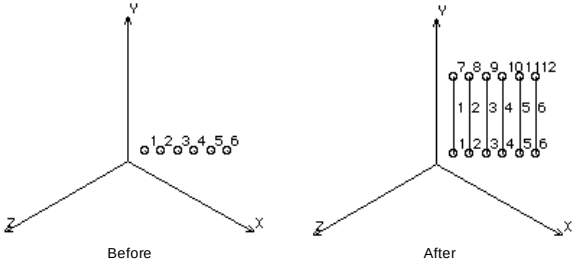
Axis symbol
Axis symbol (X, Y, Z).
(default is Z)

Value
Length of extrusion.

Example

Graphic Example:

```
            CREXTR
PT,1,10,10,0,
PTGEN,5,1,1,1,0,1
  0,5,0,
CREXTR,1,6,1,Y,40
  .,
```



CRRELOC

Geometry > CURVES > GENERATION > Relocate

The CRRELOC command relocates a pattern of curves by a specified translation and/or rotation in the currently active Cartesian coordinate system. A curve must be free from association with higher entities for a successful relocation. This command does not generate any new curves. The PTGEN (**Geometry, Curves, Generation, Generate**) command can be used to generate new curves from existing ones.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.

(defaults are 0.0)

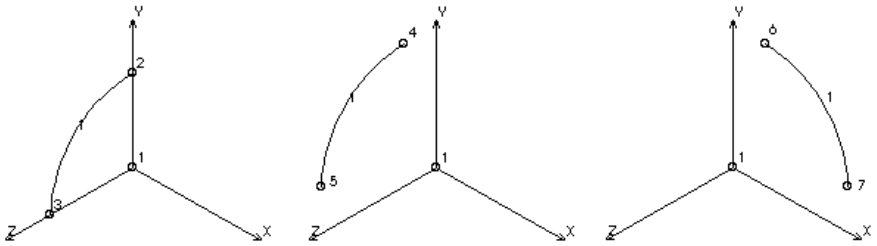
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. If a curve in the specified pattern is part of a higher entity, that curve will not be relocated.

Example

Graphic Example: CRRELOC

```
PT,1,0,0,0,
PT,2,0,50,0,
PT,3,0,0,50,
CRARC,1,2,3,1,50,
CRRELOC,1,1,1,0,0,25,20,
CRRELOC,1,1,1,1,0,90,0,
```



CRMOVE

Geometry > CURVES > GENERATION > Move

The CRMOVE command moves a pattern of curves from the current coordinate system to the destination coordinate system. This command does not generate any new curves. The CRCOPY (**Geometry, Curves, Generation, Copy**) command can be used to generate new curves from existing ones.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Destination coordinate

Destination coordinate system label.

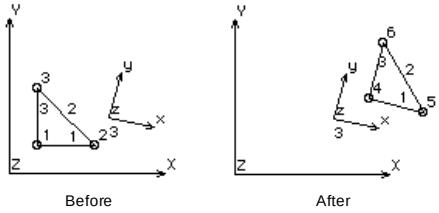
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. If a curve in the specified pattern is part of a higher entity, that curve will not be moved.
3. Both coordinate systems must be Cartesian.

Example

Graphic Example: CRMOVE

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRPCORD,1,10,10,0,30,10,0,10
  ,30,0,10,10,0,
CSANGLE,3,0,35,20,0,0,0,-15,
ACTSET,CS,0,
CRMOVE,1,3,1,3,
```



CRRESIZ

Geometry > CURVES > GENERATION > Resize

The CRRESIZ command resizes a pattern of existing curves, by scaling and translating/rotating them relative to the currently active coordinate system. This command does not generate any new curves. The CRSCALE (**Geometry, Curves, Generation, Scaling**) command can be used to generate new curves by scaling existing ones.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X-, Y-, and Z-directions.
(defaults are 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate sys-

tem. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

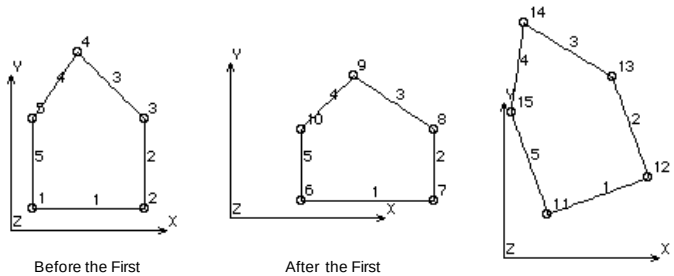
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. If a curve in the specified pattern is part of a higher entity, that curve will not be resized.

Example

Graphic Example: CRRESIZ

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRPCORD,1,5,5,0,30,5,0,30,25,0,15,40,0,5,25,0,5,5,0,
CRRESIZ,1,5,1,0,1.2,.8,0,10,0,0,
CRRESIZ,1,5,1,1,.8,1.5,0,0,0,20,
```



CRFLIP

Geometry > CURVES > GENERATION > Flip

The CRFLIP command flips a pattern of curves about a specified plane in the current coordinate system. An offset can also be specified with the flipping. This command does not generate any new curves. The CRSYM (**Geometry, Curves, Generation, Symmetry**) command can be used to generate new curves from existing ones.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.

(default is beginning curve)

Increment

Increment between curves in the pattern.
(default is 1)

Reverse direction flag

Reverse direction flag.

= 0: No	do not reverse directions of curves
= 1: Yes	reverse directions of curves
	(default is reverse direction)

Rotation/Sweep axis

Axis representing normal to the plane about which pattern of curves are to be flipped.

= X	about y-z plane
= Y	about z-x plane
= Z	about x-y plane
	(default is Z)

Offset

Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

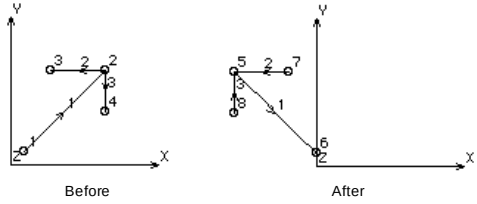
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. If a curve in the specified pattern is part of a higher entity, that curve will not be flipped.
3. A specified offset is applied to all curves in the specified pattern.

Example

Graphic Example: CRFLIP

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRPCORD,1,5,5,0,35,35,0,1
  5,35,0,15,35,0,
CR2CORD,3,35,35,0,35,20,0
  ,
CRFLIP,1,3,1,X,1,5,
```



CRGEN

Geometry > CURVES > GENERATION > Generate

The CRGEN command generates one or more patterns of curves from an existing pattern, by translating and/or rotating them about the currently active Cartesian coordinate system. The CRRELOC (**Geometry, Curves, Generation, Relocate**) command can be used to relocate existing curves without generating new ones.

Generation number

Number of patterns to be generated.

(default is 1)

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

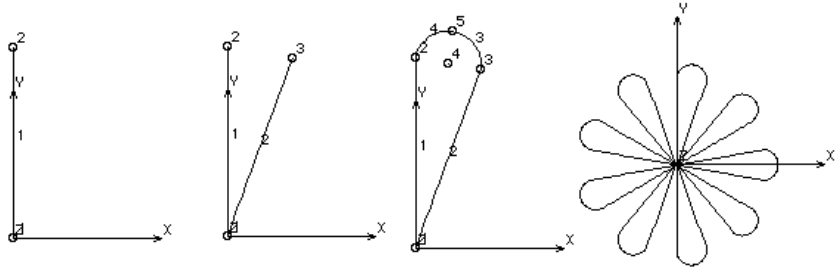
X, Y, Z-Displacement
Magnitudes of the translation in the currently active coordinate system.
(defaults are 0.0)

X, Y, Z-Rotation
Magnitudes of the rotations (degrees) in the currently active coordinate system.
(defaults are 0.0)

Example

Graphic Example: CRGEN

```
PT,1,0,0,0,
PT,2,0,70,0,
CRLINE,1,1,2,
CRGEN,1,1,1,1,1,0,0,-20.,
PLANE,Z,0,1,
CRPCIRDIA,3,3,2,180,2,
CRGEN,8,1,4,1,1,0,0,40.,
```



CRCOPY

Geometry > CURVES > GENERATION > Copy

The CRCOPY command copies a pattern of curves from the active coordinate system to the destination coordinate system. Use the CRMOVE (**Geomtery, Curves, Generation, Move**) command to move existing curves without generating new ones.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Destination coordinate
Destination coordinate system label.

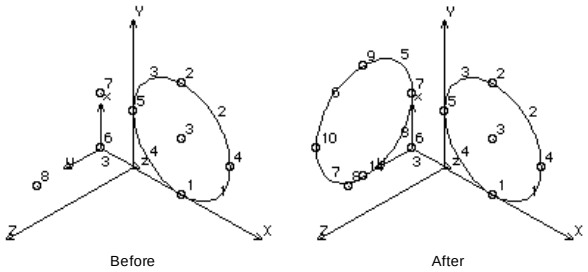
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. Both coordinate systems must be Cartesian.

Example

Graphic Example: CRCOPY

```
PT,1,30,0,0,
PT,2,30,60,0,
PLANE,Z,0,1,
CRPCIRDIA,1,1,2,
  360,4,
PT,6,-20,0,0,
PT,7,-20,30,0,
PT,8,-20,0,40,
CSYS,3,0,6,7,8,
ACTSET,CS,0,
VIEW,1,1,1,
CRCOPY,1,4,1,3,
```



CRSCALE

Geometry > CURVES > GENERATION > Scaling

The CRSCALE command generates a pattern of curves from an existing pattern, by scaling and translating/rotating them relative to the currently active Cartesian coordinate system. The coordinates of all associated keypoints are multiplied by the specified scale factors.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X, Y, and Z directions.
(defaults are 1.0)

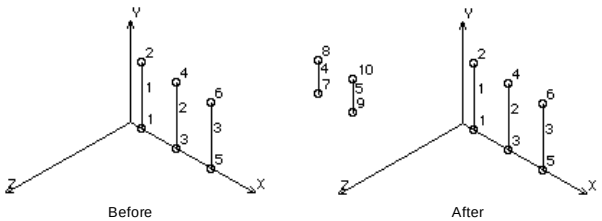
X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

Example

Graphic Example: CRSCALE

```
PT,1,10,0,0,
CREXTRUDE,1,1,1,Y
  ,50.,
CRGEN,2,1,1,1,2,3
  0,0,0,0,0,0,
CRSCALE,1,3,2,0,.
  5,.5,0,80,10,50
  ,
```



CRSYM

Geometry > CURVES > GENERATION > Symmetry

The CRSYM command generates a pattern of curves symmetric to the specified pattern about a specified plane in the current coordinate system. An offset can also be specified. Use the CRFLIP (**Geomtery, Curves, Generation, Flip**) command to flip existing curves without generating new ones.

Beginning curve

Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Rotation/Sweep axis
Axis representing normal to the plane about which pattern of curves are to be generated symmetrically.

= X	about y-z plane
= Y	about x-z plane
= Z	about x-y plane
	(default is Z)

Reverse direction flag
Reverse direction flag.

= 0: No	do not reverse directions of curves
= 1: Yes	reverse directions of curves
	(default is reverse direction)

Offset
Magnitude of the offset in the direction of the specified axis.

Notes

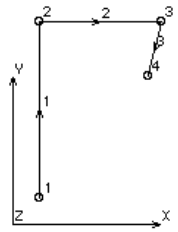
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A specified offset is applied to all curves of the specified pattern.

Example

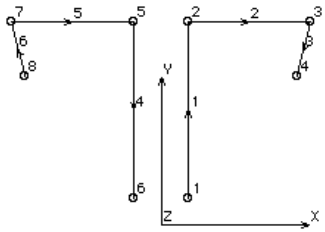
Graphic Example:

CRSYM

```
VIEW,0,0,1,
PLANE,Z,0,1,
CRPCORD,1,10,10,0
  ,10,75,0,55,75,
  0,50,55,0,50,55
  ,0,
CRSYM,1,3,1,X,1,0
  ,
```



Before



After

CRINTCC

Geometry > CURVES > GENERATION > Cr/Cr Intersect

The CRINTCC command defines segments curves based on the intersections of a primary curve with a pattern of curves.

Intersecting curve

Primary intersecting curve.

Beginning intersecting curve
Beginning intersection curve in the pattern.

Ending intersecting curve
Ending intersection curve in the pattern.
(default is beginning intersecting curve)

Increment
Increment between curves in the pattern.
(default is 1)

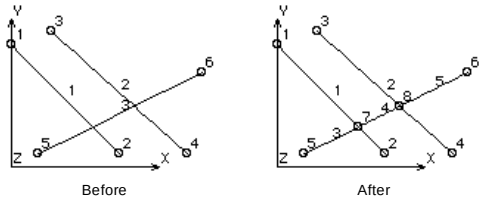
Segment type
Segmentation type.
= 0: Primary curve only segment primary curve only
= 1: Secondary curves only segment secondary curves only
= 2: All segment all curves
(default is primary curve only)

Tolerance
Tolerance.
(default depends on dimensions)

Example

Graphic Example: CRINTCC

```
PLANE,Z,0,1,
VIEW,0,0,1,
CR2CORD,1,0,45,0,40,5,0,
CR2CORD,2,15,50,0,65,5,0,
CR2CORD,3,10,5,0,70,35,0,
CRINTCC,3,1,2,1,0,
```



CRINTSC

Geometry > CURVES > GENERATION > Sf/Cr Intersect

The CRINTSC command defines points of intersection of a primary surface with a pattern of curves. Each curve is broken at its intersection with the surface.

Intersecting Surface
Primary intersecting surface.

Beginning intersecting curve
Beginning intersection curve in the pattern.

Ending intersecting curve
Ending intersection curve in the pattern.
(default is beginning intersecting curve)

Increment
Increment between curves in the pattern.

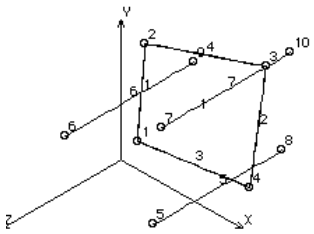
(default is 1)

Tolerance
Tolerance.
(default depends is 5.0 E-5)

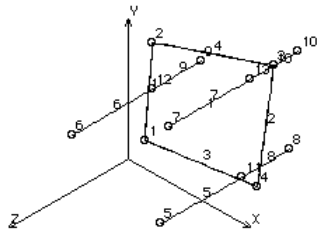
Example

Graphic Example: CRINTSC

```
PLANE,Z,0,1,
SF4CORD,1,10,15
  ,0,15,70,0,90
  ,95,0,80,25,0
  ,
PLANE,Z,50,1,
PT,5,70,25,50,
PT,6,15,45,50,
PT,7,75,80,50,
CREXTR,5,7,1,Z,
  -80.,
CRINTSC,1,5,7,1
  ,
```



Before



After

CRINTSS

Geometry > CURVES > GENERATION > Sf/Sf Intersect

The CRINTSS command creates curves resulting from the intersection of a primary surface with a specified pattern of surfaces. The curves are created on the intersecting surfaces if they are found within the specified tolerance from the primary surface.

Primary intersecting surface
Primary intersecting surface.

Beginning intersecting surface
Beginning surface in the pattern.

Ending intersecting surface
Ending surface in the pattern.
(default is beginning intersecting surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Tolerance
Convergence tolerance.
(default is 0.001)

Notes

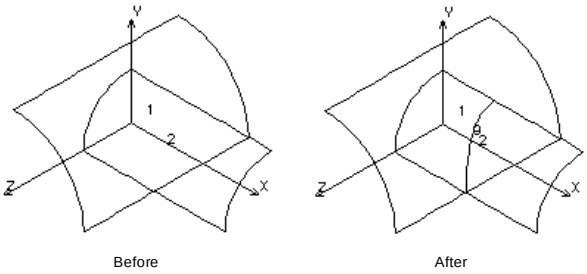
1. Entities can be picked by the mouse only if they are plotted on the screen.

2. None of the surfaces in the pattern are allowed to be parallel to, or overlapping with the primary surface.

Example

Graphic Example: CRINTSS

```
PT,1,0,0,-40,
PT,2,60,0,-40,
PT,3,0,60,-40,
PT,4,0,0,0,
PT,5,0,40,0,
PT,6,0,0,40,
CRARC,1,2,3,1,60,
CRARC,2,5,6,4,40,
SFEXTR,1,1,1,Z,140,
SFEXTR,2,2,1,X,120,
CRINTSS,1,2,2,1,0.0
  01,
```



CRONSF

Geometry > CURVES > GENERATION > Curve on Surface

The CRONSF command defines a curve on a specified surface in space between two keypoints that lie on the surface.

First point on surface
Beginning keypoint of the curve.

Second point on surface
Ending keypoint of the curve.

Underlying surface
Surface on which the curve is to be defined.

Tolerance
Error tolerance to be allowed between the curve and the surface. *(default is 0.001)*

Method used
Method to be used to create the curve.

= 0: Interpolation	the curve is created by interpolating 2 intermediate points at 1/3 distances in parametric coordinates.
= 1: End slopes	a base curve having end slopes as specified by the end points on the surface is created and is used to define the desired curve by error minimization. *(default is interpolation)*

Notes

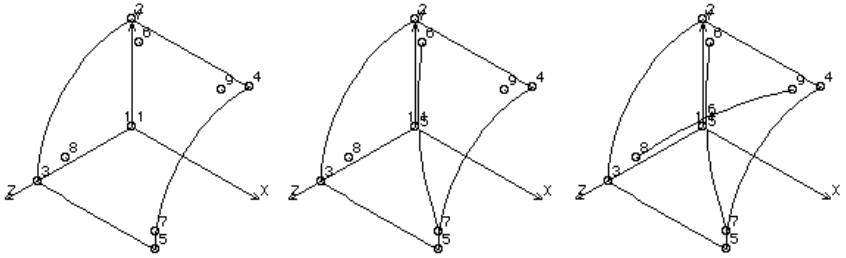
1. The second method is more accurate, but divergence is possible. The first method is less accurate but convergence is guaranteed.

2. The keypoints must lie on the surface.
3. Error tolerance may be increased to achieve convergence.

Example

Graphic Example: CRONSF

```
PT,1,0,0,0,
PT,2,0,80,0,
PT,3,0,0,80,
CRARC,1,2,3,1,80,
SFEXTR,1,1,1,X,100,
PTONSF,1,.1,.2,
PTONSF,1,.9,1.,
PTONSF,1,.8,.2,
PTONSF,1,.1,.9,
CRONSF,6,7,1,0.001,0,
CRONSF,8,9,1,0.001,1,
```



Original

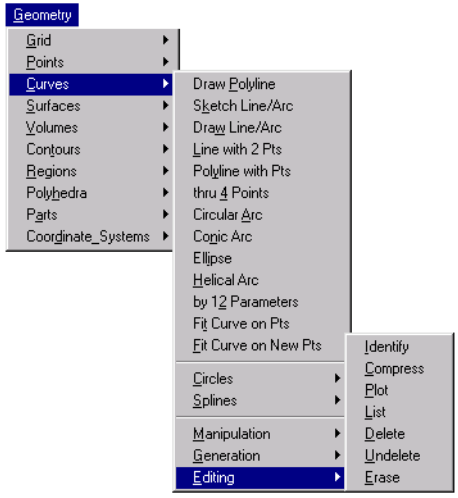
After the First CRONSF Command

After the Second CRONSF Command

▼EDITING Menu

Geometry > CURVES > EDITING

Figure 4-11 Editing Menu



CRIDENT

Geometry > CURVES > EDITING > Identify

The CRIDENT identifies a curve by highlighting it and displaying its label in dialogue area.

Curve
Select curve.

Notes:

1. Only entities plotted on the screen can be selected by the mouse.
2. Pressing the right button of the mouse keeps selecting curves in the neighborhood.

CRCOMPRESS

Geometry > CURVES > EDITING > Compress

The CRCOMPRESS command renumbers the curves in the pattern by removing all numbering gaps.

Beginning curve
Beginning curve in the pattern.
(default is 1)

Ending curve
Ending curve in the pattern.
(default is the highest curve number defined)

Example: `CRCOMPRESS, 1, 100`

This command renumbers all curves numbered 1 to 100 by removing any labeling gaps. If defined curves were numbered 11 to 50 and 76 to 100, this command renumbers curves such that curves 11 is renumbered as 1, curves 76 as 41, and curves 100 as 65.

CRPLOT

Geometry > CURVES > EDITING > Plot

The CRPLOT command plots a pattern of curves on the screen.

Beginning curve
Beginning curve in the pattern.
(default is 1)

Ending curve
Ending curve in the pattern.
(default is the highest curve defined)

Increment
Increment between curves in the pattern.
(default is 1)

Note
Use the STATUS1 table to control label plotting and color.

CRLIST

Geometry > CURVES > EDITING > List

The CRLIST command lists a pattern of curves. The command lists the type and keypoints of each curve in the pattern.

Beginning curve
Beginning curve in the pattern.
(default is 1)

Ending curve
Ending curve in the pattern.
(default is the highest curve defined)

Increment
Increment between curves in the pattern.
(default is 1)

Example: `CRLIST, 10, 20, 2`

This command lists labels and eypoints of curves 10, 12, 14, 16, 18 and 20.

CRDEL

Geometry > CURVES > EDITING > Delete

The CRDEL command deletes a pattern of curves. The deleted curves can be undeleted by using the CRUNDEL (**Geometry, Undelete, Curves**) command. Curves associated with higher entities cannot be deleted.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Note:
This command deletes curve addresses temporarily from the database so that it will be possible to undelete them using the CRUNDEL (**Edit, Delete, Curves**)command.

Example: `CRDEL, 10, 20, 2`

This command deletes curves 10, 12, 14, 16, 18 and 20 from the database.

CRUNDEL

Geometry > CURVES > EDITING > Undelete

The CRUNDEL command undeletes all curves in the specified pattern.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Note:
It is recommended that the CRUNDEL (**Edit, Undelete, Curves**) command be used immediately after an accidental deletion of curves and before the generation of any new curves.

Example: `CRUNDEL, 10, 20, 2`

This command undeletes curves 10, 12, 14, 16, 18 and 20.

CRERASE

Geometry > CURVES > EDITING > Erase

The CRERASE command erases plotted curves. The erased curves can be replotted using the CRPLOT (**Edit, Plot, Curves**) command.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Example: **CRERASE, 10, 20, 2**

This command erases curves 10, 12, 14, 16, 18 and 20 from the screen.



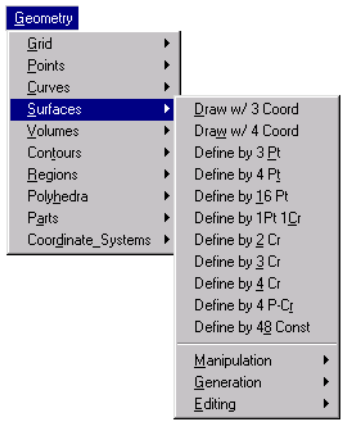

▶ SURFACES Menu

Geometry > SURFACES

This menu includes commands to generate, process and manipulate surfaces. Surfaces are 2D parametric entities, they can be planar or curved but must be bounded by 3 or 4 curves.

The short-form names of the commands in this menu start with "SF", the rest of the char-acters describe the function of the command. Surfaces can be meshed to generate 2D elements like SHELL3, SHELL4L, PLANE2D, TRIANG, MAG2D, FLOW2D, etc. Both parametric and automatic meshing are available for meshing surfaces.

Figure 4-12 Surface Menu



SF3CORD

Geometry > SURFACES > Draw w/ 3 Coord

The SF3CORD command defines a triangular planar surface by defining its corners and connecting them by straight lines. The corners are specified by their coordinates or conveniently snapped by the mouse on a predefined grid.

Surface
Surface label.
(default is the highest surface number defined + 1)

Keypoint 1 XYZ-Coordinate value
Coordinates of first keypoint.

Keypoint 2 XYZ-Coordinate value
Coordinates of second keypoint.

Keypoint 3 XYZ-Coordinate value
Coordinates of third keypoint.
(defaults are 0.0)

Notes

1. The active coordinate system must be Cartesian.
2. Only planar surfaces can be defined by this command.
3. A degenerate curve is created at the third keypoint.

To Create a planar triangular surface:

1. Click **Geometry, Grid, Plane** to define a plane.
2. Click **Geometry, Grid, Grid On** to create the desired grid on the active plane.

3. Click **Geometry, Surfaces, Draw W/3 Coord**.
4. If desired, click in the **Curve** field and enter a label. Otherwise, click the left button of the mouse to accept the default label (recommended).
5. Move the mouse pointer to the location of one of the corners and click the left button.

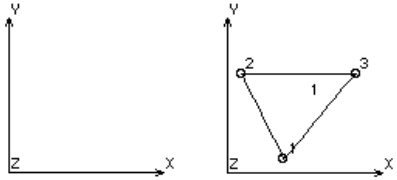
Note that you can type in other values to specify locations other than the grid intersections.

6. Move the mouse pointer to the location of the second corner and click the left button.
7. Move the mouse pointer to the location of the third corner and click the left button.
8. Click **OK**.

Example

Graphic Example: SF3CORD

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF3CORD,1,20,5,0,5,35,0,45,35,0,
```



Before After

SF4CORD

Geometry > SURFACES > Draw w/ 4 Coord

The SF4CORD command defines a quadrilateral planar surface by defining its corners and connecting them by straight lines. The corners are specified by their coordinates or conveniently snapped by the mouse on a predefined grid.

Surface
Surface label.
(default is the highest surface number defined +1)

Keypoint 1 XYZ-Coordinate value
Coordinates of first keypoint.

Keypoint 2 XYZ-Coordinate value
Coordinates of second keypoint.

Keypoint 3 XYZ-Coordinate value
Coordinates of third keypoint.

Keypoint 4 XYZ-Coordinate value
Coordinates of fourth keypoint.
(defaults are 0.0)

Notes:

1. The active coordinate system must be Cartesian.
2. Only planar surfaces with straight edges can be defined by this command.

To Create a planar quad surface:

1. Click **Geometry, Grid, Plane** to define a plane.
2. Click **Geometry, Grid, Grid On** to create the desired grid on the active plane.
3. Click **Geometry, Surfaces, Draw W/4 Coord**.
4. If desired, click in the **Curve** field and enter a label. Otherwise, click the left button of the mouse to accept the default label (recommended).
5. Move the mouse pointer to the location of one of the corners and click the left button.

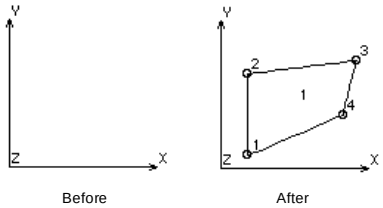
Note that you can type in other values to specify locations other than the grid intersections.

6. Move the mouse pointer to the location of each corner and click the left button.
7. Click **OK**.

Example

Graphic Example: SF4CORD

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,10,5,0,10,35,0,50,40,0,45
```



SF3PT

Geometry > SURFACES > Define by 3 Pt

The SF3PT command defines a triangular surface by joining 3 existing keypoints. If the 3 keypoints lie on a non-planar surface, you can specify it as an underlying surface. In this case, the new surface will lie on the undelying surface. If you do not specify an undelying surface, the command creates a planar surface by connecting the keypoints by straight lines.

Surface
Surface label.
(default is the highest surface number defined + 1)

Keypoint 1
First keypoint.

Keypoint 2
Second keypoint.

Keypoint 3
Third keypoint.

Underlying surface
Underlying surface.

= 0	the keypoints are connected by straight lines
= N	the keypoints are connected by curves lying on surface N

(default is 0)

Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. It is recommended that you use the PTONSF (**Geometry, Points, Generation, Point On Surface**) command to create keypoints on the underlying surface that will be used by this command. The underlying surface option will not work if the keypoints do not lie on it.
3. A degenerate curve is created at the third keypoint.

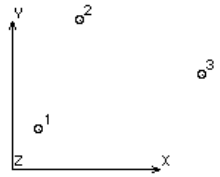
To define a surface by connecting 3 keypoints:

1. Click **Geometry, Surfaces, Define by 3 Pt**.
2. If desired, click in the **Curve** field and enter a label. Otherwise, click the left button of the mouse to accept the default label (recommended).
3. Select the 3 keypoints.
4. Specify an undelying surface, if desired.
5. Click **OK**.

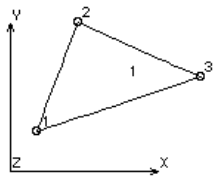
Example

Graphic Example 1: SF3PT

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,10,20,0,
PT,2,25,65,0,
PT,3,70,45,0,
SF3PT,1,1,2,3,0,
```



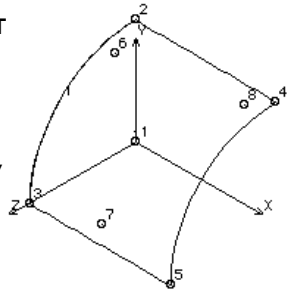
Before



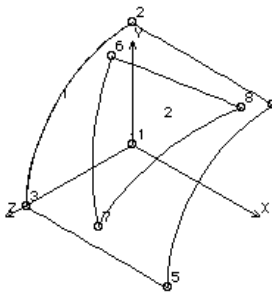
After

Graphic Example 2: SF3PT

```
PT,1,0,0,0,
PT,2,0,90,0,
PT,3,0.,0.,90,
CRARC,1,2,3,1,90,
SFEXTR,1,1,1,X,120,
PTONSF,1,.2,.1,
PTONSF,1,.9,.5,
PTONSF,1,.1,.9,
SF3PT,2,6,7,8,1,
```



Before



After

SF4PT

Geometry > SURFACES > Define by 4 Pt

The SF4PT command defines a quadrilateral surface by joining 4 existing keypoints. If the 4 keypoints lie on a non-planar surface, you can specify it as an underlying surface. In this case, the new surface will lie on the undelying surface. If you do not specify an undelying surface, the command creates a surface by connecting the keypoints by straight lines. The created surface can be planar or non-planar.

Surface

Surface label.
(default is the highest surface number defined + 1)

Keypoint 1
First keypoint.

Keypoint 2
Second keypoint.

Keypoint 3
Third keypoint.

Keypoint 4
Fourth keypoint.

Underlying surface
Underlying surface.

= 0	the keypoints are connected by straight lines
= N	the keypoints are connected by curves lying on surface N *(default is 0)*

Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. It is recommended that you use the PTONSF (**Geometry, Points, Generation, Point on Surface**) command to create keypoints on the underlying surface used by this command.

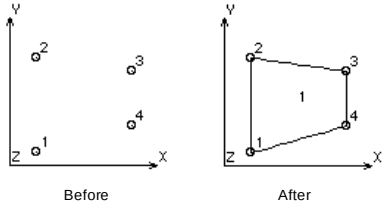
To define a surface by connecting 3 keypoints:

1. Click **Geometry, Surfaces, Define by 4 Pt**.
2. If desired, click in the **Curve** field and enter a label. Otherwise, click the left button of the mouse to accept the default label (recommended).
3. Select the 4 keypoints.
4. Specify an undelying surface, if desired.
5. Click **OK**.

Example

Graphic Example: SF4PT

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,10,5,0,
PT,2,10,40,0,
PT,3,45,35,0,
PT,4,45,15,0,
SF4PT,1,1,2,3,4,0,
```



SF16PT

Geometry > SURFACES > Define by 16 Pt

The SF16PT command creates a surface by fitting a bi-cubic polynomial to 16 existing keypoints. The command works best when the keypoints are uniformly spaced.

Surface
Surface label.
(default is the highest surface number defined + 1)

Keypoint (i)
ith keypoint. (i=1,2,.....,16)

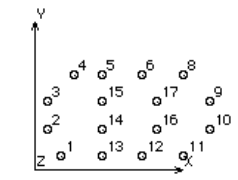
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. Keypoints 6, 7, 10 and 11 are on the interior of the surface and do not lie on its boundaries. All of the other keypoints are on the boundary.
3. Refer to the graphic example for the order in which the keypoints must be given.

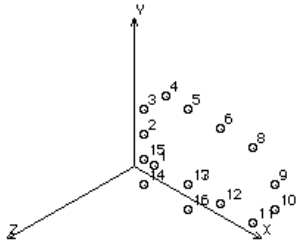
Example

Graphic Example: SF16PT

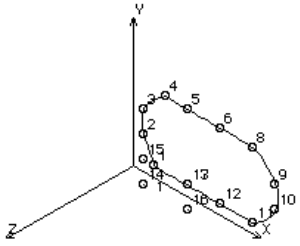
```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,10,5,0,
PT,2,5,15,0,
PT,3,5,25,0,
PT,4,15,35,0,
PT,5,25,35,0,
PT,6,40,35,0,
PT,7,45,25,0,
PT,8,55,35,0,
PT,9,65,25,0,
PT,10,65,15,0,
PT,11,55,5,0,
PT,12,40,5,0,
PT,13,25,5,0,
PTDEL,7,7,1,
PT,14,25,15,20,
PT,15,25,25,20,
PT,16,45,15,20,
PT,17,45,25,20,
VIEW,1,1,1,
VIEW,0,0,1,
SF16PT,1,1,2,3,4,13,14,15,5,12,16,17,6,11,10,9,8,
VIEW,1,1,1,
M_SF,1,1,1,4,8,12,1,1,
```



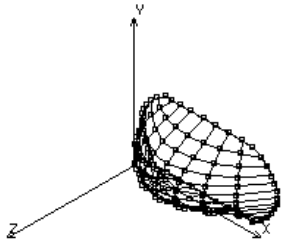
Planar View of the 16 Keypoints



3D View of the 16 Keypoints



3D View of the Generated Surface



3D View of the Surface After Meshing

Note

The surface is meshed for illustration purposese.

SFPTCR

Geometry > SURFACES > Define by 1Pt 1Cr

The SFPTCR command defines a triangular surface by specifying a base curve and a cornar (keypoint). If an underlying surface is specified, the ends of the curve and the keypoint will be joined by curves lying on the undelying surface. Otherwise they will be joined by straight lines. If you specify an underlying surface, the base curve and keypoint must lie on it. The keypoint may not be colinear with the curve.

Surface
Surface label.
(default is the highest surface number defined + 1)

Curve
Boundary curve label.

Keypoint
Vertex keypoint opposite to the boundary curve.

Underlying surface
Underlying surface.

= 0	ends of the curve are connected to the vertex keypoint by straight lines
= N	ends of the curve are connected to the vertex keypoint by curves lying on surface N *(default is 0)*

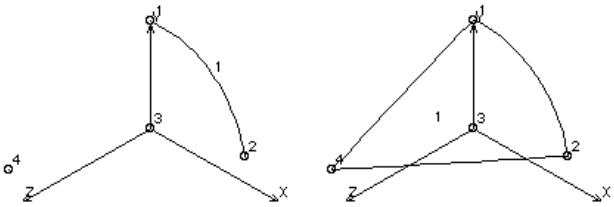
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. If an underlying surface is specified, it is recommended that you create the curve used in this command by the CRONSF (**Geometry, Curves, Generation, Curve on Surface**) command, and create the keypoint by the PTONSF command.
3. A degenerate curve is created at the vertex.

Example

Graphic Example: SFPTCR

```
PT,1,0,80,0,
PT,2,80,20,0,
PT,3,0,0,0,
CRARC,1,1,2,3,70
  .7107,
PT,4,-
  20.,10,100.,
SFPTCR,1,1,4,0,
```



SF2CR

Geometry > SURFACES > Define by 2 Cr

The SF2CR command defines a four-sided surface by specifying two edge curves. The other two edge curves of the surface are formed by joining the end points. GEOSTAR automatically aligns the curves to create a feasible surface. The curves must lie on the underlying surface if specified.

Surface
Surface label.
(default is the highest surface number defined + 1)

Curve 1
Boundary curve 1.

Curve 2
Boundary curve 2.

Underlying surface
Underlying surface.

= 0	ends of the curves are connected by straight lines
= N	ends of the curves are connected by curves lying on surface N

(default is 0)

Notes

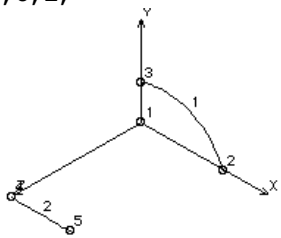
1. The Entities can be picked by the mouse only if they are plotted on the screen.
2. If an underlying surface is used, it is recommended that you create the curves used in this command by the CRONSF command to ensure that they lie on the underlying surface.
3. The command does not work if the specified curves do not lie on the specified underlying surface.

Example

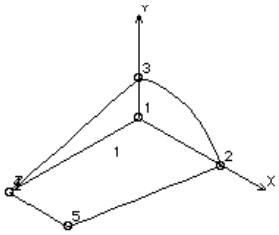
Graphic Example: SF2CR

```
PT,1,0,0,0,
PT,2,70,0,0,
PT,3,0,30,0,
CRARC,1,2,3,1,53.8516,
PT,4,0,0,110,
CREXTR,4,4,1,X,50,
```

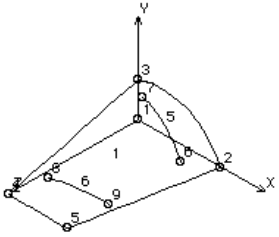
```
SF2CR,1,1,2,0,
PTONSF,1,.15,.2,
PTONSF,1,.8,.1,
PTONSF,1,.8,.8,
PTONSF,1,.1,.7,
CRONSF,6,7,1,0.00,0,
CRONSF,8,9,1,0.00,0,
SF2CR,2,5,6,1,
```



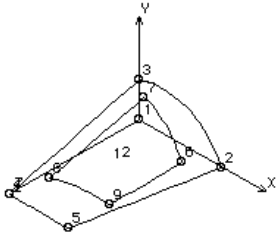
Before the First SF2CR



After the First SF2CR Command



Before the Second SF2CR Command



After the Second SF2CR Command

SF3CR

Geometry > SURFACES > Define by 3 Cr

The SF3CR command defines a 3-sided surface using three specified boundary curves. The three curves must form a closed loop. The curves must lie on the underlying surface if specified.

Surface
Surface label.
(default is the highest surface number defined + 1)

Curve 1
Boundary curve 1.

Curve 2
Boundary curve 2.

Curve 3
Boundary curve 3.

Underlying surface
Underlying surface.

= 0	surface equation is defined by the 3 curves
= N	surface is forced to lie on surface N
	(default is 0)

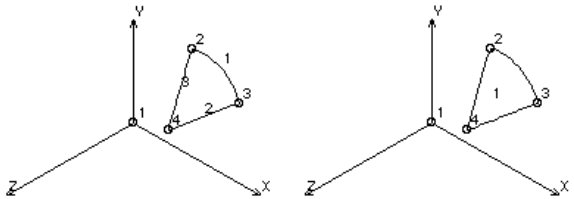
Notes

1. The Entities can be picked by the mouse only if they are plotted on the screen.
2. If an underlying surface is used, it is recommended that you create the curves used in this command by the CRONSF command to ensure that they lie on the underlying surface.
3. The command does not work if the specified curves do not lie on the specified underlying surface.
4. The curves must be selected so that each selected curve has a common key-point with the preceding curve.
5. A degenerate curve is also created.

Example

Graphic Example: SF3CR

```
PLANE,Z,0,1,
PT,1,0,0,0,
PT,2,50,80,0,
PT,3,90,60,0,
CRARC,1,2,3,1,31.
  6228,
CRPCORD,2,90,60,0
  ,30,10,0,50,80,0
  ,50,80,0,
SF3CR,1,1,2,3,0,
```



SF4CR

Geometry > SURFACES > Define by 4 Cr

The SF4CR command defines a 4-sided surface using four specified boundary curves. The four curves must form a closed loop. The curves must lie on the underlying surface if specified.

Surface
Surface label.
(default is the highest surface number defined + 1)

Curve 1
Boundary curve 1.

Curve 2
Boundary curve 2.

Curve 3
Boundary curve 3.

Curve 4
Boundary curve 4.

Underlying surface
Underlying surface.
= 0 surface equation is defined by the 4 curves
= N surface is forced to lie on surface N
(default is 0)

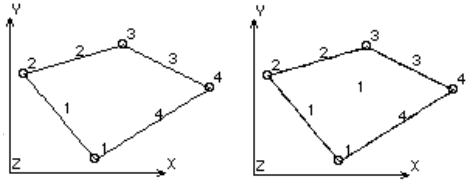
Notes

1. The Entities can be picked by the mouse only if they are plotted on the screen.
2. It is recommended that you create the curves used in this command by the CRONSF command to ensure that they lie on the underlying surface.
3. The command does not work if the specified curves do not lie on the specified underlying surface.
4. The curves must be selected so that each selected curve has a common key-point with the proceeding curve.

Example 1

Graphic Example: SF4CR

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRPCORD,1,30,5,0,5,35,0,40,
SF4CR,1,1,2,3,4,0,
```



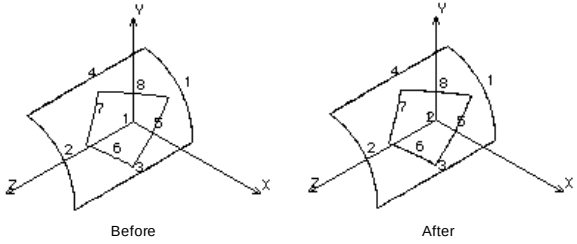
Note

The effect of the SF4CR command is the creation of surface 2 which is a plane surface in Example 1 and a curved surface in Example 2.

Example 2

Graphic Example 2: SF4CR

```
PT,1,0,0,0,
PT,2,50,10,0,
PT,3,10,60,0,
CRARC,1,2,3,1,45.2
  769,
SFEXTR,1,1,1,Z,100
  ,
PTONSF,1,.5,.1,
PTONSF,1,.1,.5,
PTONSF,1,.5,.8,
PTONSF,1,.85,.5,
CRONSF,6,7,1,0.001,0,
CRONSF,7,8,1,0.001,0,
CRONSF,8,9,1,0.001,0,
CRONSF,9,6,1,0.001,0,
SF4CR,2,5,6,7,8,1,
```



SF4PCR

Geometry > SURFACES > Define by 4 P-Cr

The SF4PCR command defines a four-sided surface that passes through 4 pre-defined curves. The boundaries of the surface are determined by the ends of the curves. The command works best when the curves are equally spaced. The curves are used to define a bi-cubic equation for the surface. An underlying surface can be specified.

Surface
Surface number.
(default is the highest surface number defined + 1)

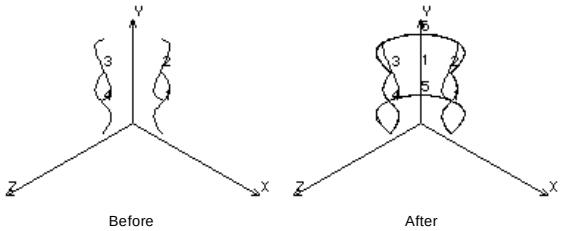
Curve (i)
ith curve. (i = 1,2,3 and 4)

Underlying surface
Underlying surface.
= 0 surface equation is defined by the 4 curves
= N surface is forced to lie on surface N
(default is 0)

Example

Graphic Example: SF4PCR

```
PT,1,25,5,0,
PT,2,20,20,0,
PT,3,30,35,0,
PT,4,25,50,0,
CR4PT,1,1,2,3,4,
CRGEN,3,1,1,1,1,0,9
  0,0,
VIEW,1,1,1,
SF4PCR,1,1,2,3,4,0,
```



SFGFORM

Geometry > SURFACES > Define by 48 Const

The SFGFORM command creates a surface by specifying its geometric parameters. A total of 48 parameters are needed to uniquely define a surface in space in this form. The parameters define a 3- or 4-sided surface by defining its corners, tangents, and twist angles. The command is not intended for on-line use. It is designed to minimize the time needed to recreate surfaces in large models. The saving is achieved by storing 48 parameters to define each surface of the final geometry. The command is internally used by the GFORM_OUT command to write the final model in a compact geometric form (neutral form) that eliminates the need to recreate intermediate geometrical entities.

Surface
Surface label.
(default is the highest surface label defined + 1)

Surface equation P(1,j,k)
k^{th} component of the j^{th} corner of the surface. (total of 12 parameters)

Surface equation P(2,j,k) and Surface equation P(3,j,k)
Parameters to define tangents to the surface at the corners (24 parameters)

Surface equation P(4,j,k)
Parameters to define needed twist angles at the corners (12 parameters)

Notes

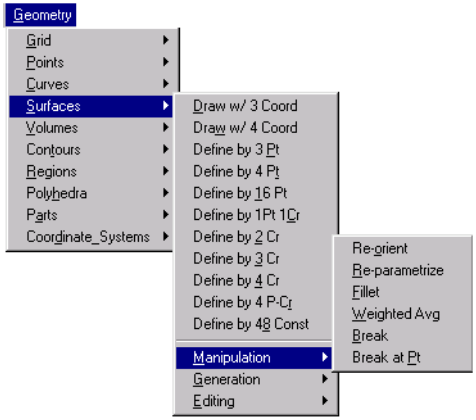
1. The GFORM_OUT (**Control, Utility, Create GFM File**) command internally implements this command to store the 48 parameters needed to regenerate each surface of the final model.
2. Use of the GFORM_OUT (**Control, Utility, Create GFM File**) command results in the creation of a neutral-type file containing all existing curves, surfaces and/or volumes in a compact form.

▼MANIPULATION Menu

Geometry > SURFACES > MANIPULATION MENU

This menu contains commands to manipulate existing surfaces.

Figure 4-13 Manipulation Menu



SFREORNT

Geometry > SURFACES > MANIPULATION MENU > Re-Orient

The SFREORNT command reverses the direction of the first parametric curve for all surfaces in the specified pattern. The second parametric curve is changed accordingly. Elements associated with the surfaces in the pattern are reoriented only if default meshing for surfaces is active. The command is useful in reorienting the top and bottom faces of shell elements for proper averaging of stresses.

Beginning surface
Beginning surface label in the pattern.

Ending surface
Ending surface label in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

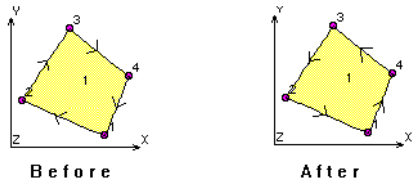
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. Default meshing can be activated using command ACTDMESH (**Control, Activate, Default Meshing**) or from the STATUS1 Table.
3. Automatic shell alignment is available by the ALIGNSHELL (**Meshing, Elements, Align Shell Elements**) command.

Example

Graphic Example: SFREORNT

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,1,40,5,0,5,20,0,25,
50,0,50,30,0,
SFREORNT,1,1,1,
```



SFREPAR

Geometry > SURFACES > MANIPULATION MENU > Re-Parametrize

The SFREPAR command replaces the current first parametric curve of a surface by the specified curve. The specified curve must be an edge of the surface.

Surface
Surface label.

Curve
Curve label.

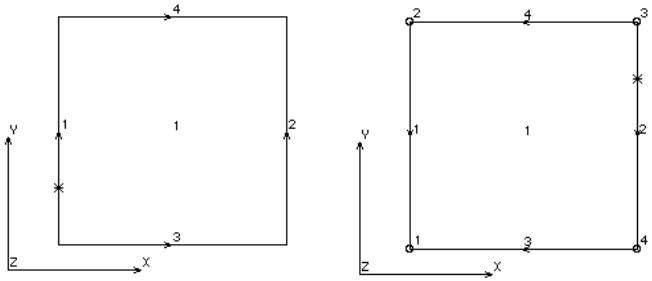
Notes

1. The entities can be picked by the mouse.
2. The second parametric curve of the specified surface is changed accordingly.

Example

Graphic Example: SFREPAR

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,1,10,5,0,10,50,0,55,50,0,55,5,0,
ACTMARK,CR,1
ACTMARK,SF,1
SFREPAR,1,2,
```



SFFILLET

Geometry > SURFACES > MANIPULATION MENU > Fillet

The SFFILLET command creates a circular surface fillet of a specified radius between 2 surfaces. An option to trim the original surfaces is provided.

Surface
Surface label.
(default is the highest surface number defined + 1)

Surface 1
First surface label.

Surface 2
Second surface label.

Radius of fillet
Radius of fillet to be created.

Trim flag
Flag to trim the filleted surfaces.

= 0: Do not trim	do not trim original surfaces
= 1: Trim	trim original surfaces *(default is trim)*

Original surface keeping flag
Original surfaces keeping flag if trimming option is selected.

= 0: Do not keep	do not keep original surfaces unless part of a higher entity
= 1: Keep	always keep original surfaces *(default is do not keep)*

Tolerance
Tolerance to define the surface fillet.
(default is 1.0E-6)

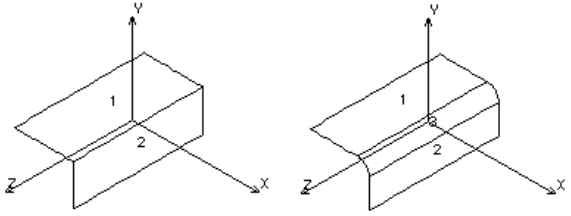
Notes

1. New keypoints at the centers of the fillets and at their ends are created. Other keypoints are created on the fillets if the angle is more than 90 degrees. If a fillet angle is more than 90 degrees, the fillet is broken into two curves.
2. The angle of the fillet is more than 90 degrees when the angle between the two surfaces is less than 90 degrees.
3. The 'original surface keeping flag' lets you keep the original surfaces even if they are not part of higher entities. A surface is always preserved if it is a part of higher entities.

Example

Graphic Example: SFFILLET

```
PLANE,Z,0,1,
CRPCORD,1,10,55,0
  ,60,55,0,60,20,0
  ,60,20,0,
SFEXTR,1,2,1,Z,11
  0,
SFFILLET,3,1,2,10
  ,1,0,
```



Before After

SFWAVRG

Geometry > SURFACES > MANIPULATION MENU > Weighted Avg

The SFWAVRG command creates a surface by the weighted averaging of two specified surfaces.

Surface
Surface number.
(default is the highest surface number defined + 1)

Surface 1
First surface label.

Surface 2
Second surface label.

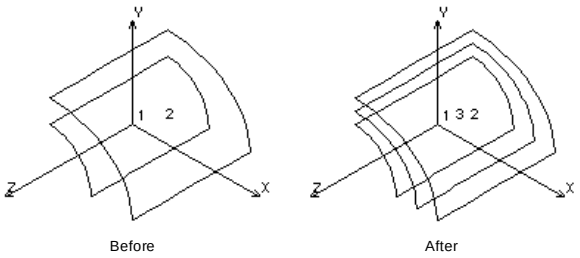
Ratio
Ratio of weight for the first surface to that of the second surface. (must be between 0.0 and 1.0)
(default is 0.5)

Note
Entities can be picked by the mouse only if they are plotted on the screen.

Example

Graphic Example: SFWAVRG

```
PLANE,Z,-30,1,
PT,1,0,0,-30,
PT,2,0,35,-30,
PT,3,35,0,-30,
CRPCIRCLE,1,1,3,35,
  90,1,
PT,4,70,0,-30,
PT,5,0,55,-30,
CRELLIPSE,2,4,5,1,1
  ,
SFEXTR,1,2,1,Z,100,
SFWAVR,3,1,2,0.5,
```



SFBRK

Geometry > SURFACES > MANIPULATION MENU > Break

The SFBRK command breaks a surface into two or four surfaces by specifying the coordinates of a point on the surface in the parametric axes.

Surface
Surface to be broken.

Parametric coordinate u
First parametric coordinate of the breaking point.
(must be between 0.0 and 1.0)
(default is 0.5)

Parametric coordinate v
Second parametric coordinate of the breaking point.
(must be between 0. and 1.)
(default is 0.5)

Original surface keeping flag
Original surface keeping flag.
= 0: Do not keep — delete the original surface
= 1: Keep original — keep the original surface
(default is do not keep)

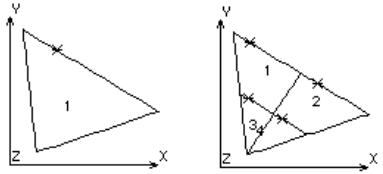
Notes
1. The surface can be picked by the mouse if is plotted on the screen.

2. It is recommended to use the STATUS1 table to turn the plotting of the orientation of surfaces (Mark Column). This resulte in displaying an asterisk (*) on the first parametric curve of each surface near the origin (plotted after the command has been issued).
3. The original surface is always preserved if it is part of a higher entity regardless of the value assigned to keep. Otherwise, the original surface is deleted unless the 'original surface keeping flag' is set to keep original.

Examples

Graphic Example 1: SFBRK

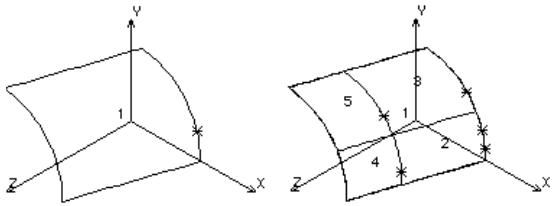
```
PLANE,Z,0,1,
VIEW,0,0,1,
SF3CORD,1,5,50,0,55,20,0,10,5,0,
SFBRK,1,.5,0.5,0,
```



Before After

Graphic Example 2: SFBRK

```
PT,1,0,0,0,
PT,2,60,0,0,
PT,3,10,60,0,
CRARC,1,2,3,1,55.226
  8,
CRGEN,1,1,1,1,0,-
  10,20,110,
SF2CR,1,1,2,0,
SFBRK,1,.4,.6,1,
```



Before After

SFPTBRK

Geometry > SURFACES > MANIPULATION MENU > Break at Pt

The SFPTBRK command breaks a surface at a specified keypoint. If the specified keypoint lies on an edge, two surfaces will be created, otherwise four surfaces will be created.

Surface
Surface to be broken.

Keypoint
Breaking keypoint.

Tolerance
Tolerance of normal distance between the keypoint and the surface.
(default is 0.0001)

Original surface keeping flag
Original surface keeping flag.

= 0: Do not keep	delete the original surface
= 1: Keep original	keep the original surface
	(default is do not keep)

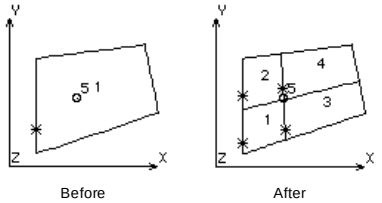
Notes

1. The surface and the keypoint can be picked by the mouse if they are plotted.
2. No surfaces are created if the selected keypoint does not lie on the surface, or if it coincides with a corner of the surface.
3. The original surface is always preserved if it is part of a higher entity regardless of the value assigned to keep.

Example

Graphic Example: SFPTBRK

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,1,10,5,0,10,40,0,50,45
  ,0,55,20,0,
PT,5,25,25,0,
SFPTBRK,1,5,0.0001,0,
```

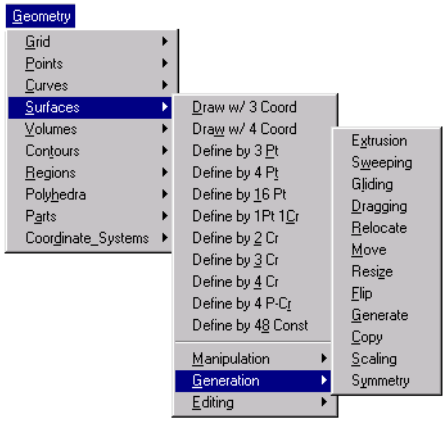


▼GENERATION Menu

Geometry > SURFACES > GENERATION

This menu contains commands to generate surfaces using existing surfaces through various operations. Surfaces are also generated by extrusion, sweeping, gliding and dragging curves.

Figure 4-14 Generation Menu



SFEXTR

Geometry > SURFACES > GENERATION > Extrusion

The SFEXTR command defines surfaces by extruding curves along a specified axis of the currently active coordinate system by a specified length. If default meshing

for surfaces is active, associated 1-D elements will be extruded to generate 2D elements .

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Axis symbol
Axis symbol (X, Y, Z).
(default is Z)

Value
Length of extrusion.

Number of layers of elements
Number of element layers in the direction of extrusion. Prompted only if default meshing for surfaces is active.
(default is 6)

Original mesh update flag
Flag for updating the original 1D mesh.

= -1: Transfer	transfer 1-D elements to the end of the extrusion
= 0: Keep	keep the original 1-D mesh as is
= 1: Delete	delete the original 1-D mesh
	(default is to delete original mesh)

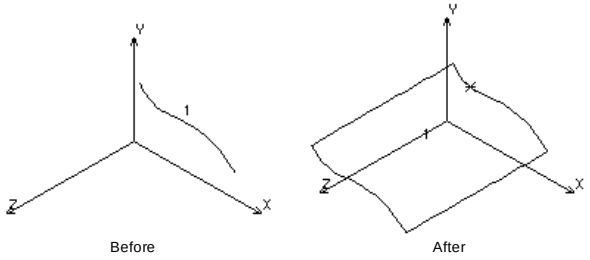
Notes

1. For proper association of nodes and elements with the generated geometric entities, do not merge nodes prior to using this command.
2. Make sure to activate a 2-D element group before issuing this command.

Example

Graphic Example: SFEXTR

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,5,45,0,
PT,2,25,20,0,
PT,3,50,55,0,
PT,4,85,20,0,
CRBEZIER,1,1,2,3,
  4,
VIEW,1,1,1,
SFEXTR,1,1,1,Z,12
  0,
```



SFSWEEP

Geometry > SURFACES > GENERATION > Sweeping

The SFSWEEP command creates surfaces by sweeping curves about a specified axis of the currently active Cartesian coordinate system by a specified angle. The number of surfaces generated by sweeping a curve is equal to the number of segments specified. For reasonable accuracy of arc representation, one segment per 90 degrees is recommended. A smaller angle is suggested for better accuracy. Associated 1-D elements are also swept to generate 2D elements if default meshing for surfaces is active.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Rotation/Sweep axis
Axis symbol (X, Y, Z).
(default is Y)

Angle of the arc
Angle of sweep in degrees.
(default is 360 degrees)

Number of segments
Number of segments (circular arcs).
(default is 1 segment for each 90 degrees)

Number of elements per segment
Number of elements per segment in the sweep direction. Prompted only if default meshing for surfaces is active.
(default is 6)

Original mesh update flag
Flag for updating the original 1-D mesh.

= -1: Transfer	transfer the 1-D elements to the end of the sweep
= 0: Keep	keep the original 1-D mesh
= 1: Delete	delete the original 1-D mesh
	(default is to delete original mesh)

Notes

1. For proper association of nodes and elements with the generated geometric entities, do not merge nodes prior to issuing this command.
2. Make sure to activate a 2-D element group before issuing this command.

Examples

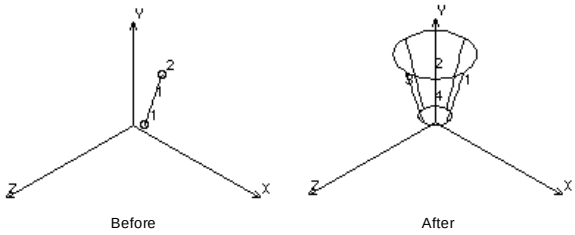
```
ACTDMESH, SF, 0,,,,
```

```
SFSWEEP, 1, 3, 1, X, 360.0, 4,
```

The above commands generate 12 new surfaces by sweeping curves 1, 2 and 3 about the x-axis the of the active coordinate system which must be Cartesian. Four surfaces are generated from each curve as it is swept 360 degrees. No 2-D elements are generated. The original 1D mesh is deleted.

Graphic Example: SFSWEEP

```
PLANE,Z,0,1,
VIEW,0,0,1,
CR2CORD,1,10,5,0,2
  5,50,0,
VIEW,1,1,1,
SFSWEEP,1,1,1,Y,36
  0,4,
```



SFGLIDE

Geometry > SURFACES > GENERATION > Gliding

The SFGLIDE command defines surfaces generated by gliding a pattern of curves along a profile. The curves in the pattern keep their original orientation as they glide along the profile curves, i.e. each curve moves parallel to its original position. Associated elements are also glided to generate 2-D elements if default meshing for surfaces is active.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Number of profile curves
Number of profile curves. Maximum is 6.
(default is 1)

Curve (i) of profile
ith profile curve. The curves must be specified in order and must form a continuous path. (i=1 - No. of profile curves)

Number of elements on curve (i)
Number of elements on the ith curve in the profile. Prompted only if default meshing of surfaces is active.
(default is 6)

Original mesh update flag
Flag for updating the original 1-D mesh.

= -1: Transfer	transfer the 1-D elements to the end of the profile
= 0: Keep	keep the original 1-D mesh
= 1: Delete	delete the original 1-D mesh

Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The profile curves must have a continuous first derivative (continuous slope). Fillets can be created to smoothen the profile if needed.
3. The locus of every single point on the gliding curves is exactly identical to the profile.
4. The GLIDE and DRAG operations are equivalent when the profile is a straight line.
5. The total number of surfaces generated is equal to the number of the profile curves multiplied by the number of curves in the pattern.
6. For proper association of nodes and elements with the generated geometric entities, do not merge nodes prior to issuing this command.

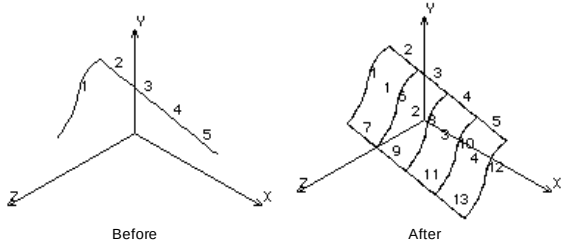
Examples

```
ACTDMESH, SF, 0

SFGLIDE, 1, 3, 1, 5, 11, 12, 13, 14, 15, 0
```

The above commands define 15 surfaces generated by gliding each of curves 1, 2, and 3 parallel to along the profile defined by curves 11, 12, 13, 14 and 15. No 2D elements are generated. The original 1D mesh is preserved.

Graphic Example: SFGLIDE

```
PLANE,X,-30,1,
PT,1,-30,40,0,
PT,2,-30,30,15,
PT,3,-30,15,20,
PT,4,-30,0,35,
CR4PT,1,1,2,3,4,
PT,5,70,20,0,
CRLINE,2,1,5,
CRBRK,2,2,1,4,0,
SFGLIDE,1,1,1,4,2,
  3,4,5,
```



SFDRAG

Geometry > SURFACES > GENERATION > Dragging

The SFDRAG command defines surfaces generated by dragging a pattern of curves along a profile defined by up to 6 curves. The surfaces are generated such that the

angles between the profile curves and any curve in the pattern remain unchanged as the curve is dragged along the profile. Associated elements are also dragged to generate 2-D elements if default meshing for surfaces is active.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Number of profile curves
Number of profile curves. Maximum is 6.
(default is 1)

Curve (i) of profile
i^{th} profile curve. The curves must be specified in order and must form a continuous path.

Number of elements on curve (i)
Number of elements on the i^{th} curve in the profile. Prompted only if default meshing for surfaces is active.
(default is 6)

Original mesh update flag
Flag for updating the original 1-D mesh.

= -1: Transfer	transfer 1-D elements to the end of the glide
= 0: Keep	keep the original 1-D mesh
= 1: Delete	delete the original 1-D mesh
	(default is delete original mesh)

Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The profile curves must have a continuous first derivative (continuous slope). Fillets can be created to smoothen the profile if needed.
3. The profile curves must be such that a curve does not intersect itself as it is dragged.
4. The GLIDE and DRAG are equivalent operations when the profile is a straight line.
5. The number of surfaces generated is equal to the number of the profile curves multiplied by the number of curves in the pattern.
6. For proper association of nodes and elements with the generated geometric entities, do not merge nodes prior to issuing this command.

Examples

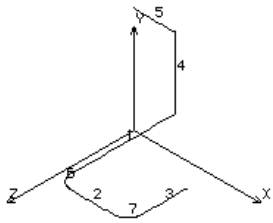
```
ACTDMESH, SF, 0
```

```
SFDRAG, 1, 3, 1, 5, 11, 12, 13, 14, 15, 0,
```

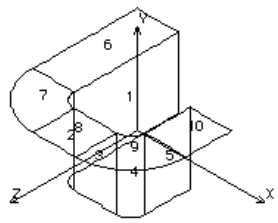
The above commands define 15 surfaces generated by dragging curves 1, 2, and 3 along curves 11, 12, 13, 14 and 15. Note that if curve 15 is normal to curve 1, for example, then the surfaces generated from curve 1 are defined such that this curve is normal to the profile at all times during the drag. No 2-D elements are generated. The original 1-D mesh is preserved.

Graphic Example: SFDRAG

```
PLANE,Z,0,1,
PT,1,35,30,0,
CREXTR,1,1,1,Z,100,
CREXTR,2,2,1,X,60,
CREXTR,3,3,1,Z,-50,
CREXTR,1,1,1,Y,60,
SCALE;
PT,6,0,90,0,
CRLINE,5,5,6,
CRFILLET,6,1,2,12,1
  ,0,
CRFILLET,7,2,3,12,1,0,
PTERASE,7,10,3,
SFDRAG,4,5,1,5,1,6,2,7,3,
```



Before



After

SFRELOC

Geometry > SURFACES > GENERATION > Relocate

The SFRELOC command relocates a pattern of surfaces by a specified translation and/or rotation in the currently active coordinate system. A surface must be free from association with higher entities for a successful relocation. This command does not generate any new surfaces. The SFGEN (**Geometry, Surfaces, Generation, Generate**) command can be used to generate new surfaces from existing ones.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
The increment between surfaces.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement

The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

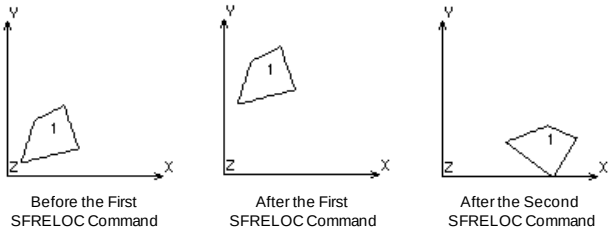
Notes

1. Surfaces can be picked by the mouse.
2. If a surface in the specified pattern is part of a higher entity, that surface is not relocated.
3. The active coordinate system must be Cartesian.

Example

Graphic Example: SFRELOC

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,1,5,5,0,
  25,10,0,20,25,0
  ,10,20,0,
SFRELOC,1,1,1,0,
  0,20,0,
SFRELOC,1,1,1,1,
  0,0,-50,
```



SFMOVE

Geometry > SURFACES > GENERATION > Move

The SFMOVE command moves a pattern of surfaces from the current coordinate system to the destination coordinate system. This command does not generate any new surfaces. The SFCOPY (**Geometry, Surfaces, Generation, Copy**) command can be used to generate new surfaces from existing ones.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces.
(default is 1)

Destination coordinate
Destination coordinate system label.
(default is 0)

Notes

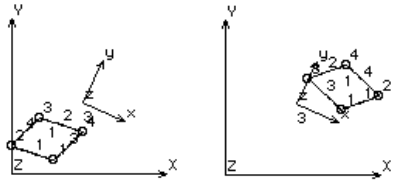
1. Surfaces can be picked by the mouse.
2. If a surface in the specified pattern is part of a higher entity, that surface is not moved.
3. Both coordinate systems must be Cartesian.

Example: `SFMOVE, 1, 6, 1, 3`

The above command moves surfaces 1 through 6 to local coordinate system 3 with the same coordinates.

Graphic Example: SFMOVE

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,1,15,5,0,0,10,0,10,20,
  0,25,15,0,
CSANGL,3,0,25,25,0,0,0,-25,
ACTSET,CS,0,
SFMOVE,1,1,1,3,
```



Before After

SFRESIZ

Geometry > SURFACES > GENERATION > Resize

The SFRESIZ command resizes a pattern of existing surfaces, by scaling and translating/rotating them relative to the currently active coordinate system. This command does not generate any new surfaces. The SFSCALE (**Geometry, Surfaces, Generation, Scaling**) command can be used to generate new surfaces by scaling existing ones.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
The increment between surfaces.
(default is 1)

Generation flag
= 0: Translation only by translations only
= 1: Rotation only by rotations only
= 2: Both by translations and rotations
(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X, Y, and Z directions.
(default is 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate sys-

tem. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

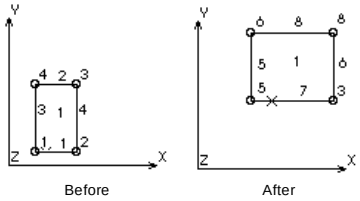
Notes

1. Surfaces can be picked by the mouse.
2. If a surface in the specified pattern is part of a higher entity, that surface is not resized.
3. The active coordinate system must be Cartesian.

Examples

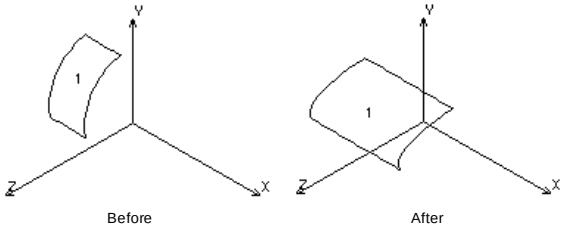
Graphic Example 1: SFRESIZ

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,1,10,5,0,25,5,0,25,3
  0,0,10,30,0,
SFRESIZ,1,1,1,0,2,1,1,0,20,0
  ,
```



Graphic Example 2: SFRESIZ

```
PLANE,X,-40,1,
PT,1,-40,0,0,
PT,2,-40,45,0,
PT,3,-40,0,30,
CRARC,1,2,3,1,38.24
  26,
SFEXTR,1,1,1,X,30,
SFRESIZ,1,1,1,0,2.5
  ,.5,1.5,50,0,0,
```



SFFLIP

Geometry > SURFACES > GENERATION > Flip

The SFFLIP command flips a pattern of surfaces about a specified plane in the current coordinate system. An offset can also be specified with the flipping. his command does not generate any new surfaces. The SFSYM (**Geometry, Surfaces, Generation, Symmetry**) command can be used to generate new surfaces from existing ones.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.

(default is beginning surface)

Increment

Increment between surfaces.
(default is 1)

Rotation/Sweep axis

Axis representing normal to plane about which pattern of surfaces are to be flipped.

= X	about y-z plane
= Y	about z-x plane
= Z	about x-y plane *(default is Z)*

Reverse direction flag

Reverse direction flag.

= 0: No	do not reverse direction of first parametric curve of each surface
= 1: Yes	reverse direction of first parametric curve of each surface *(default is reverse direction)*

Offset

Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

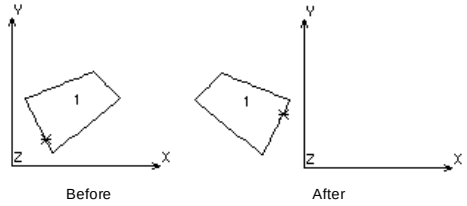
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. If a surface in the specified pattern is part of a higher entity, that surface is not flipped.
3. A specified offset is applied to all keypoints of the specified surfaces.
4. The active coordinate system must be Cartesian.

Examples

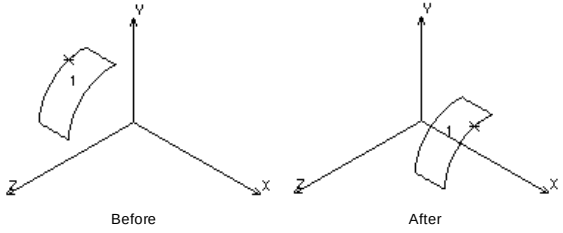
Graphic Example 1: SFFLIP

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,1,15,5,0,5,25,0,3
  0,35,0,40,25,0,
SFFLIP,1,1,1,X,1,0,
```



Graphic Example 2: SFFLIP

```
PLANE,X,-40,1,
PT,1,-40,0,0,
PT,2,-40,35,0,
PT,3,-40,0,40,
CRARC,1,2,3,1,37.58
  32,
SFEXTR,1,1,1,X,25,
SFFLIP,1,1,1,X,0,20
  ,
```



SFGEN

Geometry > SURFACES > GENERATION > Generate

The SFGEN command generates one or more patterns of surfaces from an existing pattern, by translating and/or rotating them about the currently active Cartesian coordinate system.

Generation number
Number of patterns to be generated. (must be > 0 for generation to occur)
(default is 1)

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
Magnitudes of the translation in the currently active coordinate system.
(defaults are 0.0)

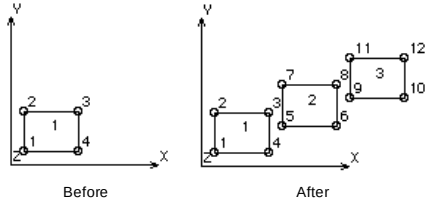
X, Y, Z-Rotation
Magnitudes of the rotations (degrees) in the currently active coordinate system.
(defaults are 0.0)

Note
The active coordinate system must be Cartesian.

Example

Graphic Example: SFGEN

```
VIEW,0,0,1,
PLANE,Z,0,1,
SF4CORD,1,5,5,0,5,20,0,25,2
  0,0,25,5,0,
SFGEN,2,1,1,1,0,25,10,0,
```



SFCOPY

Geometry > SURFACES > GENERATION > Copy

The SFCOPY command copies a pattern of surfaces from the active coordinate system to the destination coordinate system. Use the SFMOVE (**Geomtery, Surfaces, Generation, Move**) command to move existing surfaces without generating new ones.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces.
(default is 1)

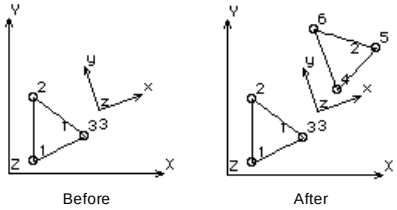
Destination coordinate
Destination coordinate system label.
(default is 0)

Note
Both coordinate systems must be Cartesian.

Example

Graphic Example: SFCOPY

```
PLANE,Z,0,1,
VIEW,0,0,1,
SF3CORD,1,10,5,0,10,30,0,30,1
  5,0
CSANGL,3,0,35,25,0,0,0,20,
ACTSET,CS,0,
SFCOPY,1,1,1,3,
```



SFSCALE

Geometry > SURFACES > GENERATION > Scaling

The SFSCALE command generates a pattern of surfaces from an existing pattern by scaling, translating or rotating them relative to the currently active coordinate system. The coordinates of all associated keypoints are multiplied by the specified scale factors.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
The increment between surfaces.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X, Y, and Z directions.
(default is 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included..
(defaults are 0.0)

Note
The active coordinate system must be Cartesian.

Example

```
SFSCALE, 1,4,1,0,0.0,0.5,0.0,5.0,0.0,0.0
```

The above command generates 4 new surfaces from the existing surfaces 1 through 4 by applying a scale factor of 0.5 in the Y-direction and translating them by 5.0 units in the X-direction.

SFSYM

Geometry > SURFACES > GENERATION > Symmetry

The SFSYM command generates a pattern of surfaces symmetric to the specified pattern about a specified plane in the current coordinate system. An offset can also be specified. Use the SFFLIP (**Geomtery, Surfaces, Generation, Flip**) command to flip existing surfaces without generating new ones.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.

(default is beginning surface)

Increment

Increment between surfaces.
(default is 1)

Rotation/Sweep axis

Axis representing normal to plane about which pattern of surfaces are to be generated symmetrically.

= X	plane y-z
= Y	plane z-x
= Z	plane x-y *(default is Z)*

Reverse direction flag

Reverse direction flag.

= 0: No	do not change surfaces directions
= 1: Yes	change directions of the surfaces by reversing the direction of the first parametric curve of each surface *(default is reverse direction)*

Offset

Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

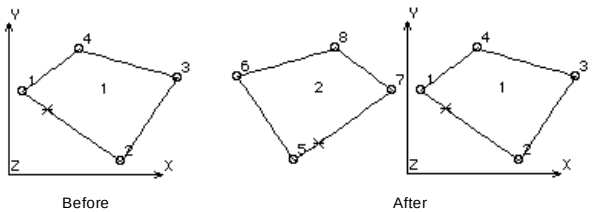
Notes

1. Surfaces can be picked by the mouse.
2. A specified offset is applied to all keypoints of the specified surfaces.
3. The active coordinate system must be Cartesian.

Example

Graphic Example: SFSYM

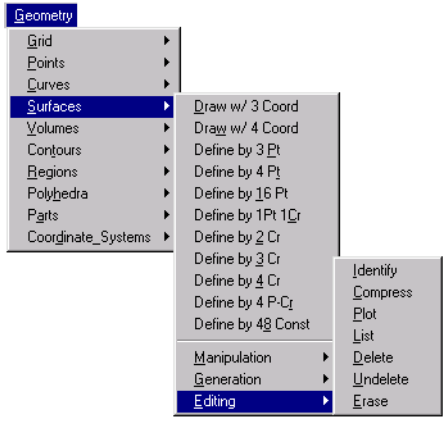
```
PLANE,Z,0,1,
VIEW,0,0,1,
SF4CORD,1,5,30,0,
  40,5,0,60,35,0,
  25,45,0,
SFSYM,1,1,1,X,1,0
  ,
```



▼EDITING Menu

Geometry > SURFACES > EDITING

Figure 4-15 Editing Menu



SFIDENT

Geometry > SURFACES > EDITING > Identify

The SFIDENT identifies a surface by highlighting its label and also displaying its label in dialogue area.

Surface
Select surface.

Notes:

1. Only entities plotted on the screen can be selected by the mouse.
2. Pressing the right button of the mouse keeps selecting surfaces in the neighborhood.

SFCOMPRESS

Geometry > SURFACES > EDITING > Compress

The SFCOMPRESS command renumbers the surfaces in the pattern by removing all numbering gaps.

Beginning surface
Beginning surface in the pattern.
(default is 1)

Ending surface
Ending surface in the pattern.
(default is the highest surface number defined)

Example

```
SFCOMPRESS, 1, 100
```

This command renumbers all surfaces numbered 1 to 100 by removing any labeling gaps. If defined surfaces were numbered 11 to 50 and 76 to 100, this command renumbers surfaces such that surface 11 is renumbered as 1, surface 76 as 41, and surface 100 as 65.

SFPLOT

Geometry > SURFACESURFACES > EDITING > Plot

The SFPLOT command plots a pattern of surfaces on the screen.

Beginning surface
Beginning surface in the pattern.
(default is 1)

Ending surface
Ending surface in the pattern.
(default is the highest surface defined)

Increment
Increment between surfaces in the pattern.
(default is 1)

Note
Use the STATUS1 table to control label plotting and color.

SFLIST

Geometry > SURFACES > EDITING > List

The SFLIST command lists a pattern of surfaces. The command lists the type and the associated curves of each surface in the pattern.

Beginning surface
Beginning surface in the pattern.
(default is 1)

Ending surface
Ending surface in the pattern.
(default is the highest surface defined)

Increment
Increment between surfaces in the pattern.
(default is 1)

Example

```
SFLIST, 10, 20, 2
```

This command lists labels and curves of surfaces 10, 12, 14, 16, 18 and 20.

SFDEL

Geometry > SURFACES > EDITING > Delete

The SFDEL command deletes a pattern of surfaces. The deleted surfaces can be undeleted by using the SFUNDEL (**Geometry, Undelete, Surfaces**) command. Surfaces associated with higher entities cannot be deleted.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Note
This command deletes surface addresses temporarily from the database so that it will be possible to undelete them using the SFUNDEL (**Edit, Delete, Surfaces**) command.

Example

```
SFDEL, 10, 20, 2
```

This command deletes surfaces 10, 12, 14, 16, 18 and 20 from the database.

SFUNDEL

Geometry > SURFACES > EDITING > Undelete

The SFUNDEL command undeletes all surfaces in the specified pattern.

Beginning surface
Beginning surface in the pattern.

Ending surface

Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Note
It is recommended that the SFUNDEL (**Edit, Undelete, Surfaces**) command be used immediately after an accidental deletion of surfaces and before the generation of any new surfaces.

Example:

```
SFUNDEL, 10, 20, 2
```

This command undeletes surfaces 10, 12, 14, 16, 18 and 20.

SFERASE

Geometry > SURFACES > EDITING > Erase

The SFERASE command erases plotted surfaces. The erased surfaces can be replotted using the SFPLOT (**Edit, Plot, Surfaces**) command.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Example:

```
SFERASE, 10, 20, 2
```

This command erases surfaces 10, 12, 14, 16, 18 and 20 from the screen.

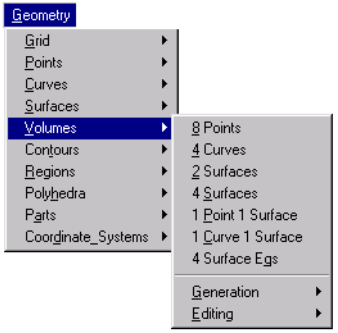



▶ VOLUMES Menu

Geometry > VOLUMES

This menu includes commands to generate, process and manipulate volumes. Volumes are 3-D parametric entities, which cannot have more than 6 surfaces and 12 curves. All the short names of commands in this menu start with "VL" . The rest of the characters describe the function of the command. Volumes can be meshed to generate brick or tetrahedral elements. PARTS are generalized 3-D geometric entities that can be meshed to generate tetrahedral elements (refer to the POLYHEDRA and PARTS menu).

Figure 4-16 Volumes Menu



VL8PT

Geometry > VOLUMES > 8 Points

The VL8PT command creates a volume from 8 keypoints. Each keypoint defines a corner.

Volume
Volume label.
(default is the highest volume number defined + 1)

Number of vertex points
Number of keypoints. (must be 8)

Vertex keypoint (i)
i^{th} vertex keypoint. (i=1,2,...,8)

Notes

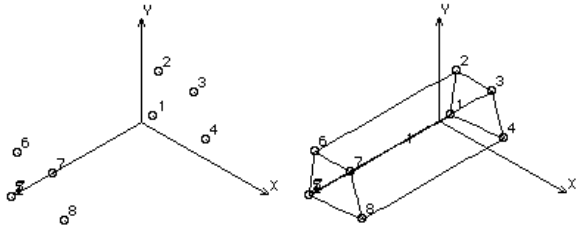
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. The number of vertex points argument is not actually needed nor used. It is only kept for consistency with earlier versions.

Example

Graphic Example: VL8PT

```
PLANE,Z,0,1,
PT,1,10,10,0,
PT,2,15,45,0,
PT,3,45,45,0,
PT,4,55,15,0,
PTGEN,1,1,4,1,0,0,0,120,
```

```
VL8PT,1,8,1,2,3,4,5,6,7,8,
```



Before After

VL4CR

Geometry > VOLUMES > 4 Curves

The VL4CR command defines a volume by specifying 4 edge curves. The volume is constructed by properly connecting the beginning keypoint of each curve to the beginning keypoints of two adjacent curves. The ending keypoints are connected similarly. However, an automatic alignment flag is provided to prevent crisscrossing and generate a feasible volume, if possible.

Volume
Volume label.
(default is the highest volume number defined +1)

Curve (i)
ith curve. (i=1,2,3,4)

Automatic alignment flag
Automatic alignment flag.

= 1: Yes	use automatic alignment
= 0: No	do not use automatic alignment
	(default is to use auto alignment)

Notes

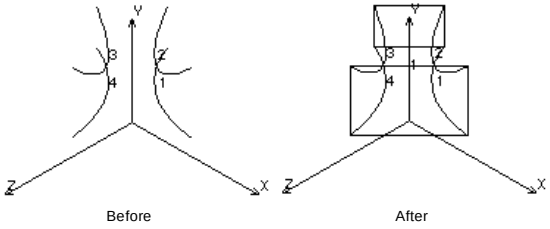
1. The Entities can be picked by the mouse only if they are plotted on the screen.
2. The curves must be specified in a cyclic order.
3. It is possible to use 3 curves to generate a volume by repeating a curve twice.
4. The automatic alignment flag, on by default, automatically aligns the curves to create a feasible volume.
5. It is possible to generate a pyramid shaped volume using curves that have a common keypoint.

Example

Graphic Example: VL4CR

```
PLANE,Z,0,1,
PT,1,30,70,0,
PT,2,20,45,0,
```

```
PT,3,25,25,0,
PT,4,50,15,0,
CR4PT,1,1,2,3,4,
CRGEN,3,1,1,1,1,0,9
  0,0,
VIEW,1,1,1,
VL4CR,1,1,2,3,4,0,
```



VL2SF

Geometry > VOLUMES > 2 Surfaces

The VL2SF command defines a six-sided volume by specifying two opposite surfaces. The volume is constructed by joining the beginning of the first parametric curve of one surface to that of the other. Two other edges are defined by connecting the ends of the parametric curves of the two surfaces. The other two corners of the surfaces, if any, are connected to form the fourth edge. An alignment flag is used to automatically make the necessary changes in orientation to define a feasible volume and prevent crisscrossing.

Volume
Volume label.
(default is the highest volume number defined + 1)

Surface 1
Boundary surface 1.

Surface 2
Boundary surface 2.

Automatic alignment flag
Automatic alignment flag.

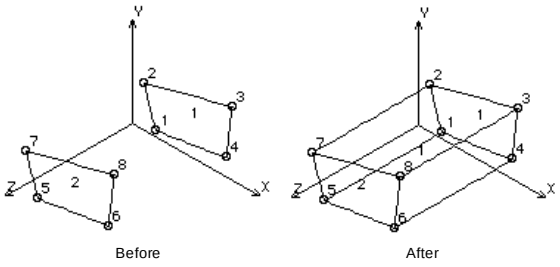
= 1: Yes	use automatic alignment
= 0: No	do not use automatic alignment *(default is to use auto alignment)*

Note
The surfaces can be picked by the mouse if they are plotted.

Example

Graphic Example: VL2SF

```
PLANE,Z,0,1,
SF4CORD,1,20,5,0,1
  0,35,0,85,55,0,80
  ,15,0,
SFGEN,1,1,1,1,0,0,
  0,100,
VL2SF,1,1,2,0,
```



VL4SF

Geometry > VOLUMES > 4 Surfaces

The VL4SF command defines a six-sided volume by specifying four adjacent surfaces that form a loop, leaving two opposite faces open. Two new surfaces are created. No new curves are formed. Each surface must have two common edges with the other surfaces. An automatic alignment flag is provided.

Volume label.
(default is the highest volume number defined + 1)

Surface 1
Boundary surface 1.

Surface 2
Boundary surface 2.

Surface 3
Boundary surface 3.

Surface 4
Boundary surface 4.

Automatic alignment flag
Automatic alignment flag.

= 1: Yes	use automatic alignment
= 0 No	do not use automatic alignment
	(default is to use auto alignment)

Notes

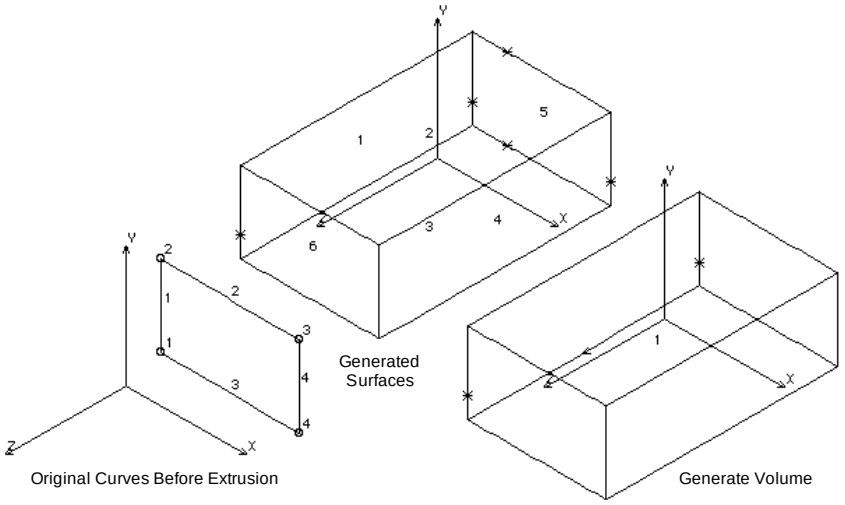
1. The surfaces can be picked by the mouse if they are plotted.
2. The orientation of the surfaces should be as shown in the graphic example, if the automatic alignment flag is not used.

Example

Graphic Example: VL4SF

```
PLANE,Z,0,1,
CRPCORD,1,15,20,0,15,55,0,75,55,0,75,55,0,
CRPCORD,3,15,20,0,75,20,0,75,55,0,75,55,0,
ACTMARK,SF,1
ACTMARK,VL,1
SFEXTR,1,4,1,Z,100,
```

VL4SF,1,1,2,4,3,1,



VLPTSF

Geometry > VOLUMES > 1 Point 1 Surface

The VLPTSF command defines a volume by specifying a keypoint and a surface. The volume is generated by connecting the keypoint to all of the keypoints of the surface.

Volume
Volume label.
(default is the highest volume number defined +1)

Keypoint
Vertex keypoint.

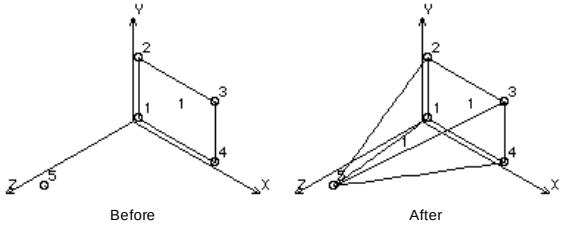
Surface
Surface label.

Note
The keypoint and the surface can be picked by the mouse if they are plotted.

Example

Graphic Example: VLPTSF

```
PLANE,Z,0,1,
SF4CORD,1,5,5,0,5,
  50,0,70,50,0,70,5
  ,0,
PT,5,35,25,110,
VLPTSF,1,5,1,
```



VLCRSF

Geometry > VOLUMES > 1 Curve 1 Surface

The VLCRSF command defines a volume by specifying a curve and a surface. An automatic alignment flag is provided to automatically align the curve and/or the surface to form a feasible volume. Otherwise, the volume is constructed by connecting the beginning and end of the curve to the beginning and end of the first parametric curve of the surface, respectively. The beginning and ending points of the curve are respectively connected to the ending and beginning points of the second parametric curve of the surface.

Volume
Volume label.
(default is the highest volume number defined +1)

Curve
Curve label.

Surface
Surface label.

Automatic alignment flag
Automatic alignment flag.

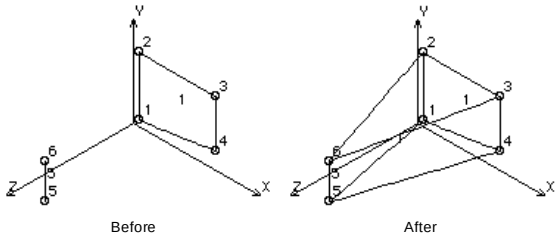
= 1: Yes	use automatic alignment
= 0: No	do not use automatic alignment *(default is to use auto alignment)*

Note
The curve and the surface can be picked by the mouse if they are plotted.

Example

Graphic Example: VLCRSF

```
PLANE,Z,0,1,
SF4CORD,1,5,5,0,5,55,
  0,70,55,0,70,15,0,
PT,5,35,15,110,
PT,6,35,45,110,
CRLINE,5,5,6,
VLCRSF,1,5,1,
```



VLGFORM

Geometry > VOLUMES > 4 Surface Eqs

The VLGFORM command creates a volume by specifying the geometric parameters of member surfaces. Forty-eight (48) parameters are needed for each surface. Two or three surfaces can be specified. The command is not intended for on-line use. The command is internally used by the GFORM_OUT (**Control, Utility, Create GFM File**) command to write the current model in a compact geometric format (also called neutral format).

Volume
Volume label.
(default is the highest volume label defined + 1)

Number of surfaces
Number of surfaces used to define the volume. (2 or 4)
(default is 2)

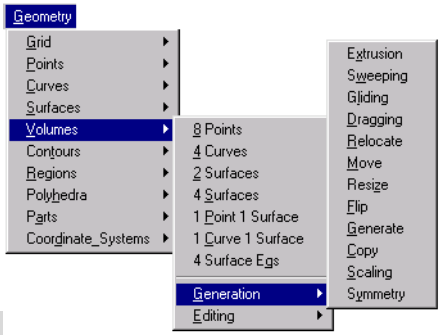
Volume surface equation (i) P(1,m,n)
Surface equations defined by 48 parameters.

▼GENERATION Menu

Geometry > VOLUMES > GENERATION

This menu contains commands to generate volumes from existing volumes through various operations. Volumes can also be generated by extruding, sweeping, gliding and dragging existing surfaces.

Figure 4-17 Generation Menu



VLEXTR

Geometry > VOLUMES > GENERATION > Extrusion

The VLEXTR command defines volumes by extruding surfaces along a specified coordinate axis of the currently active coordinate system by a specified length. Associated 2D elements are also extruded to generate 3-D elements if default meshing for volumes is active.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Axis symbol
Axis symbol (X, Y or Z).
(default is Z)

Value
Length of extrusion.

Number of layers of elements
Number of element layers in the direction of extrusion. Only prompted if default meshing for volumes is on.
(default is 6)

Original mesh update flag
Flag for handling the original 2D mesh.

= -1: Transfer	transfer mesh to the newly created surface or region at the end of the extrusion
= 0: Keep	keep the original 2D mesh
= 1: Delete	delete the original 2D mesh

(default is delete original mesh)

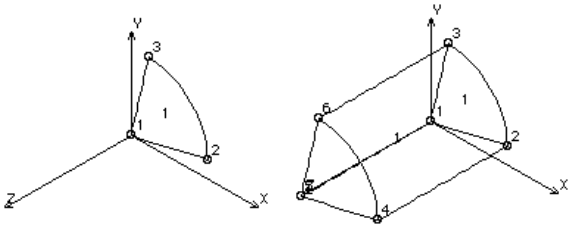
Notes

1. The active coordinate system must be Cartesian.
2. For proper association of nodes and elements with the generated geometric entities, do not merge nodes (using NMERGE) prior to this command.

Example:

Graphic Example:

```
PLANE,Z,0,1,
PT,1,0,0,0,
PT,2,65,15,0,
PT,3,15,65,0,
CRARC,1,2,3,1,50,
SFPTCR,1,1,1,0,
VLEXTR,1,1,1,Z,110,
```



VLSWEEP

Geometry > VOLUMES > GENERATION > Sweeping

The VLSWEEP command defines volumes by sweeping surfaces about a specified axis of the currently active Cartesian coordinate system by a specified angle. The number of volumes generated by sweeping a single surface depends on the number of segments specified. For reasonable accuracy of arc representation, one segment per 90 degrees is recommended. For more accuracy, a smaller angle can be used. Associated 2D elements are also swept to generate 3-D elements if default meshing for volume is active.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces.
(default is 1)

Axis symbol
Axis symbol (X, Y or Z).
(default is Y)

Angle of the arc
Angle of sweep in degrees.
(default is 360 degrees)

Number of segments
Number of segments for circular arc.
(default is 1 segment for each 90 degrees)

Number of elements per segment

Number of elements per segment in the sweep direction. Only prompted if default meshing for volumes is on.
(default is 6)

Original mesh update flag
Flag for handling the original 2D mesh.

= -1: Transfer	transfer mesh to the newly created surface or region at the end of the extrusion
= 0: Keep	keep the original 2D mesh
= 1: Delete	delete the original 2D mesh *(default is delete original mesh)*

Notes

1. For proper association of nodes and elements with the generated geometric entities, do not merge nodes (using NMERGE) prior to this command.
2. The active coordinate system must be Cartesian.

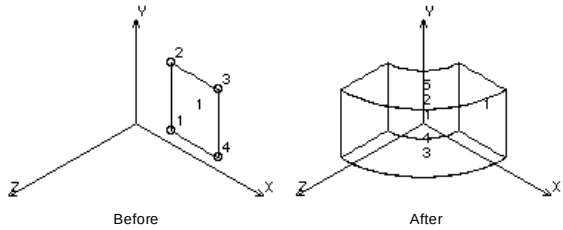
Examples

```
ACTDMESH, VL, 0
VLSWEEP, 1, 3, 1, X, 360.0, 4, , ,
```

The above commands generate 12 new volumes by sweeping curves 1 to 3 about the X-axis of the active coordinate system which must be Cartesian. Four volumes are generated from each surface. No 3-D elements are generated.

Graphic Example: VLSWEEP

```
PLANE,Z,0,1,
SF4CORD,1,30,10,0,30,
  60,0,70,60,0,70,10,
  0,
VLSWEEP,1,1,1,Y,-
  90,1,
```



Before After

VLGLIDE

Geometry > VOLUMES > GENERATION > Gliding

The VLGLIDE command defines volumes generated by moving a pattern of surfaces along profile curves. The surfaces in the pattern keep their original orientations as they glide along the profile curves, i.e. each surface moves parallel to its original position. Associated 2D elements are also glided to generate 3-D elements if default meshing for volumes is active.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Number of profile curves
Number of profile curves. (limited to 6)
(default is 1)

Profile curve (i)
ith profile curve. The curves must be specified in order and must form a continuous path.

Number of elements on curve (i)
Number of elements in the glide direction on the ith curve. Prompted only if default meshing for volumes is on.
(default is 6)

Original mesh update flag
Flag for handling the original 2D mesh.

= -1: Transfer	transfer mesh to the newly created surface or region at the end of the extrusion
= 0: Keep	keep the original 2D mesh
= 1: Delete	delete the original 2D mesh *(default is delete original mesh)*

Notes

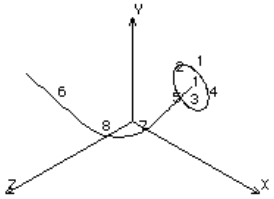
1. For proper association of nodes and elements with the generated geometric entities, do not merge nodes (using NMERGE) prior to this command.
2. The curve and the surfaces can be picked by the mouse.
3. The profile curves must have a continuous first derivative. Fillets can be created to smooth the profile if needed.
4. The locus of every single point on the gliding curves is exactly identical to the profile.
5. The GLIDE and DRAG are equivalent when the profile is a straight line.
6. The number of volumes generated is equal to the number of the profile curves multiplied by the number of surfaces in the pattern.

Example

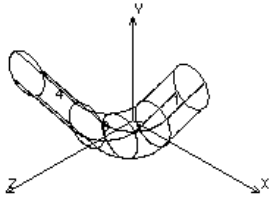
Graphic Example:

VLGLIDE

```
PT,1,50,50,0,
PT,2,60,40,70,
PT,3,-30,50,60,
PLANE,Z,0,1,
PT,4,95,85,0,
CRPCIRCLE,1,1,4,15
  ,360,4,
CRLINE,5,1,2,
CRLINE,6,2,3,
CRFILLET,7,5,6,25,1,0,
```



Before



After

```
SF4CR,1,1,2,3,4,0,
VLGLIDE,1,1,1,4,5,7,8,6,
```

VLDRAG

Geometry > VOLUMES > GENERATION > Dragging

The VLDRAG command defines volumes generated by dragging a pattern of surfaces along profile curves. The volumes are generated such that the angles between the profile and any surface in the pattern remain unchanged as the surface is dragged along the profile. Associated 2D elements are also dragged to generate 3-D elements if default meshing for volumes is active.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Number of profile curves
Number of profile curves. (limited to 6)
(default is 1)

Profile curve (i)
ith profile curve. The curves must be specified in order and must form a continuous path.

Number of elements on curve (i)
Number of elements in the glide direction on the ith curve. Prompted only if default meshing for volumes is on.
(default is 4)

Original mesh update flag
Flag for handling the original 2D mesh.

= -1: Transfer	transfer mesh to the newly created surface or region at the end of the extrusion
= 0: Keep	keep the original 2D mesh
= 1: Delete	delete the original 2D mesh
	(default is delete original mesh)

Notes

1. The curves and the surfaces can be picked by the mouse.
2. The profile curves must have a continuous first derivative (continuous slope). Fillets can be created to smooth the profile if needed.
3. The GLIDE and DRAG operations are equivalent when the profile is a straight line.
4. The number of volumes generated is equal to the number of the profile curves

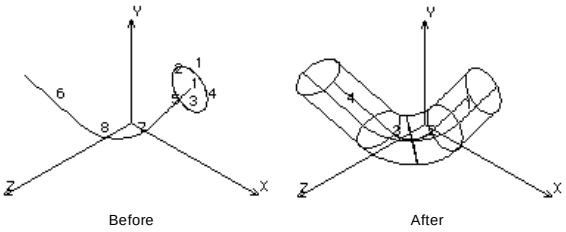
multiplied by the number of surfaces in the pattern.

5. For proper association of nodes and elements with the generated geometric entities, do not merge nodes (using NMERGE) prior to this command.

Example:

Graphic Example: VLDRAG

```
PT,1,50,50,0,
PT,2,60,40,70,
PT,3,-30,50,60,
PLANE,Z,0,1,
PT,4,95,85,0,
CRPCIRCLE,1,1,4,15,3
  60,4,
CRLINE,5,1,2,
CRLINE,6,2,3,
CRFILLET,7,5,6,25,1,0,
SF4CR,1,1,2,3,4,0,
VLDRAG,1,1,1,4,5,7,8,6,
```



VLRELOC

Geometry > VOLUMES > GENERATION > Relocate

The VLRELOC command relocates a pattern of volumes by a specified translation and/or rotation in the currently active coordinate system. This command does not generate any new volumes. The VLGEN (**Geometry, Volumes, Generation, Generate**) command can be used to generate new volumes from existing ones.

Beginning volume
Beginning volume in the pattern.

Ending volume
Ending volume in the pattern.
(default is beginning volume)

Increment
The increment between volumes.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation

The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

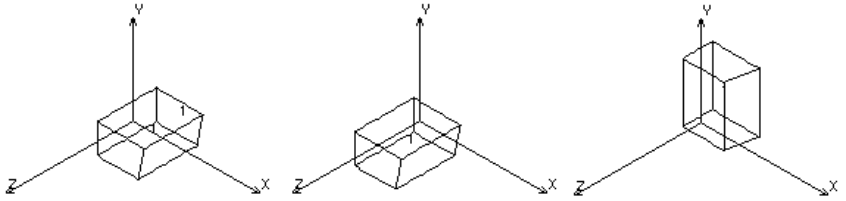
Note

The active coordinate system must be Cartesian.

Example

Graphic Example: VLRELOC

```
PLANE,Z,0,1,
SF4CORD,1,20,10,0,20,35,0,60,35,0,55,10,0,
VLEXTR,1,1,1,Z,50,
VLRELOC,1,1,1,0,-25,-20,0,
VLRELOC,1,1,1,1,-90,0,0,
```



Before the First VLRELOC Command | Before the Second VLRELOC Command | After the Second VLRELOC Command

VLMOVE

Geometry > VOLUMES > GENERATION > Move

The VLMOVE command moves a pattern of volumes from the current coordinate system to the destination coordinate system. This command does not generate any new volumes. The VLCOPY (**Geometry, Volumes, Generation, Copy**) command can be used to generate new volumes by copying existing ones.

Beginning volume
Beginning volume in the pattern.

Ending volume
Ending volume in the pattern.
(default is beginning volume)

Increment
Increment between volumes.
(default is 1)

Destination coordinate
Destination coordinate system label.
(default is 0)

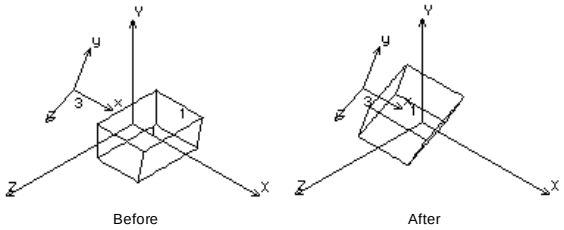
Note

Both coordinate systems must be Cartesian.

Example

Graphic Example: VLMOVE

```
PLANE,Z,0,1,
SF4CORD,1,20,10,0,
  20,35,0,60,35,0,
  55,10,0,
VLEXTR,1,1,1,Z,50,
PT,9,0,50,50,
PT,10,60,30,30,
PT,11,0,70,50,
CSYS,3,0,9,10,11,
ACTSET,CS,0,
VLMOVE,1,1,1,3,
```



VLRESIZ

Geometry > VOLUMES > GENERATION > Resize

The VLRESIZ command resizes a pattern of existing volumes, by scaling and translating/rotating them relative to the currently active coordinate system. This command does not generate any new volumes. The VLSCALE (**Geometry, Volumes, Generation, Scaling**) command can be used to generate new volumes by scaling existing ones.

Beginning volume

Beginning volume in the pattern.

Ending volume

Ending volume in the pattern.
(default is beginning volume)

Increment

The increment between volumes.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor

The scale factors in the X, Y, and Z directions.
(default are 1.0)

X, Y, Z-Displacement

The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.

(defaults are 0.0)

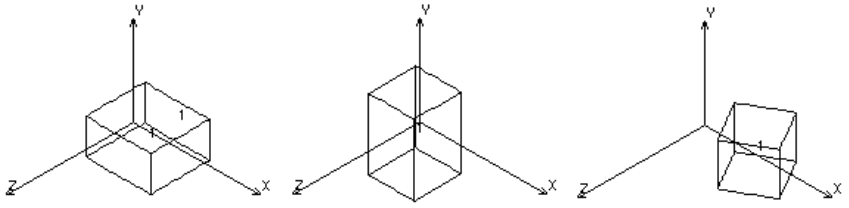
X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

Note
The active coordinate system must be Cartesian.

Example

Graphic Example: VLRESIZ

```
PLANE,Z,0,1,
SF4CORD,1,10,5,0,10,35,0,65,35,0,65,5,0,
VLEXTR,1,1,1,Z,50,
VLRESIZ,1,1,1,0,.7,2.,.8,20,0,30,
VLRESIZ,1,1,1,1,1,.6,1,0,30,0,
```



Before the First VLRESIZ Command | Before the Second VLRESIZ Command | After the Second VLRESIZ Command

VLFLIP

Geometry > VOLUMES > GENERATION > Flip

The VLFLIP command flips a pattern of volumes about a specified plane in the current coordinate system. An offset can also be specified with the flipping. This command does not generate any new volumes. The VLSYM (**Geometry, Volumes, Generation, Symmetry**) command can be used to generate new volumes from existing ones.

Beginning volume
Beginning volume in the pattern.

Ending volume
Ending volume in the pattern.
(default is beginning volume)

Increment
Increment between volumes.
(default is 1)

Rotation/Sweep axis

Axis representing normal to plane about which pattern of volumes are to be flipped.

= X	y-z plane
= Y	z-x plane
= Z	x-y plane
	(default is Z)

Reverse direction flag

Reverse direction flag.

= 0: No	do not reverse direction of first parametric curve of each volume
= 1: Yes	reverse direction of first parametric curve of each volume
	(default is not to reverse the direction)

Offset

Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

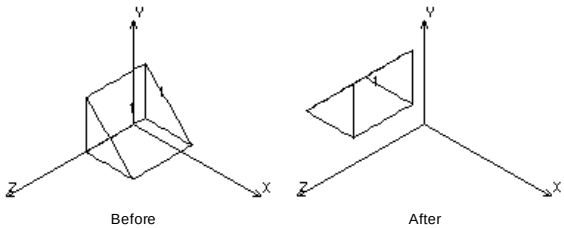
Notes

1. Volumes can be picked by the mouse.
2. A specified offset is applied to all volumes of the specified pattern.
3. The active coordinate system must be Cartesian.

Example

Graphic Example: VLFLIP

```
PLANE,Z,0,1,
SF3CORD,1,50,10,0,10
  ,10,0,10,50,0,
VLEXTR,1,1,1,Z,50,
VLFLIP,1,1,1,X,1,0,
```



VLGEN

Geometry > VOLUMES > GENERATION > Generate

The VLGEN command generates one or more patterns of volumes from an existing pattern by translating and/or rotating about the currently active Cartesian coordinate system. The VLRELOC (**Geometry, Volumes, Generation, Relocate**) command can be used to relocate existing volumes without generating new ones.

Generation number

Number of patterns to be generated. (must be > 0 for generation to occur)
(default is 1)

Beginning volume

Beginning volume in the pattern.

Ending volume

Ending volume in the pattern.

(default is beginning volume)

Increment
Increment between volumes.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
Magnitudes of the translation in the currently active coordinate system.
(defaults are 0.0)

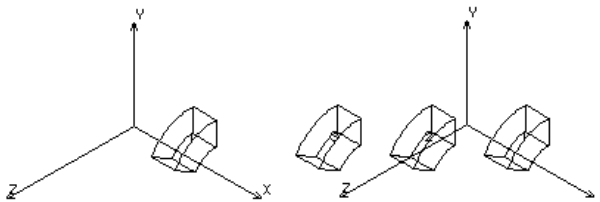
X, Y, Z-Rotation
Magnitudes of the rotations (degrees) in the currently active coordinate system.
(defaults are 0.0)

Note
The active coordinate system must be Cartesian.

Example

Graphic Example: VLGEN

```
PLANE,Z,0,1,
SF4CORD,1,50,20,0,50,40,0,70,40,0,70,20,0,
VLSWEEP,1,1,1,X,60,1,
VLGEN,2,1,1,1,0,-40,0,40,
```



VLCOPY

Geometry > VOLUMES > GENERATION > Copy

The VLCOPY command copies a pattern of volumes from the active coordinate system to the destination coordinate system. Use the VLMOVE (**Geomtery, Volumes, Generation, Move**) command to move existing volumes without generating new ones.

Beginning volume
Beginning volume in the pattern.

Ending volume
Ending volume in the pattern.
(default is beginning volume)

Increment
Increment between volumes.
(default is 1)

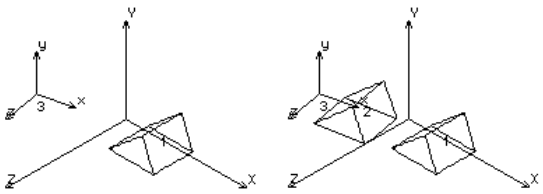
Destination coordinate
Destination coordinate system label.
(default is 0)

Note
Both coordinate systems must be Cartesian.

Example

Graphic Example: VLCOPY

```
PLANE,Z,0,1,
SF3CORD,1,20,5,0,50,3
  0,0,60,5,0,
VLEXTRUDE,1,1,1,Z,35,
CSANGL,3,0,-
  20,40,60,0,10,0,
ACTSET,CS,0,
VLCOPY,1,1,1,3,
```



VLSCALE

Geometry > VOLUMES > GENERATION > Scaling

The VLSCALE command generates a pattern of volumes from an existing pattern by scaling, translating or rotating them relative to the currently active coordinate system. The coordinates of all associated keypoints are multiplied by the specified scale factors.

Beginning volume
Beginning volume in the pattern.

Ending volume
Ending volume in the pattern.
(default is beginning volume)

Increment
Increment between volumes.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X, Y, and Z directions.
(defaults are 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

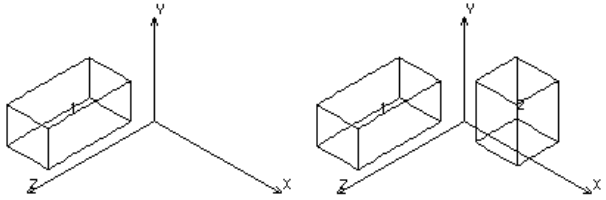
X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

Note
The active coordinate system must be Cartesian.

Example

Graphic Example: VLSCALE

```
PLANE,Z,40,1,
SF4CORD,1,20,30,40,20,60,40,-15,60,40,-15,30,40,
VLEXTRUDE,1,1,1,Z,70,
VLSCALE,1,1,1,0,1,1.5,.5,80,0,0,
```



VLSYM

Geometry > VOLUMES > GENERATION > Symmetry

The VLSYM command generates a pattern of volumes symmetric to the specified pattern about a specified plane in the active coordinate system. An offset can also be specified. Use the VLFLIP (**Geomtery, Volumes, Generation, Flip**) command to flip existing volumes without generating new ones.

Beginning volume
Beginning volume in the pattern.

Ending volume
Ending volume in the pattern.
(default is beginning volume)

Increment

Increment between volumes.
(default is 1)

Rotation/Sweep axis
Axis representing normal to plane about which pattern of volumes are to be generated symmetrically.

= X	plane y-z
= Y	plane z-x
= Z	plane x-y
	(default is Z)

Reverse direction flag
Reverse direction flag.

= 0: No	do not change vdirections of the the volumes
= 1: Yes	change directions of the volumes by reversing the direction of each first parametric curve
	(default is not to reverse the direction)

Offset
Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

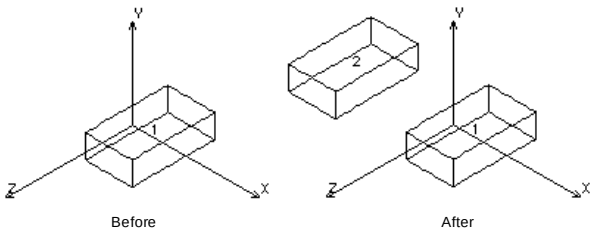
Notes

1. Volumes can be picked by the mouse.
2. A specified offset is applied to all volumes of the specified pattern.
3. If default meshing of volumes is active, it is suggested to use the default value for the reverse direction flag.
4. The active coordinate system must be Cartesian.

Example

Graphic Example: VLSYM

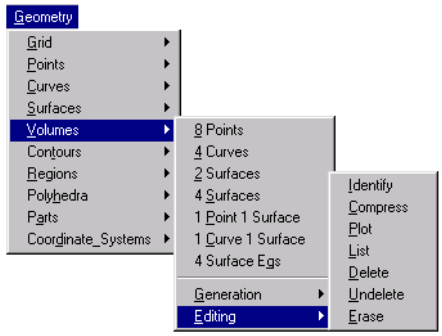
```
PLANE,Z,0,1,
SF4CORD,1,30,25,0,
  30,45,0,70,45,0,
  70,25,0,
VLEXTR,1,1,1,Z,70,
VLSYM,1,1,1,X,1,0,
```



▼EDITING Menu

Geometry > VOLUMES > EDITING

Figure 4-18 Editing Menu



VLIDENT

Geometry > VOLUMES > EDITING > Identify

The VLIDENT command identifies a volume by highlighting it and displaying its label in the dialogue area.

Volume
Select volume.

Notes

1. Only entities plotted on the screen can be selected by the mouse.
2. Pressing the right button of the mouse keeps selecting volumes in the neighborhood.

VLCOMPRESS

Geometry > VOLUMES > EDITING > Compress

The VLCOMPRESS command renumbers the volumes in a pattern by removing all numbering gaps.

Beginning volume
Beginning volume in the pattern.
(default is 1)

Ending volume
Ending volume in the pattern.
(default is the highest surface number defined)

Example:

```
VLCOMPRESS, 1, 100
```

This command renumbers all volumes numbered 1 to 100 by removing any labeling gaps. If defined volumes were numbered 11 to 50 and 76 to 100, this command renumbers volume such that volume 11 is renumbered as 1, volume 76 as 41, and volume 100 as 65.

VLPLOT

Geometry > VOLUMES > EDITING > Plot

The VLPLOT command plots a pattern of volumes on the screen.

Beginning volume
Beginning volume in the pattern.
(default is 1)

Ending volume
Ending volume in the pattern.
(default is the highest volume defined)

Increment
Increment between volumes in the pattern.
(default is 1)

Note
Use the STATUS1 table to control label plotting and color.

VLLIST

Geometry > VOLUMES > EDITING > List

The VLLIST command lists a pattern of volumes. The command lists the type and the associated curves of each volume in the pattern.

Beginning volume
Beginning volume in the pattern.
(default is 1)

Ending volume
Ending volume in the pattern.
(default is the highest volume defined)

Increment
Increment between volumes in the pattern.
(default is 1)

Example:

```
VLLIST, 10, 20, 2
```




This command lists labels and curves of volumes 10, 12, 14, 16, 18 and 20.

VLDEL

Geometry > VOLUMES > EDITING > Delete

The VLDEL command deletes a pattern of volumes. The deleted volumes can be undeleted by using the VLUNDEL (**Geometry, Undelete, Volumes**) command. Volumes associated with higher entities cannot be deleted.

Beginning volume
Beginning volume in the pattern.

Ending volume
Ending volume in the pattern.
(default is beginning volume)

Increment
Increment between volumes in the pattern.
(default is 1)

Note
This command deletes volume addresses temporarily from the database so that it will be possible to undelete them using the VLUNDEL (**Edit, Delete, Volumes**) command.

Example:

```
VLDEL, 10, 20, 2
```

This command deletes volumes 10, 12, 14, 16, 18 and 20 from the database.

VLUNDEL

Geometry > VOLUMES > EDITING > Undelete

The VLUNDEL command undeletes all volumes in the specified pattern.

Beginning volume
Beginning volume in the pattern.

Ending volume
Ending volume in the pattern.
(default is beginning volume)

Increment
Increment between volumes in the pattern.
(default is 1)

Note
It is recommended that the VLUNDEL (**Edit, Undelete, Volumes**) command be

used immediately after an accidental deletion of volumes and before the generation of any new volumes.

Example:

```
VLUNDEL, 10, 20, 2
```

This command undeletes volumes 10, 12, 14, 16, 18 and 20.

VLERASE

Geometry > VOLUMES > EDITING > Erase

The VLERASE command erases plotted volumes. The erased volumes can be replotted using the VLPLOT (**Edit, Plot, Volumes**) command.

Beginning volume
Beginning volume in the pattern.

Ending volume
Ending volume in the pattern.
(default is beginning volume)

Increment
Increment between volumes in the pattern.
(default is 1)

Example:

```
VLERASE, 10, 20, 2
```

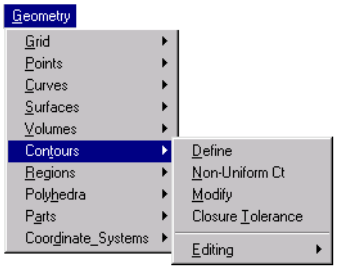
This command erases volumes 10, 12, 14, 16, 18 and 20 from the screen.




▶ CONTOURS Menu

Geometry > CONTOURS

Figure 4-19 Contours Menu



This menu includes commands to generate, process and manipulate contours. Contours are non-parametric entities built from curves that form a closed loop. A contour can have up to 250 curves. Contours are used to define regions or to specify boundary conditions. All the curves and hence the keypoints of a contour must lie in the same plane or on an underlying surface. Planar contours can be used to define planar regions. Three dimensional contours can be used to define regions on underlying surfaces (refer to the **Geometry, Regions** menu). The short names of all commands in this menu start with "CT" . The rest of the characters describe the function of the command.

CT

Geometry > CONTOURS > Define

The CT command defines a contour. A contour is a closed loop defined by a group of curves. The curves that define a contour must all lie in a plane (planar contour), or they must lie on an underlying surface. Up to 500 curves can be used to define a contour. You do not need to specify all the curves of a contour, but you must specify as many curves as needed to avoid any ambiguity. Otherwise the program will pick one of many possible contours. A curve must then be selected whenever an alternative path exists in order to uniquely define the contour. Contours are used to create regions, the specified element size is used for subsequent automatic meshing.

Contour
Contour label, maximum number allowed is 5000.
(default is maximum contour number defined +1)

Mesh flag
= 0: Element size — assign an average element size to the contour
= 1: Number of elements — assign number of elements on the contour
(default is element size)

Average element size/number of elements on the boundary
Average size of element if *mesh flag = element size* or approximate number of elements along the contour if *mesh flag = number of elements*.

Number of reference boundary curves
Number of reference curves to uniquely define the contour. (limit is 500)
(default is 1)

Curve (i)
Label of the i^{th} reference curve to define the contour. (i = 1,2,...,500)

Use selection set

Selection flag.
= 1: Yes consider curves that are in the active selection list only
= 0: No consider all curves
(default is no)

Meshing redefinition flag

Meshing redefintion flag. Prompted only if the contour being defined shares curves with other existing contours for mesh compatibility.
= 0: Previous for curves common to other contours, use same element size or number of elements as previously specified
= 1: Redefine for curves common to other contours, change the element size or number of elements to comply with the new specification
= 2: Max elements for curves common to other contours, compare old and new mesh specifications and select the option that specifies a finer mesh (more elements)
= 3: Min elements for curves common to other contours, compare old and new mesh specifications and select the option that specifies a coarser mesh (less elements)
(default is Redefine)

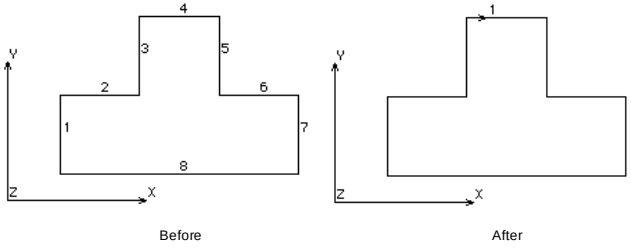
Notes

1. Curves can be selected by the mouse only if they are plotted on the screen.
2. If a large number of curves exists in the model, it is suggested to activate a curve selection list (see commands in the **Control, Select** submenu), to select curves in the neighborhood of the contour to accelerate the process.
3. If a contour cannot be created, you may increase the contour tolerance using the CTTOL (**Geometry, Contours, Closure Tolerance**) command and then try the CT command again.
4. Mesh specification can be changed by commands in the **Meshing, Mesh Density** menu.

Example

Graphic Example 1: CT

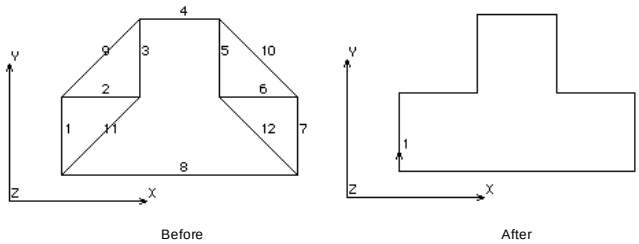
```
PLANE,Z,0,1,
VIEW,0,0,1,
CRPCORD,1,10,5,
  0,10,20,0,25,
  20,0,25,35,0,
  40,35,0,40,20
  ,0,
55,20,0,55,5,0,
  10,5,0,
CT,1,0,5,1,4
```



Graphic Example 2: CT

```
PLANE,Z,0,1,
VIEW,0,0,1,
```

```
CRPCORD,1,10,5,
  0,10,20,0,25,
  20,0,25,35,0,
  40,35,0,40,20
  ,0,55,20,0,55
  ,5,0,10,5,0,
CRLINE,9,2,4,
CRLINE,10,5,7,
CRLINE,11,1,3,
CRLINE,12,6,8,
CT,1,0,5,8,1,8,7,6,5,4,3,2,
```



CTNU

Geometry > CONTOURS > Non-Uniform Ct

The CTNU command defines a contour with nonuniform meshing density by specifying the number of elements on each curve to be used in subsequent automatic meshing. A contour is a closed path defined by a group of curves. The curves that define a contour must all lie in a plane (planar contour), or they must lie on an underlying surface. Up to 500 curves may be used to define a contour. You must specify all the curves that define the closed contour path.

Contour
Contour label (number).
(default is the highest contour number defined +1)

Number of reference boundary curves
Number of curves to define the contour. (limit is 500)

Boundary curve (i)
Label of ith curve to define the contour. (i = 1,2,...,500)

Number of elements on curve (i)
Number of elements on the ith curve. (i = 1,2,...,500)
(default is 4.0)

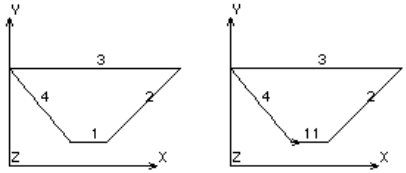
Meshing redefinition flag
Meshing redefintion flag. Prompted only if the contour being defined shares curves with other existing contours for mesh compatibility.
= 0: Previous for curves common to other contours, use same element size or number of elements as previously specified
= 1: Redefine for curves common to other contours, change the element size or number of elements to comply with the new specification
= 2: Max elements for curves common to other contours, compare old and new mesh specifications and select the option that specifies a finer mesh (more elements)
= 3: Min elements for curves common to other contours, compare old and new mesh specifications and select the option that specifies a coarser mesh (less elements)
(default is Redefine)

Notes

1. Curves can be selected by the mouse only if they are plotted on the screen.
2. Mesh specification can be changed by commands in the **Meshing, Mesh Density** menu.

Graphic Example: CTNU

```
PLANE,Z,0,1,
VIEW,0,0,1,
CRPCORD,1,25,10,0,40,10,0,70,
  40,0,0,40,0,25,10,0,
CTNU,1,4,1,3,2,6,3,8,4,5,
```



Before After

CTMODIFY

Geometry > CONTOURS > Modify

The CTMODIFY command modifies an existing contour by replacing one of its curves by one or more other existing curves. The command can also be used to redefine the number of elements on a curve by replacing it by itself and specifying the desired number of elements.

Contour
Label of contour to be modified. (between 1 and 5000, inclusive)

Curve to be modified/replaced
Label of the curve to be modified. The curve must belong to the selected contour.

Number of replacing curves
Number of replacing curves. (limit is 10)
(default is 1)

Replacing curve (i)
Label of the ith replacing curve.

Number of elements on curve (i)
Number of elements on the ith replacing curve.
(default is 4)

Meshing redefinition flag
Meshing redefintion flag. Prompted only if the contour being defined shares curves with other existing contours for mesh compatibility.
= 0: Previous for curves common to other contours, use same element size or number of elements as previously specified
= 1: Redefine for curves common to other contours, change the element size or number of elements to comply with the new specification
= 2: Max elements for curves common to other contours, compare old and new mesh specifications and select the option that specifies a finer mesh (more elements)
= 3: Min elements for curves common to other contours, compare old and new mesh specifications and select the option that specifies a coarser mesh

(less elements)
(default is Redefine)

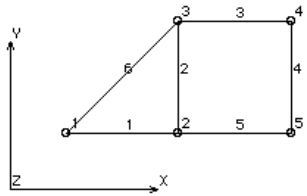
Notes

1. The replacing curves must form an alternative route without gaps, from the starting point to the ending point of the replaced curve. Erroneous results will occur if the orientation is reversed.
2. The command does not perform closed_loop check.

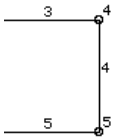
Example

Graphic Example: CTMODIFY

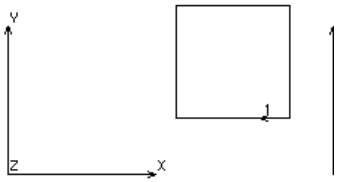
```
PLANE,Z,0,1,
VIEW,0,0,1,
CRPCORD,1,10,10,0,30,10,0,30,30,0,5
  0,30,0,50,10,0,30,10,0,30,10,0,
CRLINE,6,1,3,
CT,1,0,5,3,5,2,3,
CTMODIFY,1,2,2,1,4,6,1,
```



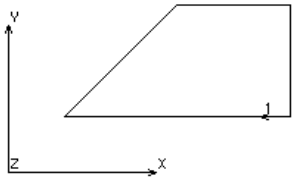
Before Defining Contour 1



Contour 1



Contour 1 Before Modification



Contour 1 After Modification

CTTOL

Geometry > CONTOURS > Closure Tolerance

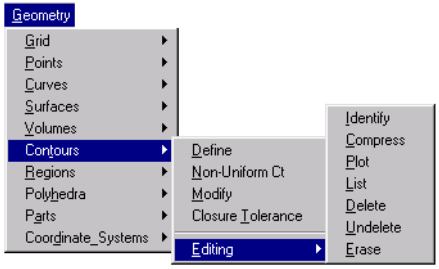
The CTTOL command sets the tolerance to be used in subsequent definition of contours. The difference between the corresponding X, Y and Z coordinates of the end keypoints of the curves of the contour must be equal to or less than the given tolerance in order for the contour to be defined.

Tolerance
Tolerance. (closing the loop of a contour)
(default is 0.0001)

▼EDITING Menu

Geometry > CONTOURS > EDITING

Figure 4-20 Editing Menu



CTIDENT

Geometry > CONTOURS > EDITING > Indentify

The CTIDENT command identifies a contour by highlighting its member curves and displaying its label in the dialogue area.

Contour
Select contour.

Notes
1. Only entities plotted on the screen can be selected by the mouse.
2. Pressing the right button of the mouse keeps selecting contours in the neighborhood.

CTCOMPRESS

Geometry > CONTOURS > EDITING > Compress

The CTCOMPRESS command renumbers the contours in the pattern by removing all numbering gaps.

Beginning contour
Beginning contour in the pattern.
(default is 1)

Ending contour
Ending contour in the pattern.
(default is the highest contour number defined)

Example:

```
CTCOMPRESS, 1, 100
```

This command renumbers all contours numbered 1 to 100 by removing any labeling gaps. If defined contours were numbered 11 to 50 and 76 to 100, this command renumbers contours such that contour 11 is renumbered as 1, contour 76 as 41, and contour 100 as 65.

CTPLOT

Geometry > CONTOURS > EDITING > Plot

The CTPLOT command plots a pattern of contours on the screen.

Beginning contour
Beginning contour in the pattern.
(default is 1)

Ending contour
Ending contour in the pattern.
(default is the highest contour defined)

Increment
Increment between contours in the pattern.
(default is 1)

Note
Use the STATUS1 table to control label plotting and color.

CTLIST

Geometry > CONTOURS > EDITING > List

The CTLIST command lists a pattern of contours. For each contour, the command lists the number of curves, the average element size, and the number of elements on each curve associated with the contour. The element specification will be used during automatic meshing.

Beginning contour
Beginning contour in the pattern.
(default is 1)

Ending contour
Ending contour in the pattern.
(default is the highest contour defined)

Increment
Increment between contours in the pattern.
(default is 1)

Example:

```
CTLIST, 10, 20, 2
```

This command lists contours 10, 12, 14, 16, 18 and 20. For each contour, the command lists the number of curves, the average element size, and the number of elements of each curve associated with the contour.

CTDEL

Geometry > CONTOURS > EDITING > Delete

The CTDEL command deletes a pattern of contours. The deleted contours can be undeleted by using the CTUNDEL (**Geometry, Undelete, Contours**) command. Contours associated with higher entities cannot be deleted.

Beginning contour
Beginning contour in the pattern.

Ending contour
Ending contour in the pattern.
(default is beginning contour)

Increment
Increment between contours in the pattern.
(default is 1)

Example:

```
CTDEL, 10, 20, 2
```

This command deletes contours 10, 12, 14, 16, 18 and 20 from the database.

CTUNDEL

Geometry > CONTOURS > EDITING > Undelete

The CTUNDEL command undeletes all contours in the specified pattern.

Beginning contour
Beginning contour in the pattern.

Ending contour
Ending contour in the pattern.
(default is beginning contour)

Increment
Increment between contours in the pattern.
(default is 1)

Note
It is recommended that the CTUNDEL (**Edit, Undelete, Contours**) command be used immediately after an accidental deletion of contours and before the genera-




tion of any new contours.

Example:

```
CTUNDEL, 10, 20, 2
```

This command undeletes contours 10, 12, 14, 16, 18 and 20.

CTERASE

Geometry > CONTOURS > EDITING > Erase

The CTERASE command erases plotted contours. The erased contours can be replotted using the CTPLOT (**Edit, Plot, Contours**) command.

Beginning contour
Beginning contour in the pattern.

Ending contour
Ending contour in the pattern.
(default is beginning contour)

Increment
Increment between contours in the pattern.
(default is 1)

Example:

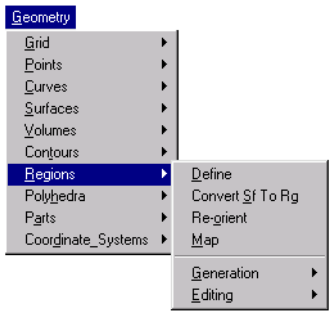
```
CTERASE, 10, 20, 2
```

This command erases contours 10, 12, 14, 16, 18 and 20 from the screen.

▶ REGIONS Menu

Geometry > REGIONS

Figure 4-21 Regions Menu



This menu includes commands to generate and process regions. Regions are non-parametric entities defined by contours. A region can be built from an outer contour and up to 119 inner contours that represent cuts or holes.

For this reason, regions represent a powerful option to generate finite element meshes for arbitrary shaped areas with cuts and/or holes. In contrast to surfaces, there is no need to subdivide into 4-sided areas. For planar regions, contours and hence the curves and keypoints of a region must lie in the same plane. Non-planar regions can also be specified. All the contours and hence the curves and keypoints of a non-planar region must all lie on one underlying surface. Regions may be meshed to generate area elements like TRIANG and SHELL3. Regions may be used in the definition of POLYHEDRA and PARTS. The short names of all commands in this menu start with "RG". The rest of the characters describe the function of the command.

RG

Geometry > REGIONS > Define

The RG command creates a region by one outer contour and up to 119 inner contours. Planar and non-planar regions may be defined. All the contours of planar region must be planar. A non-planar region may be created by specifying an underlying surface. All the contours used to create a non-planar region must entirely lie on one underlying surface. The underlying surface used for a non-planar region should be smooth.

Region
Region label.
(default is the maximum region number defined +1)

Number of contours
Number of contours that will be used to define the region. (Must be between 1 and 120)
(default is 1)

Outer contour
Outer contour label.

Inner contour (i)
Inner ith contour label. (i=1,2,...,119)

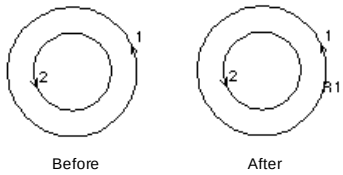
Underlying surface
Label of underlying surface.
(default is 0, for a planar region)

Notes

1. Contours may not intersect each other.
2. Contours may not share common curves.
3. The command performs close_loop check on all contours used.
4. The outer contour must be picked first and should.
5. Contours can be picked by the mouse.

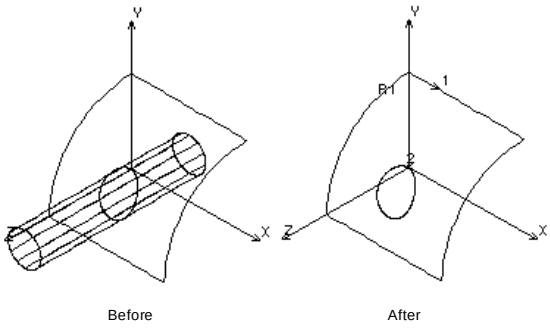
Graphic Example 1: Planar Region

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,35,30,0,
PT,2,75,30,0,
CRPCIRCLE,1,1,2,25,360,4,
CRPCIRCLE,5,1,3,15,360,4,
CT,1,0,5,1,1,
CT,2,0,5,1,7,
RG,1,2,1,2,0
```



Graphic Example 2: Region on Underlying Surface

```
PT,1,0,50,0,
CREXTR,1,1,1,X,70,
SFSWEEP,1,1,1,X,90,1,
CSANGL,3,0,35,25,0,0,
  0,0,0,
PT,7,0,10,0,
CREXTR,7,7,1,Z,100,
SFSWEEP,5,5,1,Z,360,1
  2,
CRINTSS,1,2,13,1,0.00
  1,
CT,1,0,5,1,1,0,
CT,2,0,3,1,41,0,
RG,1,2,1,2,1,
```



RGSF

Geometry > REGIONS > Convert Sf to Rg

The RGSF command creates a pattern of regions from a pattern of planar or non-planar surfaces. Each region is created by specifying the corresponding surface as the underlying surface. The command is useful for local mesh control of regions, polyhedra, and parts. If nonuniform meshing is desired for a polyhedra, or part defined by a group of surfaces and regions, then this command must be issued before meshing to convert all surfaces used in the definition of the polyhedra into

regions. You can then use commands in the **Meshing, Mesh Density** menu to specify element sizes for specific entities (local mesh control). The command automatically creates a contour from the curves associated with each surface.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Average element size
Element size to be used for subsequent meshing.

RGREORNT

Geometry > REGIONS > Re-orient

The RGREORNT command reverses the orientation of a pattern of regions. Elements associated with the region in the pattern are also reoriented if default meshing is active for regions. The command is useful in flipping the top and bottom faces of shell elements. The **Meshing, Elements, Aligh Shell Elements** can be used to perform automatic alignment of shell elements.

Beginning region
Beginning region in the pattern

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Example:

```
 ACTDMESH, RG, 1,
RGREORNT, 1, 6, 1
```

The above command changes the orientation of regions 1 to 6. Associated elements are also reoriented. This command is useful for shell elements only.




RGMAP

Geometry > REGIONS > Map

The RGMAP command maps an existing *source* region to a new or existing *target* region. This command is particularly useful when importing assemblies from solid modeling programs to map regions common to any two parts.

Source region
Existing region to be mapped.

Target region
New or existing region to which the source region is to be mapped.

Example:

```
RGMAP, 11, 16
```

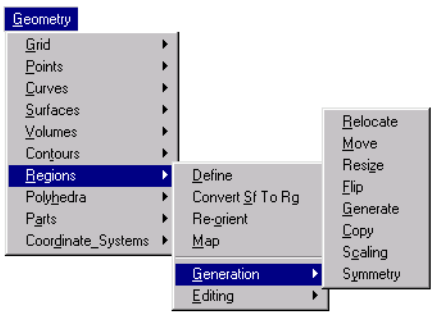
The above command maps region 11 to region 16.

▼GENERATION Menu

Geometry > REGIONS > GENERATION

This menu contains commands to change existing regions or generate new regions by operating on existing ones.

Figure 4-22 Generation Menu



RGRELOC

Geometry > REGIONS > GENERATION > Relocate

The RGRELOC command relocates a pattern of regions by a specified translation and/or rotation in the currently active coordinate system. This command does not

generate any new regions. The RGGEN (**Geometry, Regions, Generation, Generate**) command can be used to generate new regions from existing ones.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
The increment between regions.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

Note
The active coordinate system must be Cartesian.

Examples

```
RGRELOC, 1, 6, 1, 0, 5.0, 5.0, 0.0
```

The above command translates regions 1 through 6 from their current locations by X = 5., Y = 5., and Z = 0. in the currently active coordinate system.

```
RGRELOC, 1, 6, 1, 1, 30.0, 0.0, 0.0
```

The above command rotates regions 1 through 6 from their current locations by x-rotation = 30.0, y-rotation = 0.0 and z-rotation = 0.0 degrees about the currently active X, Y, and Z coordinate axes, respectively.

RGMOVE

Geometry > REGIONS > GENERATION > Move

The RGMOVE command moves a pattern of regions from the current coordinate system to the destination coordinate system. This command does not generate any

new regions. The RGCOPY (**Geometry, Regions, Generation, Copy**) command can be used to generate new regions by copying existing ones.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions.
(default is 1)

Destination coordinate
Destination coordinate system label.
(default is 0)

Note
Both coordinate systems must be Cartesian.

Example

```
RGMOVE, 1, 6, 1, 3
```

The above command moves regions 1 through 6 to local coordinate system 3 with the same coordinates.

RGRESIZ

Geometry > REGIONS > GENERATION > Resize

The RGRESIZ command resizes a pattern of existing regions, by scaling and translating/rotating them relative to the currently active coordinate system. This command does not generate any new regions. The RGSCALE (**Geometry, Regions, Generation, Scaling**) command can be used to generate new regions by scaling existing ones.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
The increment between regions.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation




(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X, Y, and Z directions.
(default are 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

Note
The active coordinate system must be Cartesian.

Example

```
RGRESIZ, 1,4,1,0,0.0,0.5,0.0,5.0,0.0,0.0
```

The above command scales regions 1 to 4 by a scale factor of 0.5 in the Y-direction after translating the regions by X = 5., Y = 0.0, and Z = 0.0 in the currently active coordinate system.

RGFLIP

Geometry > REGIONS > GENERATION > Flip

The RGFLIP command flips a pattern of regions about a specified plane in the current coordinate system. An offset can also be specified with the flipping. his command does not generate any new regions. The RGSYM (**Geometry, Regions, Generation, Symmetry**) command can be used to generate new regions from existing ones.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions.
(default is 1)

Rotation/Sweep axis
Axis representing normal to plane about which pattern of regions are to be flipped.

= X	y-z plane
= Y	z-x plane

= Z	x-y plane *(default is Z)*

Reverse direction flag
Reverse direction flag.

= 0: No	do not reverse direction of first parametric curve of each region
= 1: Yes	reverse direction of first parametric curve of each region *(default is not to reverse the direction)*

Offset
Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

Notes

1. Regions can be picked by the mouse only if they are plotted on the screen.
2. A specified offset is applied to all regions of the specified pattern.
3. The active coordinate system must be Cartesian.

Example

```
RGFLIP, 1, 6, 1, X, 1, 0.0
```

The above command flips regions 1 through 6 about the Y-Z plane. Direction of first parametric curve is reversed. No offset is specified.

RGGEN

Geometry > REGIONS > GENERATION > Generate

The RGGEN command generates one or more patterns of regions from an existing pattern by translating and/or rotating about the currently active Cartesian coordinate system. The RGRELOC (**Geometry, Regions, Generation, Relocate**) command can be used to relocate existing regions without generating new ones.

Generation number
Number of patterns to be generated. (must be > 0 for generation to occur)
(default is 1)

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
Magnitudes of the translation in the currently active coordinate system.
(defaults are 0.0)

X, Y, Z-Rotation
Magnitudes of the rotations (degrees) in the currently active coordinate system.
(defaults are 0.0)

Note
The active coordinate system must be Cartesian.

Examples

Example 1: `RGGEN, 2, 1, 4, 1, 0, 5.0, 10.0, 10.0`

The above command generates two patterns of 4 regions each at the locations obtained by translating existing regions 1 through 4 by x = 5.0, y = 10.0, and z = 10.0 for the first pattern and x = 10, y = 20., z = 20. for the second pattern, respectively.

Example 2: `RGGEN, 2, 1, 4, 1, 1, 20.0, 15.0, 10.0`

The above command generates two patterns of 4 regions each at the locations obtained by rotating existing regions 1 through 4 by x-rotation = 5.0, y-rotation = 15.0, and z-rotation = 10. degrees for the first pattern, and x-rotation =10, y-rotation = 30.0, and z-rotation = 20.0 degrees for the second pattern, respectively, about the currently active X, Y, Z axes.

RGCOPY

Geometry > REGIONS > GENERATION > Copy

The RGCOPY command copies a pattern of regions from the active coordinate system to the destination coordinate system. Use the RGMOVE (**Geomtery, Regions, Generation, Move**) command to move existing regions without generating new ones.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions.
(default is 1)

Destination coordinate
Destination coordinate system label.
(default is 0)

Note
Both coordinate systems must be Cartesian.

Example

```
RGCOPY, 1, 6, 1, 3
```

The above command copies regions 1 through 6 to local Cartesian coordinate system 3. Corresponding keypoints have identical coordinates in both systems.

RGSCALE

Geometry > REGIONS > GENERATION > Scaling

The RGSCALE command generates a pattern of regions from an existing pattern by scaling, translating or rotating them relative to the currently active coordinate system. The coordinates of all associated keypoints are multiplied by the specified scale factors.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X, Y, and Z directions.
(defaults are 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.

(defaults are 0.0)

Note

The active coordinate system must be Cartesian.

Example

```
RGSCALE, 1,4,1,0,0.0,0.5,0.0,5.0,0.0,0.0
```

The above command generates 4 new regions from the existing regions 1 through 4 by applying a scale factor of 0.5 in the Y-direction and translating them by 5.0 units in the X-direction.

RGSYM

Geometry > REGIONS > GENERATION > Symmetry

The RGSYM command generates a pattern of regions symmetric to the specified pattern about a specified plane in the active coordinate system. An offset can also be specified. Use the RGFLIP (**Geomtery, Regions, Generation, Flip**) command to flip existing regions without generating new ones.

Beginning region

Beginning region in the pattern.

Ending region

Ending region in the pattern.
(default is beginning region)

Increment

Increment between regions.
(default is 1)

Rotation/Sweep axis

Axis representing normal to plane about which pattern of regions are to be generated symmetrically.

= X	plane y-z
= Y	plane z-x
= Z	plane x-y
	(default is Z)

Reverse direction flag

Reverse direction flag.

= 0: No	do not change vdirections of the the regions
= 1: Yes	change directions of the regions by reversing the direction of each first parametric curve
	(default is not to reverse the direction)

Offset

Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

Notes

1. Regions can be picked by the mouse only if they are plotted on the screen.
2. A specified offset is applied to all regions of the specified pattern.
3. If default meshing of regions is active, it is suggested to use the default value for the reverse direction flag.
4. The active coordinate system must be Cartesian.

Example

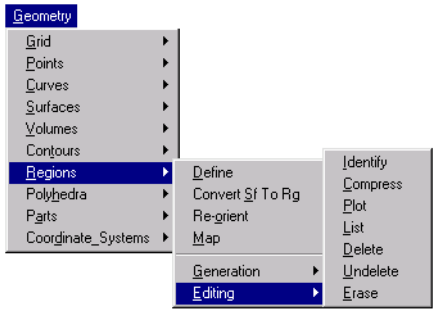
```
RGSYM, 1, 6, 1, X, 0, 20.0
```

The above command generates 6 regions that are symmetric with regions 1 through 6 relative to the y-z plane, except for an offset of 20.0 units. If a keypoint has an x-coordinate of -30., its new x-coordinate is 50.

▼Editing Menu

Geometry > REGIONS > EDITING

Figure 4-23 Editing Menu



RGIDENT

Geometry > REGIONS > EDITING > Identify

The RGIDENT command identifies a region by highlighting its boundaries and displaying its label in the dialogue area.

Region
Select region.

Notes
1. Entities can be selected by the mouse only if they are plotted on the screen.

2. Pressing the right button of the mouse keeps selecting regions in the neighborhood.

RGCOMPRESS

Geometry > REGIONS > EDITING > Compress

The RGCOMPRESS command renumbers all regions in the pattern by removing all numbering gaps.

Beginning region
Beginning region in the pattern.
(default is 1)

Ending region
Ending contour in the pattern.
(default is the highest region number defined)

Example:

```
RGCOMPRESS, 1, 100, 1
```

This command renumbers all regions numbered 1 to 100 by removing any labeling gaps. If defined regions were numbered 11 to 50 and 76 to 100, this command renumbers regions such that region 11 is renumbered as 1, region 76 as 41, and region 100 as 65.

RGPLOT

Geometry > REGIONS > EDITING > Plot

The RGPLOT command plots a pattern of regions on the screen.

Beginning region
Beginning region in the pattern.
(default is 1)

Ending region
Ending region in the pattern.
(default is the highest region defined)

Increment
Increment between regions in the pattern.
(default is 1)

Note
Use the STATUS1 table to control label plotting and color.




RGLIST

Geometry > REGIONS > EDITING > List

The RGLIST command lists a pattern of regions. For each region, the command lists the number of curves, the average element size, and the number of elements on each curve associated with the region. The element specification will be used during automatic meshing.

Beginning region
Beginning region in the pattern.
(default is 1)

Ending region
Ending region in the pattern.
(default is the highest region defined)

Increment
Increment between regions in the pattern.
(default is 1)

Example

```
RGLIST, 10, 20, 2
```

This command lists regions 10, 12, 14, 16, 18 and 20. For each region, the command lists the number of surfaces and contours, the average element size and a list of the contours associated with the region.

RGDEL

Geometry > REGIONS > EDITING > Delete

The RGDEL command deletes a pattern of regions. The deleted regions can be undeleted by using the RGUNDEL (**Geometry, Undelete, Regions**) command. Regions associated with higher entities cannot be deleted.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Note
This command deletes region addresses temporarily from the database so that it will be possible to undelete them using the RGUNDEL (**Edit, Delete, Regions**) command.

Example:

```
RGDEL, 10, 20, 2
```

This command deletes regions 10, 12, 14, 16, 18 and 20 from the database.

RGUNDEL

Geometry > REGIONS > EDITING > Undelete

The RGUNDEL command undeletes all regions in the specified pattern.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Note
It is recommended that the RGUNDEL (**Edit, Undelete, Regions**) command be used immediately after an accidental deletion of regions and before the generation of any new regions.

Example:

```
RGUNDEL, 10, 20, 2
```

This command undeletes regions 10, 12, 14, 16, 18 and 20.

RGERASE

Geometry > REGIONS > EDITING > Erase

The RGERASE command erases plotted regions. The erased regions can be replotted using the RGPLOT (**Edit, Plot, Regions**) command.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Example:

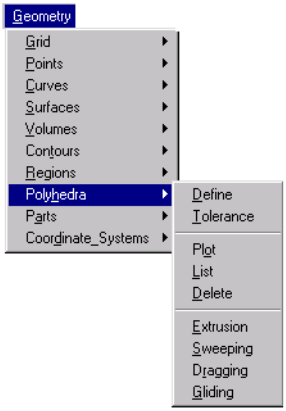
```
RGERASE, 10, 20, 2
```

This command erases regions 10, 12, 14, 16, 18 and 20 from the screen.

▶ POLYHEDRA Menu

Geometry > POLYHEDRA

Figure 4-24 Polyhedra Menu



This menu contains commands that generate and process polyhedrons. A polyhedron in GEO-STAR is the *skin* of a 3-D solid. It is a con-tinuous multi-sided closed boundary defined by a group of surfaces and/or regions that constitute the entire surface area of a 3-D enclosure. Polyhedra can be meshed to generate 2D elements like shells, or can be used to define PARTS which can be meshed to generate tetrahedral elements. The short names of all commands in this menu start with "PH". The rest of the characters describe the function of the command.

PH

Geometry > POLYHEDRA > Define

The PH command creates a polyhedron. A polyhedron is the skin of a 3-D solid. It is a continuous multi-sided closed boundary defined by a group of surfaces and/or regions that constitute the entire surface area of a 3-D enclosure. This command creates a polyhedron by specifying a reference surface or region. The polyhedron is created only if GEOSTAR can find a single closed boundary connected to the reference region or surface within the specified tolerance. The surfaces and/or regions of a polyhedron are connected to each other by common curves.

The polyhedron can be meshed using the MA_PH command to generate shell elements.

The command uses all available surfaces and regions to create the polyhedron. You can create a selection list for surfaces and/or regions before issuing this command to exclude undesired surfaces/regions from being used to create a polyhedron. The 3-D space enclosed by a polyhedron, or between a group of polyhedrons can be later defined as a PART.

Polyhedron
Label of the polyhedron to be created. Up to 200 polyhedrons can be defined.

(default is highest polyhedron defined +1)

Reference entity name

Type of the reference entity to define the polyhedron.

= SF	surfaces
= RG	regions *(default is RG)*

Surface/Region label

Reference surface or region to define the polyhedron.

Average element size

Average element size for subsequent automatic meshing of the created polyhedron or associated parts. If you celect a reference region, the average element size for the first region in the pattern is used as the default. If you celect a referenc surface, the default is generate 20 elements on the boundary of the first surface in the pattern.

Tolerance

Tolerance for closing the polyhedra, necessary to close the gaps (if any) between the keypoints of surfaces and/or regions that constitute the polyhedron. You may need to increase this tolerance if the polyhedron definition fails. The default tolerance is 20 times the value of the keypoint tolerance. See the PTTOL (**Geometry, Points, Merge Tolerance**) command for merging keypoints.

Redefine element size on bound

This flag controls whether to update element size specification on contours associated with the polyhedron.

= 0: No	do not update element size specifications on associated contour curves. The element size specified in this command will be used to mesh parts associated with the polyhedron.
= 1: Yes	update element size specifications. *(default is Yes)*

Notes

1. GEOSTAR checks all connected surfaces and regions in order to define the polyhedron. The polyhedron will not be created if the selected entity does not form a closed 3-D space with other connected surfaces and regions. Also, no polyhedron will be defined if a connected free surface or a region is found. A connected free surface or region is one with some, *but not all*, of its curves common to other surfaces/regions to form the polyhedron.
2. A polyhedron can be formed from a combination of up to 500 surfaces and/or regions. For the 256,000 node version, up to 4000 regions/surfaces are allowed.
3. Increasing the tolerance may help in creating a polyhedron.

To Create a polyhedron:

1. Click **Geometry, Polyhedra, Define**.
2. If desired, click in the **Polyhedron Lable** field and enter a label. Otherwise, click the left button of the mouse to accept the default label (recommended).
3. From the **Reference Entity** drop-down menu, select **RG** (region) or **SF** (surface).

4. Click **Continue**.
5. In the **Region or Surface** field, select the reference region/surface by the mouse or enter its label.
6. In the **Average Element Size** field, enter the desired value. This value will be used later in meshing parts assocoiated with the polyhedron.
7. In the **Tolerance** field, enter the desired value.
8. From the **Redefine Element Size on Boundary** drop-down menu, select **1: Yes**, or **0: No**.
9. Click **OK**.

PHTOL

Geometry > POLYHEDRA > Tolerance

The PHTOL command specifies the tolerance for a pattern of polyhedra. The tolerance value is used for subsequent meshing polyhedra and associated parts.

Beginning polyhedron
Beginning polyhedron in the pattern.

Ending polyhedron
Ending polyhedron in the pattern.

Increment
Increment between polyhedra in the pattern.
(default is 1)

Tolerance
Tolerance value.
(default is 0.0001)

PHPLOT

Geometry > POLYHEDRA > Plot

The PHPLOT command plots a pattern of polyhedrons on the screen.

Beginning polyhedron
Beginning polyhedron in the pattern.
(default is 1)

Ending polyhedron
Ending polyhedron in the pattern.
(default is highest polyhedron number defined)

Increment
Increment between polyhedrons in the pattern.
(default is 1)

Example:

```
PHPLOT, 1, 3, 1
```

This command plots polyhedrons 1, 2 and 3 on the screen.

PHLIST

Geometry > POLYHEDRA > EDITING > List

The PHLIST command lists a pattern of polyhedra. For each polyhedron, the command lists the number of curves, the average element size, tolerance, and a list of the associated surfaces/regions. The element specification will be used during automatic meshing.

Beginning polyhedron
Beginning polyhedron in the pattern.
(default is 1)

Ending polyhedron
Ending polyhedron in the pattern.
(default is the highest polyhedron defined)

Increment
Increment between polyhedra in the pattern.
(default is 1)

Example:

```
PHLIST, 1, 3, 2
```

This command lists polyhedra 1 and 3. For each polyhedron, the command lists the average element size, the tolerance, and the associated surfaces/regions.

PHDEL

Geometry > POLYHEDRA > Delete

The PHDEL command deletes a pattern of polyhedrons from the database. Associated lower entities are not deleted.

Beginning polyhedron
Beginning polyhedron in the pattern.

Ending polyhedron
Ending polyhedron in the pattern.
(default is beginning polyhedra)

Increment
Increment between polyhedrons in the pattern.
(default is 1)

Example:

```
PHDEL, 1, 3, 1
```

This command deletes polyhedrons 1, 2 and 3 from the database.

PHEXTR

Geometry > POLYHEDRA > Extrusion

The PHEXTR command extrudes a pattern of surfaces or regions to generate an equal number of polyhedrons. Identical regions and/or surfaces are created at the end of extrusion. Each generated polyhedron is composed of the two end surfaces or regions in addition to all the surfaces generated by the extrusion. The extrusion is performed along a specified axis of the active Cartesian coordinate system. Three dimensional elements are also generated if the primitive entity is meshed and the default meshing flag is active for polyhedra.

Base type
Name of the primitive entity.

= SF	surfaces
= RG	regions
	(default is RG)

Beginning surface/region
Beginning entity in the pattern.

Ending surface/region
Ending entity in the pattern.
(default is beginning surface/region)

Increment
Increment between entities in the pattern.
(default is 1)

Axis symbol
Axis symbol (X, Y or Z).
(default is Z)

Value
Length of extrusion.

Number of layers of elements
Number of element layers in the extrusion direction. Prompted only if default meshing for polyhedrons is active.
(default is 6)

Original mesh update flag
Flag for handling the original 2D mesh.

= -1: Transfer	transfer the 2-D mesh to the end of extrusion
= 0: Keep	delete the 2-D mesh
= 1: Delete	keep the 2-D mesh
	(default is delete original mesh)

Spacing ratio
Ratio for spacing element layers in the direction of extrusion. Use 1 for uniform

spacing. A value less than one means that element size will progressively decrease along the direction of extrusion. A value greater than one means that the element size will progressively increase in the direction of extrusion. *(default is 1.0)*

The following two arguments will be used only in conjunction with subsequent 3-D automatic meshing of polyhedra and parts.

Average element size for automesh

Average element size for subsequent automatic meshing of the created polyhedra and associated parts. For regions, the default is the average element size used for the first region in the pattern. For surfaces, the default is 20 elements on the boundary of the first surface in the pattern.

Tolerance for polyhedron closure

Tolerance for closing the polyhedra, necessary to close small gaps between the keypoints of surfaces and/or regions that constitute the polyhedron. You can increase this tolerance if the polyhedron definition fails. Default is 20 times the value specified for merging keypoints by the PTTOL (**Geometry, Points, Merge Tolerance**) command.

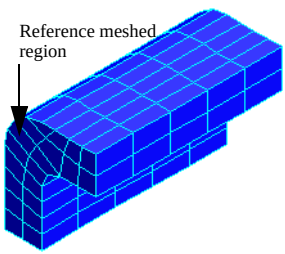
Notes

1. The active coordinate system must be Cartesian.
2. You can activate default meshing for polyhedrons using the STATUS1 table.
3. For proper association of nodes and elements with the generated geometric entities, do not merge nodes prior to issuing this command.

Example

Graphic Example:

The reference region/surface should be meshed with quad (not trinagular) elements. Also the SOLID element group should be activated before meshing.



Use the STATUS1 table to activate default meshing for polyhedra before extruding.
5 Layers of elements are specified.

PHSWEEP

Geometry > POLYHEDRA > Sweeping

The PHSWEEP command sweeps a pattern of surfaces or regions to generate a pattern of polyhedrons. Identical regions and/or surfaces are created at the end of the sweep. Each generated polyhedron is composed of the two end surfaces or regions in addition to all the surfaces generated by the sweep. The sweep is performed

about a specified axis of the active Cartesian coordinate system. Three dimensional elements are also generated if the primitive entity is meshed and default meshing is active for polyhedra.

Base type
Name of the primitive entity.

= SF	surfaces
= RG	regions
	(default is RG)

Beginning region/surface
Beginning entity in the pattern.

Ending region/surface
Ending entity in the pattern.
(default is beginning entity)

Increment
Increment between entities in the pattern.
(default is 1)

Rotation/Sweep axis
Axis symbol (X, Y or Z).
(default is Y)

Angle of the arc
Angle of sweep.
(default is 360.0)

Number of segments
Number of segments.
(default is 1 segment for each 90 degrees)

Number of elements per segment
Number of elements per segment in the sweep direction. Prompted only if default meshing for polyhedrons is active.
(default is 6)

Original mesh update flag
Flag for handling the original 2D mesh.

= -1: Transfer	transfer the original mesh to the newly created surface or region
= 0: Keep	delete the original mesh
= 1: Delete	keep the original mesh
	(default is delete original mesh)

Space ratio
Ratio for spacing element layers in the direction of sweep. Use 1 for uniform spacing. A value less than one means that element size will progressively decrease along the direction of sweep. A value greater than one means that the element size will progressively increase in the direction of sweep.
(default is 1.0)

The following two arguments will be used only in conjunction with subsequent 3-D automatic meshing of polyhedra and parts.

Average element size for automesh

Average element size for subsequent automatic meshing of the created polyhedra or parts (which will use them). For regions, default is the average element size used for the first region in the pattern. For surfaces, the default is 20 elements on the boundary of the first surface in the pattern.

Tolerance of polyhedron closure

Tolerance for closing the polyhedra, necessary to close the gaps (if any) between the keypoints of surfaces and/or regions that constitute a polyhedron. The user may need to increase this tolerance if the polyhedron definition fails. Default is 20 times the value specified for merging keypoints by the "PTTOL" command.

Notes

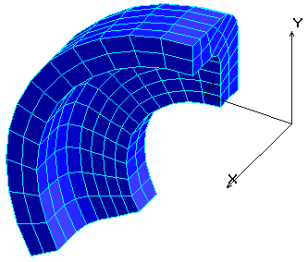
1. The active coordinate system must be Cartesian.
2. Default meshing for polyhedrons can be activated by issuing the "ACTDMESH,PH,1" command.
3. For proper association of nodes and elements with the generated geometric entities, do not merge nodes (using NMERGE) prior to this command.

Example

Graphic Example: **`PHSWEEP, RG, 1, 1, 1, X, 120., 2, 6, 1`**

The above command creates a polyhedron by sweeping region #1, 120.0 degrees about the X-axis of the active coordinate system. 6 layers of elements are specified for each segment.The original 2D mesh is deleted.

The reference region/ surface should be meshed with quad (not trinagular) elements. Also the SOLID element group should be activated before meshing.



Use the STATUS1 table to activate default meshing for polyhedra before extruding.
The region is swept 120 ^{0} (2 segments) about the X axis. 6 layers of elements are specified for each segment.

PHDRAG

Geometry > POLYHEDRA > Dragging

The PHDRAG command drags a pattern of surfaces or regions along a smooth profile to generate a pattern of polyhedrons. Identical regions or surfaces are created at the end of the profile. Each generated polyhedron is composed of the two end surfaces or regions in addition to all the surfaces generated by the drag. The primitive regions or surfaces keep their relative orientation as they move along the profile. Three dimensional elements are also generated if the primitive entity is meshed and

default meshing is active. The user is asked for the number of layers of elements along each curve in the profile.

Base type
Name of the primitive entity.

= SF	surfaces
= RG	regions
	(default is RG)

Beginning region/surface
Beginning entity in the pattern.

Ending region/surface
Ending entity in the pattern.
(default is beginning entity)

Increment
Increment between entities in the pattern.
(default is 1)

Original mesh update flag
Flag for handling the original 2D mesh.

= -1: Transfer	transfer the original mesh to the newly created surface or region
= 0: Keep	delete the original mesh
= 1: Delete	keep the original mesh
	(default is delete original mesh)

Profile curve (i)
ith profile curve. The curves must be specified in order and must form a continuous path. Selecting a curve twice terminates prompting for more curves.

Number of elements on curve (i)
Number of elements on the ith curve in the drag direction. Prompted only if default meshing for polyhedrons is active.
(default is 6)

The following two arguments will be used only in conjunction with subsequent 3-D automatic meshing of polyhedra and parts.

Average element size for automesh
Average element size for subsequent automatic meshing of the created polyhedra or parts (which will use them). For regions, default is the average element size used for the first region in the pattern. For surfaces, the default is 20 elements on the boundary of the first surface in the pattern.

Tolerance for polyhedra closure
Tolerance for closing the polyhedra, necessary to close the gaps (if any) between the keypoints of surfaces and/or regions that constitute a polyhedron. The user may need to increase this tolerance if the polyhedron definition fails. Default is 20 times the value specified for merging keypoints by the "PTTOL" command.

Notes

1. Curves of the profile and the pattern of regions or surfaces to be dragged can

be picked by the mouse.

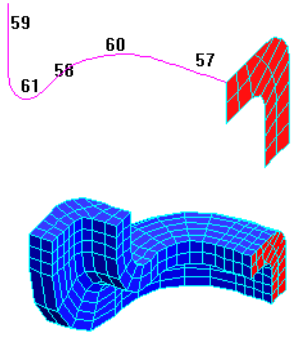
2. The profile curves must have a continuous first derivative (continuous slope). Fillets can be created to smoothen the profile if needed.
3. The GLIDE and DRAG operations are equivalent when the profile is a straight line.
4. Default meshing for polyhedrons can be activated by issuing the "ACTDMESH, PH, 1" command.
5. For proper association of nodes and elements with the generated geometric entities, do not merge nodes (using NMERGE) prior to this command.

Example

```
PHDRAG, RG, 1, 1, 1, 1, 57, 4, 60, 5, 58, 1, 61, 4, 59, 3, 5,
0.0001
```

The above command drags region 1 to generate a polyhedron. The profile is defined by curves 57, 60, 58, 61, and 59. If region 1 is meshed and default meshing is active for polyhedra, then 4 layers of 3-D elements are generated along curve 57, 5 along curve 60, 1 along curve 58, 4 along curve 61, and 3 along curve 59. The original 2D mesh is deleted.

The reference region/ surface should be meshed with quad (not trinagular) elements. Also the SOLID element group should be activated before meshing.



Use the STATUS1 table to activate default meshing for polyhedra before dragging. The region is dragged along the shown profile. The number of elements is specified for each curve profile.

PHGLIDE

Geometry > POLYHEDRA > Gliding

The PHGLIDE command glides a pattern of surfaces or regions to generate a pattern of polyhedra. Identical regions or surfaces are created at the end of the profile. Each generated polyhedron is composed of the two end surfaces/regions and the surfaces generated by the glide. The glide is performed along a specified profile of curves. The curves associated with the primitive surfaces/regions keep their original orientations as they glide along the profile curves, i.e. each curve stays parallel to its original position. 3-D elements are generated from meshed primitive surfaces/

regions if default meshing is active for polyhedra. You will be prompted to specify the number of layers of elements along each curve in the profile.

Base type
Name of the primitive entity.

= SF	surfaces
= RG	regions
	(default is RG)

Beginning surface/region
Beginning entity in the pattern.

Ending surface/region
Ending entity in the pattern.
(default is beginning surface/region)

Increment
Increment between entities in the pattern.
(default is 1)

Original mesh update flag
Flag for updating the original 2-D mesh.

= -1: Transfer	transfer the 2-D mesh to the end of the glide
= 0: Keep	delete the 2-D mesh
= 1: Delete	keep the 2-D mesh
	(default is delete original mesh)

Profile curve (i)
i^{th} profile curve. The curves must be specified in order and must form a continuous path. Selecting a curve twice terminates prompting for more curves.

Number of elements on curve (i)
Number of elements on the ith curve in the glide direction. Prompted only if default meshing for polyhedrons is active.
(default is 6)

The following two arguments will be used only in conjunction with subsequent 3-D automatic meshing of polyhedra and parts.

Average element size for automesh
Average element size for subsequent automatic meshing of the created polyhedra or parts. For regions, the default is the average element size used for the first region in the pattern. For surfaces, the default is 20 elements on the boundary of the first surface in the pattern.

Tolerance for polyhedron closure
Tolerance for closing the polyhedra, necessary to close the gaps (if any) between the keypoints of surfaces and/or regions that constitute the polyhedron. You may need to increase this tolerance if the polyhedron definition fails. Default is 20 times the value specified for merging keypoints by the "PTTOL" command.

Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.

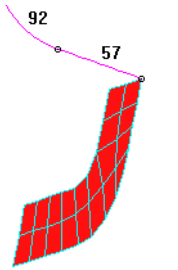
2. The profile curves must have a continuous first derivative. Fillets can be created to smoothen the profile if needed.
3. Gliding is more sensitive to the profile. No profile curve can be parallel to the original region/surface.
4. The glide and drag operations are equivalent if the profile is a straight line.
5. Default meshing for polyhedrons can be activated using the STATUS1 table.
6. For proper association of nodes and elements with the generated geometric entities, do not merge nodes prior to issuing this command.

Example

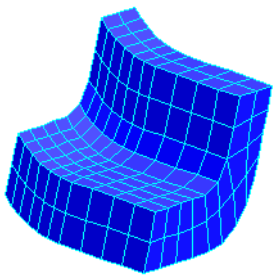
```
PHDGLIDE, RG, 1, 1, 1, 1, 57, 4, 92, 6, 3, 5, 0.0001
```

The above command glides region 1 to generate a polyhedron. The profile is defined by curves 57 and 92. If region 1 is meshed and default meshing is active for polyhedra, then 4 layers of 3-D elements are generated along curve 57 and 6 layers are created along curve 92. The original 2D mesh is deleted..

The reference region/surface should be meshed with quad (not trinagular) elements. Also the SOLID element group should be activated before meshing.



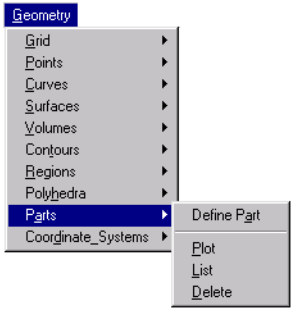
Use the STATUS1 table to activate default meshing for polyhedra before dragging. The region is dragged along the shown profile. The number of elements is specified for each curve profile.



▶ PARTS Menu

Geometry > PARTS

Figure 4-25 Parts Menu



PART

Geometry > PARTS > Define Part

The PART command creates a 3-D solid geometric entity called part. A PART in GEOSTAR is the 3-D space enclosed by a single polyhedron or between a group of POLYHEDRONS. For parts with more than one polyhedron, the outer polyhedron should be selected first. The other polyhedra represent isolated cavities entirely contained in the first one. A part can be meshed by the MA_PART (**Meshing, Auto_Mesh, Parts**)command to generate tetrahedral elements.

Part label
Label of the part to be created. Up to 100 parts can be defined.
(default is highest part defined +1)

Polyhedron (i)
List of polyhedrons that define the part. (Up to 30 polyhedrons can be listed)

PARTPLOT

Geometry > PARTS > Plot

The PARTPLOT command plots a pattern of parts on the screen.

Beginning part
Beginning part in the pattern.
(default is 1)

Ending part

Ending part in the pattern.
(default is highest part number defined)

Increment
Increment between parts in the pattern.
(default is 1)

Example:

```
PARTPLOT, 1, 3, 1
```

This command plots parts 1, 2 and 3 on the screen.

PARTLIST

Geometry > PARTS > List

The PARTLIST command lists a pattern of parts by displaying the number of polyhedra associated with the part and their labels.

Beginning part
Beginning part in the pattern.
(default is 1)

Ending part
Ending part in the pattern.
(default is the highest part number defined)

Increment
Increment between parts in the pattern.
(default is 1)

Example:

```
PARTLIST, 1, 3, 1
```

This command lists parts 1, 2 and 3 on the screen.

PARTDEL

Geometry > PARTS > Delete

The PARTDEL command deletes a pattern of parts from the database. Associated lower entities are not deleted. The polyhedra and lower entities that make up the parts will not be deleted.

Beginning part
Beginning part in the pattern.

Ending part
Ending part in the pattern.
(default is beginning part)




Increment
Increment between parts in the pattern.
(default is 1)

Example:

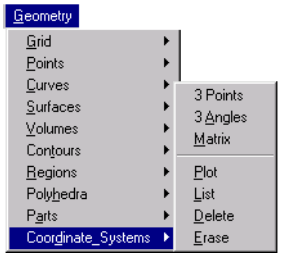
```
PARTDEL, 1, 3, 1
```

This command deletes parts 1, 2 and 3 from the database.

▶ COORDINATE_SYSTEMS Menu

Geometry > COORDINATE SYSTEMS

Figure 4-26 Coordinate Systems Menu



This menu includes commands to define coordinate systems. Three types of coordinate systems can be defined - namely, Cartesian, cylindrical, and spherical. The short names of all commands in this menu start with "CS". The rest of the characters describe the function of the command. Coordinate systems 0, 1 and 2 are readily defined. System 0 is the default and refers to the global Cartesian coordinate system. Systems 1 and 2 refer to the predefined cylindrical and spherical coordinate systems, respectively. Coordinate systems 3 through 5000 can be defined by the user and are referred to as user or local coordinate systems.

CSYS

Geometry > COORDINATE SYSTEMS > 3 Points

The CSYS command defines a local coordinate system based on 3 keypoints. The defined system becomes the active coordinate system.

Coordinate system
Coordinate system label. (must be between 3 and 5000)
(default is maximum coordinate system defined + 1)

Coordinate system type
Type of local coordinate system.

= 0: Cartesian	Cartesian coordinate system
= 1: Cylindrical	Cylindrical coordinate system
= 2: Spherical	Spherical coordinate system
	(default is Cartesian)

Keypoint at origin
Keypoint at the origin of coordinate system.

Keypoint on X-axis
Keypoint to determine the positive X-axis of the new coordinate system.

Keypoint on X-Y plane
Keypoint on the X-Y-plane of the new coordinate system.

Notes

1. Cylindrical and spherical coordinate systems can be used to apply loads and boundary conditions, define material properties, create and list nodes and keypoints. Displacements and stresses can be requested in any coordinate system. Generation of geometric entities, nodes, and elements must be done while a Cartesian coordinate system is active.
2. Coordinate systems 1 and 2 are predefined global cylindrical and spherical coordinate systems, respectively with axes coinciding with that of system 0.

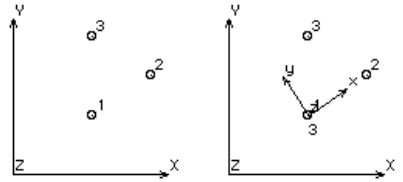
Examples

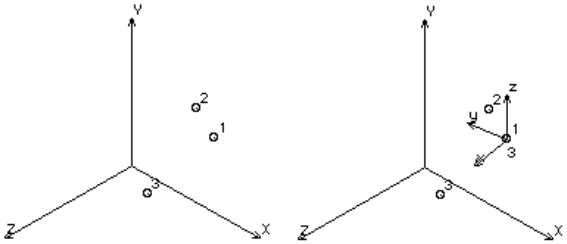
Graphic Example 1: CSYS

```
PLANE,Z,0,1
VIEW,0,0,1,
PT,1,20,15,0,
PT,2,35,25,0,
PT,3,20,0,35,0,
CSYS,3,0,1,2,3,
```

Graphic Example 2: CSYS

```
PT,1,50,40,0,
PT,2,50,60,10,
PT,3,20,0,10,
CSYS,3,0,1,3,2,
```





CSANGL

Geometry > COORDINATE SYSTEMS > 3 Angles

The CSANGL command defines a local coordinate system based on specified origin location and threerotations about global Cartesian axes. This system becomes the currently active coordinate system.

Coordinate system
Coordinate system label. (must be between 3 and 5000)
(default is maximum coordinate system defined + 1)

Coordinate system type
Type of local coordinate system.

= 0: Cartesian	Cartesian coordinate system
= 1: Cylindrical	Cylindrical coordinate system
= 2: Spherical	Spherical coordinate system
	(default is Cartesian)

X, Y, Z-Coordinate value of origin
X, Y, Z-coordinate of origin of the new local coordinate system.
(default is 0.0)

X, Y, Z-Rotation
Angles of rotation about the global Cartesian coordinate system in the right-handed basis. (must be in degrees)
(defaults are 0.0)

Rotation order
Order of applying rotations.

= XYZ in RHC	rotations are applied in the following order: x rotation, y rotation, and z rotation
= ZYX in RHC	rotations are applied in the following order: z rotation, y rotation, and x rotation
	(default is XYZ in RHC)

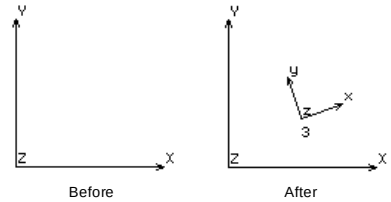
Notes

1. Cylindrical and spherical coordinate systems can be used in the application of loads and boundary conditions, definition of material properties, and the creation (and listing) of nodes and keypoints. Generation of geometric entities, nodes, and elements must be made in reference to a Cartesian coordinate system. Displacements and stresses can be requested in any coordinate system.
2. Coordinate systems 1 and 2 are predefined to be cylindrical and spherical coordinate systems respectively with axes coinciding with that of system 0.
3. Rotations follow the right-hand rule convention.

Example

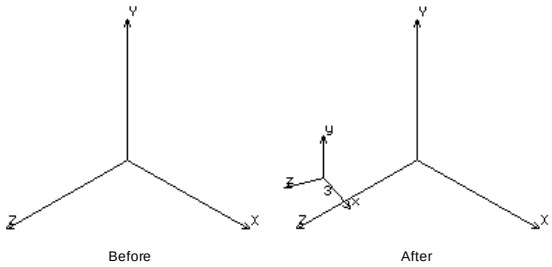
Graphic Example: CSANGL

```
PLANE,Z,0,1,
VIEW,0,0,1,
CSANGL,3,0,30,20,0,0,0,20,
```



Graphic Example 2: CSANGL

```
CSANGL,3,0,-20,0,40,0,-20,0,
```



CSMATRIX

Geometry > COORDINATE_SYSTEMS > Matrix

The CSMATRIX command defines a local coordinate system based on a specified {4x3} transformation matrix. The first three rows of this matrix define the orientations of the new axes, while the fourth row establishes its origin relative to the global Cartesian coordinate system. This system becomes the currently active coordinate system.

CSDEL

Geometry > COORDINATE_SYSTEMS >Plot

The CSPLOT command plots a pattern of coordinate systems.

Beginning Coordinate System
Beginning coordinate system label in the pattern.
(Default is 1)

Ending Coordinate System:
Ending coordinate system label in the pattern.
(Default is coordinate system with highest label defined)

Increment
Increment between coordinate system labels in the pattern.
(Default is 1)

CSLIST

Geometry > COORDINATE_SYSTEMS >List

The CSPLIST command lists a pattern of coordinate systems.

Beginning Coordinate System
Beginning coordinate system label in the pattern.
(Default is 1)

Ending Coordinate System:
Ending coordinate system label in the pattern.
(Default is coordinate system with highest label defined)

Increment
Increment between coordinate system labels in the pattern.
(Default is 1)

CSDEL

Geometry > COORDINATE_SYSTEMS >Delete

The CSPDEL command deletes a pattern of coordinate systems.

Beginning Coordinate System
Beginning coordinate system label in the pattern.

Ending Coordinate System
Ending coordinate system label in the pattern.
(Default is beginning coordinate system)

Increment
Increment between coordinate system labels in the pattern.
(Default is 1)




CSREF2D

Geometry > COORDINATE_SYSTEMS >Reference for 2D Model

The CSREF2D command sets the coordinate system to be used in defining the xy-plane for PLANE2D and TRIANG elements. All such elements must be defined entirely in the xy-plane of this coordinate system which must be Cartesian. This command should be given before running the analysis if a system other than the global coordinate system will be used. For axisymmetric elements, the y-axis of this coordinate system will be used as the axis of symmetry. If this command is not used to specify a coordinte system other than the global system, the elements must be in the global XY-plane and the global Y-axis is used as the axis of symmetry.

The stress results in the output file are printed in terms of the global coordinate system. When plotting and listing stresses, the specified reference coordinate system is used by default, otherwise the global coordinate system is used.

Reference coordinate system for 2D models
Cartesian coordinate system for defining PLANE2D and TRIANG elements..
(Default is the active coordinate system)

To instruct the program to write stresses in the output file in reference to the reference coordinate system, you must: 1) choose to write stresses in the local coordinate system when defining the PLANE2D and/or TRIANG element group(s) (Op.4), and 2) use the reference coordinate system specified in this command as the ECS for all such elements (refer to the EPROPCHANGE command), and 3) request the printing of the stresses for the desired elements using the PRINT_ELIST command.

CSREF2DLIST

Geometry > COORDINATE_SYSTEMS >Reference for 2D Model

The CSREF2DLIST command lists the current coordinate system specified by the CSREF2D command as well as its transformation matrix with respect to the global coordinate system.

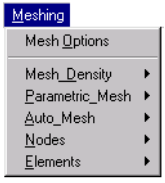
5 Meshing Menu

MESHING Menu

MESHING

This menu contains commands related to the creation of the finite element mesh (nodes and elements). Nodes and elements can be created by direct definition, or by meshing geometric entities.

Figure 5-1 Meshing Menu



The created elements are associated with the active element attributes. The element attributes in COSMOS/M are the element group (EG), material property set (MP), real constant set (RC), and the element coordinate system (ECS). It is best to activate the proper attributes before every meshing command. Meshing commands can be aborted by pressing the “Escape” key.

MESH_OPS

Meshing > Mesh_Options

This command sets options for the meshing algorithm.

Boundary parameterization

System to be used in the parameterization of the boundary curves of the model.

1:UV	2-D parametric space. Use with Pro/Engineer of Parametric Technology Corp., MicroStation Modeler of Bentley Systems, and I-DEAS of SDRC.
0:XYZ	3-D Cartesian coordinate system. Use with CV (ComputerVision), UG (UniGraphic) and CATIA.
	(default is1:UV)

UV-parametric iteration for Boundary

Controls whether iterations are to be used to generate a mesh with a uniform element size.

1: Yes	Use iterations
0: No	Do not use iterations
	(default 1: Yes)

Note:

Always use UV-Parametric iteration when XYZ is selected for boundary parameterization. Activation of UV-Parametric iteration slows down meshing and is not recommended when UV is selected for boundary curve parametrization.

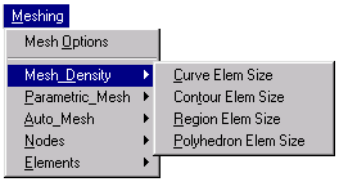



▶ MESH_DENSITY Menu

Meshing > MESH_DENSITY

This menu includes commands to set the element size for specific curves, contours, regions, and polyhedra. Use this menu when using a small uniform element size generates too many elements.

Figure 5-2 Mesh_Density Menu



CRDENSITY

Meshing > MESH_DENSITY > Curve Elem Size

The CRDENSITY command specifies the average element size for all curves in the specified pattern that are currently associated with contours. The command affects the subsequent meshing of regions, polyhedra, and parts only.

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Average element size
Local average element size to be assigned.

Tolerance
Tolerance. Used to modify mesh specification for neighboring curves. You may need to specify a higher tolerance for if geometry has been imported from other packages.
(default is 0.0001)

Example: `CRDENSITY, 1, 5, 1, 5.0,,`

This command resets the average element size to 5.0 units for any curve labeled 1 through 5 that is associated with a contour.

CTDENSITY

Meshing > MESH_DENSITY > Contour Elem Size

The CTDENSITY command resets the average element size for all curves associated with the contours in the specified pattern. The command affects the subsequent meshing of regions, polyhedra, and parts only.

Beginning contour
Beginning contour in the pattern.

Ending contour
Ending contour in the pattern.
(default is beginning contour)

Increment
Increment between contours in the pattern.
(default is 1)

Average element size
Local average element size to be assigned.

Tolerance
Tolerance. Used to modify mesh specification for neighboring curves. You may need to specify a higher tolerance if geometry has been imported from other packages.
(default is 0.0001)

Meshing redefinition flag
Meshing redefinition flag. Prompted only if the contour being defined shares curves with existing contours for mesh compatibility.

= 0: Previous — for curves common to other contours, use same element size or number of elements as previously specified

= 1:Redefine — for curves common to other contours, reset the element size as specified in this command.

= 2:Max elements — for curves common to other contours, compare old and new mesh specifications and select the option that specifies a finer mesh (more elements)

= 3:Min elements — for curves common to other contours, compare old and new mesh specifications and select the option that specifies a coarser mesh (less elements)
(default is 1:Redefine)

Example:
```
CTDENSITY, 1, 5, 1, 5.0, ,
```
This command resets the average element size for contours 1 through 5, to 5.0 for subsequent meshing of regions, polyhedra and parts.

RGDENSITY

Meshing > MESH_DENSITY > Region Elem Size

The RGDENSITY command resets the average element size for all regions in the specified pattern. The command affects the subsequent meshing of regions, polyhedra, and parts only.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Average element size
Local average element size to be assigned.

Tolerance
Tolerance. Used to modify mesh specification for neighboring curves of contours. You may need to specify a higher tolerance if geometry has been imported from other packages.
(default is 0.0001)

Redefinition criterion
Meshing redefinition flag. Used for compatibility along the boundaries. Prompted only if a conflict exists between new and old element sizes specified.

= 0:Previous	use the same element size or number of elements as previously specified for meshed boundaries
= 1:Redefine	change the element size or number of elements to comply with the new element size for meshed boundaries
= 2:Max elements	compare old and new mesh sizes and select the option that results in a finer mesh
= 3:Min elements	compare old and new mesh sizes and select the option that results in a coarser mesh

(default is 1: Redefine)

Example:
```
RGDENSITY, 1, 10, 1, 5.0, , ,
```
This command resets to 5.0, the average element size for regions 1 through 10. 5.0 for subsequent meshing of regions, polyhedra and parts. Mesh density on boundary curves is also updated.

PHDENSITY

Meshing > MESH_DENSITY > Polyhedron Elem Size

The PHDENSITY command resets the average element size for all regions associated with the polyhedra in the specified pattern. The command affects the subsequent meshing of regions, polyhedra, and parts only.

Beginning polyhedron
Beginning polyhedron in the pattern.

Ending polyhedron
Ending polyhedron in the pattern.
(default is beginning polyhedron)

Increment
Increment between polyhedrons in the pattern.
(default is 1)

Average element size
Local average element size to be assigned.

Tolerance
Tolerance. Used to modify mesh specification for neighboring curves of contours. You may need to specify a higher tolerance if geometry has been imported from other packages.
(default is 0.0001)

Example: `PHDENSITY, 1, 1, 1, 5.0,`

This command resets the average element size for all regions associated with polyhedra 1 to 5.0. Subsequent meshing of regions, polyhedra, and parts will be affected by this command.

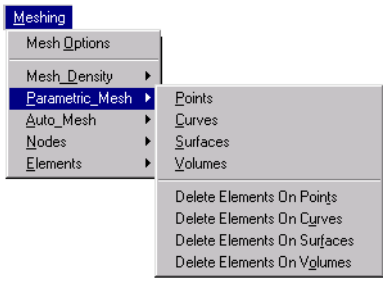



▶ PARAMETRIC_MESH Menu

Meshing > PARAMETRIC_MESH

This menu contains commands to mesh parametric entities (curves, surfaces and volumes). You can also mesh keypoints to generate one-node elements.

Figure 5-3 Parametric_Mesh Menu



M_PT

Meshing > PARAMETRIC_MESH > Points)

The M_PT command meshes a pattern of keypoints creating a 1-node element at each keypoint in the pattern. One-node elements include concentrated MASS elements, and 1-node GAP and SPRING elements. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the “Escape” key.

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.

Increment
Increment between keypoints in the pattern.
(default is 1)

Notes

1. Keypoints can be selected by the mouse only if they are plotted on the screen.
2. Activate the proper element group and real constant set before meshing.

Example: `M_PT, 11, 16, ,`
This command creates 1-node elements at keypoints 11 through 16.

To generate spring elements at a pattern of keypoints:

1. Click **PropSets, Element Group** to define 1-node spring element. Make sure to select 1 from the the number of nodes drop-down menu.
2. Click **Propsets, Real Constant** to define the stiffness of the spring in the X, Y, and Z directions.

No material property set is used for SPRING elements.

3. Click **Meshing, Parametric_Mesh, Points**.
4. Specify the pattern of keypoints.
5. Click **OK**.

To generate Mass elements at a pattern of keypoints:

1. Click **PropSets, Element Group** to define **MASS** element.
2. Click **PropSets, Real Constant** to define the masses and rotary inertias in the X, Y, and Z directions, and the thermal capacity (used in heat transfer analysis only).

No material property set is used for MASS elements.

3. Click **Meshing, Parametric_Mesh, Points**.
4. Specify the pattern of keypoints.
5. Click **OK**.




M_CR

Meshing > PARAMETRIC_MESH > Curves

The M_CR command meshes a pattern of curves creating the specified number of 1-D elements along each curve in the pattern. 1-D elements include TRUSS2D, TRUSS3D, BEAM2D, BEAM3D, PIPE, SPRING, etc. You can generate elements with uniform or non-uniform lengths. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the "Escape" key.

Beginning curve
Beginning curve in the pattern.
(default is 1)

Ending curve
Ending curve in the pattern.
(default is the highest curve label)

Increment
Increment between curves in the pattern.
(default is 1)

Number of nodes per element
Number of nodes per element.
= 3 3-node elements (used for BEAM3D)
= 2 2-node elements
(default is 3)

Number of elements on each curve
Number of elements on each curve in the pattern.
(default is 2)

Spacing ratio
Ratio of element lengths. The spacing ratio is defined as the length of the last element on each curve divided by the length of the first element on the curve. Use the STATUS1 table to activate the display of the direction of curves before meshing with nonuniform spacing.
(default is 1)

Keypoint to define principal axis
Keypoint to define the third node for BEAM3D elements. The third node defines the orientation of the beam. Prompted only if the number of nodes per element is set to 3.

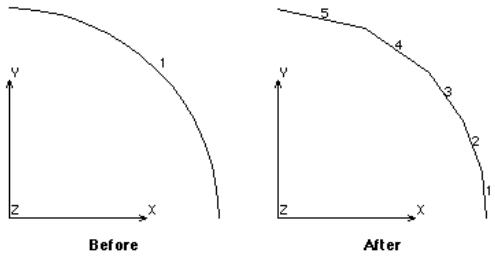
Note
The curves can be picked by the mouse only if they are plotted on the screen.

Example:
```
M_CR, 2, 3, , , 4, 1.2
```
This command creates 4 elements on curve 2, and 4 elements on curve 3. The last element on each curve is 1.2 times longer than the first ele-

ment. Elements get progressively shorter towards the end of the curve.

Graphic Example 1: M_CR

```
PT,1,0,0,0,
PT,2,70,0,0,
PT,3,0,70,0,
CRARC,1,2,3,1,70,
VIEW,0,0,1,
M_CR,1,1,1,2,5,2,
```



To mesh a pattern of curves with non-uniform BEAM3D elements:

1. Click **PropSets, Element Group** to define BEAM3D elements.
2. Click **Propsets, Real Constant** to define the properties of the cross section of the beam. You can instead select the **AISC Sect. Table** to select a standard cross-section, or select **Beam Section** to select a predefined section shape or define your own section.
3. Click **PropSets, Pick Material Library** to define the material of the beam.
4. Click the **STATUS1** button in the Geo Panel window. Check the **Mark** checkbox for curves (**CR**). The CRREPAR (**Geometry, Curves, Manipulate, Reparameterize**) can be used to reverse directions of curves.
5. Click **Edit, Plot, Curves** to plot the desired curves. An arrow is plotted to show the direction of each curve.
6. Click **Meshing, Parametric_Mesh, Curves**.
7. Specify the pattern of curves.
8. From the **Number of Nodes Per Element** drop-down menu, select **3**.
9. Specify the desired number of elements on each curve.
10. Specify the desired spacing ratio. A spacing ratio larger than 1 generates elements that are longer towards the end of the curve.
11. Click **OK**.

M_SF

Meshing > PARAMETRIC_MESH > Surfaces

The M_SF command meshes a pattern of surfaces creating 2-D elements. The generated elements can be triangular or quad. The *Number of Nodes per Element* entry determines the possible type of elements that will be generated as shown in the table below. TRIANG elements should only be used when meshing planar surfaces in the X-Y plane. The command can be used to generate elements of uniform or non-uniform sizes. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the "Escape" key.

Number of nodes	Possible Element Groups
3	TRIANG, SHELL3, SHELL3T, and SHELL3L
4	PLANE2D, MAG2D, FLOW2D, SHELL4, SHELL4T, and SHELL4L
6	TRIANG, SHELL6,
8	PLANE2D, SHELL9, and SHELL9L (without central node)
9	SHELL9 and SHELL9L (with central node)

Beginning surface
Beginning surface in the pattern.
(default is 1)

Ending surface
Ending surface in the pattern.
(default is the highest surface label)

Increment
Increment between surfaces in the pattern.
(default is 1)

Number of nodes per element (3, 4, 6, 8, 9)
Number of nodes per element. Valid entries are 3, 6 (triangular) or 4, 8 and 9 (quadrilateral). The active element attributes are assigned to the elements.
(default is 4)

Number of elements on first curve
Number of elements on the first parametric curve of the surface.
(default is 2)

Number of elements on second curve
Number of elements on the second parametric curve of the surface.
(default is 2)

Spacing ratio for first curve
Ratio of the space between the last two nodes to the space between the first two nodes on the first parametric curve of the surface.
(default is 1.0)

Spacing ratio for second curve
Spacing ratio along the second parametric curve of the surface.
(default is 1.0)

Notes

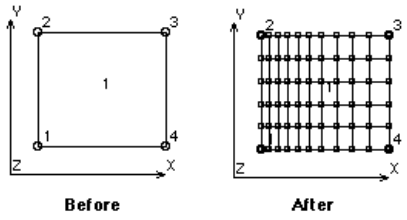
1. Surface can be selected by the mouse if they are plotted on the screen.
2. For convenience, the corresponding curve is highlighted when prompting for the number of elements.
3. Plane elements should only be used with planar surfaces in the X-Y plane.

To mesh a pattern of surfaces with uniform composite SHELL4L elements:

1. Click **PropSets, Element Group** to define **SHELL4L** elements. Specify the number of layers.
2. Define all material property sets that are used in the composite shell.
3. Click **Propsets, Real Constant** to define distance from the reference plane to the top face, temperature gradient, thickness, material set, and material angle for each layer.
4. Click **Edit, Plot, Surfaces** to plot the desired surfaces.
5. Click **Meshing, Parametric_Mesh, Surfaces**.
6. Specify the pattern of surfaces.
7. Specify the desired number of elements on each of the two parametric directions.
8. Accept the default specify ratio in each of the two parametric directions for uniform element size.
9. Click **OK**.

Graphic Example 1: M_SF

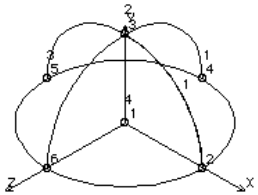
```
PLANE,Z,0,1,

VIEW,0,0,1,
```



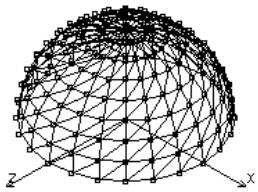
```
   SF4CORD,1,10,10,0,10,50,0,55,50,0,55,10,0,
M_SF,1,1,1,4,5,10,1,2.5,
```

Graphic Example 2: M_SF

```
   PT,1,0,0,0,

   PT,2,70,0,0,

   PT,3,0,70,0,

   CRARC,1,2,3,1,70,

   SFSWEEP,1,1,1,Y,360,4,
M_SF,1,4,1,3,8,6,1,1,
```



Before



After

You should not mesh curved surfaces with plane elements, use shells instead.

M_VL

Meshing > PARAMETRIC_MESH > Volumes

The M_VL command meshes a pattern of volumes creating 3-D elements. The command can be used to generate elements with 4, 8, or 20 nodes. Elements with 4 nodes are tetrahedral elements (TETRA4 or TETRA4R). Elements with 8 nodes are low-order brick elements (SOLID, SOLIDL, MAG3D, FLOW3D). Elements with 20 nodes are second-order bricks. Second-order tetrahedral elements (TETRA10) can be created from first-order elements using the ECHANGE (**Meshing, Elements, Element Order**) command. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the “Escape” key.

Number of nodes	Possible Element Groups
4	TETRA4 and TETRA4R
8	SOLID, SOLIDL, SOLIDPZ, FLOW3D, MAG3D (without mid-side nodes)
20	SOLID, SOLIDL and SOLIDPZ (with mid-side nodes)

Beginning volume
Beginning volume of the pattern.
(default is 1)

Ending volume
Ending volume of the pattern.
(default is the highest volume label)

Increment
Increment between volumes in the pattern.
(default is 1)

Number of nodes per element (4, 8, 20)
Number of nodes per element. Valid entries are.
= 4 first-order tetrahedral elements
= 8 first-order brick elements
= 20 second-order brick elements
(default is 8)

Number of elements on the first curve
Number of elements on the first parametric curve of the volume.
(default is 2)

Number of elements on the second curve
Number of elements on the second parametric curve of the volume.
(default is 2)

Number of elements on the third curve
Number of elements on the third parametric curve of the volume.
(default is 2)

Spacing ratio for first curve
Ratio of the length of the last element to the first element on the first parametric curve.
(default is 1.0)

Spacing ratio for second curve
Ratio of the length of the last element to the first element on the second parametric curve.
(default is 1.0)

Spacing ratio for third curve
Ratio of the length of the last element to the first element on the third parametric curve.
(default is 1.0)

Notes:

1. Entities can be picked by the mouse only if they are plotted.
2. It is recommended to check the **Mark** checkbox for curves (CR) in the STATUS1 table and plot the pattern of volumes to display the parametric coordinates of the volumes. To simplify the process, the corresponding parametric curve is highlighted when prompting for the number of elements.
3. Elements with collapsed nodes are generated by the program when it is not possible to create regular brick elements.

To mesh a pattern of volumes with uniform SOLID elements:

1 Click **PropSets, Element Group** to define **SOLID** elements.

2 Define or activate the proper material property set.

No real constant set is required for 3-D elements.

3 Click **Edit, Plot, Volumes** to plot the desired surfaces.

4 Click **Meshing, Parametric_Mesh, Volumes**.

5 Specify the pattern of volumes.

6 Specify the desired number of elements on each of the three parametric directions.

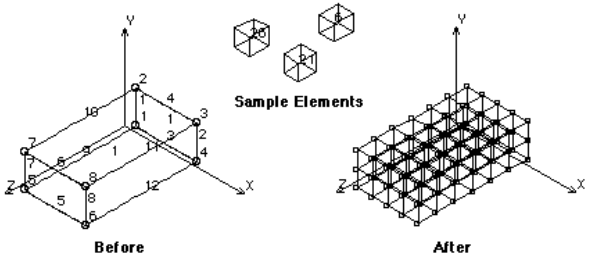
7 Accept the default specify ratio in each of the three parametric directions for uniform element size.

8 Click **OK**.

Graphic Example: M_VL

```
PLANE,Z,0,1,

VIEW,0,0,1,

SF4CORD,1,10,5,0,10,35,0,65,35,0,65,5,0,

VLEXTRUDE,1,1,1,Z,100,

VIEW,1,1,1,
```

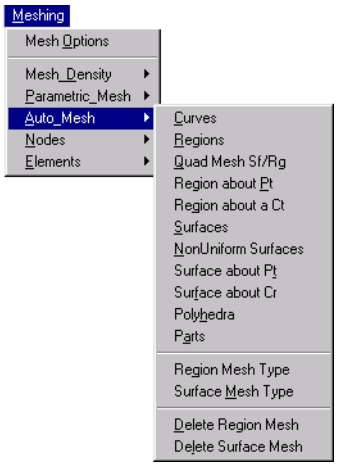
```
M_VL,1,1,1,8,2,4,6,1,1,1,
```



▶ AUTO_MESH Menu

Meshing > AUTO_MESH

Figure 5-4 Auto_Mesh Menu



This menu deals with automatic meshing of curves and the triangulation of surfaces and regions. Commands to change the generated mesh are also provided.

MA_CR

Meshing > AUTO_MESH > Curves

The MA_CR command meshes a pattern of curves with 1-D elements. The command can be used to create 1-D elements like TRUSS3D, BEAM2D, BEAM3D, and PIPE. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the "Escape" key.

Beginning curve
Beginning curve in the pattern.
(default is 1)

Ending curve
Ending curve in the pattern.
(default is the highest curve label)

Increment
Increment between curves in the pattern.
(default is 1)

Average element size
Average element length.

Number of nodes per element
Number of nodes per element.
= 3 3-node elements (BEAM3D)
= 2 2-node elements (BEAM2D, TRUSS3D, etc.)
(default is 3)

Keypoint to define principal axis
Keypoint to define the third node for BEAM3D elements. Prompted only if the number of nodes per element is set to 3.

MA_RG

Meshing > AUTO_MESH > Regions

The MA_RG command meshes a pattern of regions by generating triangular elements. The *Element Order* entry determines the possible element group to be associated with the generated elements as shown in the table below. Planar regions can be meshed with TRIANG or shell elements. TRIANG elements should only be used when meshing planar regions in the X-Y plane. The element size information is based on the information provided while creating the contours or as specified by the commands in the Mesh_Density menu. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the "Escape" key.

Element Order	Possible Element Groups
Low	TRIANG, SHELL3, SHELL3T, and SHELL3L (without mid-side nodes)
High	TRIANG and SHELL6 (with mid-side nodes)

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Number of smoothing iterations
Number of smoothing iterations. Use '0' for non-planar regions.
(default is 3)

Method
Meshing method.

= 0:Sweeping	sweeping advancing front technique (meshing advances from the boundaries to the inside)
= 1:Hierarchical	hierarchical advancing front technique (parallel layers in one direction)

(default is hierarchical)

Element order

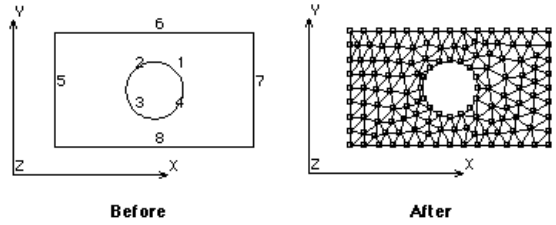
Element order

= 0: Low	low order elements (no mid-side nodes)
= 1: High	high order elements (with mid-side nodes)
	(default 0: Low)

Notes

1. The smoothing iterations option is used for the fine adjustment of the generated nodes so that every node, surrounded by a group of elements, is approximately at their center. Usually, no more than 5 iterations are needed.
2. The generated triangular elements can be replaced by a mesh of 4-, 8-node quadrilateral, or 6-node triangular elements by the use of the MARGCH command.
3. Regions can be picked by the mouse only if they are plotted.
4. Meshing problems could occur if unrealistic requirements are specified. In such cases, change the requirements or use a smaller element size all through.
5. If the hierarchical technique fails, try the sweeping technique.

Graphic Example 1: Planar Region

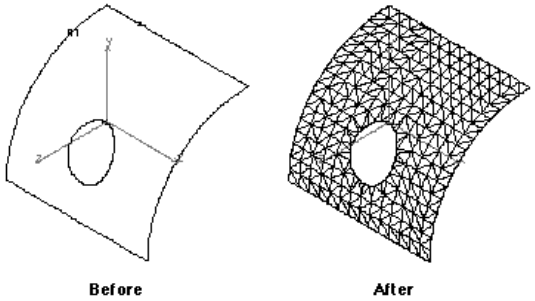



```
PLANE,Z,0,1,

VIEW,0,0,1,

PT,1,50,30,0,

PT,2,90,30,0,

CRPCIRCLE,1,1,2,10,360,4,

CRPCORD,5,15,10,0,  5,50,0,85,50,0,85,10,0,15,10,0,

CT,1,0,5,1,5,

CT,2,0,5,1,1,

RG,1,2,1,2,
MA_RG,1,1,1,3,
```

Graphic Example 2: Region on Underlying Surface

```
PT,1,0,50,0,

CREXTR,1,1,1,X,70,

SFSWEEP,1,1,1,X,90,1,

CSANGL,3,0,35,25,0,0,0,0,0,
```

```
PT,7,0,10,0,

CREXTR,7,7,1,Z
    ,100,

SFSWEEP,5,5,1,
    Z,360,12,

CRINTSS,1,2,13
    ,1,0.001,

CT,1,0,5,1,1,0
    ,

CT,2,0,3,1,41,
    0,

RG,1,2,1,2,1,

CLS;

RGPLOT,1,1,1
MA_RG,1,1,1,0,0,
```



MA_QUAD

Meshing > AUTO_MESH > Quad Mesh Sf/Rg

The MA_QUAD command meshes a pattern of surfaces or regions with quad elements. The *Number Of Nodes Per Element* entry determines the possible type of generated elements as shown in the table below. PLANE2D elements should be used only when meshing planar regions in the X-Y plane. Non-planar regions should be meshed with shell elements. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the “Escape” key.

Element Order	Possible Element Groups
Low	PLANE2D, SHELL4, SHELL4T, SHELL4L, FLOW2D, or MAG2D (without mid-side nodes)
High	PLANE2D, SHELL9, and SHELL9L (with mid-side nodes but without central node)
High w/Cen Node	SHELL9, and SHELL9L (with mid-side nodes and a central node)

Geometric entity type
Entity type.
= SF surfaces
= RG regions
(default is SF)

Beginning surface/region
Beginning surface/region in the pattern.

Ending surface/region
Ending surface/region in the pattern.
(default is beginning surface/region)

Increment
Increment between surfaces/regions in the pattern.
(default is 1)

Element order
Element order.
= 0:Low low order elements (4-nodes)
= 1:High high order elements (8-nodes)
= 2:High w/cen node high order with a node at the center (9-nodes)
(default is 0:Low)

Push boundary node flag
Flag to push mid-side nodes to the geometric boundary.
= 0:No do not push nodes to the boundary
= 1:Yes push nodes to the boundary
(default is 1:Yes)

Number of smoothing iterations
Number of smoothing iterations. Use '0' for non-planar regions.
(default is 3)

Method
Meshing method.
= 0:Sweeping sweeping advancing front technique (meshing

advances from the boundaries to the inside)

= 1:Hierarchical — hierarchical advancing front technique (parallel layers in one direction)
(default is 1: Hierarchical)

Notes

1. The smoothing iterations option is used for the fine adjustment of the generated nodes so that every node surrounded by a group of elements, is approximately, at their center. Usually, no more than 5 iterations are needed.
2. If the hierarchical technique fails, try the sweeping technique.

Example: `MA_QUAD, RG, 1, 5, 1, 6, 2;`

This command automatically meshes regions 1 through 5 to generate 9-node quad elements of average size 6.

MA_PTRG

Meshing > AUTO_MESH > Region about Pt

The MA_PTRG command meshes a planar defined by a single contour region with reference to a keypoint on the region. The command generates plane TRIANG, SHELL3, SHELL3T, or SHELL3L elements. The size of the elements will be the smallest around the reference keypoint and gets progressively larger away from it (unless other mesh control options are used). The spacing ratio parameter sets the ratio of the element size around the keypoint to the element size on the farthest boundary. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the “Escape” key.

Region
Region to be meshed.

Reference keypoint
Reference keypoint to be used as the meshing center.

Spacing ratio
Spacing ratio, defined as the average element size nearest to the reference keypoint divided by the element size on the farthest boundary. If a value greater than 1.0 is entered, its inverse will be used (smaller elements around the keypoint).
(default is 0.5)

Number of smoothing iterations
Number of smoothing iterations.
(default is 0)

Boundary node redefinition flag
Boundary nodal redefinition flag.
= 0:Redefine — redefine nodes on the boundaries to be compatible with adja-

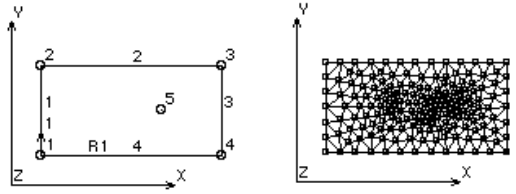
cent regions
= 1:Do not redefineignore nodal compatibility with adjacent region meshes
(default is 1: Do not redefine)

Notes

1. The smoothing iterations option is used for the fine adjustment of the generated nodes so that every node, surrounded by a group of elements is, approximately located at their center. Usually, no more than 5 iterations are needed.
2. The generated triangular elements can be changed to quadrilateral elements using the MARGCH (**Meshing, Auto_Mesh, Region Mesh Type**) command.
3. The Echange (**Meshing, Elements, Element Order**) command can be used to change the order of elements by adding or removing mid-side nodes.
4. Elements nearer to the reference keypoint will always be smaller in size. The actual "spacing ratio" value used by the program is always less than 1.0. If a larger value is specified, its inverse will be used (e.g "spacing ratio" of 10.0 is similar to 0.1).
5. Meshing problems could occur if unrealistic requirements are specified. In such cases change the requirements or use smaller elements all through.

Graphic Example 1: MA_PTRG

```
PLANE,Z,0,1,

VIEW,0,0,1,

CRPCORD,1,10,10,0,10, 40,0,70,40,0,70,10, 0,10,10,0,

CT,1,0,5,1,1,

RG,1,1,1,

PT,5,50,25,0,
MA_PTRG,1,5,.3,5,1,
```



Before **After**

MA_CTRG

Meshing > AUTO_MESH > Region about Ct

The MA_CTRG command creates a finite element mesh of triangular elements for a specified pattern of flat regions. In order for this command to work properly, each region must have *one and only one inner contour*. The mesh radiates from the inner to the outer contour. The mesh radiates from the inner contour to the outer contour. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the “Escape” key.

Element Order	Possible Element Groups
Low	TRIANG, SHELL3, SHELL3T, and SHELL3L (without mid-side nodes)
High	TRIANG and SHELL6 (with mid-side nodes)

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Number of smoothing iterations
Number of smoothing iterations.
(default is 3)

Method
Meshing method.

= 0:Sweeping	sweeping advancing front technique (meshing advances from the boundaries to the inside)
= 1:Hierarchical	hierarchical advancing front technique (parallel layers in one direction) *(default is 1:Hierarchical)*

Element order
Element order

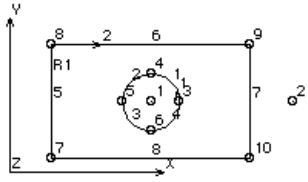
= 0: Low	low order elements (no mid-side nodes)
= 1: High	high order elements (with mid-side nodes) *(default 0: Low)*

Notes

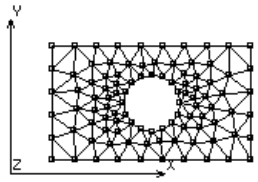
1. The smoothing iterations option is used for the fine adjustment of the generated nodes so that every node surrounded by a group of elements, is approximately located at their center. Usually, no more than 5 iterations are needed.
2. Regions can be picked by the mouse only if they are plotted.
3. The generated triangular elements can be replaced by a mesh of 4- or 8-node quadrilateral, or 6-node triangular elements by the use of the MARGCH command.
4. Meshing problems could occur if unrealistic requirements are specified. In such cases change the requirements or use a smaller element size all through.
5. The Echange (**Meshing, Elements, Element Order**) command can be used to change the order of elements by adding or removing mid-side nodes.

Graphic Example: MA_CTRG

```
PLANE,Z,0,1,

VIEW,0,0,1,

PT,1,50,25,0,

PT,2,100,25,0,

CRPCIRCLE,1,1,2,10,360,4,

CRPCORD,5,15,5,0,15,45,0,85,45,0,85,5,0,15,5,0,

CT,1,0,4,1,1,

CT,2,0,8,1,6,

RG,1,2,2,1,
MA_CTRG,1,1,1,5,
```



Before



After

MA_SF

Meshing > AUTO_MESH > Surfaces

The MA_SF command meshes a pattern of surfaces creating 3-noded triangular TRIANG or shell elements. Planar surfaces can be meshed with TRIANG or shell elements. Non-planar surfaces should be meshed with shells. TRIANG elements should only be used when meshing planar surfaces in the X-Y plane. You can specify the average element size, or the approximate number of elements on the boundary. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the "Escape" key.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Mesh flag
Flag for meshing base.

= 0:Element size	meshing is based on average element size
= 1:Number of elements	meshing is based on number of elements on boundary *(default is 0:Element size)*

Average element size/approx number of elements on the boundary
Average element size (if mesh flag = 0:Element size) or approximate number of elements on the boundary (if mesh flag = 1:Number of elements).

Method
Meshing method.

= 0:Sweeping	sweeping advancing front technique (meshing advances from the boundaries to the inside)
= 1:Hierarchical	hierarchical advancing front technique (parallel layers in one direction) *(default is 0:sweeping)*

Notes

1. The generated triangular elements can be replaced by a mesh of quadrilateral elements by the use of the MASFCH command.
2. Surfaces can be picked by the mouse if they are plotted.
3. The Echange (**Meshing, Elements, Element Order**) command can be used to change the order of elements by adding or removing mid-side nodes.

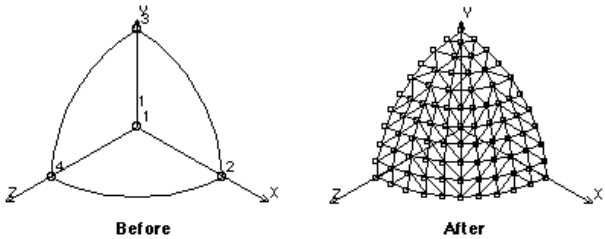
Example:

```
MA_SF, 1, 5, 1, 0, 5
```

This command automatically generates a finite element mesh of 3-node triangular elements for surfaces 1 through 5. An average element size of 5.0 is specified.

Graphic Example 1: MA_SF

```
PT,1,0,0,0,
PT,2,70,0,0,
PT,3,0,70,0,
CRARC,1,2,3,1,70,
SFSWEEP,1,1,1,Y,-90,1,
MA_SF,1,1,1,0,12,
```



MA_NUSF

Meshing > AUTO_MESH > NonUniform Surfaces

The MA_NUSF command meshes a pattern of surfaces creating 3-noded triangular TRIANG or shell elements. The program prompts for the number of elements on each of the 4 curves. Planar surfaces can be meshed with TRIANG or shell elements. Non-planar surfaces should be meshed with shells. TRIANG elements should only be used when meshing planar surfaces in the X-Y plane. The order of the generated elements can be changed using the ECHANGE (**Meshing, Elements, Change Order**) command. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the "Escape" key.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.

(default is beginning surface)

Increment

Increment between surfaces in the pattern.
(default is 1)

Number of elements on curve (i)

Number of elements on the ith side. (i=1,2,3,4)
(default is 4)

Method

Meshing method.

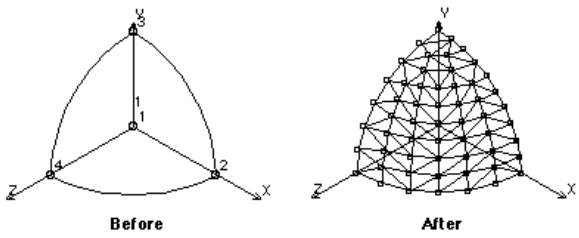
= 0:Sweeping	sweeping advancing front technique (meshing advances from the boundaries to the inside)
= 1:Hierarchical	hierarchical advancing front technique (parallel layers in one direction) *(default is 0: Sweeping)*

Notes

1. Surfaces can be picked by the mouse.
2. The generated triangular elements can be replaced by a mesh of quadrilateral elements by the use of the MASFCH command.
3. The corresponding side is conveniently highlighted when prompting for the number of elements.
4. The command can be conveniently used to mesh a pattern of surfaces when the side numbering is easily determined. Otherwise, it is recommended that you issue the command for each single surface separately.
5. The Echange (**Meshing, Elements, Element Order**) command can be used to change the order of elements by adding or removing mid-side nodes.

Graphic Example 1: MA_NUSF

```
PT,1,0,0,0,

PT,2,70,0,0,

PT,3,0,70,0,

CRARC,1,2,3,
    1,70,

SFSWEEP,1,1,
    1,Y,-90,1,
MA_NUSF,1,1,1,10,6,7,
```



MA_PTSF

Meshing > AUTO_MESH > Surface about Pt

The MA_PTSF command meshes a planar surface with respect to a reference keypoint creating 3-noded TRIANG, SHELL3, SHELL3T, or SHELL3L elements. The command prompts for the desired size of element around and away from the reference keypoint. The keypoint must be a corner point, or in the central region of the surface such that each of its parametric coordinates is between 0.10 and 0.90. Planar surfaces can be meshed with TRIANG or shell elements. Non-planar surfaces should be meshed with shells. TRIANG elements should only be used when meshing a planar surface in the X-Y plane. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the "Escape" key.

Surface
Surface to be meshed.

Keypoint
Reference keypoint.

Near element size (1.0)
Size of elements nearest to the reference keypoint.

Far element size (1.0)
Size of elements farthest from the reference keypoint.

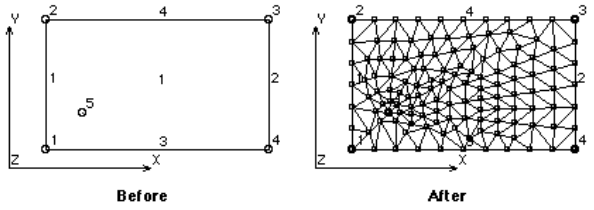
Notes

1. The surface and the reference keypoint can be picked by the mouse.
2. The element size changes progressively between the specified values.
3. No mesh is generated if the keypoint is too close to a side of the surface and it is not a corner point, i.e. if any of the parametric coordinates is outside of the range 0.10 to 0.90. In cases where such a mesh is desired, the surface can be broken to allow the desired mesh.
4. The Echange (**Meshing, Elements, Element Order**) command can be used to change the order of elements by adding or removing mid-side nodes.

Graphic Example:

MA_PTSF

```
PLANE,Z,0,1,

VIEW,0,0,1,
```



```
SF4CORD,1,10,5,0,10,40,0,70,40,0,70,5,0,

PT,5,20,15,0,
MA_PTSF,1,5,3,6,
```

MA_CRSF

Meshing > AUTO_MESH > Surface about Cr

The MA_CRSF command meshes a planar surface with respect to a reference curve creating 3-noded TRIANG, SHELL3, SHELL3T, or SHELL3L elements. The command prompts for the desired size of element around and away from the reference keypoint. The keypoint must be a corner point, or in the central region of the surface with parametric coordinates between 0.10 and 0.90. Planar surfaces can be meshed with TRIANG or shell elements. Non-planar surfaces should be meshed with shells. TRIANG elements should only be used when meshing a planar surface in the X-Y plane

Surface
Surface to be meshed.

Curve
Reference curve, it must be one of the curves of the surface.

Near element size (1.0)
Size of elements nearest to the reference curve.

Far element size (1.0)
Size of elements farthest from the reference curve.

Notes

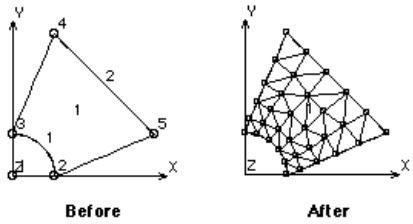
1. The surface and the reference side can be picked by the mouse if they are plotted.
2. The element size changes progressively between the specified values.
4. The Echange (**Meshing, Elements, Element Order**) command can be used to change the order of elements by adding or removing mid-side nodes.

Example: `MA_CRSF, 1, 5, 4.0, 12.0`

This command generates a triangular 3-node finite element mesh for surface 1 using curve number 5 as a reference. The elements nearest to curve 5 have an average size of 4.0, and the elements farthest from it have an average size of 12.0. Curve 5 must be a side of surface 1.

Graphic Example 1: MA_CRSF

```
PLANE,Z,0,1,
VIEW,0,0,1,
PT,1,0,0,0,
PT,2,15,0,0,
PT,3,0,15,0,
CRARC,1,2,3,1,15,
CR2CORD,2,15,50,0,50,15,0,
SF2CR,1,1,2,0,
MA_CRSF,1,1,5,10,
```



MA_PH

Meshing > AUTO_MESH > Polyhedra

The MA_PH command meshes a pattern of polyhedrons generating a mesh of 3-noded triangular shell elements. A polyhedron is the skin (surface area) of a 3-D solid. The associated element group should be compatible with the element order as shown in the table below. Plane elements may not be used when meshing polyhedra. The average element size that will be used in meshing can be listed by the PHLIST (**Edit, List, Polyhedra**) command. To change the element size, use the PHDENSITY (**Meshing, Mesh_Density, Polyhedron Element Size**) command. Other commands in the **Meshing, Mesh_Density** menu can be used to locally control the mesh on the boundary. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the "Escape" key.

Element Order	Possible Element Groups
Low	SHELL3, SHELL3T, and SHELL3L (without mid-side nodes)
High	SHELL6 (with mid-side nodes)

Beginning Polyhedron
Beginning polyhedron in the pattern.

(default is 1)

Ending Polyhedron
Ending polyhedron in the pattern.
(default is beginning polyhedron)

Increment
Increment between polyhedrons in the pattern.
(default is 1)

Element order
Element order
= 0: Low low order elements (no mid-side nodes)
= 1: High high order elements (with mid-side nodes)
(default 0: Low)

MA_PART

Meshing > AUTO_MESH > Parts

The MA_PART command meshes a pattern of parts generating tetrahedral elements. The associated element group should be compatible with the element order as shown in the table below. The average element size of associated polyhedra can be listed by the PHLIST (**Edit, List, Polyhedra**) command. To change the element size, use the PHDENSITY (**Meshing, Mesh_Density, Polyhedron Element Size**) command. Other commands in the **Meshing, Mesh_Density** menu can be used to locally control the element size on the boundary. The generated elements are associated with the active element attributes. The meshing process can be aborted by pressing the "Escape" key.

Element Order	Possible Element Groups
Low	TETRA4 and TETRA4R (without mid-side nodes)
High	TETRA10 (with mid-side nodes)

Beginning part
Beginning part in the pattern.
(default is 1)

Ending part
Ending part in the pattern.
(default is beginning part)

Increment
Increment between parts in the pattern.

(default is 1)

Hierarchy check flag
Hierarchy check flag.

= 0:No	do not check the quality of neighboring elements
= 1:Yes	check the quality of neighboring elements
	(default is 0:No)

Element order
Order of the tetrahedral elements to be generated.

= 0:Low	4-node tetrahedral
= 1:High	10-node tetrahedral
	(default is 0: Low)

Number of smoothing iterations
Number of smoothing iterations to improve initial node position selection.
(default is 4)

Number of aspect ratio check iterations
Number of iterations to check the aspect ratios of elements.

= 0	do not check
=1 to 4	check, number of iterations
	(default is 0)

Jacobian check flag
Jacobian check to improve positioning of mid side nodes on edges of high order elements.

= 0	No check.
= 1	1 Gauss point check.
= 2	4 Gauss points check.
= 3	16 Gauss points check.
= 4	29 Gauss points check.
= 5	At nodes check.
	(default is 0)

MARGCH

Meshing > AUTO_MESH > Region Mesh Type

The MARGCH command changes the mesh of a pattern of regions from from 3-node triangular elements to a 6-node triangular, or 4-, 8-, or 9-node quadrilateral elements. The command can also be used to change 4-node elements to 8- or 9-node elements. If a triangular mesh is changed to quadrilateral, all related loading and boundary conditions are deleted, and you must specify them again if desired.

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.

(default is beginning region)

Increment

Increment between regions in the pattern.
(default is 1)

Element type

Type of elements in the new mesh.

= T	triangular
= Q	quadrilateral
	(default is Q)

Total element nodes

Number of nodes in each new element. The valid numbers are 6 for triangular elements and 4, 8 or 9 for quadrilateral elements.
(default is 4 for quadrilateral and 6 for triangular elements)

Push flag

Flag to push new intermediate boundary nodes to the geometric boundary. Not prompted if 4-node quad elements are generated.

= 0:No	do not push nodes to the boundary
= 1:Yes	push nodes to the boundary
	(default is 1:Yes)

Associated element group

Element group with which the new elements will be associated.
(default is the active element group)

Shape factor

Element shape factor defined as the ratio of the shortest side to the longest side of the element. Ratios less than 0.3 allow sharp elements that might cause numerical problems and hence are not recommended. Prompted only if quad elements are generated from triangular elements.
(default is 0.4)

Quad element flag

All-quad/mixed elements option. Not prompted if triangular 6-node elements will be generated.

= 1:Mixed	allow generation of quad- and triangular-looking elements
= 2:All quad	all elements will be quad-looking
	(default is all quad)

Number of smoothing iterations if all quad

Number of smoothing iterations to improve initial node position selection.
(default is 3)

Notes

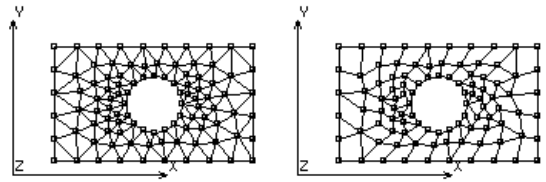
1. The *Associated Element Group* argument lets you change the element group with which the new elements will be associated.
2. The command cannot change a 4-node or a 6-node elements to 3-node elements. Use the ECHANGE (**Meshing, Elements, Element Order**) command for this purpose.
3. Regions can be picked by the mouse only if they are plotted.




4. Elements that cannot be merged with adjacent elements to form new acceptable quad elements are left unchanged unless the all quad-looking option is used. If mixed elements are allowed when changing a mesh from triangular to quad, all new elements will have the same number of nodes specified; triangular-looking elements will have collapsed nodes.

Graphic Example 1: MARGCH

```
PLANE,Z,0,1,

VIEW,0,0,1,

PT,1,50,25,0,

PT,2,100,25,0,

CRPCIRCLE,1,1,2,10,360,4,

CRPCORD,5,15,5,0,15,45,0,85,45,0,85,5,0,15,5,0,

CT,1,0,4,1,1,

CT,2,0,8,1,6,

RG,1,2,2,1,

MA_CTRG,1,1,1,5,
MARGCH,1,1,1,Q,4,1,0.4,
```



MASFCH

Meshing > AUTO_MESH > Surface Mesh Type

The MASFCH command changes the mesh of a pattern of surfaces from from 3-node triangular elements to 6-node triangular, or 4-, 8-, or 9-node quadrilateral elements. The command can also be used to change 4-node elements to 8- or 9-node elements. If a triangular mesh is changed to quadrilateral, all related loading and boundary conditions are deleted, and you must specify them again if desired.

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Element type
Type of elements in the new mesh.

= T	triangular
= Q	quadrilateral
	(default is Q)

Total element nodes
Number of nodes in each new element. The valid numbers are 6 for triangular elements and 4, 8 or 9 for quadrilateral elements.
(default is 4 for quadrilateral and 6 for triangular elements)

Push flag
Flag to push new intermediate boundary nodes to the geometric boundary. Not prompted if 4-node quad elements are generated.

= 0:No	do not push nodes to the boundary
= 1:Yes	push nodes to the boundary
	(default is 1:Yes)

Associated element group
Element group with which the new elements will be associated.
(default is the active element group)

Shape factor
Element shape factor defined as the ratio of the shortest side to the longest side of the element. Ratios less than 0.3 allow sharp elements that might cause numerical problems and hence are not recommended. Prompted only if quad elements are generated from triangular elements.
(default is 0.4)

Quad element flag
All-quad/mixed elements option. Not prompted if triangular 6-node elements will be generated.

= 1:Mixed	allow generation of quad- and triangular-looking elements
= 2:All quad	all new elements will be quad-looking
	(default is all quad)

Number of smoothing iterations if all quad
Number of smoothing iterations to improve initial node position selection.
(default is 3)

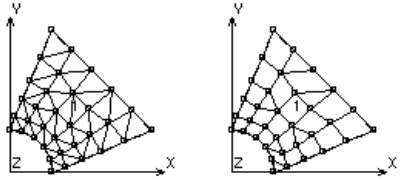
Notes

1. The *Associated Element Group* argument lets you change the element group with which the new elements will be associated.

2. This command cannot change a 4-node or a 6-node elements to 3-node elements. Use the ECHANGE (**Meshing, Elements, Element Order**) command for this purpose.
3. Surfaces can be picked by the mouse only if they are plotted.
4. Elements that cannot be merged with adjacent elements to form new acceptable quad elements are left unchanged unless the all quad-looking option is used. If mixed elements are allowed when changing a mesh from triangular to quad, all new elements will have the same number of nodes specified; triangular-looking elements will have collapsed nodes.

Graphic Example 1: MASFCH

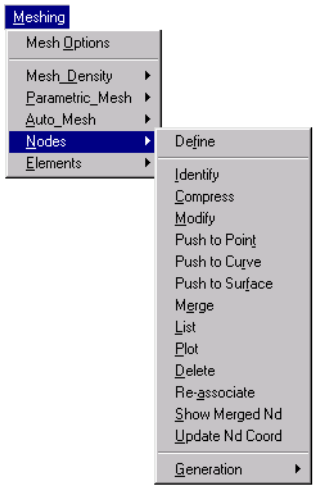
```
PLANE,Z,0,1,

VIEW,0,0,1,

PT,1,0,0,0,

PT,2,15,0,0,

PT,3,0,15,0,

CRARC,1,2,3,1,15,

CR2CORD,2,15,50,0,50,15,0,

SF2CR,1,1,2,0,

MA_CRSF,1,1,5,10,
MASFCH,1,1,1,Q,4,1,0.4,
```



▶ NODES_Menu

Meshing > NODES

Figure 5-5 Nodes Menu



This menu contains commands related to the creation of new nodes or the modification of existing ones.

ND

Meshing > NODES > Define

The ND command creates a new node at the specified location in the active coordinate system and associates it with the specified entities. The node can be specified by its coordinates or it can be snapped to the active grid. The command can also be used to modify the coordinates of an existing node.

Node
Label of node to be created.
(default is the highest node label defined + 1)

X, Y, Z-Coordinate value
Coordinate of the node in the X, Y, Z direction.
(defaults are 0.0)

Associated keypoint
Keypoint to associate the created node with.

Associated curve
Curve to associate the created node with.

Associated surface
Surface to associate the created node with.

Associated volume
Volume to associate the created node with.

Associated contour
Contour to associate the created node with.

Associated region
Region to associate the created node with.

Example: `ND, 34, 2.5, 7.2, 3.0,, 23, 3,,,`
This command creates node 34 at the location specified in the active

coordinate system. The node is associated with curve 23 and surface 3. For example, if later forces are applied to curve 23 or surface 3, they will also be applied to node 34.

NMODIFY

Meshing > NODES > Modify

The NMODIFY command modifies the coordinates of nodes in a pattern. The modification can be specified in an absolute or relative manner in any coordinate system.

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Absolute/relative flag
Absolute/relative flag.
= 0:Absolute nodes are relocated at the specified absolute coordinates
= 1:Relative nodes are moved to their new positions by the specified relative displacement values
(default is 0: Absolute)

X, Y, Z-Coordinate value
Triplet coordinates.
(defaults are 0.0)

Coordinate system
Coordinate system.
(default is 0)

Example:
```
NMODIFY, 1, 10, 1, 1, 1., -0.1, 1., 0,
```
This command modifies the coordinates of nodes 1 through 10 by adding 1.0, -0.1 and 1.0 to their x, y and z current coordinate values, respectively. The modification is done with respect to the global Cartesian coordinate system.




NPTPUSH

Meshing > NODES > Push to Point

The NPTPUSH command pushes a node to a keypoint. It replaces the coordinates of the node with that of the keypoint while preserving element connectivity.

Node to be pushed
Node to be pushed.

Keypoint
Keypoint to which node is pushed.

Association flag
Association change flag.
= 0:No do not change the present association of the node with other entities
= 1:Yes cancel the association of the node with other entities
(default is 0:No)

Notes

1. Entities can be picked by the mouse only if they are plotted.
2. This command is intended for minor node modifications. Although the command works for large distances as well, the quality of the finite element mesh can be harmed if nodes are pushed large distances. A distance exceeding half of the element size is considered large.
3. You must be careful in specifying the association flag since it could change the assigned forces, temperatures, boundary conditions or other quantities (see example below).

Example: `NPTPUSH, 1, 5`

This command pushes node number 1 to keypoint number 5. The command does not change the association of node 1. To illustrate this point, let node 1 be associated with curve number 10 before it is pushed to keypoint number 5. Now if a force of 2 units is assigned to all nodes of curve 10, this node 1 will be included (in its new position), even though it is not geometrically on curve 10 anymore.

NCRPUSH

Meshing > NODES > Push to Curve

The NCRPUSH command pushes all the nodes in the specified pattern to the specified neighboring curve.

Beginning Node
Beginning node in the pattern.

Ending Node
Ending node in the pattern.

(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Curve
Curve to which the pattern of nodes is to be pushed to.

Relative tolerance
Relative convergence tolerance.
(default is 0.001)

Association flag
Association change flag.
= 0:No do not cancel the present association of the nodes in the pattern with other entities
= 1:Yes cancel the current association of the nodes in the pattern with other entities
(default is no)

Notes

1. The nodes and the curve can be picked by the mouse only if they are plotted.
2. GEOSTAR tries to push each node to the closest point on the specified curve within a small tolerance.
3. This command is intended for minor node modifications. Convergence will not take place if the curve is too far from the nodes being pushed, and the quality of the finite element mesh can be harmed even if convergence occurs. A distance exceeding half of the element size is considered large.
3. You must be careful in specifying the association flag since it could change the assigned forces, temperatures, boundary conditions or other quantities (see example below).

Example:
```
NCRPUSH, 1, 5, 3, 8, , 0
```
This command pushes nodes 1 and 4 to the corresponding closest points on curve 8. The command does not change the association of these nodes. To illustrate this point, let node 1 be associated with curve number 10 before it is pushed to curve 8. Now if a force of 2 units is assigned to all nodes of curve 10, node 1 will be included (in its new position), even though it is not geometrically on curve 10 anymore.

NSFPUSH

Meshing > NODES > Push to Surface

The NSFPUSH command pushes all the nodes in the specified pattern to the specified neighboring surface.

Beginning Node
Beginning node in the pattern.

Ending Node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Surface
Surface to which the pattern of nodes is to be pushed to.

Relative tolerance
Relative convergence tolerance.
(default is 0.001)

Association flag
Association change flag.
= 0:No do not cancel the present association of the nodes in the pattern with other entities
= 1:Yes cancel the current association of the nodes in the pattern with other entities
(default is no)

Notes

1. The nodes and the surface can be picked by the mouse only if they are plotted.
2. GEOSTAR tries to push each node to the closest point on the specified surface within a small tolerance.
3. This command is intended for minor node modifications. Convergence will not take place if the curve is too far from the nodes being pushed, and the quality of the finite element mesh can be harmed even if convergence occurs. A distance exceeding half of the element size is considered large.
3. You must be careful in specifying the association flag since it could change the assigned forces, temperatures, boundary conditions or other quantities (see example below).

Example:
```
NCRPUSH, 1, 5, 3, 8, , 0
```
This command pushes nodes 1 and 4 to the corresponding closest points on surface 8. The command does not change the association of these nodes. To illustrate this point, let node 1 be associated with curve number 10 before it is pushed to surface 8. Now if a force of 2 units is assigned surface 10, node 1 will be included even though it does not geometrically lie on surface 10 anymore.

NMERGE

Meshing > NODES > Merge

The NMERGE command causes coinciding nodes (nodes whose coordinates are within a specified tolerance) to be merged. Merging can be

performed with respect to all nodes or with respect to the nodes in the specified pattern only.

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node of the pattern.
(default is the max. node label defined)

Increment
Increment between the nodes in the pattern.
(default is 1)

Tolerance
Magnitude of tolerance. Nodes are merged only if the difference between their corresponding X-, Y- and Z-coordinates are less than or equal to the specified tolerance.
(default is .0001)

All/Among flag
All/among merging flag.
= 0:All consider all nodes in the model
= 1:Among consider the nodes specified in the pattern only
(default is 0:All)

Echo flag
Echo flag.
= 1:On GEOSTAR issues a message whenever two nodes are merged
= 0:Off GEOSTAR does not issue messages
(default is 1:On)

Low/High flag
Low/high flag.
= 0:Low merge higher labels with lower labels of nodes such that if two nodes are merged, the node with the higher label is deleted
= 1:High merge lower labels with higher labels of nodes such that if two nodes are merged, the node with the lower label is deleted
(default is 0:Low)

Notes

1. If you plan to generate finite element meshes from existing meshed geometric entities by operations like extrusion, sweeping, flipping, moving, etc., then it is recommended (for proper association of the mesh with geometric entities) to postpone the use of the NMERGE command until all such operations have been performed.
2. Nodes can be selected by the mouse.
3. Element connectivity, load, and boundary conditions are automatically updated after merging.
4. If two nodes are merged and both of them have prescribed loads or boundary conditions, then the highest magnitude for each component is kept.

5. If the loads or boundary conditions on the two merged nodes are specified in different coordinate systems, the prescribed conditions for the kept node are enforced and the prescribed conditions for the deleted node are ignored.
6. If merging with respect to the nodes in the pattern is specified, then other nodes (nodes not specified in the pattern) are not checked for merging.
7. If a selection for nodes is active, then only nodes in the selection set will be considered.
8. Merging coinciding nodes ensures the continuity of the model. If two coinciding nodes are not merged, then instability problems may occur due to discontinuity. There are many situations however, where coinciding nodes should not be merged.

Example: `NMERGE, 1, 64, 1, 0.0001, 0, 1, 0`

The nodes in the set of 1 through 64 are merged if any two or more nodes are within the specified tolerance. Let node 7 and 15 be spaced within 0.0001. Then node 15 is merged with node 7. Similarly, if nodes 41, 53 and 62 are spaced within 0.0001, then nodes 53 and 62 are merged with node 41. Nodes 15, 53, and 62 are deleted.

SHOW_MERGE

Meshing > NODES > Show merged Nd

The SHOW_MERGE command may be used to display the free edges of a model, or the nodes that will be merged using a given tolerance. The command does not merge any nodes but will show the nodes that will be merged if the NMERGE command is issued with the same tolerance. The command is very useful in showing disconnected parts of a model.

Edge plot criterion
Display criterion.
0: Free edgedisplay free edges. Edges with coinciding nodes are also plotted. The user may then use the NMERGE command to merge such nodes if desired
1: Merge tolerancedisplay discontinuities by connecting nodes whose x-, y- and z-coordinates are within the tolerance to be specified

Echo flag
Display labels of nodes within the merging tolerance.
= 0:Off do not display
= 1:On display
(default is 1: On)

Merge tolerance
Merge tolerance. Prompted only if “edge plot criterion” is 1.
(default is 0.001)

NREASSOC

Meshing > NODES > Re-associate

The NREASSOC command reassociates nodes with surfaces or regions. A node is reassociated only if it is associated with a face of an element which is associated with the surface or the region. The command is useful in cases where multiple boundary curves sharing end keypoints exist since association may be lost when nodes on such curves are merged.

Entity type
Geometric entity type.
= 0: Surface
= 1: Region
(default is 0: Surface)

Beginning surface/region
Beginning node in the pattern.

Ending surface/region
Ending node in the pattern.
(default is beginning surface/region)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `NREASSOC, RG, 1,5,1`

The above command re-associates all nodes which are not currently associated with regions 1 through 5 but are associated with some faces of elements which in turn are associated with these regions.

NDUPDATE

Meshing > NODES > Update Nd Coord

The NDUPDATE command updates the nodal coordinates, for plotting, to the deformed locations or restores them to the initial locations. The command is only effective when used after a successful run by STAR or NSTAR in which updating the nodal coordinates has been requested by the A_STATIC command for STAR or the A_NONLINEAR command for NSTAR. The updated nodal coordinates will always be used for analysis purposes in subsequent runs.

Geometry update
Flag to update or restore nodal coordinates.
= 0: Restore restore original nodal coordinates

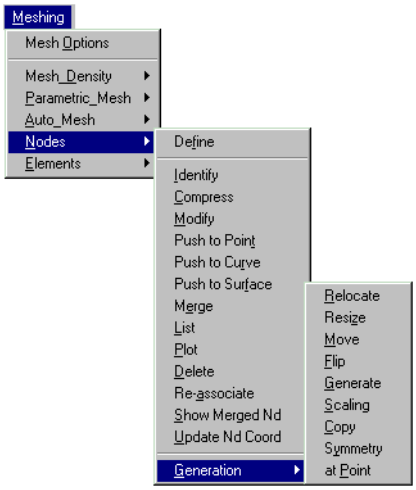
= 1: Update update nodal coordinates
(default is 1: Update)

▼GENERATION Menu

Meshing > NODES > GENERATION MENU

This menu contains commands related to the generation of new nodes using existing ones through a number of operations.

Figure 5-6 Generation Menu



NDRELOC

Meshing > NODES > GENERATION MENU > Relocate)

The NDRELOC command relocates a pattern of nodes by a specified translation and/or rotation about the currently active Cartesian coordinate system. Nodes associated with elements are not relocated unless the hierarchy check flag is deactivated.

Beginning node
Beginning node in the pattern.
Ending node

Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Generation flag
Generation flag.

0: Translation only	for translation
1: Rotation only	for rotation
2: Both	for translation and rotation
	(default is 0: Translation only)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations (degrees) in the currently active Cartesian coordinate system.
(defaults are 0.0)

Hierarchy check flag
Hierarchy check flag.

0: No	do not check hierarchy. Relocate all specified nodes
1: Yes	check hierarchy. Only relocate nodes that are not used in element connectivity
	(default is 1: Yes)

Notes

1. Nodes can be picked by the mouse.
2. The relocated nodes will be free from any association with geometric entities.

Example 1:
```
NDRELOC, 1, 6, 1, 0, 5.0, 5.0, 0.0, 0,
```
The above command translates nodes 1 through 6 from their current locations by X = 5.0, Y = 5.0 and Z = 0.0.

Example 2:
```
NDRELOC, 1, 6, 1, 1, 30.0, 10.0, 10.0, 0,
```
The above command rotates nodes 1 through 6 from their current locations by x-rotation = 30.0, y-rotation = 10.0 and z-rotation = 10.0 degrees about the currently active X, Y, and Z coordinate axes respectively.

NDRESIZE

Meshing > NODES > GENERATION MENU > Resize

The NDRESIZE command resizes a pattern of existing nodes, by scaling and translating or rotating them relative to an active Cartesian coordinate

system. Note that this command does not generate additional nodes but scales existing ones. Nodal coordinates are scaled as specified by the x, y and z scale factors.

The command is particularly useful in two situations:

1. Changing the units of a model.
2. Overcoming precision problems when modeling parts with small dimensions. Artificial dimensions can be used to build the geometry and generate the mesh and then use scale factors to resize.

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Generation flag
Generation flag.

0:Translation only	for translation
1: Rotation only	for rotation
	(default is 0: Translation only)

X, Y, Z-Scale factor
The scale factors in the X-, Y-, and Z-directions.
(defaults are 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted for only if the generation flag is set to translation.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations (degrees) in the currently active Cartesian coordinate system. Prompted for only if the generation flag is set to rotation.
(defaults are 0.0)

Hierarchy check flag
Hierarchy check flag.

0: No	do not check hierarchy. Resize all specified nodes
1: Yes	check hierarchy. Only resize nodes that are not used in element connectivity
	(default is check hierarchy)

Note
The active coordinate system must be Cartesian.

Example: `NDRESIZE, 1,1000, 1, 1./25.4, 1./25.4, 1./25.4,,,,`

The above command multiplies the nodal coordinates of nodes 1 through 1000 in the x, y and z directions by 1./25.4. Such factors would change the dimensions of the mesh from millimeters to inches.

NDMOVE

Meshing > NODES > GENERATION MENU > Move

The NDMOVE command moves a pattern of nodes from the current coordinate system to the destination coordinate system.

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Destination Coordinate System
Destination coordinate system label.
(default is 0)

Hierarchy check flag
Hierarchy check flag.

0: No	do not check hierarchy. Move all specified nodes
1: Yes	check hierarchy. Only move nodes which are not used in element connectivity *(default is 1: Yes)*

Notes

1. Nodes can be picked by the mouse.
2. The moved nodes will be free from any association with geometric entities.

Example: `NDMOVE, 1, 6, 1, 3, 0,`

The above command moves nodes 1 through 6 to local coordinate system 3 with the same coordinates.

NDFLIP

Meshing > NODES > GENERATION MENU > Flip

An offset can be specified with the flipping. Only floating nodes (nodes that have not been used to define elements) can be flipped. Flipping elements automatically

flips all associated nodes. This command does not generate any new nodes. The NDSYM (**Meshing, Nodes, Generation, Symmetry**) command can be used to generate new nodes from existing ones.

Beginning Node
Beginning node in the pattern.

Ending Node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Axis normal to plane
Axis representing normal to plane about which the pattern of nodes is to be flipped.

= X	Y-Z plane
= Y	Z-X plane
= Z	X-Y plane
	(default is Z)

Offset
Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

Hierarchy check flag
Hierarchy check flag.

0: No	do not check hierarchy. Flip all specified nodes
1: Yes	check hierarchy. Only flip nodes that are not used in element connectivity
	(default is 1: Yes)

Notes

1. Nodes can be picked by the mouse.
2. The flipped nodes will be free from any association with geometric entities.
3. If an offset is specified, then it is applied to all nodes of the specified pattern.

Example: `NDFLIP, 1, 6, 1, X, 0.0, 0,`

The above command flips nodes 1 through 6 about the Y-Z plane. No offset is specified.

NDGEN

Meshing > NODES > GENERATION MENU > Generate

The NDGEN command generates one or more patterns of nodes from an existing pattern, by translating and/or rotating about the currently active Cartesian coordinate system. The NDRELOC (**Meshing, Elements, Generation, Relocate**) command can be used to relocate existing nodes without generating new ones.

Generating new elements using the ELGEN (**Meshing, Elements, Generation, Generate**) command automatically generates new nodes.

Generation number
Number of patterns to be generated. Must be > 0 for generation to occur.
(default is 1)

Beginning Node
Beginning nodes in the pattern.

Ending Node
Ending nodes in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Generation flag

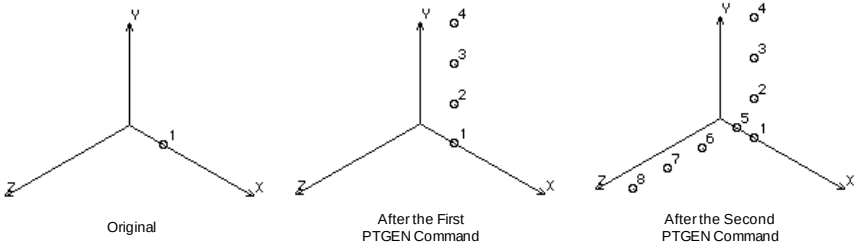
= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
Magnitudes of the translation in the currently active coordinate system.
(defaults are 0.0)

X, Y, Z-Rotation
Magnitudes of the rotations (degrees) in the currently active coordinate system.
(defaults are 0.0)

Graphic Example: NDGEN

```
  ND,1,30,0,0,
  ND GEN,3,1,1,1,0,0,30.,0,
NDGEN,1,1,4,1,2,-15,0,0,90,0,0,
```



NDSCALE

Meshing > NODES > GENERATION MENU > Scaling

The NDSCALE command generates a pattern of new nodes from an existing pattern, by scaling their coordinates, and translating or rotating them relative to the active Cartesian coordinate system. Nodal coordinates are scaled as specified by the x, y and z scale factors. Offsets can also be specified.

Beginning Node
Beginning node in the pattern.

Ending Node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Generation flag
Generation flag.

0: Translation only	for translation
1: Rotation only	for rotation
	(default is 0: Translation only)

X, Y, Z-Scale factor
The scale factors in the X, Y, and Z directions.
(defaults are 1.0.)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted for only if generation flag = translation.
(defaults are 0.)

X, Y, Z-Rotation
The magnitudes of rotations (degrees) in the currently active Cartesian coordinate system. Prompted for only if generation flag = rotation.
(defaults are 0.)

Note
The active coordinate system must be Cartesian.

Example: `NDSCALE, 1,4,1,0,0.0,0.5,0.0,5.0,0.0,0.0`
The above command generates 4 new nodes from nodes 1 through 4 by applying a scale factor of 0.5 in the Y-direction and translating them by 5.0 units in the x-direction.

NDCOPY

Meshing > NODES > GENERATION MENU > Copy

The NDCOPY command copies a pattern of keypoints from the active coordinate system to the destination coordinate system. Both coordinate systems must be Cartesian. Use the NDMOVE (**Meshing, Nodes, Generation, Move**) command to move existing nodes without generating new ones. Generating new elements using the ELCOPY (**Meshing, Elements, Generation, Copy**) command automatically generates new nodes.

Beginning node
Beginning node in the pattern.

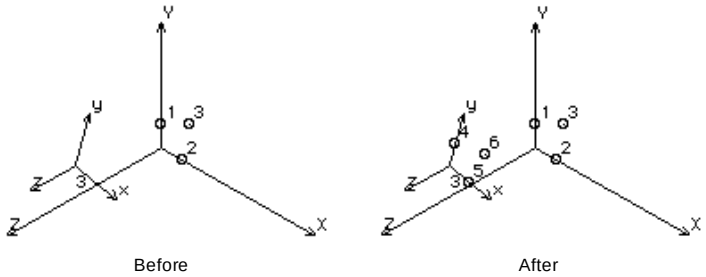
Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Destination coordinate
Destination coordinate system.
(default is 0)

Graphic Example: NDCOPY

```
PLANE,Z,0,1,
ND,1,0,15,0,
ND,2,15,0,0,
ND,3,20,25,0,
CSANGL,3,0,-50,-30,10,0,0,-15,
ACTSET,CS,0,
NDCOPY,1,3,1,3,
```



Before After

NDSYM

Meshing > NODES > GENERATION MENU > Symmetry

The PTSYM command generates a pattern of keypoints by symmetry about the specified plane of the active coordinate system which must be Cartesian. An offset can also be specified. Use the NDFLIP (**Meshing, Nodes, Generation, Flip**) command to flip existing nodes without generating new ones. Generating elements by symmetry using the ELSYM (**Meshing, Elements, Generation, Symmetry**) command automatically generates the associated nodes.

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Rotation/Sweep axis
Axis representing normal to the plane of symmetry.
= X y-z plane
= Y x-z plane
= Z x-y plane
(default is Z)

Offset
Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

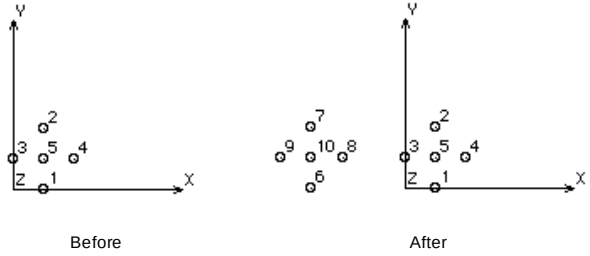
Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A specified offset is applied to all nodes of the specified pattern.

Graphic Example:

NDSYM

```
PLANE,Z,0,1,
VIEW,0,0,1,
ND,1,10,0,0,
ND,2,10,20,0,
ND,3,0,10,0,
ND,4,20,10,0,
ND,5,10,10,0,
NDSYM,1,5,1,
  X,-20,
```



NDPT

Meshing > NODES > GENERATION MENU > at Point

The NDPT command creates a node at the location of a keypoint.

Node label
Node label.
(default is highest node defined + 1)

Keypoint
Keypoint label.

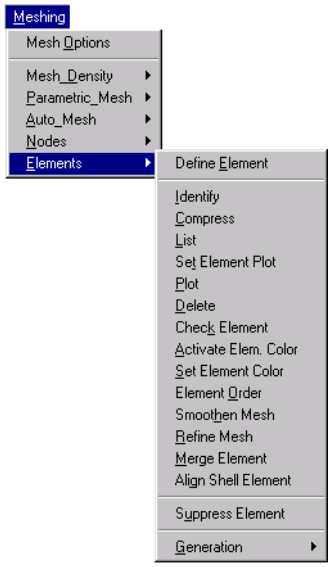
Example: `NDPT, , 10`
This command creates a new node at the location of keypoint number 10.




▶ ELEMENTS_Menu

Meshing > ELEMENTS

Figure 5-7 Elements Menu



This menu contains commands to define, identify, compress, list, plot, delete, and set colors for elements.

EL

Meshing > ELEMENTS > Define Element

The EL command creates a new element or modifies an existing one using the specified nodes. The element can be associated with a valid existing entity.

Element
Label of element to be created.
(default is max. element label defined + 1)

Associated Primitive name
Type of entity to associate the created element with.
= PT for 1-node elements
= CR or CT 1-D elements
= SF or RG 2-D elements
= VL 3-D elements
(default is SF)

Associated primitive label
Label of the entity to associate the created element with. Use zero if the element is not associated with a particular entity.
(default is 0)

Total nodes
Number of nodes in the element.

Node (i)
ith node label. Refer to Chapter 4 of the COSMOS/M User Guide for valid numbering order.

Geometry association of element face (i)
Label of geometric entity associated with face i of the element. (i = 1,2,....,6)
(default is 0, these six entries are intended for internal use by the GFORM_OUT command)

Notes

1. Entities can be selected by the mouse only if they are selected.
2. The '*Associated Primitive Name*' is used to associate elements. A BEAM3D element for example, must be associated with a curve not a keypoint, surface or volume. SHELL elements on the other hand, should be associated with a surface or a region and solid elements must be associated with a volume.
3. The '*Geometry Association Of Element Face (i)*' entries represent geometric association of element faces. The type of the geometric entity could be a curve, a surface or a region depending on the element group and the face number. For example, faces 5 and 6 of shell elements are associated with a surface or a region while the other faces are associated with curves. An input of 0 means no geometric association for the faces of the element to be created.
4. The association of element faces with geometric entities is intended for internal use to create neutral gfm files. You may enter the 0 default values when using the EL command to manually create elements.

Example 1: `EL, 34, SF, 3, 4, 10, 11, 12, 13, 1, 0, 0, 0, 3, 0,`

This command creates 2-D element number 34 and associates it with surface 3. The element has 4 nodes. Face 1 is associated with curve 1 and face 5 is associated with surface 3.

Example 2: `EL, ,CR, 0, 3, 1, 2, 3;`

This command creates a new 3-node 1-D element (BEAM3D) connecting nodes 1 and 2, node 3 is used to determine the element orientation. The element is not associated with any particular curve.

Example 3: `EL, ,SF, 0, 3, 1, 2, 3;`

This command creates a new 2-D element connecting nodes 1, 2, and 3. The element is not associated with any particular surface.

Example 4: `EL, ,PT, 0, 3;`

This command creates a 1-node element (e.g. MASS, GAP or SPRING) connecting nodes 1, 2, and 3. The element is not associated with any particular keypoint.

SETEPLOT

Meshing > ELEMENTS > Set Element Plot

The SETEPLOT command sets flags to be used in subsequent element plotting. Special element plotting is useful for plotting elements with mid-side nodes, mass elements, truss elements, and beam elements. For plotting truss/beam elements, the program uses the area/(width and depth) of the element when available. Otherwise, some values based on the dimensions of the model are assumed.

The command can also be used in postprocessing axisymmetric and asymmetric models in a 3-D mode and the plotting of material directions for PLANE2D, TRIANG, SHELL3, SHELL3T, SHELL3L, SHELL4, SHELL4T, SHELL4L, SHELL6, SHELL6T, SHELL9L, TETRA4, TETRA10, SOLID, SOLIDL, and SOLIDPZ elements.

Element plot order

Order of plotting higher order elements.

0: Linear linear (straight element edges. Mid-side nodes are ignored in plotting the edges)

1: Bilinearbilinear (curved edges. Mid-side nodes are connected to corner nodes by straight lines).

(default is 0)

Plot beam orientation

Beam orientation plotting flag.

0: No beam elements orientations are not plotted

1: Yes beam elements orientations are shown by plotting the local element coordinate system on each beam element

(default is no)

Special element plot

Flag for special plotting of some element types.

0: No deactivate special elements plotting

1: Yes activate special elements plotting (Beams, Trusses, Pipes, Elbows, 3-D current sources, and 3-D postprocessing of asymmetric models)

(default is 0: No)

Plot material direction

Flag for plotting material directions for shell, plane, solid, and tetrahedral elements supporting orthotropic material properties.

0: No do not plot material directions

1: Yes plot material directions

(default is 0: No)

Layer number

Layer of elements for plotting the material angle. Used for layered elements only.

(default is 1)

Reference

Flag to specify elements for plotting the material angle.

= -1: geom; plot the material angle for each group of elements with similar geometyric association (surfaces/regions for area elements and volume for volume elements).

= 0: no reference; plot the material angles for every element. If a selection list is active, the plot will be generated for the selected elements only.

= n:el n plot the material angles for element n only.

(default is 0)




ACTECLR

Meshing > ELEMENTS > Activate Elem. Color

The ACTECLR command activates or deactivates element coloring based on element attributes for subsequent element plots. Default colors may be selected, or you can select your own colors using the SETECLR (**Meshing, Elements, Set Element Color**) command.

Color flag
Flag to activate/deactivate element coloring based on element attributes.
0: No deactivate
1: Yes activate
(default is 1: Yes)

Set label
Admissible property set names are:
MP: Material Property
RC: Real Constant
EG: Element Group
(default is EG: Element Group)

Default colors
Default color flag.
1: On use default colors
0: Off use colors defined by the SETECLR(**Meshing, Elements, Set Element Color**) command, if available. Otherwise, default colors will be used.
(default is 1: On)

Notes

1. Note that once the flag to deactivate the color selection is used, the command will not prompt for the remaining parameters.
2. Colors set by the SETECLR (**Meshing, Elements, Set Element Color**) command will not be disturbed by the activation or deactivation of default colors.

SETECLR

Meshing > ELEMENTS > Set Element Color

The SETECLR command assigns a specified color to all elements that belong to a pattern of property sets. A property set can be EG (element group), RC (real constant), or MP (material property).

First set label

Beginning set label in the pattern.

Last set label
Ending set label in the pattern.

Increment
Increment between set labels in the pattern.
(default is 1)

Property set name
Property set name. Admissible set names are:
EG: Element Group
RC: Real Constant
MP: Material Property
(default is EG: Element Group)

Color of elements
Color code to be assigned to these elements in subsequent plotting. (1 to 16, refer to the STATUS1 Table)
(default is 5: Dark Green)

Note
The color code used by GEOSTAR is shown in the STATUS1 Table.

Example:
```
ACTECLR, 1, RC,0,
EPLOT;
SETECLR, 1, 1, 1, RC, 10,
EPLOT; (or REPAINT)
ACTECLR, 0,
EPLOT; (or REPAINT)
ACTECLR, 1, RC, 1,
EPLOT; (or REPAINT)
```
The first EPLOT command plots elements in different colors indicating different associated real constant sets. Default colors are used. The second EPLOT command is similar except that elements associated with real constant set 1 are plotted in color 10. The third EPLOT command plots all elements in the same color. The fourth EPLOT command acts exactly like the first one.

ECHECK

Meshing > ELEMENTS > Check Element

The ECHECK command checks the aspect ratio for the elements specified in the pattern. You can request to list/highlight (or both) the elements whose aspect ratios exceed the specified value. The command can also be used to fix the connectivity, for example a SHELL4 element with nodal connectivity 1 1 3 4 will be changed to 3 4 1 1. This condition is not encountered when elements are generated by merging commands. The

command deletes degenerate elements automatically. Degenerate elements are 1-D elements whose length is less than PTTOL, 2-D elements whose area is less than $(PTTOL)^2$, and 3-D elements whose volume is less than $(PTTOL)^3$. The default value for PTTOL is 0.00005 or as specified by the PTTOL (**Geometry, Points, Point Merge Tolerance**) command.

Beginning element
Beginning element of the pattern.
(default is 1)

Ending element
Ending element of the pattern.
(default is the highest element label)

Increment
Increment between the elements in the pattern.
(default is 1)

Aspect ratio
Aspect ratio.
(default is 5.0)

Check element node connectivity order
Flag to check element connectivity and fix the order in which nodes are defined.
0: No do not check connectivity
1: Yes check and fix element connectivity
(default is no)

Element check option
Flag to control the display of elements with high aspect ratios.
0: None do not display
1: Draw draw elements with high aspect ratios
2: List list elements with high aspect ratios
3: Both draw and list elements with high aspect ratios
(default is 0: None)

Aspect ratio option
Criterion for calculating aspect ratios.
0: Emax/Emin Longest edge/shortest edge.
1: Emax^2/A or sqrt(Emax^3/V)
For area elements: $(\text{longest edge})^2/\text{Area}$
For volume elements: Square root of $[(\text{longest edge})^3/\text{volume}]$
2: Emax/Hmin Longest edge/shortest height. Height is the distance from a corner node to the midpoint of the opposite edge/face.
3: Emax*Amax/V (Largest edge)x(area of largest face)/Volume
Used for volume elements only.
For area elements, Emax/Hmin is used.
4: Emax/Radius Longest edge/Radius of largest sphere inscribed in the element. Used for tetrahedral elements only. Emax*Amax/V is used for other volume elements. Emax/Hmin is used for area elements.

Sort list

flag to sort the listed elements with respect to aspect ratio.
0: No do not sort the list (list relevant elements sequentially).
1: Yes sort the relevant elements based on their aspect ratios.
(default is no)

Threshold percentage

Threshold percentage for sorted lists. x% means list only those elements whose aspect ratios are within x% of the heighest aspect ratio (and meet the other criteria specified in the command).
(default is 100)

Notes

1. It is recommended to issue this command before running the analysis.
2. A more vigorous check is performed by the R_CHECK (**Analysis, Run Check**) command.

3) The method used to calculate the aspect ratio is also listed.

Example: `ECHECK,1,,1,2.5,1,3,0,1,50`

This command checks the aspect ratio of all elements. The elements whose aspect ratios exceed 2.5 will be highlightd and listed on the screen. Emax/Emin will be used to calculate the aspect ratio. Because of the specified threshold, only aspect ratios exceeding 2.5 and whose aspect ratios are within 50% of the highest aspect ratio will be listed. For example if the highest aspect ratio is 10, then only elements with aspect ratios larger than 9 will be listed or highlighted.

The command also deletes degenerate elements. Element connectivities are checked and fixed. For example a PLANE2D element with a collapsed node must be defined such that the third and fourth nodes collapse. If such an element is defined as connecting nodes 17, 17, 18, and 19, this command will change its connectivity to 18, 19, 17, and 17. The connectivity of SOLID and other elements are also checked and fixed.

ECHANGE

Meshing > ELEMENTS > Element Order

The ECHANGE command changes the order of a pattern of existing elements. The order can be increased or decreased by adding or removing mid-side nodes. When the order is decreased for an element, all mid-side nodes are removed. Refer to the ADAPTIVE (**Analysis, Static, Adaptive Method**) command for specifying the polynomial order to be used in conjunction with the P-method.

Beginning Element

Beginning element in the pattern.

Ending Element
Ending element in the pattern.
(default is highest element label defined)

Increment
Increment between elements in the pattern.
(default is 1)

Order flag
Flag to decrease or increase the order.
0: Low decrease the order
1: High increase the order
(default is 1: High)

Push boundary node
Flag to push generated boundary nodes to the geometric boundary (prompted for if "order flag" = high only).
0: No do not push nodes to the geometric boundary
1: Yes push nodes to the geometric boundary
(default is 0: No)

Notes

1. The command can be used to change 3-node elements to 6-node elements, 4-node elements to 8-node elements and 8-node (solid) elements to 20-node elements. The element group associated with the elements must be redefined if necessary.
2. It is recommended to activate the *Push Boundary Node* flag to improve the accuracy of the model.

ESMOOTH

Meshing > ELEMENTS > Smoothen Mesh

The ESMOOTH command smoothens an existing mesh of 3-noded, 2-D triangular elements like TRIANG, and plane SHELL3, SHELL3T, and SHELL3L elements. The nodes of the mesh are slightly moved around to improve the quality of the mesh.




Beginning Element
Beginning element in the pattern.

Ending Element
Ending element in the pattern.
(default is highest element label defined)

Increment
Increment between elements in the pattern.
(default is 1)

Boundary crossing
Cross boundary flag.

1: Yes refinement process can cross geometric boundaries
0: No refinement process cannot cross geometric boundaries (i.e. nodes on curves can only be relocated on these curves)
(default is 0: No)

Number of smoothing iterations
Number of iterations to smoothen the mesh.
(default is 3)

EREFINE

Meshing > ELEMENTS > Refine Mesh

The EREFINE command refines an existing mesh of 3-node 2-D elements (TRIANG or plane SHELL3, SHELL3T, or SHELL3L), and 4-node tetrahedral elements (TETRA4 and TETRA4R). The specified elements are refined and elements in the neighborhood are redefined to generate a compatible mesh. Flags to smoothen the refined mesh and the whole model are provided. The part of the model to be refined can conveniently be selected by commands in the Control, Selection menu. You can then issue this command to refine *all* elements which will then mean *all selected* elements.

Beginning Element
Beginning element in the pattern.

Ending Element
Ending element in the pattern.
(default is highest element label defined)

Increment
Increment between elements in the pattern.
(default is 1)

Push boundary node
Push the new boundary nodes to the corresponding boundary.
1: Yes push new boundary nodes to the boundary
0: No do not push boundary nodes to the boundary
(default is 0: No)

Boundary crossing
Cross boundary flag.
1: Yes refinement process can cross geometric boundaries
0: No refinement process cannot cross geometric boundaries (i.e. nodes on curves can only be relocated on these curves)
(default is 0:No)

Number of local smoothing iterations
Number of iterations to smoothen the area around the new mesh.
(default is 3)




Number of global smoothing iterations
Number of iterations to smoothen the global mesh as a result of the local refinement. Nodes are relocated to obtain a better mesh.
(default is 0)

Notes

1. The NMERGE (**Meshing, Nodes, Merge**) command must be used to merge coincident nodes before issuing the EREFINE command, otherwise incompatibility errors may occur across boundaries with unmerged coincident nodes.
2. To refine high order elements like 6-node TRIANG and TETRA10, first use the ECHANGE (**Meshing, Elements, Element Order**) command to lower the order, refine, and then raise the order back using the ECHANGE command.




EMERGE

Meshing > ELEMENTS > Merge Element

The EMERGE command merges elements with identical nodal connectivity in a pattern. The command is useful after accidental duplicate meshing of geometric entities. The NMERGE must be given first to merge coincident nodes in order for the EMERGE command to be effective. Merging can be performed with respect to all elements or with respect to the elements in the specified pattern only.

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.
(default is the max. element label defined)

Increment
Increment between elements in the pattern.
(default is 1)

All/Among flag
All/among merging flag.
0: All consider all elements in the model
1: Among consider the elements specified in the pattern only
(default is 0: All)

Echo flag
Echo flag.
1: On GEOSTAR issues a message whenever two elements are merged
0: Off GEOSTAR does not issue messages
(default is 1: On)

Low/High flag
Low/high flag.
0: Low merge higher labels with lower labels of elements, such that if two elements are merged, the element with higher label is deleted
1: High merge lower labels with higher labels of elements, such that if two elements are merged, the element with lower label is deleted
(default is 0: Low)

Example:
```
EMERGE;
```
The above commands merges accessible elements with identical nodal connectivity.

ALIGNSHELL

Meshing > ELEMENTS > Align Shell Element

The ALIGNSHELL command aligns a pattern of shell elements such that adjacent elements will have consistent top and bottom faces. The first element in the pattern is considered the reference. Only elements with common edges are aligned. The user must use the NMERGE command to merge coincident nodes. Elements whose edges make 90 degrees are not aligned.

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.
(default is the highest element label defined)

Increment
Increment between the elements in the pattern.
(default is 1)

Reverse direction flag
Flag to reverse the top and bottom faces of shell elements.
0: No do not reverse
1: Yes reverse
(default is 0: No)

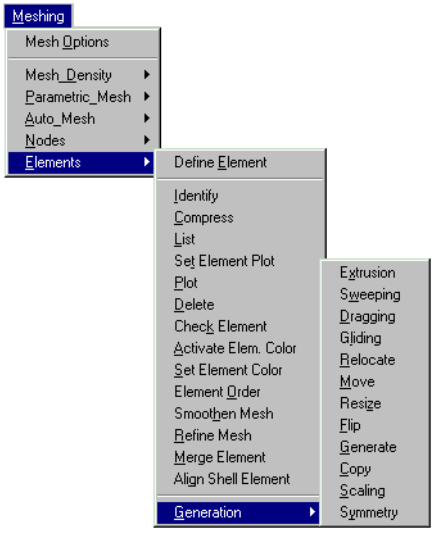



▼GENERATION Menu

Meshing > ELEMENTS > GENERATION MENU

This menu contains commands dealing with the generation of elements.

Figure 5-8 Generation Menu



ELEXTR

Meshing > ELEMENTS > GENERATION MENU > Extrusion

The ELEXTR command creates 1-D, 2-D or 3-D elements by extruding existing 1-node, 1-D or 2-D elements respectively. The extrusion is done along the specified axis of the active coordinate system which must be Cartesian.

Beginning Element
Beginning element in the pattern.

Ending Element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Axis symbol
Axis symbol (X, Y, Z).
(default is Z)

Total length of extrusion
Length of extrusion.

Number of layers of elements
Number of layers in the extrusion direction.
(default is 6)

Original mesh update flag
Flag for updating the original lower order elements.

-1: Transfer	transfer elements to the end of extrusion
0: Keep	keep the original elements
1: Delete	delete the original elements *(default is delete original mesh)*

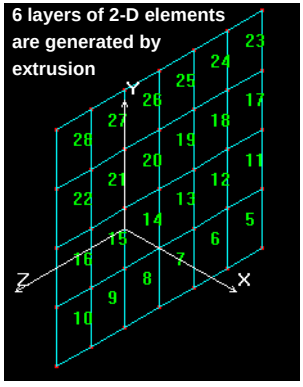
Spacing ratio
Spacing ratio in the direction of extrusion.
(default is 1.0 for uniform spacing)

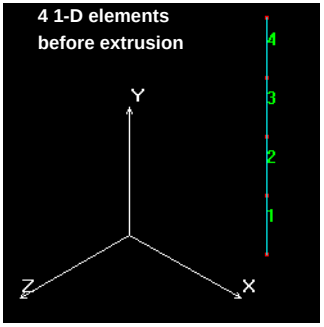
Notes

1. Elements can be picked by the mouse.
2. The user should activate the appropriate 2-D quad element group, material property, and real constant sets before issuing the command.
3. The generated elements will be free from geometrical associations.
4. The active coordinate system must be Cartesian.

Graphic Example: `ELEXTR, 1, 4, 1, Z, 60, 6, 1, 1`

The above command generates 24 new elements by extruding elements 1 to 4 in the Z-direction of the active coordinate system. The total length of extrusion is 60 in 6 layers. The 1-D original elements are deleted.





ELSWEEP

Meshing > ELEMENTS > GENERATION MENU > Sweeping

The ELSWEEP command creates new elements by sweeping existing ones about a specified axis of the currently active coordinate system by a specified angle. The command creates 1-D elements from 1-node elements, 2-D elements from 1-D elements, or 3-D elements from 2-D elements.

Beginning Element
Beginning element in the pattern.

Ending Element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Axis symbol
Axis symbol (X, Y, Z).
(default is Y)

Angle of the arc
Angle of sweep in degrees.
(default is 90 degrees)

Number of layers of elements
Handling in the sweep direction.
(default is 6)

Original mesh update flag
Flag for treatment of the original lower order elements.

-1: Transfer	transfer elements to the end of sweep
0: Keep	keep the original elements
1: Delete	delete the original elements
	(default is delete original mesh)

Spacing ratio
Spacing ratio in the direction of sweep.
(default is 1.0 for uniform spacing)

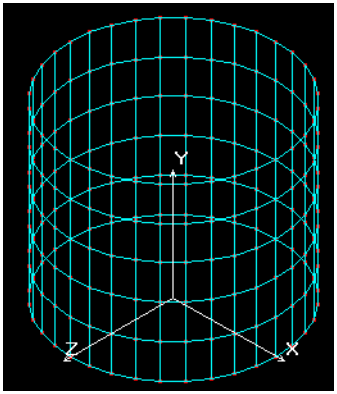
Example: `ELSWEEP, 1, 3, 1, X, 90.0, 20, 0,`

The above command generates 60 new elements by sweeping elements 1 through 3 about the X-axis of the active coordinate system. The original elements are kept.

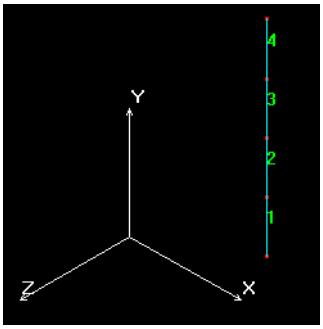
Graphic Example: `ELSWEEP, 1, 5, 1, Y, 360, 36, 1, 1`

The above command generates 5x36=180 new elements by extruding

elements 5 1-D elements about the Y axis for 360^0. The 1-D original elements are deleted.



2-D elements are generated by sweeping 1-D elements



5 1-D elements before sweeping

ELDRAG

Meshing > ELEMENTS > GENERATION MENU > Dragging

The ELDRAG command creates new elements by dragging a pattern of elements along a profile formed by a group of curves. The elements are generated such that the angles between the profile curves and any element in the pattern remain unchanged as the element is dragged along the profile. The command creates 2-D elements from 1-D elements, or 3-D elements from 2-D elements.

Beginning Element
Beginning element in the pattern.

Ending Element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Number of profile curves
Number of profile curves (limited to 6).
(default is 1)

Original mesh update flag

Flag for treatment of the original lower order elements.

-1:Transfer	transfer elements to the end of sweep
0: Keep	keep the original elements
1: Delete	delete the original elements
	(default is delete original mesh)

Select profile curve (i)

The ith profile curve. The curves must be specified in order and must form a continuous path.

Number of elements on curve (i)

Number of elements on the ith profile curve.
(default is 4)

Notes

1. The curves can be picked by the mouse.
2. The profile curves must have a continuous first derivative (continuous slope). Fillets can be created to smoothen the profile if needed.
3. The profile curves must be such that an element does not intersect itself as it is dragged.
4. The GLIDE and DRAG are equivalent operations when the profile is a straight line.
5. The number of elements generated is equal to the number of elements in the pattern multiplied by the sum of "number of elements on curve (i)".

Example:

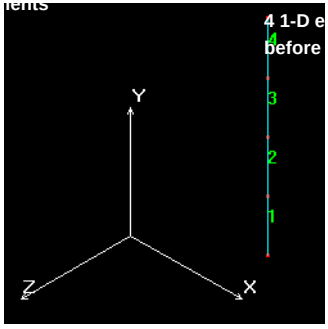
```
ELDRAG, 1, 3, 1, 5, , 11, 2, 12, 3, 13, 3, 14, 3,
15, 3
```

The above command creates 42 (3x14) new elements generated by dragging elements 1, 2, and 3 along the profile defined by curves 11, 12, 13, 14 and 15.

Example:

```
ELEXTR, 1, 4, 1, Z, 60, 6, 1, 1
```

The above command generates 24 new elements by extruding elements 1 to 4 in the Z-direction of the active coordinate system. The total length of extrusion is 60 in 6 layers. The 1-D original elements are deleted.



ELGLIDE

Meshing > ELEMENTS > GENERATION MENU > Gliding

The ELGLIDE command creates new elements by gliding existing ones along a specified profile formed by a group of curves. The command creates 2-D elements from 1-D elements or 3-D elements from 2-D elements. The elements in the pattern keep their original orientation as they glide along the profile curves.

Beginning Element
Beginning element in the pattern.

Ending Element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Number of profile curves
Number of profile curves (limited to 6).
(default is 1)

Original mesh update flag
Flag for treatment of the original lower order elements.

-1: Transfer	transfer elements to the end of sweep
0: Keep	keep the original elements
1: Delete	delete the original elements
	(default is delete original mesh)

Select profile curve (i)
The ith profile curves. The curves must be specified in order and must form a continuous path (i = 1 to 6).

Number of elements on curve (i)
Number of elements on the ith profile curve.
(default is 4)

Notes

1. The curves can be picked by the mouse.
2. The profile curves must have a continuous first derivative (continuous slope). Fillets can be created to smoothen the profile if needed.
3. The locus of every single node on the gliding elements is exactly identical to the profile.
4. The GLIDE and DRAG are equivalent when the profile is a straight line.
5. The user should activate the appropriate element group, material property and real constant sets before issuing the command.
6. The generated elements will be free from geometrical associations.




Example: `ELGLIDE, 1, 3, 1, 5, 11, 5, 12, 5, 13, 5, 14, 6, 15, 6`

The above command defines 81 new elements generated by moving each of elements 1, 2, and 3 parallel to its original position along the profile defined by curves 11, 12, 13, 14 and 15. Elements 1, 2 and 3 are kept.

ELRELOC

Meshing > ELEMENTS > GENERATION MENU > Relocate

The ELRELOC command relocates a pattern of elements by a specified translation and/or rotation about the currently active Cartesian coordinate system. This command does not generate any new elements. The ELGEN (**Meshing, Elements, Generation, Generate**) command can be used to generate new elements from existing ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending elements in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

Notes

1. Elements can be picked by the mouse only if they are plotted on the screen.
2. If a curve in the specified pattern is part of a higher entity, that curve will not be relocated.

Example 1: `ELRELOC, 1, 6, 1, 0, 5.0, 5.0, 0.0`

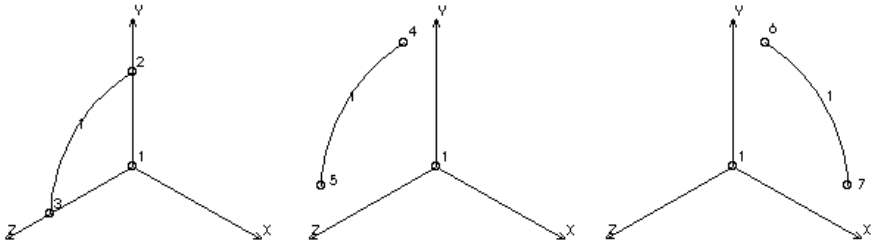
The above command translates curves 1 through 6 from their current locations by X = 5.0, Y = 5.0, and Z = 0.0

Example 2: `ELRELOC, 1, 6, 1, 1, 30.0, 0.0, 0.0`

The above command rotates curves 1 through 6 from their current locations by x_rot = 30.0, y_rot = 0.0 and z_rot = 0.0 degrees about the currently active X, Y, and Z coordinate axes, respectively. Note that new keypoints are generated.

Graphic Example: CRRELOC

```
PT,1,0,0,0,
PT,2,0,50,0,
PT,3,0,0,50,
CRARC,1,2,3,1,50,
CRRELOC,1,1,1,0,0,25,20,
CRRELOC,1,1,1,1,0,90,0,
```



Original

After the First CRRELOC Command

After the Second CRRELOC Command

ELMOVE

Meshing > ELEMENTS > GENERATION MENU > Move

The ELMOVE command moves a pattern of elements from the current coordinate system to the destination coordinate system. This command does not generate any new elements. The ELCOPY (**Meshing, Elements, Generation, Copy**) command can be used to generate new elements from existing ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.

(default is 1)

Destination coordinate
Destination coordinate system label.

Notes

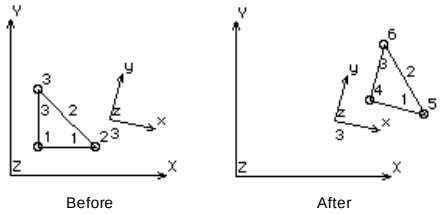
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. Both coordinate systems must be Cartesian.

Example: `ELMOVE, 1, 6, 1, 3`

The above command moves elements 1 through 6 to local coordinate system 3 with the same coordinates.

Graphic Example: ELMOVE

```
PLANE,Z,0,1,

VIEW,0,0,1,

ELCORD,1,10,10,0,30,10,0,1
    0,30,0,10,10,0,

CSANGLE,3,0,35,20,0,0,0,-
    15,

ACTSET,CS,0,
ELMOVE,1,3,1,3,
```



ELRESIZ

Meshing > ELEMENTS > GENERATION > Resize

The ELRESIZ command resizes a pattern of existing elements, by scaling and translating/rotating them relative to the currently active coordinate system. This command does not generate any new elements. The ELSCALE (**Meshing, Elements, Generation, Scaling**) command can be used to generate new elements by scaling existing ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X-, Y-, and Z-directions.
(defaults are 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

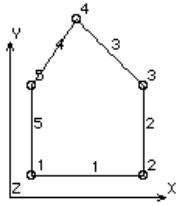
X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

Note
Entities can be picked by the mouse only if they are plotted on the screen.

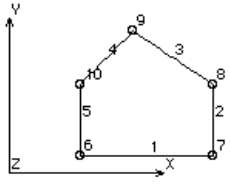
Example: `ELRESIZ, 1,4,1,0,0.0,0.5,0.0,5.0,10.0,10.0`
The above command scales down elements 1 to 4 by a scale factor of 0.5 in Y-direction after translating the elements by X = 5., Y = 10. and Z = 10. in the currently active coordinate system.

Graphic Example: ELRESIZ

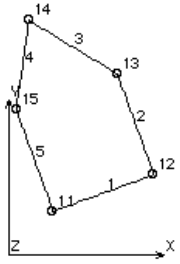
```
PLANE,Z,0,1,

VIEW,0,0,1,

ELPCORD,1,5,5,0,30,5,0,30,25,0,15,40,0,5,25,0,5,5,0,
ELRESIZ,1,5,1,0,1.2,.8,0,10,0,0,
ELRESIZ,1,5,1,1,.8,1.5,0,0,0,20,
```



Before the First CRRESIZ Command



After the First CRRESIZ Command



After the Second CRRESIZ Command

ELFLIP

Meshing > ELEMENTS > GENERATION > Flip

The ELFLIP command flips a pattern of elements about a specified plane in the current coordinate system. An offset can also be specified with the flipping. his command does not generate any new elements. The ELSYM (**Meshing, Elements, Generation, Symmetry**) command can be used to generate new elements from existing ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Reverse direction flag
Reverse direction flag.

= 0: No	do not reverse directions of elements
= 1: Yes	reverse directions of elements *(default is reverse direction)*

Rotation/Sweep axis
Axis representing normal to the plane about which pattern of elements are to be flipped.

= X	about y-z plane
= Y	about z-x plane
= Z	about x-y plane *(default is Z)*

Offset
Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

Notes

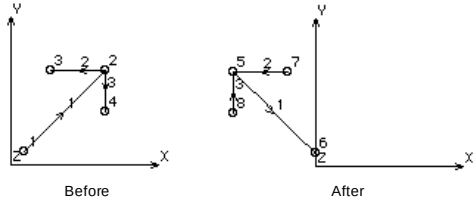
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A specified offset is applied to all elements in the specified pattern.

Example: `ELFLIP, 1, 6, 1, X,, 0, 0.0`

The above command flips elements 1 through 6 about the Y-Z plane (normal to Y-Z plane is X-axis). No offset is specified.

Graphic Example: ELFLIP

```
PLANE,Z,0,1,

VIEW,0,0,1,

ELCORD,1,5,5,0,35,35,0,1
    5,35,0,15,35,0,

EL2CORD,3,35,35,0,35,20,0,
ELFLIP,1,3,1,X,1,5,
```



ELGEN

Meshing > ELEMENTS > GENERATION > Generate

The ELGEN command generates one or more patterns of elements from an existing pattern, by translating and/or rotating them about the currently active Cartesian coordinate system. The ELRELOC (**Meshing, Elements, Generation, Relocate**) command can be used to relocate existing elements without generating new ones.

Generation number
Number of patterns to be generated.
(default is 1)

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
Magnitudes of the translation in the currently active coordinate system.
(defaults are 0.0)

X, Y, Z-Rotation
Magnitudes of the rotations (degrees) in the currently active coordinate system.
(defaults are 0.0)

Example 1: `ELGEN, 2, 1, 4, 1, 0, 5.0, 10.0, 10.0`

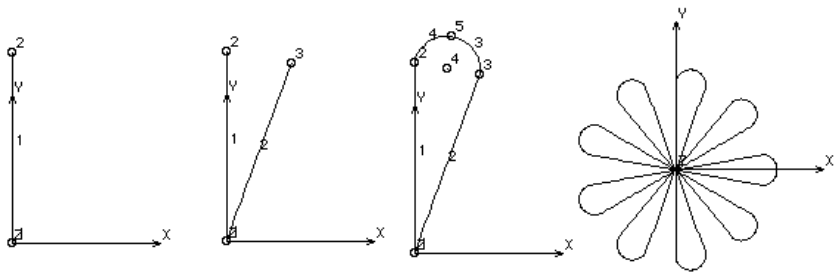
The above command generates two patterns of 4 elements each at the locations obtained by translating existing elements 1 through 4 by x = 5.0, y = 10.0 and z = 10.0 for the first pattern and x = 10, y = 20., z = 20. for the second pattern, respectively.

Example 2: `ELGEN, 2, 1, 4, 1, 1, 20.0, 15.0, 10.0`

The above command generates two patterns of 4 elements each at the locations obtained by the rotating existing elements 1 through 4 by x-rotation = 5.0, y-rotation = 15.0 and z-rotation = 10. degrees for the first pattern, and x-rotation = 10, y-rotation = 30.0 and z-rotation = 20.0 degrees for the second pattern, respectively, about the currently active X, Y, Z axes.

Graphic Example: ELGEN

```
PT,1,0,0,0,

PT,2,0,70,0,

ELLINE,1,1,2,
ELGEN,1,1,1,1,1,0,0,-20.,

PLANE,Z,0,1,

ELPCIRDIA,3,3,2,180,2,
ELGEN,8,1,4,1,1,0,0,40.,
```



ELCOPY

Meshing > ELEMENTS > GENERATION > Copy

The ELCOPY command copies a pattern of elements from the active coordinate system to the destination coordinate system. Use the ELMOVE

(**Geometry, Elements, Generation, Move**) command to move existing elements without generating new ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Destination coordinate
Destination coordinate system label.

Notes

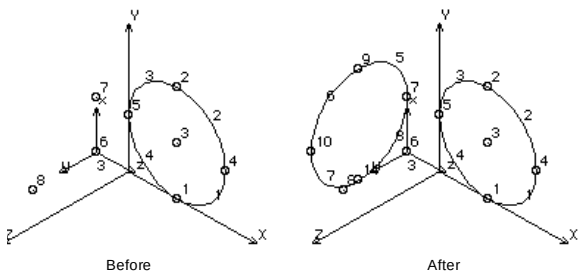
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. Both coordinate systems must be Cartesian.

Example: `ELCOPY, 1, 6, 1, 3`

The above command copies elements 1 through 6 to local Cartesian coordinate system 3. Corresponding keypoints will have identical coordinates in both systems.

Graphic Example: ELCOPY

```
PT,1,30,0,0,

PT,2,30,60,0,

PLANE,Z,0,1,

ELPCIR-
    DIA,1,1,2,
    360,4,

PT,6,-20,0,0,

PT,7,-20,30,0,

PT,8,-20,0,40,

CSYS,3,0,6,7,8,

ACTSET,CS,0,

VIEW,1,1,1,
ELCOPY,1,4,1,3,
```



ELSCALE

Meshing > ELEMENTS > GENERATION > Scaling

The ELSCALE command generates a pattern of elements from an existing pattern, by scaling and translating/rotating them relative to the currently active Cartesian coordinate system. The coordinates of all associated keypoints are multiplied by the specified scale factors.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X, Y, and Z directions.
(defaults are 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

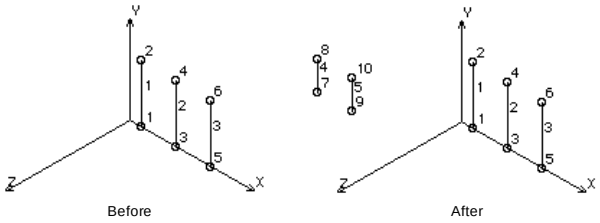
Example: `ELSCALE, 1,4,1,0,0.0,0.5,0.0,5.0,0.0,0.0`

The above command generates 4 new elements from the existing elements 1 through 4 by applying a scale factor of 0.5 in the Y-direction and translating them by 5.0 units in the X-direction.

Graphic Example: ELSCALE

```
PT,1,10,0,0,

ELETRUDE,1,1,1,Y,50.,

ELGEN,2,1,1,1,2,30,0,0,0,0,0,
```

```
ELSCALE,1,3,2,0,.5,.5,0,80,10,50,
```



ELSYM

Meshing > ELEMENTS > GENERATION > Symmetry

The ELSYM command generates a pattern of elements symmetric to the specified pattern about a specified plane in the current coordinate system. An offset can also be specified. Use the ELFLIP (**Geometry, Elements, Generation, Flip**) command to flip existing elements without generating new ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Rotation/Sweep axis
Axis representing normal to the plane about which pattern of elements are to be generated symmetrically.

= X	about y-z plane
= Y	about x-z plane
= Z	about x-y plane
	(default is Z)

Reverse direction flag
Reverse direction flag.

= 0: No	do not reverse directions of elements
= 1: Yes	reverse directions of elements
	(default is reverse direction)

Offset
Magnitude of the offset in the direction of the specified axis.

Notes

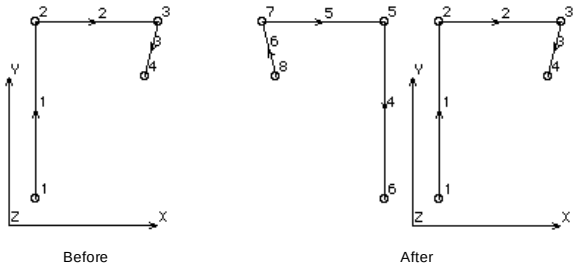
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A specified offset is applied to all elements of the specified pattern.

Example: `ELSYM, 1, 6, 1, X, 0, 0.0`

The above command generates 6 elements that are symmetric with elements 1 through 6, about the Y-Z plane. No offset is specified and the new elements have the same direction as the elements they are generated from.

Graphic Example: ELSYM

```
VIEW,0,0,1,

PLANE,Z,0,1,

ELCORD,1,10,
    10,0,10,7
    5,0,55,75
    ,0,50,55,
    0,50,55,0
    ,
ELSYM,1,3,1,X,
  1,0,
```



The ELMOVE command moves a pattern of elements from the current coordinate system to the destination coordinate system. This command does not generate any new elements. The ELCOPY (**Meshing, Elements, Generation, Copy**) command can be used to generate new elements from existing ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Destination coordinate
Destination coordinate system label.

Notes

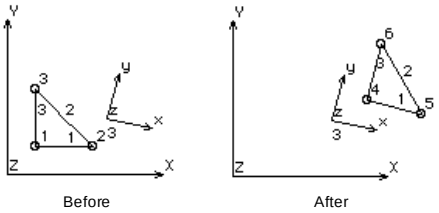
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. Both coordinate systems must be Cartesian.

Example: `ELMOVE, 1, 6, 1, 3`

The above command moves elements 1 through 6 to local coordinate system 3 with the same coordinates.

Graphic Example: ELMOVE

```
PLANE,Z,0,1,
VIEW,0,0,1,
ELPCORD,1,10,10,0,30,10,0,
10,30,0,10,10,0,
CSANGLE,3,0,35,20,0,0,0,-
  15,
ACTSET,CS,0,
ELMOVE,1,3,1,3,
```



ELRESIZ

Meshing > ELEMENTS > GENERATION > Resize

The ELRESIZ command resizes a pattern of existing elements, by scaling and translating/rotating them relative to the currently active coordinate system. This command does not generate any new elements. The ELSCALE (**Meshing, Elements, Generation, Scaling**) command can be used to generate new elements by scaling existing ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X-, Y-, and Z-directions.
(defaults are 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordi-

nate system. Prompted only if rotations are included.
(defaults are 0.0)

Note

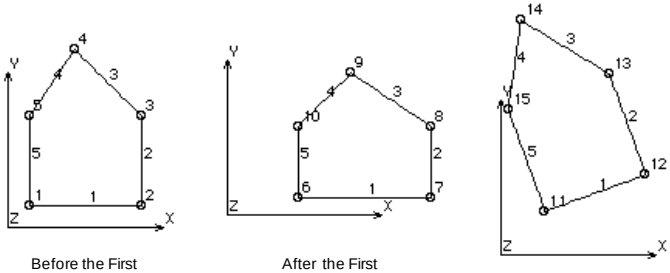
Entities can be picked by the mouse only if they are plotted on the screen.

Example: `ELRESIZ, 1,4,1,0,0.0,0.5,0.0,5.0,10.0,10.0`

The above command scales down elements 1 to 4 by a scale factor of 0.5 in Y-direction after translating the elements by X = 5., Y = 10. and Z = 10. in the currently active coordinate system.

Graphic Example: ELRESIZ

```
PLANE,Z,0,1,
VIEW,0,0,1,
ELPCORD,1,5,5,0,30,5,0,30,25,0,15,40,0,5,25,0,5,5,0,
ELRESIZ,1,5,1,0,1.2,.8,0,10,0,0,
ELRESIZ,1,5,1,1,.8,1.5,0,0,0,20,
```



Before the First CRRESIZ Command

After the First CRRESIZ Command

After the Second CRRESIZ Command

ELFLIP

Meshing > ELEMENTS > GENERATION > Flip

The ELFLIP command flips a pattern of elements about a specified plane in the current coordinate system. An offset can also be specified with the flipping. his command does not generate any new elements. The ELSYM (**Meshing, Elements, Generation, Symmetry**) command can be used to generate new elements from existing ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment

Increment between elements in the pattern.
(default is 1)

Reverse direction flag
Reverse direction flag.

= 0: No	do not reverse directions of elements
= 1: Yes	reverse directions of elements
	(default is reverse direction)

Rotation/Sweep axis
Axis representing normal to the plane about which pattern of elements are to be flipped.

= X	about y-z plane
= Y	about z-x plane
= Z	about x-y plane
	(default is Z)

Offset
Magnitude of the offset in the direction of the specified axis.
(default is 0.0)

Notes

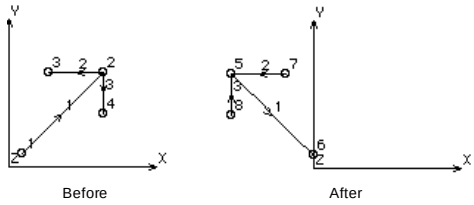
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A specified offset is applied to all elements in the specified pattern.

Example: `ELFLIP, 1, 6, 1, X,, 0, 0.0`

The above command flips elements 1 through 6 about the Y-Z plane (normal to Y-Z plane is X-axis). No offset is specified.

Graphic Example: ELFLIP

```
PLANE,Z,0,1,
VIEW,0,0,1,
ELPCORD,1,5,5,0,35,35,0,15
  ,35,0,15,35,0,
EL2CORD,3,35,35,0,35,20,0,
ELFLIP,1,3,1,X,1,5,
```



ELGEN

Meshing > ELEMENTS > GENERATION > Generate

The ELGEN command generates one or more patterns of elements from an existing pattern, by translating and/or rotating them about the currently active Cartesian coordinate system. The ELRELOC (**Meshing, Elements, Generation, Relocate**) command can be used to relocate existing elements without generating new ones.

Generation number
Number of patterns to be generated.
(default is 1)

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Displacement
Magnitudes of the translation in the currently active coordinate system.
(defaults are 0.0)

X, Y, Z-Rotation
Magnitudes of the rotations (degrees) in the currently active coordinate system.
(defaults are 0.0)

Example 1: `ELGEN, 2, 1, 4, 1, 0, 5.0, 10.0, 10.0`

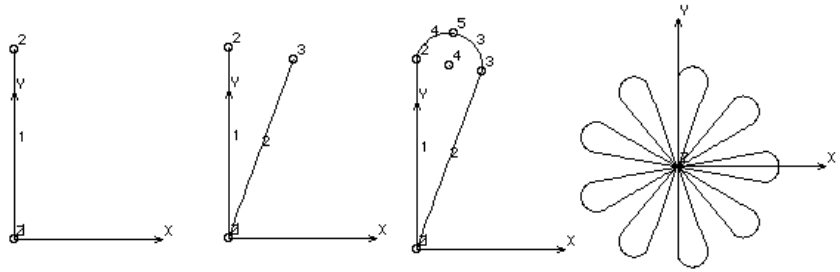
The above command generates two patterns of 4 elements each at the locations obtained by translating existing elements 1 through 4 by x = 5.0, y = 10.0 and z = 10.0 for the first pattern and x = 10, y = 20., z = 20. for the second pattern, respectively.

Example 2: `ELGEN, 2, 1, 4, 1, 1, 20.0, 15.0, 10.0`

The above command generates two patterns of 4 elements each at the locations obtained by the rotating existing elements 1 through 4 by x-rotation = 5.0, y-rotation = 15.0 and z-rotation = 10. degrees for the first pattern, and x-rotation = 10, y-rotation = 30.0 and z-rotation = 20.0 degrees for the second pattern, respectively, about the currently active X, Y, Z axes.

Graphic Example: ELGEN

```
ND,1,0,0,0,
ND,2,0,70,0,
EL,1,1,2,
ELGEN,1,1,1,1,1,0,0,-20.,
PLANE,Z,0,1,
ELPCIRDIA,3,3,2,180,2,
ELGEN,8,1,4,1,1,0,0,40.,
```



ELCOPY

Meshing > ELEMENTS > GENERATION > Copy

The ELCOPY command copies a pattern of elements from the active coordinate system to the destination coordinate system. Use the ELMOVE (**Geometry, Elements, Generation, Move**) command to move existing elements without generating new ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Destination coordinate
Destination coordinate system label.

Notes

1. Entities can be picked by the mouse only if they are plotted on the screen.
2. Both coordinate systems must be Cartesian.

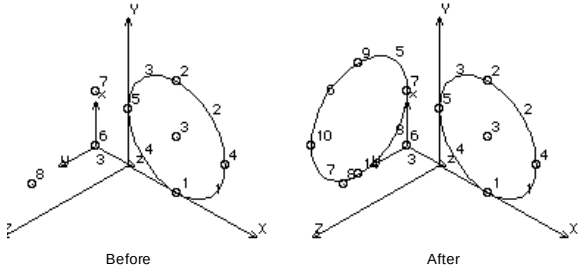
Example: `ELCOPY, 1, 6, 1, 3`

The above command copies elements 1 through 6 to local Cartesian coordinate system 3. Corresponding keypoints will have identical coordinates in both systems.

Graphic Example: ELCOPY

```
PT,1,30,0,0,
```

```
PT,2,30,60,0,
PLANE,Z,0,1,
ELPCIRDIA,1,1,2,
360,4,
PT,6,-20,0,0,
PT,7,-20,30,0,
PT,8,-20,0,40,
CSYS,3,0,6,7,8,
ACTSET,CS,0,
VIEW,1,1,1,
ELCOPY,1,4,1,3,
```



ELSCALE

Meshing > ELEMENTS > GENERATION > Scaling

The ELSCALE command generates a pattern of elements from an existing pattern, by scaling and translating/rotating them relative to the currently active Cartesian coordinate system. The coordinates of all associated keypoints are multiplied by the specified scale factors.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Generation flag

= 0: Translation only	by translation only
= 1: Rotation only	by rotation only
= 2: Both	by translation and rotation
	(default is translation only)

X, Y, Z-Scale factor
The scale factors in the X, Y, and Z directions.
(defaults are 1.0)

X, Y, Z-Displacement
The magnitudes of translations in the currently active Cartesian coordinate system. Prompted only if translations are included.
(defaults are 0.0)

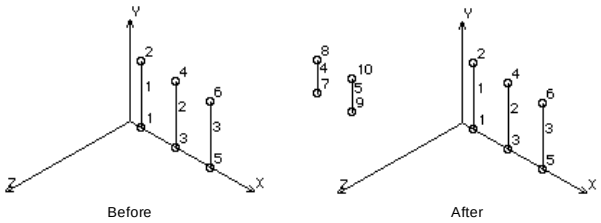
X, Y, Z-Rotation
The magnitudes of rotations in degrees in the currently active Cartesian coordinate system. Prompted only if rotations are included.
(defaults are 0.0)

Example: `ELSCALE, 1,4,1,0,0.0,0.5,0.0,5.0,0.0,0.0`

The above command generates 4 new elements from the existing elements 1 through 4 by applying a scale factor of 0.5 in the Y-direction and translating them by 5.0 units in the X-direction.

Graphic Example: ELSCALE

```
PT,1,10,0,0,
ELEXTRUDE,1,1,1,Y,
  50.,
ELGEN,2,1,1,1,2,30
  ,0,0,0,0,0,
ELSCALE,1,3,2,0,.5
  ,.5,0,80,10,50,
```



ELSYM

Meshing > ELEMENTS > GENERATION > Symmetry

The ELSYM command generates a pattern of elements symmetric to the specified pattern about a specified plane in the current coordinate system. An offset can also be specified. Use the ELFLIP (**Geometry, Elements, Generation, Flip**) command to flip existing elements without generating new ones.

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Rotation/Sweep axis
Axis representing normal to the plane about which pattern of elements are to be generated symmetrically.

= X	about y-z plane
= Y	about x-z plane
= Z	about x-y plane
	(default is Z)

Reverse direction flag
Reverse direction flag.

= 0: No	do not reverse directions of elements
= 1: Yes	reverse directions of elements
	(default is reverse direction)

Offset
Magnitude of the offset in the direction of the specified axis.

Notes

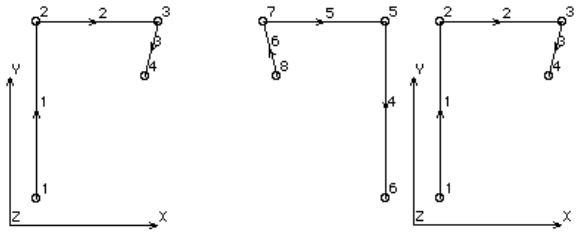
1. Entities can be picked by the mouse only if they are plotted on the screen.
2. A specified offset is applied to all elements of the specified pattern.

Example: `ELSYM, 1, 6, 1, X, 0, 0.0`

The above command generates 6 elements that are symmetric with elements 1 through 6, about the Y-Z plane. No offset is specified and the new elements have the same direction as the elements they are generated from.

Graphic Example: ELSYM

```
VIEW,0,0,1,
PLANE,Z,0,1,
ELPCORD,1,10,10,0,
  10,75,0,55,75,0,
  50,55,0,50,55,0,
ELSYM,1,3,1,X,1,0,
```



Before After

6 PropSets Menu

This menu contains commands to define, list and delete element attributes. The element attributes in COSMOS/M are the element group (EG), material property set (MP), real constant set (RC), and the element coordinate system (ECS). The element group defines the element type and associated details to be used in the analysis. The material property set contains a group of material properties to be associated with an element. The real constant set defines the geometric and other properties related to an element. The element coordinate system defines the coordinate system to be used for material angles definition and output stress directions.

Figure 6-1 PropSets Menu

EGROUP

Propsets > Element Group

The EGROUP command defines an element group and specifies the related options. A maximum of 5000 element groups can be defined. The defined element group becomes the default and remains so until another group is created, or the ACTSET (Control, Activate, Set Entity) command is used to activate another group. The prompted options depend on the selected element group (refer to the Element Library for detailed information). It is recommended to make sure that the proper element group and other element attributes are active before each meshing operation.

Element group
Element group number.
(default is active element group number + 1)

Element Name

Select one of the following element groups:

- Point Elements
- Line Elements
- Area Elements
- Volume Elements

Option(1) .... option (8)
Option flags for this element group.

Example: `EGROUP,1,SHELL4,,,,,,,,`

This command defines element group 1 to be SHELL4, the 4-node thin shell. All default options are accepted.

Point Elements

BUOY	1-node spherical buoyant element (usually used with the IMPIPE element)
GAP	1-node gap element
MASS	general mass element
SPRING	1-node spring element

Line Elements

BEAM2D	two dimensional elastic beam element
BEAM3D	three dimensional elastic beam element
BOUND	boundary element
CLINK	convection link
ELBOW	elastic curved pipe (elbow) element
ELINK	electrical link
FLOW2D	two dimensional fluid flow element
FLOW3D	three dimensional fluid flow element
GAP	2-node gap element
GENSTIF	general stiffness element
HLINK	hydraulic link
IMPIPE	2-node immersed pipe or cable element
PIPE	elastic straight pipe element
RBAR	rigid bar element
RLINK	radiation link

SHELLAX	axisymmetric shell element
SPRING	2-node spring element
TRUSS2D	two dimensional truss/spar element
TRUSS3D	three dimensional truss/spar element

Area Elements

MAG2D	2D magnetic element
MAG3D	3D magnetic element
PLANE2D	4- to 8-node two dimensional plane element
SHELL3	3-node thin shell element
SHELL4	4-node thin shell element
SHELL6	6-node thin shell element
SHELL9	9/8-node shell element
SHELL3L	multi layer 3-node shell/plate element
SHELL4L	multi layer 4-node shell/plate element
SHELL9L	composite 9/8-node shell element
SHELL3T	3-node thick shell element
SHELL4T	4-node thick shell element
SHELL6T	6-node thick shell element
TRIANG	3- to 6-node triangular element

Volume Elements

SOLID	8- to 20-node 3D solid element
SOLIDL	composite 8-node solid element
SOLIDPZ	8- to 20-node Piezoelectric solid element for frequency analysis
TETRA4	4-node tetrahedral solid element
TETRA4R	4-node tetrahedral solid element with translational and rotational degrees of freedom
TETRA10	10-node tetrahedral solid element




MPROP

Propsets > Material Property

The MPROP command can be used to define any material property used by GEO-STAR. The properties that can be defined are listed below. Some properties get associated with the active time curve, temperature curve, or BH curve as detailed in the notes below. The defined material property set becomes active and remains so until another material property set is created, or the ACTSET (Control, Activate, Set Entity) command is used to activate another set. Refer to the *Units* topic in the for units to be used in defining each property. It is recommended to make sure that the proper material property set and other element attributes are active before each meshing operation.

Material property set

Material property set number. (between 1 and 90)
(default is the active set number + 1)

Property value

Value(s) of the material property. CREEPC needs three values to be entered, CREEPX needs seven.
(defaults are 0.0)

Material property name

Name of the material property. The admissible property names are:

ALPH1	first power coefficient for the Ogden material model
ALPH2	second power coefficient for the Ogden material model
ALPH3	third power coefficient for the Ogden material model
ALPH4	fourth power coefficient for the Ogden material model
ALPX	coefficient of thermal expansion in the first material direction
ALPY	coefficient of thermal expansion in the second material direction.
ALPZ	coefficient of thermal expansion in the third material direction.
BETA	coefficient of volumetric expansion
C	specific heat in HSTAR and incompressible fluid flow. For compressible fluid flow, C is specific heat at constant pressure
COHESN	cohesion strength
CREEPC	creep constants for the classical creep law (three constants are needed)
CREEPX	creep constants for the exponential creep law (seven constants are needed)
CREEPTC	creep temperature-dependency constant.
DAMP	material damping coefficient
DCij	i=1,3 and j=i,3. A total of 6 constants. To define the upper half of the dielectric material matrix (used for Piezoelectric applications in STAR only)
DENS	mass density

ECONX	electric conductivity in the global Cartesian X-direction
ECONY	electric conductivity in the global Cartesian Y-direction
ECONZ	electric conductivity in the global Cartesian Z-direction
EMIS	emissivity
ENTHALPY	enthalpy
ETAN	tangent modulus
EX	elasticity modulus in the first material direction
EY	elasticity modulus in the second material direction
EZ	elasticity modulus in the third material direction
FRCANG	friction angle
G1 to G8	shear relaxation moduli used in visco-elasticity
GAMMA	ratio of specific heats
GXY	shear modulus relating first and second material directions
GXZ	shear modulus relating first and third material directions
GYZ	shear modulus relating second and third material directions
HC	convection film coefficient
K1 to K8	bulk relaxation moduli used in visco-elasticity
KC	Thermal contact resistance
KX	thermal conductivity in the x-direction of the coordinate system defined by ECS (for each element), which must be 0, -1, or user-defined. For 3D elements, the global Cartesian system is always used
KY	thermal conductivity in the y-direction of the coordinate system defined by ECS (for each element), which must be 0, -1, or user-defined. For 3D elements, the global Cartesian system is always used
KZ	thermal conductivity in the z-direction of the coordinate system defined by ECS (for each element), which must be 0, -1, or user-defined. For 3D elements, the global Cartesian system is always used
MCij	i=1,6 and j=i,6. A total of 21 constants *For STAR:* To define the upper half of the anisotropic material stiffness matrix (option 1), or compliance matrix (option 2). *For NSTAR:* To define 20 additional material properties that may be used in a user-defined material model (MC66 is not used. Refer to the Advanced Modules Manual).
MOONEY_A	first material constant for Mooney-Rivlin hyperelastic material model
MOONEY_B	second material constant for Mooney-Rivlin hyperelastic material model
MOONEY_C	third material constant for Mooney-Rivlin hyperelastic mateial model
MOONEY_D	fourth material constant for Mooney-Rivlin hyperelastic material model

MOONEY_E	fifth material constant for Mooney-Rivlin hyperelastic model
MOONEY_F	sixth material constant for Mooney-Rivlin hyperelastic material model
MPERM	magnetic permeability
MPERM_R	real part of relative permeability
MPERM_I	imaginary part of relative permeability
MU1	first constant for the Ogden material model
MU2	second constant for the Ogden material model
MU3	third constant for the Ogden material model
MU4	fourth constant for the Ogden material model
NUXY	Poisson's ratio, relating strain in the second material direction to strain in the first material direction
NUXZ	Poisson's ratio, relating strain in the third material direction to strain in the first material direction
NUYZ	Poisson's ratio, relating strain in the third material direction to strain in the second material direction
PCij	i=1,6 and j=1,3. A total of 18 constants to define the material matrix (used in STAR only)
PERMIT	permittivity (dielectric constant)
PERMIT_R	real part of relative permittivity
PERMIT_I	imaginary part of permittivity
PMAGR	coercivity of a permanent magnet in the radial direction of the global cylindrical system
PMAGT	coercivity of a permanent magnet in the circumferential direction of the global cylindrical system
PMAGX	coercivity of a permanent magnet in the global Cartesian X-direction
PMAGY	coercivity of a permanent magnet in the global Cartesian Y-direction
PMAGZ	coercivity of a permanent magnet in the global Cartesian Z-direction
REFTMP	glassy transition temperature for visco-elasticity
SIGXC	compressive strength in the first material direction
SIGXT	tensile strength in the first material direction
SIGXYC	Compressive shear strength in the plane defined by the first and second material
SIGXY	shear strength in the plane defined by the first and second material directions
SIGYLD	yield stress
SIGYT	tensile strength in the second material direction
TAUG1 to TAUG8	time values associated with G1 through G8, respectively and are used in visco-elasticity
TAUK1 to TAUK8	time values associated with K1 through K8, respectively and are used in visco-elasticity

VC1 and VC2	constants for the Williams-Landel-Ferry equation used in viscoelasticity (refer to the Advanced Modules Manual)
VISC	dynamic viscosity

Property value

Value(s) of the material property. CREEPC needs three values to be entered, CREEPX needs seven.
(defaults are 0.0)

GEOSTAR will prompt for more material properties and values until terminated by null entry for material property name.

Notes

1. Some material properties are associated with the active temperature curve. The following is a listing of these properties for different modules:

   a. For STAR, DSTAR, and ASTAR:

   ALPX ALPY ALPZ DENS EX EY EZ GXY
   GYZ GXZ NUXY NUYZ NUXZ MCij DCij PCij
   DAMP SIGXT SIGYT SIGXC SIGYC SIGXY

   b. For NSTAR:

   ALPX ALPY ALPZ EX EY EZ GXY GYZ
   GXZ NUXY NUYZ NUXZ MCij ETAN SIGYLD

   G1, G2,..., G8 and K1, K2,..., K8 are used in the viscoelastic material model in NSTAR and are temperature-dependent using the William-Landel-Ferry function, and time-dependent based on TAUG1 through TAUG8, and TAUK1 through TAUK8. (Refer to the Advanced Modules Manual.)

   c. For HSTAR:

   C DENS KX KY KZ EMIS HC ECONX ENTHALPY

   EMIS and HC are also associated with the active time curve. The input value is multiplied by corresponding values from both the time and temperature curves. Note that HSTAR uses the average element face temperature and the ambient temperature to enter the temperature curve for HC.

   d. For ESTAR:

   MPERM is associated with the active B-H curve. MPERM should be defined as unity and MPERM values will be as given in the B-H curve.

3. It is good practice to use the following sequence of commands to associate material properties with a desired temperature, time curve, or both:

   a. Activate the curve to be used for association:

   ACTSET, TP, temperature curve number, or ACTSET, TC, time curve number

   b. Define material properties:

   ..... (Material properties defined here will be associated with the active curve(s), if that capability is supported.) .....

   c. Activate temperature or time curve zero:

ACTSET, TP, 0,
ACTSET, TC, 0,

d. Define material properties:
..... (Material properties defined here have fixed values and will not be associated with any temperature or time curves.) .....

4. For NSTAR, the following table shows the temperature dependent properties used for different material models in association with certain element groups:

Property	Material Model	Element Groups
ALPX	all material models	all element groups
EX, NUXY, and GXY	elastic isotropic	all element groups
	elastic orthotropic	PLANE2D, SOLID, SHELL3L, SHELL4L, TRIANG, and TETRA'S
EX, NUXY, SIGYLD, and ETAN	von Mises plasticity	TRUSS2D, TRUSS3D, SOLID, TRIANG, PLANE2D, and TETRA'S
EY, EZ, GXZ, GYZ, NUXZ, NUYZ, ALPY, and ALPZ	elastic orthotropic	PLANE2D, SOLID, SHELL3L, SHELL4L, TRIANG, and TETRA'S
VISC	elastic (fluid viscosity)	IMPIPE

5. For the Mooney-Rivlin material model:
   a. The sum of MOONEY_A and MOONEY_B must be greater than zero.
   b. The Mooney-Rivlin model should be used if Poisson's ratio is greater than 0.48.
   c. The Mooney-Rivlin strain energy density function is expressed as:

$$W = W1 + W2$$

$$W1 = A(I - 3) + B(II - 3) + X [1./(III)2 - 1] + Y(III - 1)2$$

$$W2 = C(I - 3) (II - 3) + D(I - 3)2 + E(II - 3)2] + F(I - 3)3$$

Where I, II, III are invariants of the right Cauchy-Green deformation tensor which can be expressed in terms of principal stretch ratios. A, B, C, D, E, and F are Mooney-Rivlin material constants; and:

$$X = A/2. + B$$

$$Y = [A(5 \times nu - 2.0) + B(11 \times nu - 5.0)] / 2.0(1 - 2 \times nu)$$

nu = Poisson's ratio

6. For orthotropic materials, the following assumptions are made:
   a. NUXY defaults to 0.3, NUXZ and NUYZ default to 0.0.
   b. EY and EZ default to EX.
   c. GXY, if not explicitly given, will be calculated from
      GXY = EX /[2(1 + NUXY)] for isotropic materials, and from
      GXY = (EX.EY) / (EX + EY + 2.EY.NUXY) for orthotropic materials.
   d. GYZ and GXZ default to GXY.

Example:

```
ACTSET (Control, Activate, Set Entity), TP, 3,
MPROP (PropSets, Material Property), 1, EX, 0.8,
ACTSET, TP, 0,
MPROP, 2, EY, 3.0E6
```

The first command activates temperature curve `3` defined by the CURDEF (LoadsBC, Function_Curve, Time/Temp Curve) command. The second command defines `EX` for property set number `1` to be `0.8` times the value obtained from temperature curve `3`. The third command deactivates the association with temperature curve `3`. and the fourth command defines a fixed value of `3.0E6` for `EY` in material property set `2`.

The `EX` value to be used for an element associated with material property set `1` will be calculated by:

a. averaging the temperatures at the nodes of that element,
b. finding the value of temperature curve `3` at that average temperature,
c. multiplying the obtained value by `0.8`, and assigning the resulting value for `EX` to material property set `1`,
d. `EY` for material property set number `2` will have a fixed value of `3.0E6`.

RCONST

Propsets > Real Constant

The RCONST command defines a real constant set for an element group. The defined real constant set becomes active and remains so until another real constant set is created, or the ACTSET (Control, Activate, Set Entity) command is used to activate another set. It is recommended to define the associated element group before defining the real constant set. The user is then prompted for the geometric and other properties related the associated element group. For example, the first real constant in the set is thickness for shells and cross-sectional area for trusses (refer to the Element Library for details).

Associated element group
Element group associated with this real constant set.
(default is the active element group)

Real constant set

Real constant set number. (between 1 and 5000)
(default is the active number set + 1)

Start location of the real constant

Starting location of the first constant.
(default is the highest location defined + 1)

Number of real constants to be entered

Number of constants that will be entered.
(default depends on element group)

Real constant (i)

The value of the ith real constant.
(defaults are 0.0)

Notes

1. A maximum of 10 real constants can be specified each time the command is issued. Repeat the command as many times as necessary to define all real constants. The default *Start location of the real constant* is equal to the highest location defined + 1.
2. The *Start location of the real constant and the Number of real constants to be entered* entries let you modify some of the real constants without having to re-enter all constants.

Example: `RCONST,2,1,3,2,1.2,3.1`

This command defines real constant set 1 for element group 2 to have its third and fourth real constants as `1.2` and `3.1` respectively.

PICK_MAT

Propsets > Pick Material Lib

The PICK_MAT command defines a material property set by picking a material from the COSMOS/M material library. The material library is contained in the `pickmat.lib` file located in the COSMOS/M installation folder. It is recommended to make sure that the proper material property set and other element attributes are active before each meshing operation.

Material property set

Material set label. (between 1 and 90)
(default is the active material set number + 1)

Material name

Name of the selected material. The available materials are:

A_STEEL	ALLOY STEEL
ACRYLIC	ACRYLIC (Medium-High Impact)
AIR	AIR
ALUMINUM	ALUMINUM ALLOY
AL_1345	ALUMINUM 1345 ALLOY

AL_1350	ALUMINUM 1350 ALLOY
AL_2014	ALUMINUM 2014 ALLOY
AL_2018	ALUMINUM 2018 ALLOY
AL_2024	ALUMINUM 2024 ALLOY
AL_3003	ALUMINUM 3003 ALLOY
AL_6061	ALUMINUM 6061 ALLOY
AL_7079	ALUMINUM 7079 ALLOY
AL_BRONZE	ALUMINUM BRONZE
BRASS	SILICON BRASS & BRONZE
BRONZE	BRONZE
CA_STEEL	CAST ALLOY STEEL (Below 8% Content)
COBALT	COBALT
COPPER	COPPER
CS_STEEL	CAST STAINLESS STEEL (CF-8M or CF-20)
GC_IRON	GRAY CAST IRON (ASTM CLASS40)
GLASS	GLASS
GOLD	PURE GOLD
IRON	IRON
LEAD	PURE LEAD
MAGNES	MAGNESIUM ALLOY (Wrought or Cast)
MC_IRON	MALLEABLE CAST IRON (ASTM - A220)
MN_BRONZE	MANGANESE BRONZE
MOLYBDENUM	MOLYBDENUM
MONEL	MONEL 400
NICKEL	NICKEL
D_NICKEL	DURANICKEL 301
NYLON	NYLON 6/10
PC_STEEL	PLAIN CARBON STEEL
PORCELAIN	CERAMIC PORCELAIN
RUBBER	RUBBER
SILVER	PURE SILVER
STEEL	STEEL
ST_1020	STEEL, AISI C1020 (Hot Worked)
ST_304	STEEL, AISI 304 (Sheet)
ST_ST	STAINLESS STEEL
T_BRONZE	TIN BEARING BRONZE
TITANIUM	TITANIUM
TUNGSTEN	TUNGSTEN
VANADIUM	VANADIUM
WATER	WATER
W_COPPER	WROUGHT COPPER
WS_STEEL	WROUGHT STAINLESS STEEL




ZIRCONIUM	ZIRCONIUM *(default is A_STEEL)*

Unit label

Units in which properties are to be defined.

= FPS	British system of units. (inch for distance, pound for force and second for time)
= SI	International system of units. (meter for distance, kilogram for mass and second for time)
= MKS	Metric system of units. (centimeter for distance, kilogram for force and second for time) *(default is FPS)*

Notes

1. The FPS and MKS are gravitational systems, while the SI system is an absolute system. An absolute system is independent of the numerical value of gravity since the unit of mass is a basic unit and the unit of weight is a derived unit. A gravitational system is dependent on the numerical value of gravity since the unit of weight is basic and the unit of mass is derived. The SI system is location independent and can be used anywhere. The other two systems are location dependent.
2. The values and units are written to the session file.
3. The user can add materials to the library by editing the **pickmat.lib** file.

Example: **PICK_MAT, 2, COPPER, FPS**

This command assigns the basic material properties of copper to material set **2**. The command opens the file **pickmat.lib** to read properties in the FPS (English) system of units.

USER_MAT

Propsets > User Material Lib

The USER_MAT command defines a material property set by picking a material from a user-created material library. The user material library must be placed in the COSMOS/M subdirectory and must be named **usermat.lib**. The existing revision of the **usermat.lib**.contains a material library for electromagnetic properties. If you are not using COSMOS/HFS, you can delete or add to the entries in the file. You may edit this file by any text editor, but the exact same format must be used. Another approach for creating a material library is to open a new GEOSTAR problem, use the commands to define all desired material property sets, and close GEOSTAR. All the defined material will be in the session file that can be loaded to other problems using the FILE, Load command.

Material property set

Material set label. (between 1 and 90)
(default is the active material set number + 1)

Material name

Enter the name of the material. The available materials are:

Material Name	Description
AIR	Air Er=1.0 Tand=0.0
CELLULAR_FEP	Er=1.50, Tand=0.0007
CELLULAR_TFE	Er=1.40, Tand=0.0002
Duroid_5500	Rogers Corp. Ceramic/PTFE Er=2.5, Tand=N/A
DUROID_5870	Rogers Corp. PTFE/Random Glass Er=2.33, Tand=0.0012
DUROID_5880	Rogers Corp. PTFE/Random Glass Er=2.33, Tand=0.0009
DUROID_6002	Rogers Corp. Er=2.94, Tand=N/A
DUROID_6006	Rogers Corp. Er=6.00, Tand=0.0025
DUROID_6010.2	Rogers Corp. Er=10.2, Ceramic/PTFE Tand=N/A
DUROID_6010.5	Rogers Corp. Er=10.5, Tand=0.0028
DUROID_6010.5	Rogers Corp. Er=10.5, Tand=0.0028
E002	Mitsubishi Plastic Er=5.1, Tand=0.022
EPSILAM10	Keene Corp. Er=10.2, Tand=0.0020
EPSILAM6	Keene Corp. Er=6.00, Tand=0.0018
ETH_PROP	Ethylene Propylene Er=2.24, Tand=0.00046
FEP	Er=2.10, Tand=0.00007
FR-4	Norplex/Oak Epoxy/Glass Er=4.3, Tand=0.020
G-10	Generic Epoxy/Glass Er=4.3, Tand=0.008
GAAS	Gallium Arsenide Er=13.1, Tand=0.0016, Cond=0.008 S/m
GE	Germanium Er=16.0, Tand=N/A
INSP	Er=15.9, Tand=N/A
K002	Mitsubishi Plastic Er=3.6, Tand=0.0035
K012	Mitsubishi Plastic Er=3.5, Tand=0.0021
LOSSLESS	Make Material Lossless,Tand=0.0
PAPER_492	Norplex/Oak Er=4.8, Tand=0.045
POLYMIDE	Norplex/Oak Er=4.2, Tand=0.015
POLYMIDFLESMD	Rogers Corp. Er=3.40, Tand=0.003
RO2800	Rogers Corp. Er=2.88, Tand=N/A
SAPPHIRE	Sapphire Er=10.0, Tand=0.0001, Cond=5.5E-04 S/m
SI	Silicon Er=11.7, Tand=0.005, Cond=4.39E-04 S/m
SI_RUBBER	Er=2.08, Tand=N/A




Material Name	Description
SIO	Silicon Monoxyde Er=6.0, Tand=0.007
SIO2	Silicon Dioxyde Er=3.9, Tand=0.007
SOLID_PTFE	Er=2.10, Tand=0.0002
SPLINE_PTFE	Er=1.35, Tand=0.0002
TA2O5	Tantalum Quintoxyde Er=25.0, Tand=0.003
TEFLON	Er=1.96, Tand=N/A
TEFLONI	Er=2.60, Tand=0.005
TMM-10	Rogers Corp. Er=9.80, Tand=0.0017
TMM-13	Rogers Corp. Er=12.85, Tand=0.0019
TMM-3	Rogers Corp. Er=3.24, Tand=0.0018
TMM-4	Rogers Corp. Er=4.50, Tand=0.0018
TMM-6	Rogers Corp. Er=6.50, Tand=0.0018

Unit label

Units in which properties are to be defined.

= FPS British system of units. (inch for distance, pound for force and second for time)

= SI International system of units. (meter for distance, kilogram for mass and second for time)

= MKS Metric system of units. (centimeter for distance, kilogram for force and second for time)
(default is FPS)

For more information, visit www.cosmosm.com, or www.electromagnetic-works.com.

R_MATLIB

Propsets > Material Browser

The R_MATLIB command launches a material browser which can be used to define material property sets. The library is produced and maintained by CenTOR Software and it provides a large library of materials and associated temperature curves. After selecting the desired materials and exiting the browser, the selected materials and associated temperature curves will be automatically loaded into the database. Control will then be transferred back to GEOSTAR. The MPROP (PropSets, Material Property) command can be used to add or modify any material properties in the set. The browser is an add-on utility that must be acquired before executing this command. For more information, visit www.cosmosm.com, or www.centor.com.




PICK_SEC

Propsets > AISC Sect. Table

The PICK_SEC command assigns a selected section from the AISC section tables to a real constant set to be used with BEAM3D elements. The defined real constant set becomes the default and remains so until another real constant is created, or the ACTSET (Control, Activate, Set Entity) command is used to activate another set.

Associated element group
Element group associated with this section.
(default is the active element group)

Real constant set
Real constant set label.

Section name
Name of AISC section. The admissible sections are: WBEAM, MBEAM, HPBEAM, SBEAM, CBEAM, STUBE, MCBEAM, ANGLE, RANGLE, USRANGL, PIPE, and USRBEAM.

AISC dimension 1
Normal depth of the section in the y-direction as used in the AISC (for an angle section, it is the length of the leg in the y-direction).

AISC dimension 2
Nominal weight per foot of the section according to AISC, for all sections except angles. For an angle section, this argument is used as the length in the x-direction as used in AISC.

AISC dimension 3
Thickness of the section according to AISC. Used for ANGLE sections only.

Notes

1. AISC dimension 1, AISC dimension 2 and AISC dimension 3 are the identification parameters for the section according to AISC. The unit of length is inches and that of weight is lb/ft. This command should only be used in conjunction with the FPS system.
2. AISC dimension 3 is required only for ANGLE, RANGLE, and STUBE sections.
3. The definitions of local x, y, and z coordinates are different in GEOSTAR and the AISC code.
4. The weight of the section (used for sections other than angles) is based on the density of steel. If the user is using some other material, the weight should be based on steel as in the AISC code. This does not affect any material properties since it is used only to define section properties.




EPROPCHANGE

Propsets > Change El-Prop

The EPROPCHANGE command can be used to change an element attribute associated with a pattern of elements. The command also assigns to the specified elements based on the changed attribute. The element attributes that can be changed by this command are: EG (element group), RC (real constant), MP (material property), and EC (element coordinate system).

Beginning Element
Beginning element in the pattern.

Ending Element
Ending element in the pattern.
(default is the maximum number of elements defined)

Increment
Increment between the elements.
(default is 1)

Property set name (element attribute)
Set name. Admissible set names are:
EG: Element Group
RC: Real Constant
MP: Material Property
ECS: Element Coordinate System
(default is EG: Element Group)

Set label to be assigned
Set label to be assigned.
(default is the previously assigned set label + 1)

Color of elements
Color of elements.
(default is 2)

Example: `EPROPCHANGE, 1, 16, 1, MP, 3, 10`

Suppose that elements 1 through 16 were assigned material property set 2 by mistake. The above command changes the material of these elements to be material set 3 and assigns the red color to it. Subsequent plotting of elements with active element coloring based on materials, will show these elements in red. See the EpropSet (Meshing, Elements, Activate Elem Color) command for details.




EPROPSET

Propsets > New Property Set

The EPROPSET command controls the procedure for assigning attributes for new elements generated from existing ones by various operations on geometric entities with active default meshing. The element attributes include element groups, element coordinate systems, material property and real constant sets. New elements can be generated by directly operating on a pattern of elements or through their association with geometric entities. New elements created by the EL command or through meshing of geometric entities will still assume the active attributes.

Property set name

Property set name.
EG: Element Group
MP: Material Property
RC: Real Constant
ECS: Element Coordinate System
(default is EG: Element Group)

Set flag

Assignment flag. (-1=act, 0=source, n=source+n)
= -1 assign the active set
= 0 assign the source set
= n increment the source set by n
(default is -1)

Example: Suppose you have meshed a surface whose elements are assigned element group 1, material set 1, and real constant set 1.

```
EPROPSET, EG, 0,
EPROPSET, RC, 3,
EPROPSET, MP, 2,
EPROPSET, ECS, -1,
ACTDMESH, SF, 1,
SFGEN, 2, 1, 1, 1, 0, 10., 10., 0.,
```

The SFGEN (Geometry, Surfaces, Generation, Generate) command generates two new surfaces, the elements on the first surface are associated with element group 1 (same as the source), real constant set 4 (i.e., 1+3), and material property set 3 (i.e., 1+2). The elements on the second surface are associated with element group 1 (same as the source), real constant set 7 (i.e., 1+3+3), and material property set 5 (i.e., 1+2+2). The active value for ECS is assumed for all new elements.




BMSECDEF

Propsets > Beam Section

The BMSECDEF command defines a cross-section for beam elements. The command substitutes the general RCONST (PropSets, Real Constant) command to define real constant sets. You can select one of several predefined shapes, or you can define an arbitrarily shaped cross-section. The cross-sectional area, moments of inertia, center of gravity, and the shear center are automatically calculated by the program. It should be noted that the command *must* be used to define the real constant set for non-rectangular beam elements with material nonlinearity, since the RCONST command assumes a rectangular cross-section in this case. For beam elements with geometric nonlinearity (large displacement), you have the option to use the RCONST command, this command, or the PICK_SEC (PropSets, AISC Sect. Table) command.

Associated Element group

Associated element group (must be a beam element group).
(default is the active element group)

Real Constant set

Label for the real constant set to be defined.

Section number

Shape of the cross-section:
= 0; User-Defined Section
= 1; Rectangular section (solid)
= 2; Circular Section (solid)
= 3; Pipe Section
= 4; Box Section
= 5; I-Section

The following options will be available only when BEAM3D (UnSymmetric) element group is active.

= 6; Trapezoidal Section
= 7; Open Channel Section
= 8; Z-Section
= 9; T-Section
=10; L-Section
(defaults is 1: Rectangular Section)

Number of user section points

Number of points to define the user-defined section. Prompted only if “0” is selected for section number.

Start location of the section parameters

Starting location for storing section parameters.
(default is the highest defined location + 1)

No. of beam section constants

Number of beam section constants to be entered.

(defaults depends on "section number")

bs (i)

Geometric real constants (i=1,2,....n). The user is prompted for the required constants depending on the selected "section number". Refer to Figure 6-2. The description of the constants for various sections is given below.

r (i)

Non-geometric real constants. Refer to the linear and nonlinear BEAM2D and BEAM3D elements in Chapter 4 of the COSMOS/M User Guide for detailed definitions of the particular non-geometric real constants. Note that the command prompts for all the properties that are not automatically calculated. Gaps in numbering the r (..) may exist, since the command may calculate non-consecutive real constants. For example, the command calculates values corresponding to real constants 1 through 5, and 8 for BEAM3D elements, but the values corresponding to real constants 6, 7, 9,..etc., should be entered.

0 : User-defined Section

0. User-defined section (3D unsymmetrical beam):
   (section type 0)
   c(1) y-coordinate of the section at node-1
   c(2) z-coordinate of the section at node-1
   c(3) Thickness of the section at node-1
   .....
   .....
   .....
   This pattern can be repeated to define all needed properties.

1.Rectangular cross-section:

a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
   c(1) Height of the beam (H)
   c(2) Width of the beam (B)

b. Symmetric tapered BEAM3D elements:
   c(1) Height of the beam at node-1 (H1)
   c(2) Width of the beam at node-1 (B1)
   c(3) Height of the beam at node-2 (H2)
   c(14) Width of the beam at node-2 (B2)

2.Circular cross-section:

a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
   c(1) Radius of the beam (R)

b. Symmetric tapered BEAM3D elements:
   c(1) Radius of the beam at node-1 (R1)
   c(2) Radius of the beam at node-2 (R2)

3.Pipe cross-section:

a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
   c(1) Outside diameter of the beam (D)
   c(2) Thickness of the beam (T)

b. Symmetric tapered BEAM3D elements:
   c(1) Outside diameter of the beam at node-1 (D1)
   c(2) Thickness of the beam at node-1 (T1)
   c(3) Outside diameter of the beam at node-2 (D2)
   c(4) Thickness of the beam at node-2 (T2)

4.Box cross-section:

a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
   c(1) Height of the beam (H)
   c(2) Width of the beam (B)
   c(3) Thickness associated with height (TB)
   c(4) Thickness associated with width (TH)

b. Symmetric tapered BEAM3D elements:
   c(1) Height of the beam at node-1 (H1)
   c(2) Width of the beam at node-1 (B1)
   c(3) Thickness of the web at node-1 (TB1)
   c(4) Thickness of the flange at node-1 (TH1)
   c(5) Height of the beam at node-2 (H2)
   c(6) Width of the beam at node-2 (B2)
   c(7) Thickness of the web at node-2 (TB2)
   c(8) Thickness of the flange node-2 (TH2)

5.I-section:

a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
   c(1) Height of the beam (H)
   c(2) Width of the beam (B)
   c(3) Thickness of the flange (TH)
   c(4) Thickness of the web (TB)

b. Symmetric tapered BEAM3D:
   c(1) Height of the beam at node-1 (H1)
   c(2) Width of the beam at node-1 (B1)
   c(3) Thickness of the flange at node-1 (TH1)
   c(4) Thickness of the web at node-1 (TB1)
   c(5) Height of the beam at node-2 (H2)
   c(6) Width of the beam at node-2 (B2)
   c(7) Thickness of the flange at node-2 (TH2)
   c(8) Thickness of the web at node-2 (TB2)

6.Trapezoidal (unsymmetrical BEAM3D):

c(1) Height of the beam (H)
c(2) Bottom width of the beam (B1)
c(3) Top width of the beam (B2)




7.Open channel (unsymmetrical BEAM3D):

c(1) Height of the beam (H)
c(2) Width of the beam (B)
c(3) Thickness of the flange (TH)
c(4) Thickness of the web (TB)

8.Z-section (unsymmetrical BEAM3D):

c(1) Height of the beam (H)
c(2) Width of the beam (B)
c(3) Thickness of the flange (TH)
c(4) Thickness of the web (TB)

9.T-section (unsymmetrical BEAM3D):

c(1) Height of the beam (H)
c(2) Width of the beam (B)
c(3) Thickness of the flange (TH)
c(4) Thickness of the web (TB)

10.L-section (unsymmetrical BEAM3D):

c(1) Height of the beam (H)
c(2) Width of the beam (B)
c(3) Thickness associated with height (TH)
c(4) Thickness associated with width (TB)

The beam section constants shown above are required to define the geometry of the cross-section. Beside the geometric constants, some other real constants are also required to fully define the element. The following is a list of the required non-geometric constants:

1. BEAM2D elements
   r(4) End-release code (node-1)
   r(5) End-release code (node-2)
   r(6) Shear factor in the element y-axis
   r(7) Temperature difference in the element y-axis
2. Symmetric BEAM3D elements
   r(6) End-release code (node-1)
   r(7) End-release code (node-2)
   r(9) Shear factor in the element y-axis
   r(10) Shear factor in the element z-axis
   r(11) Temperature difference in the element y-axis
   r(12) Temperature difference in the element z-axis
3. Unsymmetrical BEAM3D elements
   r(6) End-release code (node-1)
   r(7) End-release code (node-2)
   r(9) Shear factor in the element y-axis
   r(10 Shear factor in the element z-axis
   r(11) Temperature difference in the element y-axis
   r(12) Temperature difference in the element z-axis




r(15) x-distance of section centroid relative to nodal point location at node-1
r(16) x-distance of section centroid relative to nodal point location at node-2
r(17) y-distance of section centroid relative to nodal point location at node-1*
r(18) y-distance of section centroid relative to nodal point location at node-2*
r(19) z-distance of section centroid relative to nodal point location at node-1*
r(20) z-distance of section centroid relative to nodal point location at node-2*
r(25) y-distance of the point where stresses are to be calculated
r(26) z-distance of the point where stresses are to be calculated

* r(17) through r(20) are input for predefined sections (shape-1 to shape-10) only

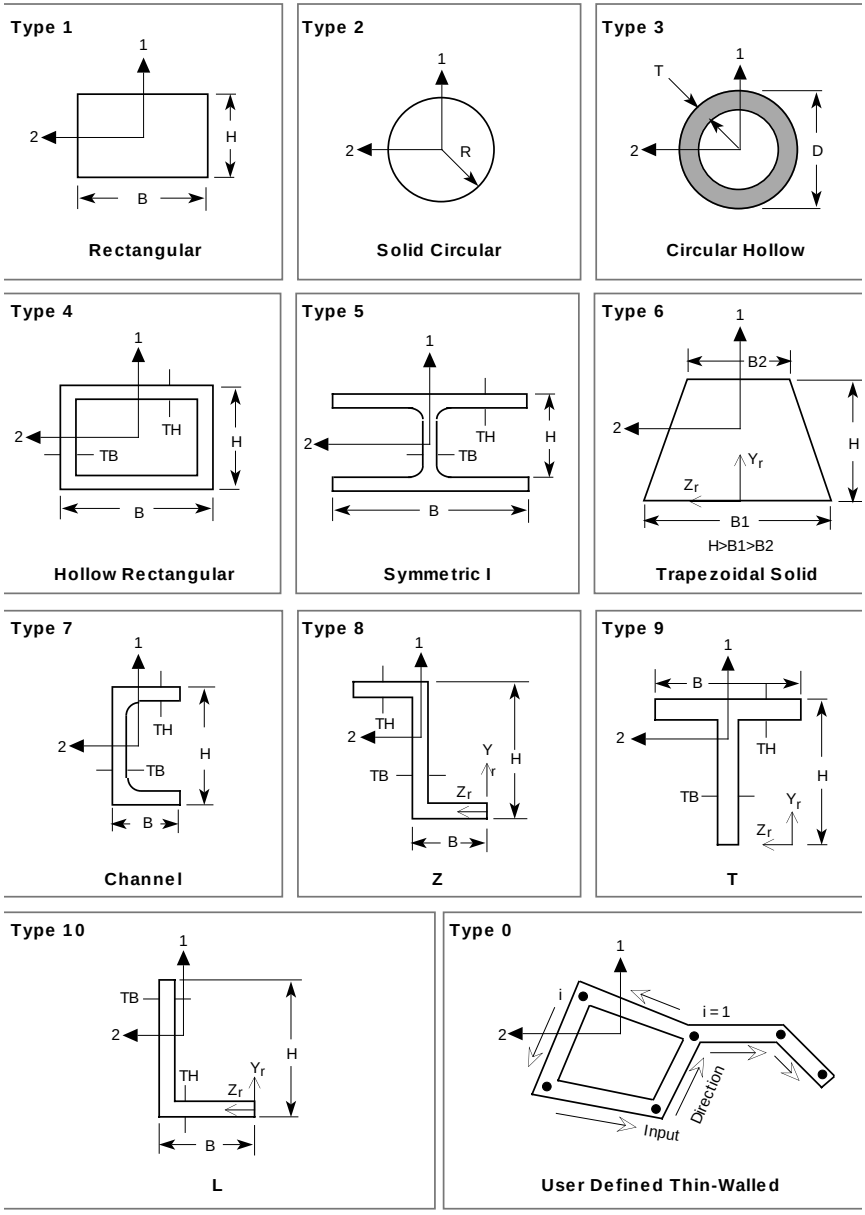
4. Symmetric tapered BEAM3D elements
   r(11) End-release code (node-1)
   r(12) End-release code (node-2)
   r(15) Shear factor in the element y-axis
   r(16) Shear factor in the element z-axis
   r(17) Temperature difference in the element y-axis
   r(18) Temperature difference in the element z-axis

Notes

1. The BMSECDEF and RCONST commands can be used alternately. Both commands share the same database locations and any information provided by one of them will overwrite the previous input defined by the other.

2. Precautions for user-defined section:
   a. The wall thickness of the user-defined section should be small compared to the total length of the section.
   b. The beam section must have a continuous outline such that the end point of one segment should be the beginning point of the next.
   c. The first node must be input with a 'zero' thickness.
   d. The input with 'zero' thickness must follow the original geometry (trace back) in order to maintain a continuous outline.
   e. The closed section must constitute a single cell. Multi-cell sections are not supported.
   f. The input with 'zero' thickness cannot appear in the closed section.
   g. The closed section must be input in a counter-clockwise order.
   h. If the section includes a closed cell, then the segment input must start with it.

Figure 6-2 Identification Parameters of the Built-in BEAM3D Section Library (Viewed by Looking in the Negative X-direction





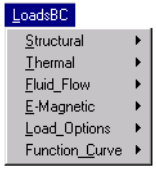

7 LoadsBC Menu

LoadsBC Menu

Geo Panel: LoadsBC

This menu contains commands to specify various types of loadings and boundary conditions. The commands are grouped into five menus according to their type. It should be mentioned that these divisions refer to the type of boundary condition and not necessarily the analysis. For example, structural analysis utilizes the temperature menu given under "THERMAL", and the fluid flow analysis uses the "PRESSURE" menu from "STRUCTURAL" and most of the menus under "THERMAL". Boundary conditions are always applied to nodes or elements but geometry association can be conveniently used. Commands for defining, deleting and listing loads and boundary conditions are provided. Except for the LOAD OPTIONS menu, the first one, two or three letters indicate the type of loading or boundary condition to be operated on, and the rest refer to the geometry association and the function. For example, the NTSF command applies a temperature at nodes associated with a pattern of surfaces and the DRDEL command deletes prescribed displacements at nodes associated with a pattern of regions. Loads and boundary conditions are associated with the active loadcase. Depending on the analysis type, some load and boundary conditions are also associated with the active curve (refer to Chapter 3 of the COSMOS/M User Guide).

Figure 7-1 LoadsBC Menu

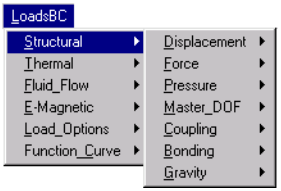


▶ STRUCTURAL Menu

Geo Panel: **LoadsBC > STRUCTURAL**

This menu contains commands to specify structural loading and boundary conditions like forces, pressure, and displacements.

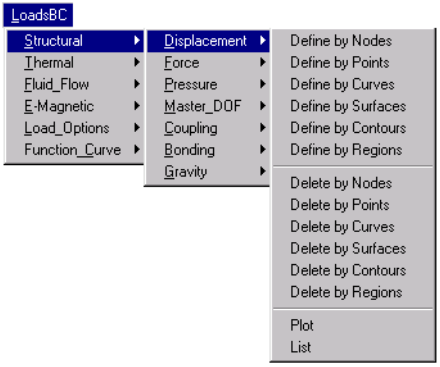
Figure 7-2 Structural Menu



▼DISPLACEMENT Menu

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT**

Figure 7-3 Displacement Menu



DND

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > Define by Nodes**

The DND command specifies a displacement boundary condition value for a pattern of nodes. The prescribed value of the displacement is interpreted in the active coordinate system and can be zero or nonzero. Up to 5 coordinate systems can be used to define constraints at a node. If a contradiction is found, the analysis module will give a message before stopping. For NSTAR, a nonzero value must be associated with a time curve and the force-control method must be used. FFE Static uses one coordinate system, the one used for the last constraint, to define constraints at a node.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Displacement label
Label of the degree of freedom (interpreted in the active coordinate system).

= UX	translation in the x-direction
= UY	translation in the y-direction
= UZ	translation in the z-direction
= RX	rotation about the x-axis
= RY	rotation about the y-axis
= RZ	rotation about the z-axis
= AL	all 6 degrees of freedom
= AU	all translations
= AR	all rotations
= SX	symmetrical B.C. about the y-z plane (UX = RY = RZ = 0.0)
= SY	symmetrical B.C. about the x-z plane (UY = RX = RZ = 0.0)
= SZ	symmetrical B.C. about the x-y plane (UZ = RX = RY = 0.0)
= AX	anti-symmetrical B.C. about the y-z plane (RX = UY = UZ = 0.0)
= AY	anti-symmetrical B.C. about the x-z plane (RY = UX = UZ = 0.0)
= AZ	anti-symmetrical B.C. about the x-y plane (RZ = UX = UY = 0.0)

Value
Displacement value. Use radians for rotations. The specified value is associated with the active time curve for NSTAR.
(default is 0.0)

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Displacement labels
Additional labels. (degrees of freedom)

Note
Additional degrees of freedom are assigned the same prescribed value.

Example:
```
DND, 1, UX, , 3, 1, RZ,
```
This command specifies zero X-displacement and Z-rotation to nodes 1, 2 and 3. Note that if the active coordinate system is cylindrical, then "UX" refers to the radial direction.

DPT

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > Define by Points

The DPT command specifies a displacement boundary condition value for all nodes associated with a pattern of keypoints. The prescribed value of the displacement is interpreted in the active coordinate system and can be zero or nonzero. Up to 5 coordinate systems can be used to define constraints at a node. If a contradiction is found, the analysis module will give a message before stopping. For NSTAR, a nonzero value must be associated with a time curve and the force-control method must be used. FFE Static uses one coordinate system, the one used for the last constraint, to define constraints at a node.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Displacement label
Label of the degree of freedom (interpreted in the active coordinate system).

= UX	translation in the x-direction
= UY	translation in the y-direction
= UZ	translation in the z-direction
= RX	rotation about the x-axis
= RY	rotation about the y-axis
= RZ	rotation about the z-axis
= AL	all 6 degrees of freedom
= AU	all translations
= AR	all rotations
= SX	symmetrical B.C. about the y-z plane (UX = RY = RZ = 0.0)
= SY	symmetrical B.C. about the x-z plane (UY = RX = RZ = 0.0)
= SZ	symmetrical B.C. about the x-y plane (UZ = RX = RY = 0.0)
= AX	anti-symmetrical B.C. about the y-z plane (ROTX = UY = UZ = 0.0)
= AY	anti-symmetrical B.C. about the x-z plane (RY = UX = UZ = 0.0)
= AZ	anti-symmetrical B.C. about the x-y plane (RZ = UX = UY = 0.0)

Value
Displacement value. Use radians for rotations. The specified value is associated with the active time curve for NSTAR.
(default is 0.0)

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment

Increment between keypoints in the pattern.
(default is 1)

Displacement labels
Additional labels. (degrees of freedom)

Note
Additional degrees of freedom are assigned the same prescribed value.

Example:
```
DPT, 1, UY, ,3, 1, UX, RZ,
```
This command specifies zero X- ,Y-displacements and zero Z-rotation to all the nodes associated with keypoints 1, 2 and 3.

DCR

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > Define by Curves**

The DCR command specifies a displacement boundary condition value for all nodes associated with a pattern of curves. The prescribed value of the displacement is interpreted in the active coordinate system and can be zero or nonzero. Up to 5 coordinate systems can be used to define constraints at a node. If a contradiction is found, the analysis module will give a message before stopping. For NSTAR, a nonzero value must be associated with a time curve and the force-control method must be used. FFE Static uses one coordinate system, the one used for the last constraint, to define constraints at a node.

(Refer to the DPT command for syntax and details.)

DSF

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > Define by Surfaces**

The DSF command specifies a displacement boundary condition value for all nodes associated with a pattern of surfaces.

(Refer to the DPT command for syntax and details.)

DCT

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > Define by Contours**

The DCT command specifies a displacement boundary condition value for all nodes associated with a pattern of contours.

(Refer to the DPT command for syntax and details.)

DRG

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > Define by Regions

The DRG command specifies a displacement boundary condition value for all nodes associated with a pattern of regions.

(Refer to the DPT command for syntax and details.)

DNDEL

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > Delete by Nodes

The DNDEL command deletes previously defined displacement boundary conditions in the specified direction, at all nodes in the specified pattern, regardless of the coordinate system used.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Displacement label
Label of the degree of freedom. (refer to the DND command)

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Displacement labels
Additional labels to be deleted. (degrees of freedom)

Example: `DNDEL, 1, UX, 3, 1, RZ,`

This command deletes previously specified X-translational and Z-rotational boundary conditions for nodes 1,2 and 3.

DPDEL

Geo Panel: LoadsBC > STRUCTURAL > DISPLACEMENT > Delete by Points

The DPDEL command deletes previously defined displacement boundary conditions at all the nodes associated with any of the keypoints in the specified pattern regardless of the coordinate system used to define the constraint.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.
(default is 1)

Displacement label
Label of the degree of freedom. (refer to the DND command)

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Displacement labels
Additional labels to be deleted. (degrees of freedom)

Example: `DPDEL, 1, ALL, 3, 1`
This command deletes all prescribed displacement boundary conditions at all nodes associated with keypoints 1, 2 or 3.

DCDEL

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > Delete by Curves**

The DCDEL command deletes previously defined displacement boundary conditions at all the nodes associated with any of the curves in the specified pattern regardless of the coordinate system used to define the constraint.

(Refer to the DPDEL command for syntax and details.)

DSDEL

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > Delete by Surfaces**

The DSDEL command deletes previously defined displacement boundary conditions at all the nodes associated with any of the surfaces in the specified pattern regardless of the coordinate system used to define the constraint.

(Refer to the DPDEL command for syntax and details.)

DCTDEL

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > Delete by Contours**

The DCTDEL command deletes previously defined displacement boundary conditions at all the nodes associated with any of the contours in the specified pattern regardless of the coordinate system used to define the constraint.




(Refer to the DPDEL command for syntax and details.)

DRDEL

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > Delete by Regions**

The DRDEL command deletes previously defined displacement boundary conditions at all the nodes associated with any of the regions in the specified pattern regardless of the coordinate system used to define the constraint.

(Refer to the DPDEL command for syntax and details.)

DPLOT

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > Plot**

The DPLOT command plots symbols to identify the degrees of freedom for which a prescribed displacement value has been specified. Symbols are plotted for a pattern of nodes and specified degrees of freedom.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Displacement label
Label of the degree of freedom. (refer to the DND command)

Ending node
Ending node in the pattern.
(default is the highest node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Additional Displacement labels
Additional displacement labels to be plotted.

Note

An arrow in a given direction indicates a prescribed translation in that direction. Two arrows indicate a prescribed rotation and three arrows indicate both translation and rotation.

Example: `DPLOT, , UX, 6, ,`

The above command plots the prescribed translation symbol in the X-direction at nodes 1 through 6.

DLIST

Geo Panel: **LoadsBC > STRUCTURAL > DISPLACEMENT > List**

The DLIST command lists prescribed displacement boundary conditions for a pattern of nodes. The degree of freedom, coordinate system, and the associated time curve are listed. Up to 5 coordinate systems are listed for each node.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is the highest node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `DLIST, ,10 , 2,`

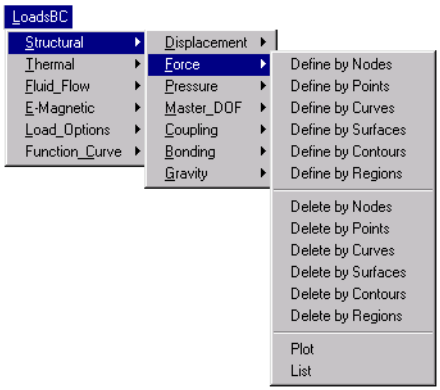
This command lists all prescribed displacement boundary conditions at nodes 1, 3, 5, 7 and 9.




▼FORCE Menu

Geo Panel: **LoadsBC > STRUCTURAL > FORCE**

Figure 7-4 Force Menu



FND

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Define by Nodes**

The FND command applies forces (or moments) at a pattern of nodes. The specified force (or moment) value is interpreted in the active coordinate system. For example if a cylindrical coordinate system is active and a force is applied in the X Force direction, the program interprets that as a force in the radial direction of the cylindrical system. An applied force or moment at a node will only be effective if the corresponding DOF is active at that node. It is meaningless, for example, to apply a moment at a node connected to SOLID elements only, or apply an Z force to a node connected to PLANE2D elements only.

Entry & Option Description

Beginning node

Beginning node in the pattern.
(default is 1)

Force label

Direction of force or moment.
= FX force in the x-direction
= FY force in the y-direction
= FZ force in the z-direction
= MX moment about the x-axis
= MY moment about the y-axis

= MZ moment about the z-axis
= AL for all forces and moments
= AF force in all directions
= AM moment about all axes
(default is FX)

Value
Force or moment value.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Phase angle
Phase angle in degrees. (prompted only if the PD_ATYPE command was issued and harmonic analysis was specified)

Example: `FND, 1, X Force, 2.5, 3, 1,`
This command applies a force of magnitude 2.5 at nodes 1, 2, and 3 in the x-direction of the active coordinate system.

FPT

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Define by Points**

The FPT command applies forces (or moments) at nodes associated with a pattern of keypoints. The specified force (or moment) value is interpreted in the active coordinate system. For example, if a cylindrical coordinate system is active and a force is applied in the X Force direction, the program interprets that as a force in the radial direction of the cylindrical system. An applied force or moment at a node will only be effective if the corresponding DOF is active at that node. It is meaningless, for example, to apply a moment at a node connected to SOLID elements only, or apply a Z force to a node connected to PLANE2D elements only.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Force label
Direction of force or moment.
= FX force in the x-direction
= FY force in the y-direction
= FZ force in the z-direction
= MX moment about the x-axis
= MY moment about the y-axis
= MZ moment about the z-axis

= AL for all forces and moments
= AF force in all directions
= AM moment about all axes
(default is FX)

Value
Force or moment value.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Phase angle
Phase angle in degrees. (prompted only if the PD_ATYPE command was issued and harmonic analysis was specified)

Example: `FPT, 1, Y Force, 10.0 ,3, 1,`
This command applies a force of 10.0 units in the y-direction at all nodes associated with keypoints 1, 2, and 3.

FCR

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Define by Curves**

The FCR command applies forces (or moments) at nodes associated with a pattern of curves. The specified force (or moment) value is interpreted in the active coordinate system. For example, if a cylindrical coordinate system is active and a force is applied in the X Force direction, the program interprets that as a force in the radial direction of the cylindrical system. An applied force or moment at a node will only be effective if the corresponding DOF is active at that node. It is meaningless, for example, to apply a moment at a node connected to SOLID elements only, or apply a Z force to a node connected to PLANE2D elements only.

(Refer to the FPT command for syntax and details.)

FSF

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Define by Surfaces**

The FSF command applies forces (or moments) at nodes associated with a pattern of surfaces. The specified force (or moment) value is interpreted in the active coordinate system. For example, if a cylindrical coordinate system is active and a force is applied in the X Force direction, the program interprets that as a force in the radial direction of the cylindrical system. An applied force or moment at a node

will only be effective if the corresponding DOF is active at that node. It is meaningless for example to apply a moment at a node connected to SOLID elements only, or apply a Z force to a node connected to PLANE2D elements only.

(Refer to the FPT command for syntax and details.)

FCT

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Define by Contours**

The FCT command applies forces (or moments) at nodes associated with a pattern of contours. The specified force (or moment) value is interpreted in the active coordinate system. For example, if a cylindrical coordinate system is active and a force is applied in the X Force direction, the program interprets that as a force in the radial direction of the cylindrical system. An applied force or moment at a node will only be effective if the corresponding DOF is active at that node, it is meaningless for example to apply a moment at a node connected to SOLID elements only, or apply a Z force to a node connected to PLANE2D elements only.

(Refer to the FPT command for syntax and details.)

FRG

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Define by Regions**

The FRG command applies forces (or moments) at nodes associated with a pattern of regions. The specified force (or moment) value is interpreted in the active coordinate system. For example, if a cylindrical coordinate system is active and a force is applied in the X Force direction, the program interprets that as a force in the radial direction of the cylindrical system. An applied force or moment at a node will only be effective if the corresponding DOF is active at that node. It is meaningless for example to apply a moment at a node connected to SOLID elements only, or apply a Z force to a node connected to PLANE2D elements only.

(Refer to the FPT command for syntax and details.)

FNDEL

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Delete by Nodes**

The FNDEL command deletes forces and/or moments at a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Force label
Direction of force or moment.
= FX force in the x-direction
= FY force in the y-direction
= FZ force in the z-direction
= MX moment about the x-axis
= MY moment about the y-axis
= MZ moment about the z-axis
= AL for all forces and moments
= AF force in all directions
= AM moment about all axes
(default is FX)

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `FNDEL, 1, AL, 3, 1,`
This command deletes all forces applied at nodes 1, 2 and 3.

FPDEL

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Delete by Points**

The FPDEL command deletes forces and/or moments defined at nodes associated with a pattern of keypoints.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Force label
Direction of force or moment.
= FX force in the x-direction
= FY force in the y-direction
= FZ force in the z-direction
= MX moment about the x-axis
= MY moment about the y-axis
= MZ moment about the z-axis
= AL for all forces and moments
= AF force in all directions
= AM moment about all axes
(default is FX)

Ending keypoint
Ending keypoint in the pattern.

(default is beginning keypoint)

Increment

Increment between keypoints in the pattern.
(default is 1)

Example: `FPDEL, 1, Y Force, 3, 1,`

This command deletes all Y-forces applied at nodes associated with keypoints 1, 2 and 3. Moments are not deleted.

FCDEL

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Delete by Curves**

The FCDEL command deletes forces and/or moments defined at nodes associated with a pattern of curves.

(Refer to the FPDEL command for syntax and details.)

FSDEL

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Delete by Surfaces**

The FSDEL command deletes forces and/or moments defined at nodes associated with a pattern of surfaces.

(Refer to the FPDEL command for syntax and details.)

FCTDEL

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Delete by Contours**

The FCTDEL command deletes forces and/or moments defined at nodes associated with a pattern of contours.

(Refer to the FPDEL command for syntax and details.)

FRDEL

Geo Panel: **LoadsBC > STRUCTURAL > FORCE > Delete by Regions**

The FRDEL command deletes forces and/or moments defined at nodes associated with a pattern of regions.

(Refer to the FPDEL command for syntax and details.)

FPLOT

Geo Panel: LoadsBC > STRUCTURAL > FORCE > Plot

The FPLOT command plots symbols to identify the degrees of freedom at which forces or moments are applied. Symbols are plotted for a specified pattern of nodes and specified degrees of freedom.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Force label
Direction of force or moment.
= FX force in the x-direction
= FY force in the y-direction
= FZ force in the z-direction
= MX moment about the x-axis
= MY moment about the y-axis
= MZ moment about the z-axis
= AL for all forces and moments
(default is AL)

Ending node
Ending node in the pattern.
(default is the highest node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `FPLOT, , X Force, 6, ,`

The above command plots the prescribed force symbol in the X-direction at nodes 1 through 6.

FLIST

Geo Panel: LoadsBC > STRUCTURAL > FORCE > List

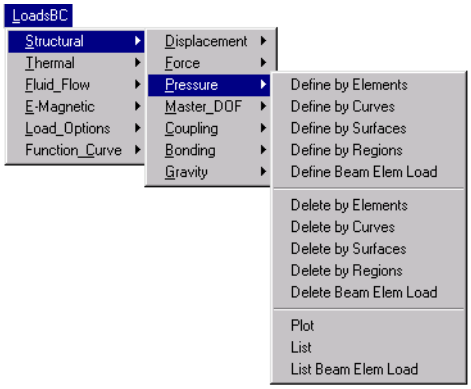
The FLIST command lists all forces and moments applied at a pattern of nodes.

(Refer to the FPLOT command for syntax.)

▼PRESSURE Menu

Geo Panel: **LoadsBC > STRUCTURAL > PRESSURE**

Figure 7-5 Pressure Menu



PEL

Geo Panel: **LoadsBC > STRUCTURAL > PRESSURE > Define by Elements**

The PEL command applies pressure (force per unit area) to the specified face number of a pattern of elements. The pressure direction can be in the x, y, or z direction of the active coordinate system, or normal to the face of the element. A normal positive pressure value on a face corresponds to inward pressure. Refer to Chapter 4 of COSMOS/M User Guide for face numbering of different elements. The active coordinate system can be Cartesian, cylindrical, or spherical.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Pressure magnitude
Magnitude of applied pressure.

Face number
Face number for the elements in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between the elements in the pattern.
(default is 1)

Pressure direction
Pressure direction.

= 1: X	pressure in the x-direction
= 2: Y	pressure in the y-direction
= 3: Z	pressure in the z-direction
= 4: Normal direction	pressure normal to the face *(default is normal direction)*

Example: `PEL,1,1.,4,3,,`

This command defines 1 unit of pressure on the fourth face of elements 1, 2, and 3 in the inward direction.

PCR

Geo Panel: **LoadsBC > STRUCTURAL > PRESSURE > Define by Curves**

The PCR command applies pressure (force per unit area) to the faces of elements associated with a pattern of curves. The pressure direction can be in the x, y, or z direction of the active coordinate system (must be Cartesian), or normal to the faces of elements. Positive pressure points downward when node 1 of the element is to the left and node 2 to the right. The active coordinate system can be Cartesian, cylindrical, or spherical.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Pressure magnitude
Magnitude of applied pressure at the start of the curve.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Pressure at end of direction 1
Magnitude of applied pressure at the end of the curve.
(default is pressure magnitude)

Pressure direction
Pressure direction. Active coordinate system must be Cartesian.

= 1: X	pressure in the x-direction
= 2: Y	pressure in the y-direction
= 3: Z	pressure in the z-direction
= 4: Normal direction	normal pressure *(default is normal direction)*

Example: `PCR, 1, -1.0, 3, , 5.0,`

This command defines a linearly varying normal pressure for curves 1 through 3. The pressure is -1.0 at the start and 5.0 at the end of each curve, element pressures are calculated accordingly.

PSF

Geo Panel: **LoadsBC > STRUCTURAL > PRESSURE > Define by Surfaces**

The PSF command applies pressure (force per unit area) to the faces of elements associated with a pattern of surfaces. The pressure can be specified in the x, y, or z directions of the active coordinate system (must be Cartesian), or normal to the faces of elements. An inward normal pressure is positive. For shell elements, the pressure is applied to face number 5. The active coordinate system can be Cartesian, cylindrical, or spherical.

Entry & Option Description

Beginning surface
Beginning surface in the pattern.

Pressure magnitude
Magnitude of applied pressure per unit area at the origin of the parametric coordinates of the surface.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between the surfaces in the pattern.
(default is 1)

Pressure at the end of direction 1
Magnitude of applied pressure per unit area at the end of the first parametric curve.
(default is pressure magnitude)

Pressure at the end of direction 2
Magnitude of applied pressure per unit area at the end of the second parametric curve.
(default is pressure magnitude)

Pressure direction
Pressure direction. Active coordinate system must be Cartesian.

= 1: X	pressure in the x-direction
= 2: Y	pressure in the y-direction
= 3: Z	pressure in the z-direction
= 4: Normal direction	normal pressure
	(default is normal direction)

Example: `PSF, 1, 1.0, 3, 1, 2.0, 2.0,,`

This command defines pressure in the inward normal direction for surfaces 1, 2, and 3. The pressure varies linearly such that its magnitude is 1.0 at the origins of the parametric curves of the surfaces and 2.0 at both ends of the parametric curves of each surface. Element pressures are interpolated linearly.

PRG

Geo Panel: **LoadsBC > STRUCTURAL > PRESSURE > Define by Regions**

The PRG command applies pressure (force per unit area) to the faces of elements associated with a pattern of regions. The pressure can be specified in the x, y, or z direction of the active coordinate system (must be Cartesian), or normal to the faces of elements. An inward normal pressure is positive. For shell elements, the pressure is applied to face number 5. The active coordinate system can be Cartesian, cylindrical, or spherical.

Entry & Option Description

Beginning region
Beginning region in the pattern.

Pressure magnitude
Pressure value.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Unused option
Unused option.

Pressure direction
Pressure direction. Active coordinate system must be Cartesian.

= 1: X	pressure in the x-direction
= 2: Y	pressure in the y-direction
= 3: Z	pressure in the z-direction
= 4: Normal direction	normal pressure *(default is normal direction)*

Example: `PRG, 1, -1.5, 3, 1,, 4,`

This command specifies 1.5 units of pressure in outward normal direction for elements associated with regions 1, 2 and 3.

PBEL

Geo Panel: **LoadsBC > STRUCTURAL > PRESSURE > Define Beam Elem Load**

The PBEL command specifies beam element loading. The command can be used to specify concentrated forces as well as linearly varying distributed loads on BEAM2D and BEAM3D elements. The loading can be anywhere along the element, and can be specified in local or global element coordinate systems.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Force label
Loading label.
= FX force in the x-direction
= FY force in the y-direction
= FZ force in the z-direction
= MX moment about the x-axis
= MY moment about the y-axis
= MZ moment about the z-axis
(default is FY)

Value
Magnitude of loading at first foot distance.
(default is 0.0)

Foot distance
Location 1, specified in element parametric coordinate.
(default is 0.50 i.e. at the center)

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Value
Magnitude of loading at second foot distance.
(default is value)

Foot distance
Location 2, specified in element parametric coordinate.
(default is first foot distance)

Notes

1. Local element coordinates for BEAM3D elements are defined as follows:
   The x-axis runs from node 1 to node 2 of the element. The y-axis lies in the plane defined by the 3 nodes and runs from node 1 towards node 3 normal

to the x-direction. The z-axis is defined by the right-hand rule.

2. BEAM2D elements are always in the global x-y plane, the local element coordinates are defined as follows:

   The x-axis runs from node 1 to node 2 of the element. The y-axis is in the x-y plane such that a right hand system is observed with the z-axis of the global coordinate system.

3. If the user wishes to specify loading in the global Cartesian coordinates, this coordinate system must be activated first using the following command:

   ACTSET, ECS, 0,

   Otherwise, local element coordinates will be used. The element local coordinate system can be activated by issuing the following command:

   ACTSET, ECS, -1,

4. Concentrated forces are specified by equating second foot distance to first foot distance (default option).

5. The program interprets the units according to whether first foot distance and second foot distance are equal or not. If first foot distance is equal to second foot distance, the program interprets the given values as units of force or moment per unit length. If first foot distance is not equal to second foot distance, GEOSTAR interprets the given values as forces or moments.

Example 1:
```
ACTSET, ECS, 0,
PBEL, 1, Y Force, 100.0, 0.10, 5, 2, 200., 0.80,
```
The global element coordinate system is activated and a pressure is applied in the y-direction (of the global system) to elements 1, 3, and 5. The pressure varies linearly from 100 units at the location specified by 10% of the element length (starting from node 1 towards node 2 of the element), to 200 units at the 80% location measured in the same way.

Example 2:
```
ACTSET, ECS, -1,
PBEL, 1, Z axis moment, 50.0, 0.5, 1, 1, 50.0, 0.5,
```
The local element coordinate system is activated and a concentrated moment about the z-direction (of the local coordinate system) is applied at the midpoint of the beam element.

PEDEL

Geo Panel: **LoadsBC > STRUCTURAL > PRESSURE > Delete by Elements**

The PEDEL command deletes previously defined pressures on a specified face for all the elements in the specified pattern.

Entry & Option Description

Beginning element

Beginning element in the pattern.

Face number

Face number of the element on which the pressure is to be removed.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements of the pattern.
(default is 1)

Example: `PEDEL, 1, 2, 3, 1`

This command deletes the pressure applied on face two of elements 1, 2 and 3.

PCDEL

Geo Panel: **LoadsBC > STRUCTURAL > PRESSURE > Delete by Curves**

The PCDEL command deletes previously defined pressures on all the curves in the specified pattern.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves of the pattern.
(default is 1)

Example: `PCDEL,1,3,1`

This command deletes pressures on curves 1,2 and 3.

PSDEL

Geo Panel: **LoadsBC > STRUCTURAL > PRESSURE > Delete by Surfaces**

The PSDEL command deletes previously defined pressures on all the surfaces in the specified pattern.

(Refer to the PCDEL command for syntax and details.)




PRDEL

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > Delete by Regions

The PRDEL command deletes previously defined pressures for all the elements associated with any of the regions in the specified pattern.

(Refer to the PCDEL command for syntax and details.)

PBEDEL

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > Delete Beam Elem Load

The PBEDEL command deletes the specified components of beam loading previously defined by the PBEL command, for a specified pattern of beam elements.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Force label
Loading label.
= FX force in the x-direction
= FY force in the y-direction
= FZ force in the z-direction
= MX moment about the x-axis
= MY moment about the y-axis
= MZ moment about the z-axis
= AF force in all directions
= AM moment about all axes
= AL all directions of force and moment
(default is FY)

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `PBEDEL, 2, ALL, 2, 1,`

This command deletes all beam loading components for element number 2 from the database.

PPLOT

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > Plot

The PPLOT command plots the specified pressures for the elements specified in a pattern.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is the highest element number defined)

Increment
Increment between the elements in the pattern.
(default is 1)

Example: `PPLOT,,6,,`
The above command plots specified pressures for element numbers 1 through 6.

PLIST

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > List

The PLIST command lists currently defined element pressures for the specified pattern.

(Refer to the PPLOT command for syntax.)

PBELIST

Geo Panel: LoadsBC > STRUCTURAL > PRESSURE > List Beam Elem Load

The PBELIST command lists beam loadings previously specified by the PBEL command, for a specified pattern of beam elements. The command lists the loadings by showing the values of the parameters as specified by the PBEL command as well as the coordinate system used to specify the loadings.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.

(default is highest element number defined)

Increment

Increment between elements in the pattern.
(default is 1)

Example: `PBELIST, 1, 10, 1,`

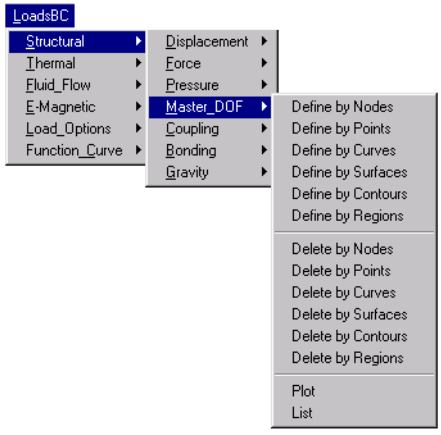
This command results in listing beam loadings for elements 1 through 10.

▼MASTER_DOF Menu

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF**

The Master DOF commands are used specially for reducing the size of dynamic problems and limiting the degrees of freedom to those designated as "master" or dynamic degrees of freedom. This task is accomplished with the matrix condensation technique known as the Guyan Reduction Algorithm. The Master DOF can only be applied in the global Cartesian coordinate system and their number is limited to a maximum of 125. To activate the Guyan reduction, specify in the A_FREQ command to use the GS method (Guyan reduction with Subspace) or the GJ method (Guyan reduction) with Jacobi method.

Figure 7-6 Master_DOF Menu



MDOFND

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Define by Nodes**

The MDOFND command defines master degrees of freedom for use with the Guyan Reduction technique to extract natural frequencies and corresponding mode shapes at a pattern of nodes. Master degrees of freedom are specified in the global Cartesian coordinate system. The use of the Guyan Reduction technique may be activated by choosing GS or GJ for the method of extraction prompt in the A_FREQUENCY command. The GS and GJ options refer to using Guyan Reduction with the Subspace and Jacobi methods, respectively.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Master DOF label
Label of the master degree of freedom.

= UX	translation in the x-direction
= UY	translation in the y-direction
= UZ	translation in the z-direction
= RX	rotation about the x-axis
= RY	rotation about the y-axis
= RZ	rotation about the z-axis
= AL	all 6 degrees of freedom
= AU	all translations
= AR	all rotations
= AX	Y-Z anti-symmetrical BC
= AY	X-Z anti-symmetrical BC
= AZ	X-Y anti-symmetrical BC
= SX	Y-Z symmetrical BC
= SY	X-Z symmetrical BC
= SZ	X-Y symmetrical BC
	(default is UX)

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Master DOF labels
Additional labels. (master degrees of freedom)

Note

Additional master degrees of freedom are assigned to all the nodes in the pattern.

Example:
```
MDOFND, 1, UX, 3, 1, RZ,
```

This command designates the X-translation and Z-rotation degrees of freedom at nodes 1, 2, and 3 as master degrees of freedom.

MDOFPT

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Define by Points**

The MDOFPT command defines master degree of freedom at nodes associated with a pattern of keypoints. The specified Master DOF are used by the Guyan Reduction technique to extract natural frequencies and corresponding mode shapes. Master degrees of freedom are specified in the global Cartesian coordinate system. Use of the Guyan Reduction technique may be activated by choosing GS or GJ for the method of extraction prompt in the A_FREQUENCY command. The GS and GJ options refer to using Guyan Reduction with the Subspace and Jacobi methods, respectively.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Master DOF label
Label of the master degree of freedom.

= UX	translation in the x-direction
= UY	translation in the y-direction
= UZ	translation in the z-direction
= RX	rotation about the x-axis
= RY	rotation about the y-axis
= RZ	rotation about the z-axis
= AL	all 6 degrees of freedom
= AU	all translations
= AR	all rotations
= AX	Y-Z anti-symmetrical BC
= AY	X-Z anti-symmetrical BC
= AZ	X-Y anti-symmetrical BC
= SX	Y-Z symmetrical BC
= SY	X-Z symmetrical BC
= SZ	X-Y symmetrical BC
	(default is UX)

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Master DOF labels
Additional labels. (master degrees of freedom)

Note

Additional master degrees of freedom are assigned to all nodes associated with the keypoints in the pattern.

Example: `MDOFPT, 1, UX, 3, 1, RZ,`

This command designates the X-translation and Z-rotation degrees of freedom at nodes associated with keypoints 1, 2, and 3 as master degrees of freedom.

MDOFCR

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Define by Curves**

The MDOFCR command defines master degree of freedom at nodes associated with a pattern of curves. The specified Master DOF are used by the Guyan Reduction technique to extract natural frequencies and corresponding mode shapes. Master degrees of freedom are specified in the global Cartesian coordinate system. Use of the Guyan Reduction technique may be activated by choosing GS or GJ for the method of extraction prompt in the A_FREQUENCY command. The GS and GJ options refer to using Guyan Reduction with the Subspace and Jacobi methods, respectively.

(Refer to the MDOFPT command for syntax and details.)

MDOFSF

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Define by Surfaces**

The MDOFSF command defines master degree of freedom at nodes associated with a pattern of surfaces. The specified Master DOF are used by the Guyan Reduction technique to extract natural frequencies and corresponding mode shapes. Master degrees of freedom are specified in the global Cartesian coordinate system. Use of the Guyan Reduction technique may be activated by choosing GS or GJ for the method of extraction prompt in the A_FREQUENCY command. The GS and GJ options refer to using Guyan Reduction with the Subspace and Jacobi methods, respectively.

(Refer to the MDOFPT command for syntax and details.)

MDOFCT

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Define by Contours**

The MDOFCT command defines master degree of freedom at nodes associated with a pattern of contours. The specified Master DOF are used by the Guyan Reduction technique to extract natural frequencies and corresponding mode shapes.

Master degrees of freedom are specified in the global Cartesian coordinate system. Use of the Guyan Reduction technique may be activated by choosing GS or GJ for the method of extraction prompt in the A_FREQUENCY command. The GS and GJ options refer to using Guyan Reduction with the Subspace and Jacobi methods, respectively.

(Refer to the MDOFPT command for syntax and details.)

MDOFRG

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Define by Regions**

The MDOFRG command defines master degree of freedom at nodes associated with a pattern of regions. The specified Master DOF are used by the Guyan Reduction technique to extract natural frequencies and corresponding mode shapes. Master degrees of freedom are specified in the global Cartesian coordinate system. Use of the Guyan Reduction technique may be activated by choosing GS or GJ for the method of extraction prompt in the A_FREQUENCY command. The GS and GJ options refer to using Guyan Reduction with the Subspace and Jacobi methods, respectively.

(Refer to the MDOFPT command for syntax and details.)

MDOFNDEL

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Delete by Nodes**

The MDOFNDEL command deletes previously defined master degrees of freedom at a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Master DOF label
Label of the master degree of freedom. (refer to the MDOFND command)

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Master DOF labels
Additional labels. (master degrees of freedom)

Example: `MDOFNDEL, 1, UX, 3, 1, RZ,`

This command deletes Master DOF defined at nodes 1, 2, and 3.

MDOFPDEL

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Delete by Points**

The MDOFPDEL command deletes master degrees of freedom defined at nodes associated with a pattern of keypoints.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Master DOF label
Label of the master degree of freedom. (refer to the MDOFND command)

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Master DOF labels
Additional labels for master degrees of freedom to be deleted.

Example:
```
MDOFPDEL, 1, UX, 3, 1, RZ,
```
This command deletes master degrees of freedom defined at sides associated with keypoints 1, 2, and 3.

MDOFCDEL

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Delete by Curves**

The MDOFCDEL command deletes master degrees of freedom defined at nodes associated with a pattern of curves.

(Refer to the MDOFPDEL command for syntax and details.)

MDOFSDEL

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > Delete by Surfaces**

The MDOFSDEL command deletes master degrees of freedom defined at nodes associated with a pattern of surfaces.

(Refer to the MDOFPDEL command for syntax and details.)

MDOFCTDEL

Geo Panel: LoadsBC > STRUCTURAL > MASTER_DOF > Delete by Contours

The MDOFCTDEL command deletes master degrees of freedom defined at nodes associated with a pattern of contours.

(Refer to the MDOFPDEL command for syntax and details.)

MDOFRDEL

Geo Panel: LoadsBC > STRUCTURAL > MASTER_DOF > Delete by Regions

The MDOFRDEL command deletes master degrees of freedom defined at nodes associated with a pattern of regions.

(Refer to the MDOFPDEL command for syntax and details.)

MDOFPLOT

Geo Panel: LoadsBC > STRUCTURAL > MASTER_DOF > Plot

The MDOFPLOT command plots symbols at Master DOF in the specified pattern of nodes. One arrow indicates a translational Master DOF and two arrows indicate a rotational DOF.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Master DOF label
Label of the master degree of freedom.

= UX	translation in the x-direction
= UY	translation in the y-direction
= UZ	translation in the z-direction
= RX	rotation about the x-axis
= RY	rotation about the y-axis
= RZ	rotation about the z-axis
= AL	all 6 degrees of freedom
= AU	all translations
= AR	all rotations
= AX	Y-Z anti-symmetrical BC
= AY	X-Z anti-symmetrical BC
= AZ	X-Y anti-symmetrical BC
= SX	Y-Z symmetrical BC
= SY	X-Z symmetrical BC
= SZ	X-Y symmetrical BC

(default is AL)

Ending node
Ending node in the pattern.
(default is the highest node label defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Master DOF labels
Additional labels for master degrees of freedom to be deleted.

Example: `MDOFPLOT;`

This command plots all defined master degrees of freedom for the active set of nodes.

MDOFLIST

Geo Panel: **LoadsBC > STRUCTURAL > MASTER_DOF > List**

The MDOFLIST command lists master degrees of freedom defined at nodes in the specified pattern.

(Refer to the MDOFPLOT command for syntax and details.)

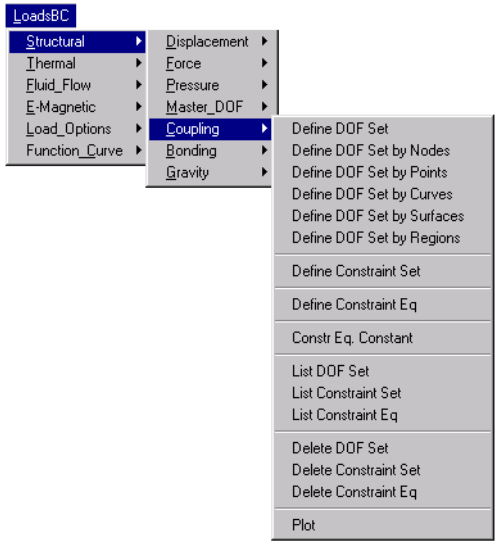



▼COUPLING Menu

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING**

This menu contains commands to define, list and delete coupling specifications including coupling DOF's, coupling equations and coupling constraints.

Figure 7-7 Coupling Menu



CPDOF

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Define DOF Set**

The CPDOF command defines a coupling set to impose equal displacement (in the specified direction of the active coordinate system) or temperature. The set is defined by specifying a master node, displacement degree of freedom or temperature, and up to 9 slave nodes. The displacement component or temperature of the slave nodes will be equal to that of the master node after running the analysis. Any contradicting conditions will be ignored. The first node in the set is the master node. A maximum total of 10,000 coupling sets can be defined.

✍ FFE Static and FFE Thermal do not support this command.

Entry & Option Description

Cp set number
Coupling set number. Maximum is 10,000.

(default is max. coupled set defined + 1)

Coupling label
Displacement component or temperature. The valid labels are UX, UY, UZ, RX, RY, RZ, and TEMP.
(default is UX)

Number of coupled nodes
Number of nodes to be coupled. (must be between 2 and 10, inclusive)

Node (i)
ith node to be coupled. (i = 1,2,...,10). The first node is the master node and the rest of the nodes are slave nodes.

Notes

1. A different label for a defined set redefines the previous dof associated with it.
2. Nodes can be picked by the mouse.
3. The CPDOF and CPCNS commands use the same coupling set in the database and will overwrite each other.
4. If a slave node appears in several different coupling sets, the slave node will be coupled with the first defined master node (i.e., the master node in the coupling set with the lowest set label).
5. The same constraints (boundary conditions) imposed on the coupled degrees of freedom of the master node will also be imposed on the slave nodes in the same coupling set. In nonlinear analysis, slave nodes will follow the master node even when different prescribed displacement and/or time curve have been specified.
6. In a coupling set, if the master node is free at the coupled degree of freedom, the constraints of the slave nodes at the same degree of freedom will be released.

Example: `CPDOF, 1, UX, 4, 2, 5, 8, 11`

This command defines set 1 and forces equal displacements in the x-direction of the active coordinate system, for nodes 2, 5, 8, and 11. For linear analysis, if the active coordinate system is cylindrical for example, then radial equal radial displacement is specified for these nodes.

CPDOFND

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Define DOF Se by Nodes**

The CPDOFND command defines a number of coupling sets to impose equal displacement (in the specified direction of the active coordinate system) or temperature. You specify the DOF, master node, and a pattern of nodes. The actual number of coupling sets created by this command depends on the number of nodes in the specified pattern. The displacement component or temperature of the slave nodes will be equal to that of the master node after running the analysis. Any contradict-

ing conditions will be ignored. A maximum total of 10,000 coupling sets can be defined.

✍ FFE Static and FFE Thermal do not support this command.

Entry & Option Description

Starting Cp set number
The first coupling set label to be created. The maximum is 10,000.
(default is max. coupled set defined + 1)

Coupling label
Displacement component or temperature. The valid labels are UX, UY, UZ, RX, RY, RZ, and TEMP.
(default is UX)

Master Node
Label of the master node.

Beginning Node
Label of the beginning node in the pattern.

Ending Node
Label of the ending node in the pattern.

Increment
Increment between nodes in the pattern.
(default is 1)

Notes:

1) If a slave node is the same as the master node, it will be ignored.
2) If a node selection list is active, it will be used.
3) Use the CPDOFLIST command to list existing coupled DOF sets.
4. The same constraints (boundary conditions) imposed on the coupled degrees of freedom of the master node will also be imposed on the slave nodes in the same coupling set. In nonlinear analysis, slave nodes will follow the master node even when different prescribed displacement and/or time curve have been specified.
5. In a coupling set, if the master node is free at the coupled degree of freedom, the constraints of the slave nodes at the same degree of freedom will be released.

Example: **CPDOFND, 10, UY, 3, 101, 118, 1**

This command forces the y-displacement (in the active coordinate system) of nodes 101 through 118 to be equal to that of node 3 (master node). The first coupling set created by this command is labeled 10. The number of coupling sets to be created by this command depends on the actual number of slave nodes. The command starts defining the next coupling set when the previous set is filled with 9 slave nodes.




CPDOFPT

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Define DOF Set by Points**

The CPDOFPT command defines a number of coupling sets to impose equal displacement (in the specified direction of the active coordinate system) or temperature. You specify the DOF, master node, and a pattern of keypoints. The actual number of coupling sets created by this command depends on the number of nodes associated with the specified pattern of keypoints. The displacement component or temperature of the slave nodes will be equal to that of the master node after running the analysis. Any contradicting conditions will be ignored. A maximum total of 10,000 coupling sets can be defined.

✍ FFE Static and FFE Thermal do not support this command.

Entry & Option Description

Starting Cp set number
The first coupling set label to be created. The maximum is 10,000.
(default is max. coupled set defined + 1)

Coupling label
Displacement component or temperature. The valid labels are UX, UY, UZ, RX, RY, RZ, and TEMP.
(default is UX)

Master Node
Label of the master node.

Beginning Keypoint
Label of the beginning keypoint in the pattern.

Ending Keypoint
Label of the ending keypoint in the pattern.

Increment
Increment between keypoints in the pattern.
(default is 1)

Notes:

1) If a slave node is the same as the master node, it will be ignored.
2) If a keypoint selection list is active, it will be used.
3) Use the CPDOFLIST command to list existing coupled DOF sets.

Example: **CPDOFPT, 10, UX, 5, 100, 120, 1**

This command forces the x-displacement (in the active coordinate system) of nodes 100 through 120 to be equal to that of node 5 (master node). The first coupling set created by this command is labeled 10. The number of coupling sets to be created by this command depends on the actual number of slave nodes. The command starts defining the next coupling set when the previous set is filled with 9 slave nodes.

CPDOFCR

Geo Panel: LoadsBC > STRUCTURAL > COUPLING > Define DOF Set by Curves

The CPDOFCR command defines a number of coupling sets to impose equal displacement (in the specified direction of the active coordinate system) or temperature. You specify the DOF, master node, and a pattern of curves. The actual number of coupling sets created by this command depends on the number of nodes associated with the specified pattern of curves. The displacement component or temperature of the slave nodes will be equal to that of the master node after running the analysis. Any contradicting conditions will be ignored. A maximum total of 10,000 coupling sets can be defined.

✍ FFE Static and FFE Thermal do not support this command.

Refer to the CPDOFPT command (LoadsBC > STRUCTURAL > COUPLING > Define DOF Set by Points) for more information.

CPDOFSF

Geo Panel: LoadsBC > STRUCTURAL > COUPLING > Define DOF Set by Surfaces

The CPDOFSF command defines a number of coupling sets to impose equal displacement (in the specified direction of the active coordinate system) or temperature. You specify the DOF, master node, and a pattern of surfaces. The actual number of coupling sets created by this command depends on the number of nodes associated with the specified pattern of surfaces. The displacement component or temperature of the slave nodes will be equal to that of the master node after running the analysis. Any contradicting conditions will be ignored. A maximum total of 10,000 coupling sets can be defined.

✍ FFE Static and FFE Thermal do not support this command.

Refer to the CPDOFPT command (LoadsBC > STRUCTURAL > COUPLING > Define DOF Set by Points) for more information.

CPDOFRG

Geo Panel: LoadsBC > STRUCTURAL > COUPLING > Define DOF Set by Regions

The CPDOFRG command defines a number of coupling sets to impose equal displacement (in the specified direction of the active coordinate system) or temperature. You specify the DOF, master node, and a pattern of regions. The actual number of coupling sets created by this command depends on the number of nodes associated with the specified pattern of regions. The displacement component or temperature of the slave nodes will be equal to that of the master node after running

the analysis. Any contradicting conditions will be ignored. A maximum total of 10,000 coupling sets can be defined.

✍ FFE Static and FFE Thermal do not support this command.

Refer to the CPDOFPT command (LoadsBC > STRUCTURAL > COUPLING > Define DOF Set by Points) for more information.

CPCNS

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Define Constraint Set**

The CPCNS command defines a point-to-point, point-to-curve or point-to-surface constraint and assigns a set number to it. The command works with STAR, DSTAR, ASTAR, and NSTAR.

Entry & Option Description

Cp set number

Constraint set number.
(default is max. constraint set defined + 1)

Constraint type

Constraint type.
= 1: PT-PT point-to-point constraint
= 2: PT-CR point-to-curve constraint
= 3: PT:-SF point-to-surface constraint
(default is 1: PT-PT)

Coupling type

Coupling type. The interpretation depends on the constraint type value as shown in the table below.

	Constraint		
	type = 1	type = 2	type = 3
Coupling type = 1	TIE	ATTACH	ATTACH
Coupling type = 2	MATCH	SLIDE	SLIDE
Coupling type = 3	RIGID	ROTATE	---

Number of coupled nodes

Number of nodes. Not prompted for constraint type = 1.
For constraint type 1 number of coupled nodes is set to 2.
For constraint type 2 number of coupled nodes must be 3 or 4.
For constraint type 3 number of coupled nodes must be between 4 and 10.
(default is 2, 3 and 9 for coupling type 1, 2 and 3 respectively)

Node 1

Primary node (node to be constrained).

Node (i)

Node(s) to define a point, line, curve, or surface. (i>1)

Notes

1. The TIE constraint enforces a constant distance between two nodes.
2. The MATCH constraint enforces equal displacements for two nodes.
3. The RIGID constraint enforces the two nodes to rotate by the same amount and have a rigid body displacement.
4. The ATTACH constraint enforces rotational and translational compatibility between a point and a curve (or line), or between a point and a surface.
5. The SLIDE constraint restricts a point to slide along a curve or a surface.
6. The ROTATE constraint enforces a point to rotate about a line, or a curve, keeping translational compatibility with the axis of rotation.
7. The CPDOF and CPCNS commands use the same coupling set in the database and will overwrite each other.

Example 1: `CPCNS, 4, 1, 3, ,5, 6`

This command specifies that nodes 5 and 6 are constrained to move as a rigid bar. The constraint is assigned a set number 4. Number of nodes assumes the default value 2 since constraint type is 1.

Example 2: `CPCNS, 1, 2, 1, 4, 10, 12, 15, 18`

This command specifies that node 10 is attached to a curve, defined by nodes 12, 15 and 18. The constraint is assigned a set number 1.

Example 3: `CPCNS, 3, 3, 1, 5, 1, 2, 3, 4, 6`

This example specifies that node 1 is attached to a surface, defined by nodes 2, 3, 4, and 6. (Remember that a fifth or a ninth node must be specified for a surface).

CPEQN

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Define Constraint Eq**

The CPEQN command defines a constraint equation by specifying nodes, displacement labels and the corresponding coefficients. If the right hand side is not zero, then the CPEQNVAL command should be used to specify its value. Constraint equations are valid only for STAR, DSTAR, and NSTAR, and are considered in the active coordinate system. For NSTAR, the value in the right hand side of the equation should be associated with a time curve (refer to the CPEQNVAL command).

Entry & Option Description

Cq set number
Equation number.
(default is max. equation number defined + 1)

Starting term
Starting term. (1,2,....,10)

Number of terms
Number of terms in the equation.

Node (i)
ith node to be included in the equation.

Displacement label for node (i)
Degree of freedom corresponding to the ith node. Valid labels are UX, UY, UZ, RX, RY and RZ.
(default is UX)

Coefficient for node (i)
Coefficient corresponding to the ith node.
(default is 1)

Note
The CPEQN can be repeated to define (or redefine) up to 10 terms in a constraint equation by using an appropriate value for the starting term argument.

Example:
```
CPEQN, 1, 1, 2, 5, UX, 2.0, 11, UX, -3.0
CPEQN, 1, 3, 2, 6, UY, 1.5, 12, UZ,  1.2
```
These two commands define the constraint equation no. 1 as:
2.0 x UX(node 5) - 3.0 x UX(node 11) + 1.5 x UY(node 6) + 1.2 x UZ(node 12) = 0 in the active coordinate system.

CPEQNVAL

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Constr Eq Constant**

The CPEQNVAL command defines the constant on the right hand side of the specified constraint equation. For NSTAR, the specified constant value is associated with the active time curve. If the specified value is not associated with a time curve, or a control method other than the force-control method is used, then the right hand side of the constraint equation is assumed to be zero in all solution steps.

Entry & Option Description

Cq set number
Constraint equation number.
(default is max. equation number defined + 1)

Equation constant
Value of the right hand side.
(default is 0.0)

Note
This command is only needed if the right hand side of the corresponding constraint equation is not zero.

Example:
```
CPEQN, 1, 1, 2, 5, UX, 2.0, 11, UX, -3.0
CPEQN, 1, 3, 2, 6, UY, 1.5, 12, UZ,  1.2
```

```
ACTSET, TC, 10,
CPEQNVAL, 1, 30.
```

These four commands define the constraint equation no. 1 as:

2.0 x UX(node 5) - 3.0 x UX(node 11) + 1.5 x UY(node 6) + 1.2 x UZ(node 12) = 30

where the value 30 is associated with time curve number 10 for nonlinear analysis using the force-control method.

CPDOFLIST

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > List DOF Set**

The CPDOFLIST lists all defined coupled degrees-of-freedom in a pattern of coupling sets.

Entry & Option Description

Beginning set
Beginning set in the pattern.
(default is 1)

Ending set
Ending set in the pattern.
(default is beginning set)

Increment
Increment between sets in the pattern.
(default is 1)

Example: `CPDOFLIST, 5, 10, 2`

This command results in the listing of all coupled degrees-of-freedom in coupling sets 5, 7 and 9.

CPCNSLIST

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > List Constraint Set**

The CPCNSLIST command lists all the constraints in a pattern of constraint sets, defined by the CPCNS command. This can include point-to-point, point-to-curve and point-to-surface constraints.

(Refer to the CPDOFLIST command for syntax.)

CPEQNLIST

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > List Constraint Eq**

The CPEQNLIST command lists a pattern of constraint equations previously defined by the CPEQN command. The value of the right hand sides and associated time curves are also listed.

Entry & Option Description

Beginning equation
Beginning equation in the pattern.
(default is 1)

Ending equation
Ending equation in the pattern.
(default is highest equation number defined)

Increment
Increment between equations in the pattern.
(default is 1)

Example: `CPEQNLIST, 5, 10, 2`
This command results in the listing of constraint equations 5, 7 and 9, previously defined by the CPEQN command.

CPDOFDEL

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Delete DOF Set**

The CPDOFDEL command deletes a pattern of sets of coupled degrees-of-freedom, previously defined by the CPDOF command.

Entry & Option Description

Beginning set
Beginning set in the pattern.

Ending set
Ending set in the pattern.
(default is beginning set)

Increment
Increment between sets in the pattern.
(default is 1)

Example: `CPDOFDEL, 5, 10, 2`
This command results in the deletion of all coupled degrees-of-freedom, in coupling sets 5, 7 and 9.

CPCNSDEL

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Delete Constraint Set**

The CPCNSDEL command deletes a pattern of constraints, previously defined by the CPCNS command.

(Refer to the CPDOFDEL command for syntax.)

CPEQNDEL

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Delete Constraint Eq**

The CPEQNDEL command deletes a pattern of existing constraint equations, previously defined by the CPEQN command.

(Refer to the CPDOFDEL command for syntax.)

CPDOFPLOT

Geo Panel: **LoadsBC > STRUCTURAL > COUPLING > Plot**

The CPDOFPLOT plots all defined coupled degrees-of-freedom in a pattern of coupling sets.

Entry & Option Description

Beginning set
Beginning set in the pattern.
(default is 1)

Ending set
Ending set in the pattern.
(default is the highest set number defined)

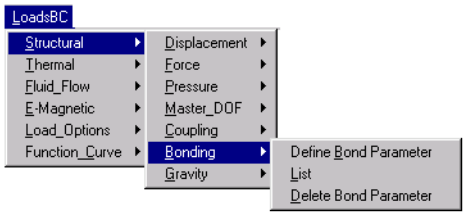
Increment
Increment between sets in the pattern.
(default is 1)

Example: `CPDOFPLOT, 5, 10, 2`
This command results in the plotting of all coupled degrees-of-freedom, in coupling sets 5, 7 and 9.

▼BONDING Menu

Geo Panel: LoadsBC > STRUCTURAL > BONDING

Figure 7-8 Bonding Menu



BONDDEF

Geo Panel: LoadsBC > STRUCTURAL > BONDING > Define Bond Parameter

The BONDDEF command bonds faces of elements associated with selected geometric entities. The user specifies a primary bond entity (curve, surface, or region) and a pattern of target entities (curves, surfaces, or regions). All the geometric entities must have been meshed before issuing this command in order to generate the bond information. Element edges/faces associated with the primary geometric entity are bonded with edges/faces associated with the pattern of secondary entities. The command is useful in connecting parts with incompatible mesh at the interface. The command should only be used if it is too hard to generate a compatible mesh. The utility is only available for STAR, HSTAR, and NSTAR. Refer to Chapter 8 of the COSMOS/M Basic System Manual for details, examples, and limitations.

Entry & Option Description

Bonding set
Bond set number.
(default is highest set number + 1)

Primary geometric entity type
Primary bond geometric entity type.
= 0: Curve curve
= 1: Surface surface
= 2: Region region
(default is curve)

Primary entity
Label of primary bond entity.

Secondary geometric entity type
Target geometric bond type.
= 0: Curve curve
= 1: Surface surface

= 2: Region region
(default is region)

Beginning entity
Beginning entity label in the pattern of bond target entities.

Ending entity
Ending entity label in the pattern of bond target entities.

Increment
Increment between entity labels in the pattern of bond target entities.

Direction flag
Flag to specify unidirectional or bidirectional bonding. Refer to Chapter 8 of the COSMOS/M Basic System Manual for details.
= 1: Uni dir unidirectional bonding
= 2: Bi dir bidirectional bonding
(default is uni dir)

Note
For the BONDDEF command to work properly, the nodes associated with the target entity must be located (within a small tolerance) on the source entity also. It is suggested that the NCRPUSH or the NSFPUSH command be used to push nodes associated with the source entity to the target entities.

BONDDEL

Geo Panel: **LoadsBC > STRUCTURAL > BONDING > Delete Bond Parameter**

The BONDDEL command deletes a pattern of bond sets previously defined by the BONDDEF command.

Entry & Option Description

Beginning set
Beginning bond set in the pattern.

Ending set
Ending bond set in the pattern.

Increment
Increment between bond sets in the pattern.
(default is 1)

Example: `BONDDEL, 1, 2, 1,`
This command deletes bond sets 1 and 2.

BONDLIST

Geo Panel: **LoadsBC > STRUCTURAL > BONDING > List**

The BONDLIST command lists a pattern of bond sets previously defined by the BONDDEF command.

Entry & Option Description

Beginning set
Beginning bond set in the pattern.
(default is 1)

Ending set
Ending bond set in the pattern.
(default is the highest bond set defined)

Increment
Increment between bond sets in the pattern.
(default is 1)

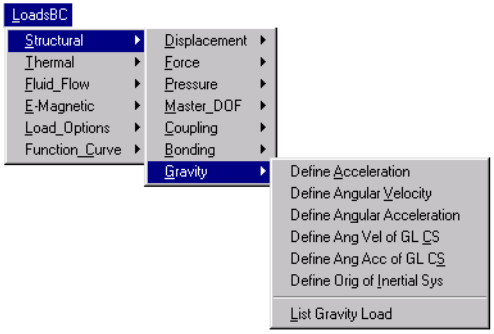
Example: `BONDLIST;`
This command lists all defined bond sets.

▼GRAVITY Menu

Geo Panel: **LoadsBC > STRUCTURAL > GRAVITY**

This menu contains commands related to gravity and centrifugal loading in STAR and NSTAR. The corresponding special loading flags in the A_STATIC for STAR, and A_NONLINEAR for NSTAR, must be activated before running the analysis to consider such effects. Masses must be defined by specifying density as a material property, or through the use of concentrated MASS elements. The gravity value must be associated with a time curve for NSTAR.

Figure 7-9 Gravity Menu

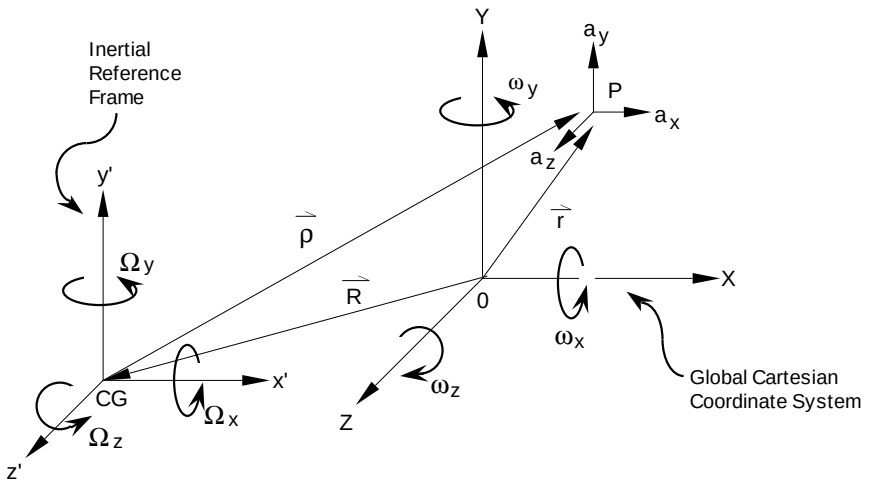


Loads Due to Motion about a Rotating Coordinate System

For centrifugal loading the program calculates the absolute acceleration and multiplies it by the masses to find the forces. The absolute acceleration is calculated as follows:

The forces resulting from the motion of a body defined in a coordinate system that rotates relative to an inertial system are given in terms of the total acceleration which requires several terms for its complete description. Consider a fixed frame of reference denoted by x′y′z′ with the origin at CG, and the coordinate system XYZ with its origin at 0 rotating and translating with respect to x′y′z′ as shown in Figure 7-10.

Figure 7-10



Coordinate system XYZ with origin at 0 rotates about x'y'z' with angular velocity and acceleration Ω and $\dot{\Omega}$

The structure is defined in the XYZ system which is assumed to be the global Cartesian coordinate system and parallel to x′y′z′ at the instance of time considered. The total acceleration of point P relative to the inertial coordinate system x′y′z′ is given by:

$$a_T = a + \omega \times (\omega \times r) + \dot{\omega} \times r + \Omega \times (\Omega \times \rho) + \dot{\Omega} \times \rho + 2\Omega \times (\omega \times r)$$

where:

x	indicates vector cross product
a_T	total (absolute) acceleration of point P

r	location vector of point P from 0
ρ	location vector of point P from CG = r - R




and:

Parameter	COSMOS/M Command	Description
a	ACEL	Acceleration of point P relative to 0 (global Cartesian)
ω	OMEGA	Angular velocity of point P about 0
$\dot{\omega}$	DOMEGA	Angular acceleration of point P about 0
Ω	CGOMEGA	Angular velocity of point 0 about CG
$\dot{\Omega}$	DCGOMEGA	Angular acceleration of point 0 about CG
R	CGLOC	Location vector of CG from 0

It is assumed in the above acceleration equations that R is constant in magnitude.

All of the above parameters are input in terms of their components relative to the global Cartesian coordinate system.

ACEL

Geo Panel: **LoadsBC > STRUCTURAL > GRAVITY > Define Acceleration**

The ACEL command defines the acceleration of the structure in terms of its global Cartesian coordinates. For gravity effects, this command should be used to specify the gravitational acceleration in the appropriate direction. The specified value is associated with the active load case.

Entry & Option Description

x-comp of structure acceleration
x-component of the structure acceleration in the global Cartesian coordinates. *(default is 0.0)*

y-comp of structure acceleration
y-component of the structure acceleration in the global Cartesian coordinates. *(default is 0.0)*

z-comp of structure acceleration
z-component of the structure acceleration in the global Cartesian coordinates. *(default is 0.0)*

Example: `ACEL, 0.0, , -386.4`

This command specifies a gravitational acceleration of 386.4 in the negative global z-direction.

OMEGA

Geo Panel: **LoadsBC > STRUCTURAL > GRAVITY > Define Angular Velocity**

The OMEGA command defines the components of the angular velocity of the structure about the global Cartesian coordinate system. Units are in radians/time. The specified values are associated with the active load case.

(Refer to the ACEL command for syntax.)

DOMEGA

Geo Panel: **LoadsBC > STRUCTURAL > GRAVITY > Define Angular Acceleration**

The DOMEGA command defines the angular acceleration of the structure about the global Cartesian system. Units are in radians/time/time. The specified values are associated with the active load case.

(Refer to the ACEL command for syntax.)

CGOMEGA

Geo Panel: **LoadsBC > STRUCTURAL > GRAVITY > Define Ang Vel of GL CS**

The CGOMEGA command defines the angular velocity of the global Cartesian coordinate system with respect to the inertial reference frame which is a Cartesian coordinate system parallel to the global system with origin at the center of gravity of the model. The specified value is associated with the active load case.

(Refer to the ACEL command for syntax.)

DCGOMEGA

Geo Panel: **LoadsBC > STRUCTURAL > GRAVITY > Define Ang Acc of GL CS**

The DCGOMEGA command defines the angular acceleration of the global Cartesian coordinate system with respect to the inertial reference frame which is a Cartesian coordinate system parallel to the global system with origin at the center of gravity of the model. The specified value is associated with the active load case.

(Refer to the ACEL command for syntax.)

CGLOC

Geo Panel: LoadsBC > STRUCTURAL > GRAVITY > Define Orig of Inertial Sys

The CGLOC command defines the coordinates of the center of gravity with respect to the global Cartesian coordinate system.

(Refer to the ACEL command for syntax.)

GRVLIST

Geo Panel: LoadsBC > STRUCTURAL > GRAVITY > List Gravity Load

The GRVLIST command lists the x, y and z components of all commands related to gravity loading and movements about a fixed or rotating coordinate system. Namely: ACEL, OMEGA, DOMEGA, CGOMEGA, DCGOMEGA and CGLOC commands. (Refer to the help on each command for details).

Example: `GRVLIST`

This command lists the x, y, and z components of all the gravity related commands mentioned above.

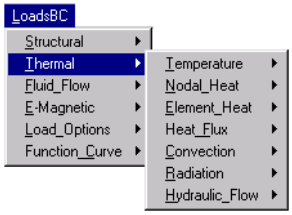



▶ THERMAL Menu

Geo Panel: **LoadsBC > THERMAL**

This menu contains commands to specify thermal-loading and boundary conditions like temperature, heat flow, heat flux, convection, radiation, etc.

Figure 7-11 Thermal Menu

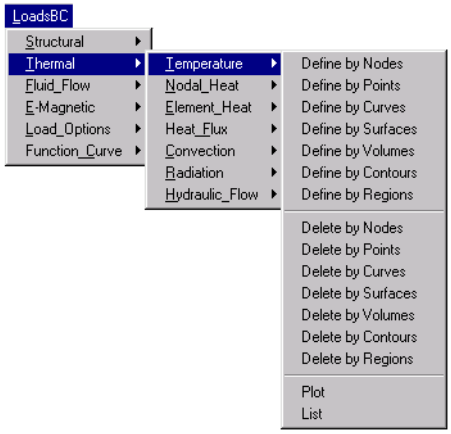


▼TEMPERATURE Menu

Geo Panel: **LoadsBC > THERMAL > TEMPERATURE**

This menu contains commands to specify, list and delete nodal temperatures.

Figure 7-12 Temperature Menu



NTND

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Define by Nodes

The NTND command specifies a temperature value at a pattern of nodes for thermal, structural, and fluid flow problems. The defined temperatures will be associated with the active primary load case. For linear static analysis problems, thermal loading may be applied to some or all primary load cases. For nonlinear, linearized buckling, and frequency analysis considering in-plane loading effects, only the active load case at the time the analysis is performed will be considered. Note that the thermal loading flag must be activated for thermal loading effects to be considered.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Value
Temperature value.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `NTND,1, 5.0, 3, 1`
This command specifies 5.0 units of temperature at nodes 1, 2 and 3.

NTPT

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Define by Points

The NTPT command specifies a temperature value at all nodes associated with any of the keypoints in the specified pattern. The defined temperatures will be associated with the active primary load case. For linear static analysis problems, thermal loading may be applied to some or all primary load cases. For nonlinear, linearized buckling, and frequency analysis considering in-plane loading effects, only the active load case at the time the analysis command is issued will be considered.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Value
Temperature value.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `NTPT, 1, 5.0, 3, 1,`
This command specifies 5.0 units of temperature at all nodes associated with keypoints 1, 2 and 3.

NTCR

Geo Panel: **LoadsBC > THERMAL > TEMPERATURE > Define by Curves**

The NTCR command specifies a temperature value at all nodes associated with any of the curves in the specified pattern. The defined temperatures will be associated with the active primary load case. For linear static analysis problems, thermal loading may be applied to some or all primary load cases. For nonlinear, linearized buckling, and frequency analysis considering in-plane loading effects, only the active load case at the time the analysis command is issued will be considered.

(Refer to the NTPT command for syntax and details.)

NTSF

Geo Panel: **LoadsBC > THERMAL > TEMPERATURE > Define by Surfaces**

The NTSF command specifies a temperature value at all nodes associated with any of the surfaces in the specified pattern. The defined temperatures will be associated with the active primary load case. For linear static analysis problems, thermal loading may be applied to some or all primary load cases. For nonlinear, linearized buckling, and frequency analysis considering in-plane loading effects, only the active load case at the time the analysis command is issued will be considered.

(Refer to the NTPT command for syntax and details.)

NTVL

Geo Panel: **LoadsBC > THERMAL > TEMPERATURE > Define by Volumes**

The NTVL command specifies a temperature value at all nodes associated with any of the volumes in the specified pattern. The defined temperatures will be associated with the active primary load case. For linear static analysis problems, thermal loading may be applied to some or all primary load cases. For nonlinear, linearized buckling, and frequency analysis considering in-plane loading effects, only the active load case at the time the analysis command is issued will be considered.

(Refer to the NTPT command for syntax and details.)




NTCT

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Define by Contours

The NTCT command specifies a temperature value at all nodes associated with any of the contours in the specified pattern. The defined temperatures will be associated with the active primary load case. For linear static analysis problems, thermal loading may be applied to some or all primary load cases. For nonlinear, linearized buckling, and frequency analysis considering in-plane loading effects, only the active load case at the time the analysis command is issued will be considered.

(Refer to the NTPT command for syntax and details.)

NTRG

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Define by Regions

The NTRG command specifies a temperature value at all nodes associated with any of the regions in the specified pattern. The defined temperatures will be associated with the active primary load case. For linear static analysis problems, thermal loading may be applied to some or all primary load cases. For nonlinear, linearized buckling, and frequency analysis considering in-plane loading effects, only the active load case at the time the analysis command is issued will be considered.

(Refer to the NTPT command for syntax and details.)

NTNDEL

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Delete by Nodes

The NTNDEL command deletes previously defined temperatures at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `NTNDEL, 1, 3, 1`
This command deletes the temperatures applied at nodes 1, 2 and 3.




NTPDEL

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Delete by Points

The NTPDEL command deletes previously defined temperatures for all the nodes associated with any of the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `NTPDEL, 1, 4, 1`
This command deletes temperatures for all the nodes associated with any of keypoints 1 through 4.

NTCDEL

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Delete by Curves

The NTCDEL command deletes previously defined temperatures at all nodes associated with any of the curves in the specified pattern.

(Refer to the NTPDEL command for syntax and details.)

NTSDEL

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Delete by Surfaces

The NTSDEL command deletes previously defined temperatures for all the nodes associated with any of the surfaces in the specified pattern.

(Refer to the NTPDEL command for syntax and details.)

NTVDEL

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Delete by Volumes

The NTVDEL command deletes previously defined temperatures for all the nodes associated with any of the volumes in the specified pattern.

(Refer to the NTPDEL command for syntax and details.)

NTCTDEL

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Delete by Contours

The NTCTDEL command deletes previously defined temperatures for all the nodes associated with any of the contours in the specified pattern.

(Refer to the NTPDEL command for syntax and details.)

NTRDEL

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Delete by Regions

The NTRDEL command deletes previously defined temperatures for all the nodes associated with any of the regions in the specified pattern.

(Refer to the NTPDEL command for syntax and details.)

NTPLOT

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > Plot

The NTPLOT command plots a predefined symbol at every node with prescribed nodal temperatures in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is max. node number defined)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `NTPLOT, , , ,`
The above command plots a symbol at nodes with defined nodal temperatures.

NTLIST

Geo Panel: LoadsBC > THERMAL > TEMPERATURE > List

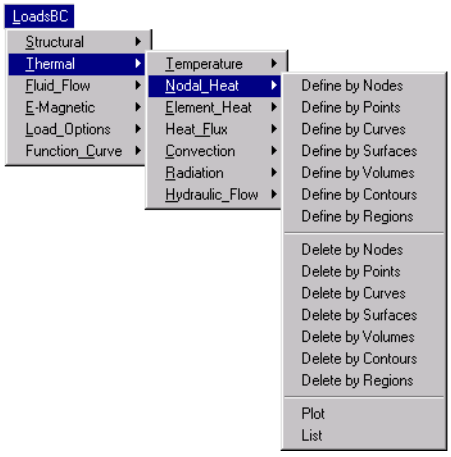
The NTLIST command lists prescribed nodal temperatures in the specified pattern.

(Refer to the PTLIST command for syntax and details.)

▼NODAL_HEAT Menu

Geo Panel: **LoadsBC > THERMAL > NODAL_HEAT**

Figure 7-13 Nodal_Heat Menu



QND

Geo Panel: **LoadsBC > THERMAL > NODAL_HEAT > Define by Nodes**

The QND command specifies rate of heat generation at all nodes in the specified pattern for thermal analysis.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Value
Magnitude of rate of heat generation.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `QND, 1, 5.0, 3, 1`
This command specifies 5.0 units of rate of heat generation at nodes 1,2 and 3.

QPT

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Define by Points

The QPT command specifies a rate of heat generation at all nodes associated with any of the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Value
Magnitude of rate of heat generation.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example:
```
QPT, 1, 5.0, 3, 1,
```
This command specifies 5.0 units of rate of heat generation at all nodes associated with keypoints 1, 2 and 3.

QCR

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Define by Curves

The QCR command specifies rate of heat generation at all nodes associated with any of the curves in the specified pattern.

(Refer to the QPT command for syntax and details.)

QSF

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Define by Surfaces

The QSF command specifies a rate of heat generation at nodes associated with any of the surfaces in the specified pattern.

(Refer to the QPT command for syntax and details.)

QVL

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Define by Volumes

The QVL command specifies the rate of heat generation at all nodes associated with any of the volumes in the specified pattern.

(Refer to the QPT command for syntax and details.)

QCT

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Define by Contours

The QCT command specifies the rate of heat generation at all nodes associated with any of the contours in the specified pattern.

(Refer to the QPT command for syntax and details.)

QRG

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Define by Regions

The QRG command specifies the rate of heat generation at all nodes associated with any of the regions in the specified pattern.

(Refer to the QPT command for syntax and details.)

QNDEL

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Delete by Nodes

The QNDEL command deletes previously defined rates of heat generation at all nodes of the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `QNDEL, 1, 3, 1`
This command deletes the rates of heat generation applied at nodes 1, 2 and 3.

QPDEL

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Delete by Points

The QPDEL command deletes previously defined rates of heat generation at all nodes associated with any of the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `QPDEL, 1, 4, 1`

This command deletes rates of heat generation at all nodes associated with any of keypoints 1 through 4.

QCDEL

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Delete by Curves

The QCDEL command deletes previously defined rates of heat generation at all nodes associated with any of the curves in the specified pattern.

(Refer to the QPDEL command for syntax and details.)

QSDEL

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Delete by Surfaces

The QSDEL command deletes previously defined rates of heat generation at all nodes associated with any of the surfaces in the specified pattern.

(Refer to the QPDEL command for syntax and details.)

QVDEL

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Delete by Volumes

The QVDEL command deletes previously defined rates of heat generation at all nodes associated with any of the volumes in the specified pattern.

(Refer to the QPDEL command for syntax and details.)

QCTDEL

Geo Panel: LoadsBC > THERMAL > NODAL_HEAT > Delete by Contours

The QCTDEL command deletes previously defined rates of heat generation for all nodes associated with any of the contours in the specified pattern.

(Refer to the QPDEL command for syntax and details.)

QRDEL

Geo Panel: **LoadsBC > THERMAL > NODAL_HEAT > Delete by Regions**

The QRDEL command deletes previously defined rates of heat generation for all nodes associated with any of the regions in the specified pattern.

(Refer to the QPDEL command for syntax and details.)

QPLOT

Geo Panel: **LoadsBC > THERMAL > NODAL_HEAT > Plot**

The QPLOT command plots a predefined symbol at each node with prescribed rate of heat generation in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is the maximum node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `QPLOT;`
The above command plots a symbol at nodes for which a rate of heat generation has been specified.

QLIST

Geo Panel: **LoadsBC > THERMAL > NODAL_HEAT > List**

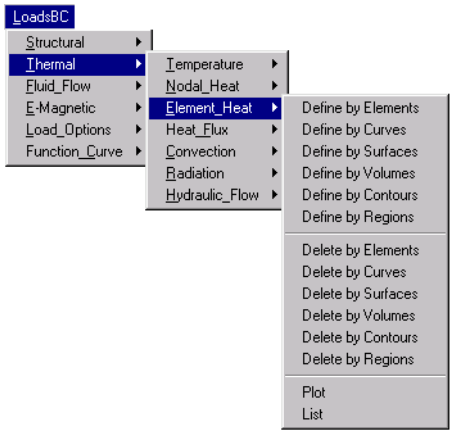
The QLIST command lists prescribed rates of heat generation for a specified pattern of nodes.

(Refer to the QPLOT command for syntax and details.)

▼ELEMENT_HEAT Menu

Geo Panel: **LoadsBC > THERMAL > ELEMENT_HEAT**

Figure 7-14 Element_Heat Menu



QEL

Geo Panel: **LoadsBC > THERMAL > ELEMENT_HEAT > Define by Elements**

The QEL command applies heat generation (per unit volume, per unit time) to elements in the specified pattern for thermal and fluid flow analyses.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Value
Magnitude of element heat generation rate per unit volume.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between the elements in the pattern.
(default is 1)

Example: `QEL,1,5.,3,1`
This command applies heat generation rate of 5. to elements 1, 2 and 3.

QECR

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Define by Curves

The QECR command applies heat generation (per unit volume, per unit time) to all 1D elements associated with any of the curves in the specified pattern.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Value
Magnitude of heat generation rate per unit length.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Example: `QECR, 1, 5.0, 3, 1,`
This command applies heat generation rate of 5.0 units to all 1D elements associated with curves 1, 2 and 3.

QESF

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Define by Surfaces

The QESF command applies heat generation (per unit volume, per unit time) to all 2D elements associated with any of the surfaces in the specified pattern.

(Refer to the QECR command for syntax and details.)

QEVL

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Define by Volumes

The QEVL command applies heat generation (per unit volume, per unit time) to all 3D elements associated with any of the volumes in the specified pattern.

(Refer to the QECR command for syntax and details.)

QECT

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Define by Contours

The QECT command applies heat generation (per unit volume, per unit time) to all 1D elements associated with any of the contours in the pattern.

(Refer to the QECR command for syntax and details.)




QERG

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Define by Regions

The QERG command applies heat generation (per unit volume, per unit time) to all 2D elements associated with any of the regions in the specified pattern.

(Refer to the QECR command for syntax and details.)

QEDEL

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Delete by Elements

The QEDEL command deletes previously defined rates of heat generation at all elements in the specified pattern.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `QEDEL, 1, 3, 1`
This command deletes the rates of heat generation applied to elements 1, 2 and 3.

QECDEL

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Delete by Curves

The QECDEL command deletes previously defined rates of heat generation for all elements associated with any of the curves in the specified pattern.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)




Example: `QECDEL, 1, 3, 1`

This command deletes rates of heat generation for all the elements associated with curves 1, 2 and 3.

QESDEL

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Delete by Surfaces

The QESDEL command deletes previously defined rates of heat generation for all elements associated with any of the surfaces in the specified pattern.

(Refer to the QECDEL command for syntax and details.)

QEVDEL

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Delete by Volumes

The QEVDEL command deletes previously defined rates of heat generation for all elements associated with any of the volumes in the specified pattern.

(Refer to the QECDEL command for syntax and details.)

QECTDEL

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Delete by Contours

The QECTDEL command deletes previously defined rates of heat generation for all elements associated with any of the contours in the specified pattern.

(Refer to the QECDEL command for syntax and details.)

QERDEL

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Delete by Regions

The QERDEL command deletes previously defined rates of heat generation for all elements associated with any of the regions in the specified pattern.

(Refer to the QECDEL command for syntax and details.)

QEPLOT

Geo Panel: LoadsBC > THERMAL > ELEMENT_HEAT > Plot

The QEPLOT command plots a predefined symbol at all elements with prescribed rates of heat generation for a pattern of elements. The symbol is shown in the STATUS2 Table.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is the highest element number defined)

Increment
Increment between the elements in the pattern.
(default is 1)

Example: `QEPLOT, , 6, ,`

The above command plots a predefined symbol to identify elements with prescribed rates of heat generation for elements 1 through 6.

QELIST

Geo Panel: **LoadsBC > THERMAL > ELEMENT_HEAT > List**

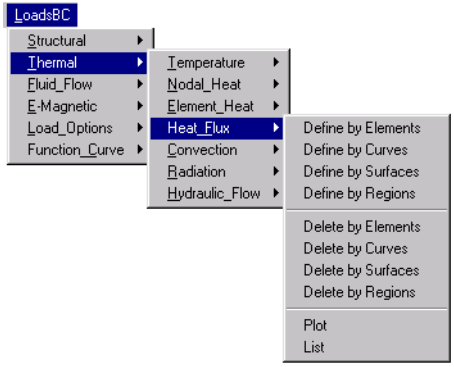
The QELIST command lists prescribed rates of heat generation for a pattern of elements.

(Refer to the QEPLOT command for syntax and details.)

▼HEAT_FLUX Menu

Geo Panel: **LoadsBC > THERMAL > HEAT_FLUX**

Figure 7-15 Heat_Flux Menu



HXEL

Geo Panel: **LoadsBC > THERMAL > HEAT_FLUX > Define by Elements**

The HXEL command applies heat flux on a specified face for a pattern of elements.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Heat flux
Heat flux value.

Face number
Face number on which the heat flux is to be specified. (Refer to Chapter 4 of the COSMOS/M User Guide.)

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

HXCR

Geo Panel: **LoadsBC > THERMAL > HEAT_FLUX > Define by Curves**

The HXCR command applies heat flux on all elements associated with a pattern of curves.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Heat flux
Flux value.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

HXSF

Geo Panel: **LoadsBC > THERMAL > HEAT_FLUX > Define by Surfaces**

The HXSF command applies heat flux on all elements associated with a pattern of surfaces.

(Refer to the HXCR command for syntax and details.)

HXRG

Geo Panel: LoadsBC > THERMAL > HEAT_FLUX > Define by Regions

The HXRG command applies heat flux on all elements associated with a pattern of regions.

(Refer to the HXCR command for syntax and details.)

HXEDEL

Geo Panel: LoadsBC > THERMAL > HEAT_FLUX > Delete by Elements

The HXEDEL command deletes the heat flux on a specified face for a pattern of elements.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Face number
Face number. (Refer to Chapter 4 of the COSMOS/M User Guide).

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `HXEDEL, 1, 2, 5, 1`
This command deletes prescribed heat flux on face 2 of elements 1 through 5.

HXCDEL

Geo Panel: LoadsBC > THERMAL > HEAT_FLUX > Delete by Curves

The HXCDEL command deletes the heat flux on elements associated with a pattern of curves.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment

Increment between curves in the pattern.
(default is 1)

HXSDEL

Geo Panel: LoadsBC > THERMAL > HEAT_FLUX > Delete by Surfaces

The HXSDEL command deletes the heat flux on elements associated with a pattern of surfaces.

(Refer to the HXCDEL command for syntax and details.)

HXRDEL

Geo Panel: LoadsBC > THERMAL > HEAT_FLUX > Delete by Regions

The HXRDEL command deletes the heat flux on elements associated with a pattern of regions.

(Refer to the HXCDEL command for syntax and details.)

HXPLOT

Geo Panel: LoadsBC > THERMAL > HEAT_FLUX > Plot

The HXPLOT command plots a predefined symbol at elements with prescribed heat flux. The symbol is shown in the STATUS2 Table.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `HXPLOT, 1, 10, 1`
This command plots the heat flux symbol at elements 1 through 10 for which heat flux is specified.

HXLIST

Geo Panel: LoadsBC > THERMAL > HEAT_FLUX > List

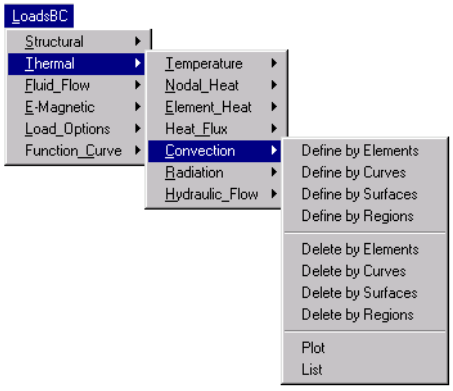
The HXLIST command lists the heat flux for a pattern of elements.

(Refer to the HXPLOT command for syntax and details.)

▼CONVECTION Menu

Geo Panel: **LoadsBC > THERMAL > CONVECTION**

Figure 7-16 Convection Menu



CEL

Geo Panel: **LoadsBC > THERMAL > CONVECTION > Define by Elements**

The CEL command specifies a convection film coefficient and an ambient temperature for faces of elements in the specified pattern for use in thermal and fluid flow analyses.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Convection coefficient
Magnitude of convection coefficient.

Ambient temperature
Adjacent ambient temperature.

Face number
Face of the elements on which film coefficient is applied.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment

Increment between elements in the pattern.
(default is 1)

Time curve for ambient temperature
Time curve associated with the ambient temperature.

Notes

1. Refer to Chapter 4 of the COSMOS/M User Guide for the definition of element faces for different element types.
2. The convection coefficient is associated with the active time curve.

Example: `CEL, 3, 100., 30.0, 1, 10, 1, 1`

This command applies convection coefficient of 100.0 and bulk temperature of 30.0 on face 1 of elements 3 through 10.

CECR

Geo Panel: **LoadsBC > THERMAL > CONVECTION > Define by Curves**

The CECR command specifies a convection coefficient and an ambient temperature to elements associated with a pattern of curves that are sides of surfaces.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Convection coefficient
Magnitude of convection coefficient.

Ambient temperature
Adjacent ambient temperature.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between the curves in the pattern.
(default is 1)

Time curve for ambient temperature
Time curve associated with the ambient temperature.

Note

The convection coefficient is associated with the active time curve.

Example: `CECR, 1, 100.0, 30, 3, 1, 2`

This command applies convection coefficient of 100 with bulk temperature of 30 on curves 1, 2 and 3.

CESF

Geo Panel: LoadsBC > THERMAL > CONVECTION > Define by Surfaces

The CESF command specifies a convection coefficient and an ambient temperature for element faces associated with a pattern of surfaces.

(Refer to the CECR command for syntax and details.)

CERG

Geo Panel: LoadsBC > THERMAL > CONVECTION > Define by Regions

The CERG command specifies a convection coefficient and an ambient temperature for all elements associated with any of the regions in the specified pattern.

(Refer to the CECR command for syntax and details.)

CEDEL

Geo Panel: LoadsBC > THERMAL > CONVECTION > Delete by Elements

The CEDEL command deletes previously defined convection coefficients and ambient temperatures for the specified face for a pattern of elements.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Face number
Face number of the elements for which the previously specified convection coefficient is to be deleted.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Note

Refer to Chapter 4 of the COSMOS/M User Guide for the definition of element faces for different element types.

Example: `CEDEL, 3, 2, 10, 1`

This command deletes the convection coefficients and ambient temperatures on face 2 of elements 3 through 10.




CECDEL

Geo Panel: LoadsBC > THERMAL > CONVECTION > Delete by Curves

The CECDEL command deletes previously defined convection coefficients for elements associated with a pattern of curves.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Example: `CECDEL, 1, 10, 1`
This command deletes convection coefficients and ambient temperatures for elements associated with curves 1 through 10.

CESDEL

Geo Panel: LoadsBC > THERMAL > CONVECTION > Delete by Surfaces

The CESDEL command deletes previously defined convection coefficients and ambient temperatures for all elements associated with a pattern of surfaces.

(Refer to the CECDEL command for syntax and details.)

CERDEL

Geo Panel: LoadsBC > THERMAL > CONVECTION > Delete by Regions

The CERDEL command deletes previously defined convection coefficients and ambient temperatures for all the elements associated with any of the regions in the specified pattern.

(Refer to the CECDEL command for syntax and details.)

CEPLOT

Geo Panel: LoadsBC > THERMAL > CONVECTION > Plot

The CEPLOT command plots a predefined symbol at elements with prescribed convection coefficients for a pattern of elements. The symbol is shown in the STATUS2 Table.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is the maximum element number defined)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `CEPLOT;`

The above command plots a predefined symbol at elements with prescribed convection coefficients.

CELIST

Geo Panel: **LoadsBC > THERMAL > CONVECTION > List**

The CELIST command lists element convection coefficients and ambient temperatures for a pattern of elements.

(Refer to the CEPLOT command for syntax and details.)

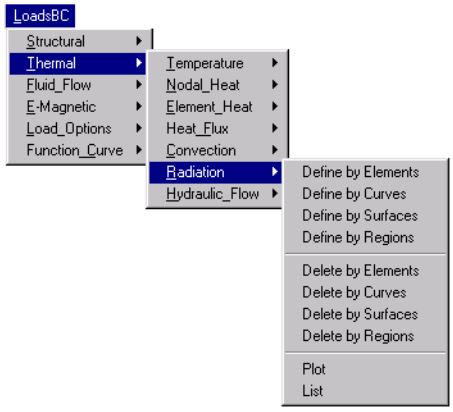



▼RADIATION Menu

Geo Panel: **LoadsBC > THERMAL > RADIATION**

This menu contains commands to define, list and delete coupling specifications including coupling DOF's, coupling equations and coupling constraints.

Figure 7-17 Radiation Menu



REL

Geo Panel: **LoadsBC > THERMAL > RADIATION > Define by Elements**

The REL command specifies temperature dependent emissivity, view factor, and the source temperature on a specified element face for a pattern of elements. The prescribed emissivity is associated with the active temperature curve while the source temperature is associated with the specified time curve.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Emissivity
Emissivity of the radiating face.
(default is 1.0)

View factor
View factor between this face and the radiation source/sink.
(default is 1.0)

Source temperature
Source temperature.

Face number

Face number on which the radiation is to be specified. (Refer to Chapter 4 of the COSMOS/M User Guide.)

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Time curve for source temperature
Time curve to be associated with the source temperature.

RECR

Geo Panel: LoadsBC > THERMAL > RADIATION > Define by Curves

The RECR command specifies emissivity, view factor, and source temperature for elements associated with a specified pattern of curves. The prescribed emissivity is associated with the active temperature curve while the source temperature is associated with the specified time curve.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Emissivity
Emissivity of the radiating face.
(default is 1.0)

View factor
View factor between this face and the radiation source/sink.
(default is 1.0)

Source temperature
Source temperature.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Time curve for source temperature
Time curve to be associated with the source temperature.

RESF

Geo Panel: **LoadsBC > THERMAL > RADIATION > Define by Surfaces**

The RESF command specifies emissivity, view factor, and source temperature for elements associated with a specified pattern of surfaces. The prescribed emissivity is associated with the active temperature curve while the source temperature is associated with the specified time curve.

(Refer to the RECR command for syntax and details.)

RERG

Geo Panel: **LoadsBC > THERMAL > RADIATION > Define by Regions**

The RERG command specifies emissivity, view factor, and source temperature for elements associated with a specified pattern of regions. The prescribed emissivity is associated with the active temperature curve while the source temperature is associated with the specified time curve.

(Refer to the RECR command for syntax and details.)

REDEL

Geo Panel: **LoadsBC > THERMAL > RADIATION > Delete by Elements**

The REDEL command deletes the emissivity, view factor, and the source temperature on element faces for a pattern of elements.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Face number
Face number. (Refer to Chapter 4 of the COSMOS/M User Guide.)

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)




RECDEL

Geo Panel: LoadsBC > THERMAL > RADIATION > Delete by Curves

The RECDEL command deletes the emissivity, view factor, and the source temperature for elements associated with a pattern of curves.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

RESDEL

Geo Panel: LoadsBC > THERMAL > RADIATION > Delete by Surfaces

The RESDEL command deletes the emissivity, view factor, and the source temperature for elements associated with a pattern of surfaces.

(Refer to the RECDEL command for syntax and details.)

RERDEL

Geo Panel: LoadsBC > THERMAL > RADIATION > Delete by Regions

The RERDEL command deletes the emissivity, view factor, and the source temperature for elements associated with a pattern of regions.

(Refer to the RECDEL command for syntax and details.)

REPLOT

Geo Panel: LoadsBC > THERMAL > RADIATION > Plot

The REPLOT command plots a predefined symbol at elements with prescribed radiation boundary conditions.

Entry & Option Description

Beginning element

Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is the highest element number defined)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `REPLOT, 1, 10, 1`
This command plots the radiation symbol at elements 1 through 10 for which radiation was specified.

RELIST

Geo Panel: **LoadsBC > THERMAL > RADIATION > List**

The RELIST command lists the emissivity, view factors and the ambient temperatures for a pattern of elements.

(Refer to the REPLOT command for syntax and details.)

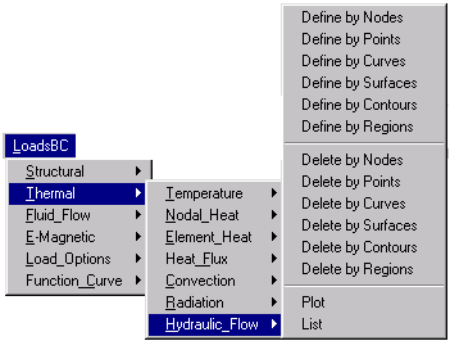



▼HYDRAULIC_FLOW Menu

Geo Panel: **LoadsBC > THERMAL > HYDRAULIC_FLOW**

This menu contains commands to define, list, delete, and plot main flow rates used by HSTAR with the HLINK element.

Figure 7-18 Hydraulic_Flow Menu



HFND

Geo Panel: **LoadsBC > THERMAL > HYDRAULIC_FLOW > Define by Nodes**

The HFND command specifies a mass flowrate value at a pattern of nodes for thermal problems using the HLINK element.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Value
Mass flowrate value.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example:
```
HFND, 1, 5.0, 1, 1
```
This command specifies 5.0 units of mass flowrate at node 1.

HFPT

Geo Panel: LoadsBC > THERMAL > HYDRAULIC_FLOW > Define by Points

The HFPT command specifies a mass flowrate value at nodes associated with a pattern of keypoints for thermal problems using the HLINK element.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Value
Mass flowrate value.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `HFPT, 1, 5.0, 1, 1`

This command specifies 5.0 units of mass flowrate at node(s) associated with keypoint 1.

HFCR

Geo Panel: LoadsBC > THERMAL > HYDRAULIC_FLOW > Define by Curves

The HFCR command specifies a mass flowrate value at nodes associated with a pattern of curves for thermal problems using the HLINK element.

Entry & Option Description

Beginning keypoint
Beginning curve in the pattern.

Value
Mass flowrate value.

Ending keypoint
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curve in the pattern.
(default is 1)

Example: `HFCR, 1, 5.0, 1, 1`

This command specifies 5.0 units of mass flowrate at node(s) associated with curve 1.

HFSF

Geo Panel: **LoadsBC > THERMAL > HYDRAULIC_FLOW > Define by Surfaces**

The HFSF command specifies a mass flowrate value at nodes associated with a pattern of surfaces for thermal problems using the HLINK element.

(Refer to the HFCR command for syntax and details.)

HFCT

Geo Panel: **LoadsBC > THERMAL > HYDRAULIC_FLOW > Define by Contours**

The HFCT command specifies a mass flowrate value at nodes associated with a pattern of contours for thermal problems using the HLINK element.

(Refer to the HFCR command for syntax and details.)

HFRG

Geo Panel: **LoadsBC > THERMAL > HYDRAULIC_FLOW > Define by Regions**

The HFRG command specifies a mass flowrate value at nodes associated with a pattern of regions for thermal problems using the HLINK element.

(Refer to the HFCR command for syntax and details.)

HFNDEL

Geo Panel: **LoadsBC > THERMAL > HYDRAULIC_FLOW > Delete by Nodes**

The HFNDEL command deletes previously defined mass flowrate at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment

Increment between nodes in the pattern.
(default is 1)

Example: `HFNDEL, 1, 1, 1`
This command deletes the mass flowrate applied at node 1.

HFPDEL

Geo Panel: **LoadsBC > THERMAL > HYDRAULIC_FLOW > Delete by Points**

The HFPDEL command deletes previously defined mass flowrate at nodes associated with keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `HFPDEL, 1, 1, 1`
This command deletes the mass flowrate applied at node(s) associated with keypoint 1.

HFCDEL

Geo Panel: **LoadsBC > THERMAL > HYDRAULIC_FLOW > Delete by Curves**

The HFCDEL command deletes previously defined mass flowrate at nodes associated with curves in the specified pattern.

(Refer to the HFPDEL command for syntax and details.)

HFSDEL

Geo Panel: **LoadsBC > THERMAL > HYDRAULIC_FLOW > Delete by Surfaces**

The HFSDEL command deletes previously defined mass flowrate at nodes associated with surfaces in the specified pattern.

(Refer to the HFPDEL command for syntax and details.)




HFCTDEL

Geo Panel: LoadsBC > THERMAL > HYDRAULIC_FLOW > Delete by Contours

The HFCTDEL command deletes previously defined mass flowrate at nodes associated with contours in the specified pattern.

(Refer to the HFPDEL command for syntax and details.)

HFRDEL

Geo Panel: LoadsBC > THERMAL > HYDRAULIC_FLOW > Delete by Regions

The HFRDEL command deletes previously defined mass flowrate at nodes associated with regions in the specified pattern.

(Refer to the HFPDEL command for syntax and details.)

HFLIST

Geo Panel: LoadsBC > THERMAL > HYDRAULIC_FLOW > List

The HFLIST command lists prescribed nodal mass flowrate for a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is max. node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Example:
```
HFLIST, 1, 10, 1,
```
The above command lists all the specified mass flowrate values for nodes 1 through 10.

HFPLOT

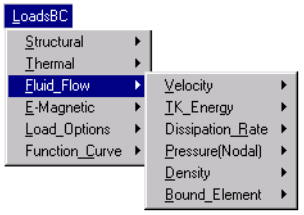
Geo Panel: LoadsBC > THERMAL > HYDRAULIC_FLOW > Plot

The HFPLOT command plots a symbol at every node in the pattern with prescribed mass flowrate.

(Refer to the HFLIST command for syntax.)

▶ FLUID_FLOW Menu

Figure 7-19 Fluid_Flow Menu



Geo Panel: **LoadsBC > FLUID_FLOW**

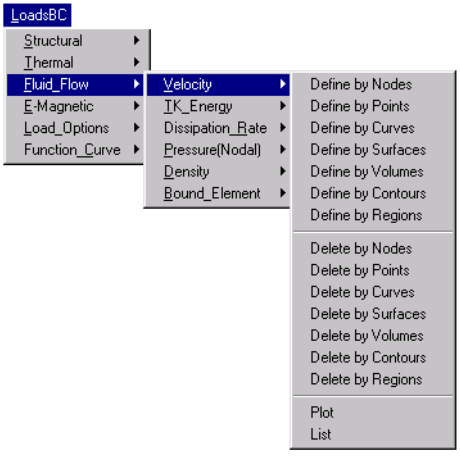
This menu contains commands to specify fluid flow loading and boundary conditions like velocity, dissipation rate, etc.

▼VELOCITY Menu

Geo Panel: **LoadsBC > FLUID_FLOW > VELOCITY**

This menu contains commands to specify, list and delete nodal temperatures.

Figure 7-20 Velocity Menu



VND

Geo Panel: **LoadsBC > FLUID FLOW > VELOCITY > Define by Nodes**

The VND command defines velocities at a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Velocity label
Label indicates the direction of the velocity. Labels refer to the global coordinate

system.
= VX velocity in the x-direction
= VY velocity in the y-direction
= VZ velocity in the z-direction
= AL velocity in the x-, y- and z-directions
(default is VX)

Value
Velocity value.
(default is 0.0)

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between the nodes in the pattern.
(default is 1)

Velocity labels
Additional labels of velocity components.

Example:
```
VND, 1, X velocity, 5., 3, 1, Z velocity,
```
This command specifies 5 units of velocity along the X- and Z-directions at nodes 1, 2 and 3.

VPT

Geo Panel: **LoadsBC > FLUID_FLOW > VELOCITY > Define by Points**

The VPT command specifies velocities for nodes associated with any of the keypoints in the specified pattern. Velocities are specified in the global coordinate system.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Velocity label
Label indicates the direction of the velocity. Labels refer to the global coordinate system.
= VX velocity in the x-direction
= VY velocity in the y-direction
= VZ velocity in the z-direction
= AL velocity in the x-, y- and z-directions
(default is VX)




Value
Velocity value.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between the keypoints in the pattern.
(default is 1)

Velocity labels
Additional labels of velocity components.

Example: `VPT, 1, Y velocity, -5.0, 3, 1,`
This command specifies 5.0 units of velocity along negative Y-direction to all the nodes associated with keypoints 1, 2 or 3.

VCR

Geo Panel: **LoadsBC > FLUID_FLOW > VELOCITY > Define by Curves**

The VCR command specifies velocities for nodes associated with any of the curves in the specified pattern. Velocities are specified in the global coordinate system.

(Refer to the VPT command for syntax and details.)

VSF

Geo Panel: **LoadsBC > FLUID_FLOW > VELOCITY > Define by Surfaces**

The VSF command specifies velocities for nodes associated with any of the surfaces in the specified pattern. Velocities are specified in the global coordinate system.

(Refer to the VPT command for syntax and details.)

VVL

Geo Panel: **LoadsBC > FLUID_FLOW > VELOCITY > Define by Volumes**

The VVL command specifies velocities for nodes associated with any of the volumes in the specified pattern. Velocities are specified in the global coordinate system.

(Refer to the VPT command for syntax and details.)




VCT

Geo Panel: LoadsBC > FLUID_FLOW > VELOCITY > Define by Contours

The VCT command specifies velocities for all nodes associated with any of the contours in the specified pattern. Velocities are specified in the global coordinate system.

(Refer to the VPT command for syntax and details.)

VRG

Geo Panel: LoadsBC > FLUID_FLOW > VELOCITY > Define by Regions

The VRG command specifies velocities for all nodes associated with any of the regions in the specified pattern. Velocities are specified in the global coordinate system.

(Refer to the VPT command for syntax and details.)

VNDEL

Geo Panel: LoadsBC > FLUID_FLOW > VELOCITY > Delete by Nodes

The VNDEL command deletes previously defined velocities for all the nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Velocity label
Label indicates the direction of the velocity component to be deleted. Labels refer to the global coordinate system.
= VX velocity in the x-direction
= VY velocity in the y-direction
= VZ velocity in the z-direction
= AL all components
(default is VX)

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Velocity labels

Additional labels of velocity components to be deleted.

Example: `VNDEL, 1, X velocity, 3, 1`

This command deletes the velocities applied in the X-direction at nodes 1, 2 and 3.

VPDEL

Geo Panel: **LoadsBC > FLUID_FLOW > VELOCITY > Delete by Points**

The VPDEL command deletes previously defined velocities for all nodes associated with any of the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Velocity label
Label indicates the direction of the velocity component to be deleted. Labels refer to the global coordinate system.
= VX velocity in the x-direction
= VY velocity in the y-direction
= VZ velocity in the z-direction
= AL all components
(default is VX)

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Velocity labels
Additional labels of velocity components to be deleted.

Example: `VPDEL, 1, Y velocity, 4, 1`

This command deletes the velocity component in the Y-direction for all the nodes associated with keypoints 1 through 4.

VCDEL

Geo Panel: **LoadsBC > FLUID_FLOW > VELOCITY > Delete by Curves**

The VCDEL command deletes previously defined velocities for all nodes associated with any of the curves in the specified pattern.

(Refer to the VPDEL command for syntax and details.)

VSDEL

Geo Panel: LoadsBC > FLUID_FLOW > VELOCITY > Delete by Surfaces

The VSDEL command deletes previously defined velocities for all nodes associated with any of the surfaces in the specified pattern.

(Refer to the VPDEL command for syntax and details.)

VVDEL

Geo Panel: LoadsBC > FLUID_FLOW > VELOCITY > Delete by Volumes

The VVDEL command deletes previously defined velocities for all nodes associated with any of the volumes in the specified pattern.

(Refer to the VPDEL command for syntax and details.)

VCTDEL

Geo Panel: LoadsBC > FLUID_FLOW > VELOCITY > Delete by Contours

The VCTDEL command deletes previously defined velocities for all the nodes associated with any of the contours in the specified pattern.

(Refer to the VPDEL command for syntax and details.)

VRDEL

Geo Panel: LoadsBC > FLUID_FLOW > VELOCITY > Delete by Regions

The VRDEL command deletes previously defined velocities for all nodes associated with any of the regions in the specified pattern.

(Refer to the VPDEL command for syntax and details.)

VPLOT

Geo Panel: LoadsBC > FLUID_FLOW > VELOCITY > Plot

The VPLOT command plots velocity using a predefined symbol for nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Velocity label
Label indicates the direction of the velocity component to be plotted. Labels

refer to the global coordinate system.
= VX velocity in the x-direction
= VY velocity in the y-direction
= VZ velocity in the z-direction
= AL all components

Ending node
Ending node in the pattern.
(default is the highest node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Velocity labels
Additional labels of velocity components.

Example: `VPLOT, , X velocity, 6, ,`
This command plots the prescribed velocity symbol at nodes 1 through 6 for which X velocity was specified.

VLIST

Geo Panel: **LoadsBC > FLUID_FLOW > VELOCITY > List**

The VLIST command lists currently defined nodal velocities for a specified pattern of nodes.

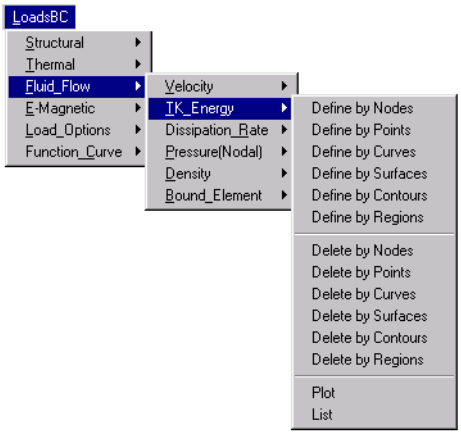
(Refer to the VPLOT command for syntax and details.)




▼TK_ENERGY Menu

Geo Panel: **LoadsBC > FLUID_FLOW > TK_ENERGY**

Figure 7-21 TK_Energy Menu



TKEND

Geo Panel: **LoadsBC > FLUID_FLOW > TK_ENERGY > Define by Nodes**

The TKEND command specifies turbulence kinetic energy value for a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Value
Turbulence kinetic energy value.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `TKEND, 1, 5.0, 3, 1`
This command specifies 5.0 units of TKE at nodes 1, 2 and 3.

TKEPT

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Define by Points

The TKEPT command specifies turbulence kinetic energy value for nodes associated with a pattern of keypoints.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Value
Turbulence kinetic energy value.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between the keypoints in the pattern.
(default is 1)

Example:
```
TKEPT, 1, 5.0, 3, 1,
```
This command specifies 5.0 units of turbulence kinetic energy at the keypoints 1, 2 and 3.

TKECR

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Define by Curves

The TKECR command specifies turbulence kinetic energy value at all nodes associated with a pattern of curves.

(Refer to the TKEPT command for syntax and details.)

TKESF

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Define by Surfaces

The TKESF command specifies a turbulence kinetic energy value at all nodes associated with the surfaces in the specified pattern.

(Refer to the TKEPT command for syntax and details.)




TKECT

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Define by Contours

The TKECT command specifies a turbulence kinetic energy value at all nodes associated with a pattern of contours.

(Refer to the TKEPT command for syntax and details.)

TKERG

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Define by Regions

The TKERG command specifies a turbulence kinetic energy value at all nodes associated with a pattern of regions.

(Refer to the TKEPT command for syntax and details.)

TKENDEL

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Delete Nodes

The TKENDEL command deletes previously defined turbulence kinetic energy at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `TKENDEL, 1, 3, 1`

This command deletes the turbulence kinetic energy applied at nodes 1, 2 and 3.




TKEPDEL

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Delete by Points

The TKEPDEL command deletes previously defined turbulence kinetic energy at all nodes associated with a pattern of keypoints.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `TKEPDEL, 1, 3, 1`
This command deletes the turbulence kinetic energy applied at keypoints 1, 2 and 3.

TKECDEL

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Delete by Curves

The TKECDEL command deletes previously defined turbulence kinetic energy at all nodes associated with a pattern of curves.

(Refer to the TKEPDEL command for syntax and details.)

TKESDEL

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Delete by Surfaces

The TKESDEL command deletes previously defined turbulence kinetic energy at all nodes associated with a pattern of surfaces.

(Refer to the TKEPDEL command for syntax and details.)




TKECTDEL

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Delete by Contours

The TKECTDEL command deletes previously defined turbulence kinetic energy at all nodes associated with a pattern of contours.

(Refer to the TKEPDEL command for syntax and details.)

TKERDEL

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Delete by Regions

The TKERDEL command deletes previously defined turbulence kinetic energy at all nodes associated with a pattern of regions.

(Refer to the TKEPDEL command for syntax and details.)

TKEPLOT

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > Plot

The TKEPLOT command plots a predefined symbol at nodes with prescribed nodal turbulence kinetic energy for a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is max. node number defined)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `TKEPLOT, ,6 , ,`

The above command plots a predefined symbol at nodes 1 through 6 for which turbulence kinetic energy was prescribed.

TKELIST

Geo Panel: LoadsBC > FLUID_FLOW > TK_ENERGY > List

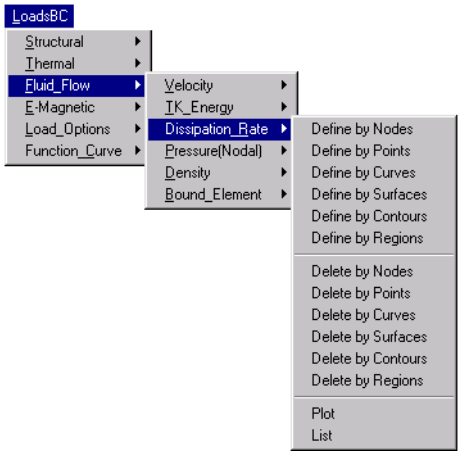
The TKELIST command lists prescribed nodal turbulence kinetic energy for a pattern of nodes.

(Refer to the TKEPLOT command for syntax and details.)

▼DISSIPATION_RATE Menu

Geo Panel: **LoadsBC > FLUID_FLOW > DISSIPATION_RATE**

Figure 7-22 Dissipation_Rate Menu



EPSND

Geo Panel: **LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Define by Nodes**

The EPSND command specifies a dissipation rate for a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Value
Dissipation rate.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `EPSND, 1, 5.0, 3, 1`
This command specifies 5.0 units of EPS at nodes 1, 2 and 3.

EPSPT

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Define by Points

The EPSPT command specifies a dissipation rate for nodes associated with a pattern of keypoints.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Value
Dissipation rate.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between the keypoints in the pattern.
(default is 1)

Example: `EPSPT, 1, 5.0, 3, 1,`

This command specifies 5.0 units of dissipation rate at keypoints 1, 2 and 3.

EPSCR

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Define by Curves

The EPSCR command specifies a dissipation rate at all nodes associated with a pattern of curves.

(Refer to the EPSPT command for syntax and details.)

EPSSF

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Define by Surfaces

The EPSSF command specifies a dissipation rate at all nodes associated with a pattern of surfaces.

(Refer to the EPSPT command for syntax and details.)

EPSCT

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Define by Contours

The EPSCT command specifies a dissipation rate at all nodes associated with a pattern of contours.

(Refer to the EPSPT command for syntax and details.)

EPSRG

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Define by Regions

The EPSRG command specifies a dissipation rate at all nodes associated with a pattern of regions.

(Refer to the EPSPT command for syntax and details.)

EPSNDEL

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Delete by Nodes

The EPSNDEL command deletes previously defined dissipation rates at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `EPSNDEL, 1, 3, 1`
This command deletes the dissipation rates applied at nodes 1, 2 and 3.

EPSPDEL

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Delete by Points

The EPSPDEL command deletes previously defined dissipation rates at all nodes associated with a pattern of keypoints.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `EPSPDEL, 1, 3, 1`
This command deletes dissipation rates applied at keypoints 1, 2 and 3.

EPSCDEL

Geo Panel: **LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Delete by Curves**

The EPSCDEL command deletes previously defined dissipation rates at all nodes associated with a pattern of curves.

(Refer to the EPSPDEL command for syntax and details.)

EPSSDEL

Geo Panel: **LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Delete by Surfaces**

The EPSSDEL command deletes previously defined dissipation rates at all nodes associated with a pattern of surfaces.

(Refer to the EPSPDEL command for syntax and details.)

EPSCTDEL

Geo Panel: **LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Delete by Contours**

The EPSCTDEL command deletes previously defined dissipation rates at all nodes associated with a pattern of contours.

(Refer to the EPSPDEL command for syntax and details.)




EPSRDEL

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Delete by Regions

The EPSRDEL command deletes previously defined dissipation rates at all nodes associated with a pattern of regions.

(Refer to the EPSPDEL command for syntax and details.)

EPSPLOT

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > Plot

The EPSPLOT command plots a symbol at nodes with prescribed nodal dissipation rates for a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is max. node number defined)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `EPSPLOT, ,6 , ,`

The above command plots a predefined symbol at nodes 1 through 6 for which a dissipation rate was prescribed.

EPSLIST

Geo Panel: LoadsBC > FLUID_FLOW > DISSIPATION_RATE > List

The EPSLIST command lists prescribed nodal dissipation rates for a pattern of nodes.

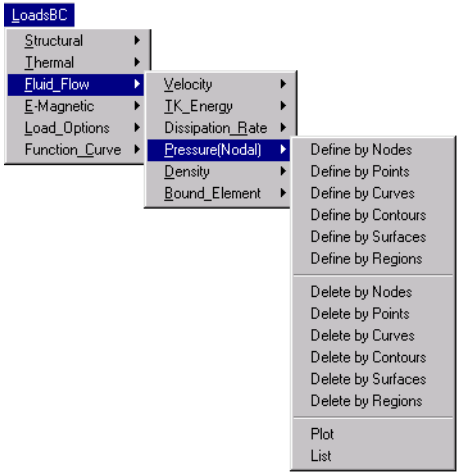
(Refer to the EPSPLOT command for syntax and details.)




▼PRESSURE Menu

Geo Panel: **LoadsBC > FLUID_FLOW > PRESSURE**

Figure 7-23 Pressure Menu



NPRND

Geo Panel: **LoadsBC > FLUID_FLOW > PRESSURE > Define by Nodes**

The NPRND command specifies pressure boundary conditions for a pattern of nodes. This command is only used by the FLOWPLUS module. For FLOWSTAR, pressure is specified on elements through the EP family of commands.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Value
Value of pressure.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `NPRND, 1, 5.0, 3, 1`
This command specifies 5.0 units of pressure at nodes 1, 2 and 3.

NPRPT

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Define by Points

The NPRPT command specifies pressure for nodes associated with a pattern of keypoints. This command is only used by the FLOWPLUS module. For FLOWSTAR, pressure is specified on elements through the EP family of commands.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Value
Value of pressure.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between the keypoints in the pattern.
(default is 1)

Example:
```
NPRPT, 1, 5.0, 3, 1,
```
This command specifies 5.0 units of pressure at nodes associated with keypoints 1, 2 and 3.

NPRCR

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Define by Curves

The NPRCR command specifies pressure for nodes associated with a pattern of curves. This command is only used by the FLOWPLUS module. For FLOWSTAR, pressure is specified on elements through the EP family of commands.

(Refer to the NPRPT command for syntax and details.)

NPRCT

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Define by Contours

The NPRCT command specifies pressure for nodes associated with a pattern of contours. This command is only used by the FLOWPLUS module. For FLOWSTAR, pressure is specified on elements through the EP family of commands.

(Refer to the NPRPT command for syntax and details.)

NPRSF

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Define by Surfaces

The NPRSF command specifies pressure for nodes associated with a pattern of surfaces. This command is only used by the FLOWPLUS module. For FLOWSTAR, pressure is specified on elements through the EP family of commands.

(Refer to the NPRPT command for syntax and details.)

NPRRG

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Define by Regions

The NPRRG command specifies pressure for nodes associated with a pattern of regions. This command is only used by the FLOWPLUS module. For FLOWSTAR, pressure is specified on elements through the EP family of commands.

(Refer to the NPRPT command for syntax and details.)

NPRNDEL

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Delete by Nodes

The NPRNDEL command deletes previously defined pressure values at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `NPRNDEL, 1, 3, 1`
This command deletes prescribed pressure at nodes 1, 2 and 3.




NPRPDEL

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Delete by Points

The NPRPDEL command deletes previously defined pressure values at nodes associated with a pattern of keypoints.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `NPRPDEL, 1, 3, 1`

This command deletes prescribed pressure applied at nodes associated with keypoints 1, 2 and 3.

NPRCDEL

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Delete by Curves

The NPRCDEL command deletes previously defined pressure values at nodes associated with a pattern of curves.

(Refer to the NPRPDEL command for syntax and details.)

NPRCTDEL

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Delete by Contours

The NPRCTDEL command deletes previously defined pressure values at nodes associated with a pattern of contours.

(Refer to the NPRPDEL command for syntax and details.)




NPRSDEL

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Delete by Surfaces

The NPRSDEL command deletes previously defined pressure values at nodes associated with a pattern of surfaces.

(Refer to the NPRPDEL command for syntax and details.)

NPRRDEL

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Delete by Regions

The NPRRDEL command deletes previously defined pressure values at nodes associated with a pattern of regions.

(Refer to the NPRPDEL command for syntax and details.)

NPRPLOT

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > Plot

The NPRPLOT command plots a predefined symbol at nodes with prescribed pressure values for a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is max. node number defined)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `NPRPLOT, ,6 , ,`

The above command plots a predefined symbol at nodes 1 through 6 for which pressure was prescribed.

NPRLIST

Geo Panel: LoadsBC > FLUID_FLOW > PRESSURE > List

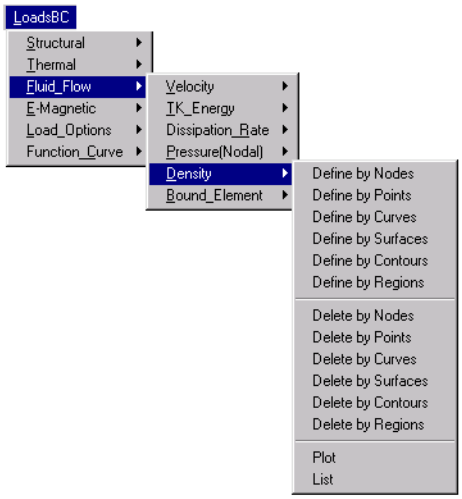
The NPRLIST command lists prescribed pressure for a pattern of nodes.

(Refer to the NPRPLOT command for syntax and details.)

▼DENSITY Menu

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY**

Figure 7-24 Density Menu



DNSND

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > Define by Nodes**

The DNSND command specifies a value for the density at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Value
Value of density.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `DNSND, 1, 5.0, 3, 1`

This command specifies a value of 5.0 units for the density at nodes 1, 2 and 3.

DNSPT

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > Define by Points**

The DNSPT command specifies the density at all nodes associated with a pattern of keypoints.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Value
Value of density.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between the keypoints in the pattern.
(default is 1)

Example: `DNSPT, 1, 5.0, 3, 1,`

This command specifies 5.0 units of density at nodes associated with keypoints 1, 2 and 3.

DNSCR

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > Define by Curves**

The DNSCR command specifies a value for the density at all nodes associated with a pattern of curves.

(Refer to the DNSPT command for syntax and details.)

DNSSF

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > Define by Surfaces**

The DNSSF command specifies a value for the density at all nodes associated with a pattern of surfaces.

(Refer to the DNSPT command for syntax and details.)




DNSCT

Geo Panel: LoadsBC > FLUID_FLOW > DENSITY > Define by Contours

The DNSCT command specifies a value for the density at all nodes associated with a pattern of contours.

(Refer to the DNSPT command for syntax and details.)

DNSRG

Geo Panel: LoadsBC > FLUID_FLOW > DENSITY > Define by Regions

The DNSRG command specifies a value for the density at all nodes associated with a pattern of regions.

(Refer to the DNSPT command for syntax and details.)

DNSNDEL

Geo Panel: LoadsBC > FLUID_FLOW > DENSITY > Delete by Nodes

The DNSNDEL command deletes previously defined density values at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `DNSNDEL, 1, 3, 1`

This command deletes the predefined density values at nodes 1, 2 and 3.

DNSPDEL

Geo Panel: LoadsBC > FLUID_FLOW > DENSITY > Delete by Points

The DNSPDEL command deletes previously defined density values at all nodes associated with a pattern of keypoints.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `DNSPDEL, 1, 3, 1`
This command deletes the density applied at keypoints 1, 2 and 3.

DNSCDEL

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > Delete by Curves**

The DNSCDEL command deletes previously defined density values at all nodes associated with a pattern of curves.

(Refer to the DNSPDEL command for syntax and details.)

DNSSDEL

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > Delete by Surfaces**

The DNSSDEL command deletes previously defined density values at all nodes associated with a pattern of surfaces.

(Refer to the DNSPDEL command for syntax and details.)

DNSCTDEL

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > Delete by Contours**

The DNSCTDEL command deletes previously defined density values at all nodes associated with a pattern of contours.

(Refer to the DNSPDEL command for syntax and details.)

DNSRDEL

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > Delete by Regions**

The DNSRDEL command deletes previously defined density values at all nodes associated with a pattern of regions.

(Refer to the DNSPDEL command for syntax and details.)

DNSPLOT

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > Plot**

The DNSPLOT command plots a predefined symbol at nodes with prescribed density values for a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is max. node number defined)

Increment
Increment between the nodes in the pattern.
(default is 1)

Example: `DNSPLOT, ,6 , ,`

The above command plots a predefined symbol at nodes 1 through 6 for which density was prescribed.

DNSLIST

Geo Panel: **LoadsBC > FLUID_FLOW > DENSITY > List**

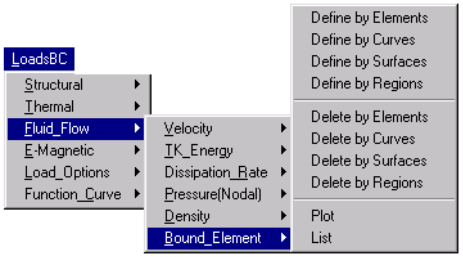
The DNSLIST command lists prescribed density values for a pattern of nodes.

(Refer to the DNSPLOT command for syntax and details.)

▼BOUND_ELEMENT Menu

Geo Panel: **LoadsBC > FLUID_FLOW > BOUND_ELEMENT**

Figure 7-25 Bound_Element Menu



BEL

Geo Panel: **LoadsBC > FLUID_FLOW > BOUND_ELEMENT > Define by Elements**

For compressible Euler fluid flow analysis (FLOWSTAR), the BEL command specifies zero normal velocity on faces of a pattern of elements.

For electromagnetic analysis (ESTAR), the BEL command applies infinite boundaries at the specified faces of a pattern of elements. Infinite elements are only available for electro-static and magnetostatic analyses.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Face number
Face of the elements to which infinite (Euler) boundary is applied.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Note
The node labels (local) on each face are as follows:

Face Number	Local Node Labels	
1	1	2
2	2	3
3	3	4
4	4	1

Example: `BEL, 3, 2, 10, 1`

For FLOWSTAR, this command applies Euler boundary condition on face 2 of elements 3 through 10.

For ESTAR, this command applies infinite boundaries to face 2 of elements 3 through 10.

BECR

Geo Panel: **LoadsBC > FLUID_FLOW > BOUND_ELEMENT > Define by Curves**

For fluid flow analysis (FLOWSTAR), the BECR command specifies Euler boundary condition (zero normal velocity) to faces of elements associated with a pattern of curves.

For electromagnetic analysis (ESTAR), the BECR command applies infinite boundaries to MAG2D elements associated with a pattern of curves. Infinite elements are only available for electrostatic and magnetostatic analyses.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between the curves in the pattern.
(default is 1)

Example: `BECR, 1, 3, 1`

For FLOWSTAR, this command applies Euler boundary condition to elements associated with curves 1, 2 and 3.

For ESTAR, this command applies infinite boundaries at all faces of MAG2D elements associated with curves 1, 2 and 3.

BESF

Geo Panel: LoadsBC > FLUID_FLOW > BOUND_ELEMENT > Define by Surfaces

For 3D electromagnetic analysis (ESTAR), the BESF command applies infinite boundaries to the faces of MAG3D elements associated with a pattern of surfaces. Infinite elements are only available for electrostatic and magnetostatic analyses.

Entry & Option Description

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between the surfaces in the pattern.
(default is 1)

Example: `BESF, 1, 3, 1`

This command applies infinite boundaries to all faces of MAG3D elements associated with surfaces 1, 2 and 3.

BERG

Geo Panel: LoadsBC > FLUID_FLOW > BOUND_ELEMENT > Define by Regions

For 3D electromagnetic analysis (ESTAR), the BERG command applies infinite boundaries to the faces of MAG3D elements associated with a pattern of regions. Infinite elements are only available for electrostatic and magnetostatic analyses.

Entry & Option Description

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Example: `BERG, 1, 3, 1`

This command applies infinite boundaries to all faces of MAG3D elements associated with regions 1, 2 and 3.

BEDEL

Geo Panel: LoadsBC > FLUID_FLOW > BOUND_ELEMENT > Delete by Elements

For compressible Euler fluid flow analysis (FLOWSTAR), the BEDEL command deletes the zero normal velocity conditions on the specified face for a pattern of elements.

For electromagnetic analysis (ESTAR), the BEDEL command deletes infinite boundaries at the specified face of a pattern of elements.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Face number
Face number.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Note

Face numbering is as shown in the BEL command.

Example: `BEDEL, 3, 2, 10, 1`

For compressible Euler flow, this command deletes the Euler boundary condition on face 2 of elements 3 through 10.

For electromagnetic analysis, this command deletes the infinite boundary condition on face 2 of elements 3 through 10.

BECDEL

Geo Panel: LoadsBC > FLUID_FLOW > BOUND_ELEMENT > Delete by Curves

For compressible Euler fluid flow, the BECDEL command deletes previously defined Euler boundary condition for all elements associated with a pattern of curves.

For electromagnetic analysis, the BECDEL command deletes previously defined infinite boundaries applied to faces of elements associated with a pattern of curves.

Entry & Option Description

Beginning curve

Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Example: `BECDEL, 1, 10, 1`

For compressible Euler flow, this command deletes the Euler boundary condition on elements associated with curves 1 through 10.

For electromagnetic analysis, this command deletes infinite boundaries applied to elements associated with curves 1 through 10.

BESDEL

Geo Panel: **LoadsBC > FLUID_FLOW > BOUND_ELEMENT > Delete by Surfaces**

For electromagnetic analysis, the BESDEL command deletes previously defined infinite boundaries applied to faces of elements associated with a pattern of surfaces.

Entry & Option Description

Beginning surface
Beginning surface in the pattern.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)




Example: `BESDEL, 1, 10, 1`

This command deletes infinite boundaries applied to all faces of MAG3D elements associated with surfaces 1 through 10.

BERDEL

Geo Panel: **LoadsBC > FLUID_FLOW > BOUND_ELEMENT > Delete by Regions**

For electromagnetic analysis, the BERDEL command deletes previously defined infinite boundaries applied to faces of elements associated with a pattern of regions.

Entry & Option Description

Beginning region
Beginning region in the pattern.

Ending region
Ending region in the pattern.
(default is beginning region)

Increment
Increment between regions in the pattern.
(default is 1)

Example: `BERDEL, 1, 5, 3`

This command deletes infinite boundaries applied to all faces of MAG3D elements associated with regions 1 and 4.

BEPLOT

Geo Panel: **LoadsBC > FLUID_FLOW > BOUND_ELEMENT > Plot**

For compressible Euler flow, the BEPLOT command plots a symbol on element faces with Euler boundary condition.

For electromagnetic analysis, the BEPLOT command plots a symbol on element faces with infinite boundaries.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is the maximum element number defined)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `BEPLOT;`

For compressible fluid flow, the above command plots a predefined symbol on all faces of elements with prescribed Euler boundary condition.

For electromagnetic analysis, the above command plots a predefined symbol on all faces of elements with infinite boundaries.

BELIST

Geo Panel: **LoadsBC > FLUID_FLOW > BOUND_ELEMENT > List**

For compressible Euler flow, the BELIST command lists element faces with Euler boundary condition (zero normal velocity).

For electromagnetic analysis, the BELIST command lists element faces with infinite boundaries.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is the maximum element number defined)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `BELIST, 1, 6, 2,`

For compressible fluid flow, the above command lists all faces of elements with Euler boundary condition (zero normal velocity) for elements 1, 3, and 5.

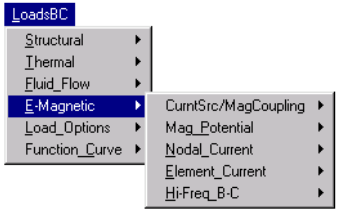
For electromagnetic analysis, the above command lists all faces with infinite boundaries for elements 1, 3, and 5.

▶ E-MAGNETIC Menu

Geo Panel: **LoadsBC > E-MAGNETIC**

This menu contains commands to specify electromagnetic loading and boundary conditions like current, magnetic potential, etc.

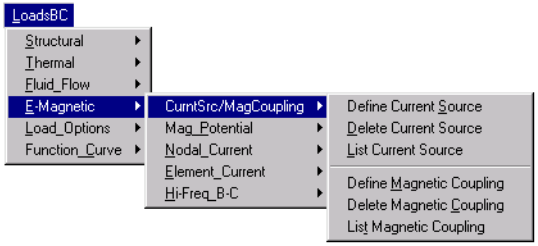
Figure 7-26 E-Magnetic Menu



▼CURNT SRC/MAG COUPLING Menu

Geo Panel: **LoadsBC > E-MAGNETIC > Curnt Src/Mag Coupling**

Figure 7-27 Curnt Src/Mag Coupling Menu



JSDEF

Geo Panel: **LoadsBC > E-MAGNETIC > Curnt Src/Mag Coupling > Define Current Source**

The JSDEF command defines 3D current sources for magnetostatic analysis.

Entry & Option Description

Current source number
Source number.
(default is highest source defined +1)

Source type
Source type.
= Cyl cylindrical coil
= Arc arc type coil
= Bar bar type current source
(default is cylindrical)

Local coordinate system label

Label of corresponding local coordinate system. (must be greater than 2)

Radial thickness/length in x-direction
Radial thickness for source types 1 and 2, or thickness in the x-direction for source type 3.

Axial thickness/length in y-direction
Axial thickness for source types 1 and 2, or thickness in the y-direction for source type 3.

Mean radius/length of bar
Mean radius for source types 1 and 2, or length of the bar in the z-direction for source type 3.

Intgr points in radial/x-direction
Number of integration points along radial thickness/length in x-direction.

Intgr points in axial/y-direction
Number of integration points along axial thickness/length in y-direction.

Applied current value
Current amplitude in terms of Amp_turns or Abamp_turns.

Angle of the arc
Angle of the arc in degrees. (applicable only to type 2 source)
(default is 90 degrees)

Note

Every 3D current source should be associated with a defined local coordinate system to determine its orientation. Refer to Figure 7-28 for convention used in defining current sources. Symbols for 3D current sources are also plotted on the screen.

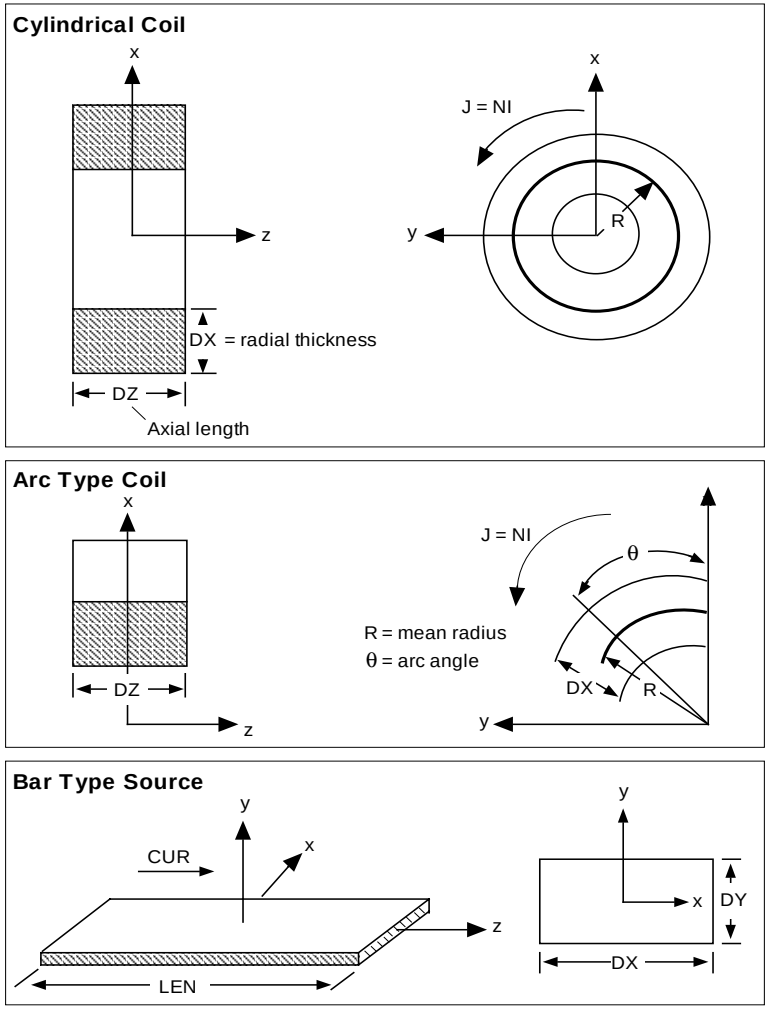
Example:

```
JSDEF, 2, 1, 3, 1., 5., 2., 4, 12, 10.
```

The above command defines a cylindrical coil with label 2, its orientation is given by local coordinate system number 3, the radial thickness is 1.0, the axial length is 5.0, and the mean radius is 2.0. Integration points 4 and 12 are to be chosen along the radial and axial directions, respectively. The current amplitude is 10.0 amp_turns.




Figure 7-28



JSDEL

Geo Panel: **LoadsBC > E-MAGNETIC > Curnt Src/Mag Coupling > Delete Current Source**

The JSDEL command deletes a pattern of 3D current sources from the database.

Entry & Option Description

Beginning current source number
Beginning current source in the pattern.

Ending current source number
Ending current source in the pattern.
(default is beginning current source number)

Increment
Increment between sources in the pattern.
(default is 1)

Example: `JSDEL, 1, 5, 2`

This command deletes 3D current sources 1, 3, and 5 from the database.

JSLIST

Geo Panel: **LoadsBC > E-MAGNETIC > Curnt Src/Mag Coupling > List Current Source**

The JSLIST command lists information about a pattern of defined 3D current sources on the screen.

(Refer to the JSDEL command for syntax.)

MCPDEF

Geo Panel: **LoadsBC > E-MAGNETIC > Curnt Src/Mag Coupling > Define Magnetic Coupling**

The MCPDEF command equates the magnetic potential values at the corresponding nodes in two patterns.

Entry & Option Description

Pattern 1 beginning node
Beginning node in the first pattern.

Pattern 1 ending node
Ending node in the first pattern.

Pattern 1 increment
Increment between nodes in the first pattern.
(default is 1)

Pattern 2 beginning node
Beginning node in the second pattern.

Pattern 2 ending node
Ending node in the second pattern.

Pattern 2 increment
Increment between nodes in the second pattern.
(default is 1)

Example: `MCPDEF, 3, 9, 3, 12, 14, 1`
This command instructs the program to equate magnetic potential at node 3 to that at node 12, 6 to 13, and 9 to 14.

MCPDEL

Geo Panel: **LoadsBC > E-MAGNETIC > Curnt Src/Mag Coupling > Delete Magnetic Coupling**

The MCPDEL command deletes magnetic coupling for a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `MCPDEL, 3, 9, 3,`
This command deletes magnetic potential coupling at node 3, 6 and 9.

MCPLIST

Geo Panel: **LoadsBC > E-MAGNETIC > Curnt Src/Mag Coupling > List Magnetic Coupling**

The MCPLIST command lists magnetic potential coupling at a pattern of nodes.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.

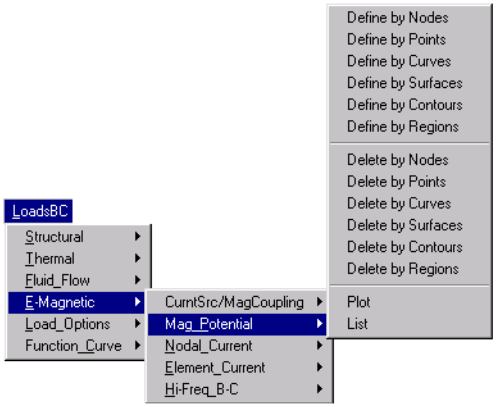
Increment
Increment between nodes in the pattern.
(default is 1)

Example: `MCPLIST, 3, 9, 3,`
This command lists magnetic potential coupling at node 3, 6 and 9.

▼MAG_POTENTIAL Menu

Geo Panel: **LoadsBC > E-MAGNETIC > MAG_POTENTIAL**

Figure 7-29 Mag_Potential Menu



NPND

Geo Panel: **LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Define by Nodes**

The NPND command defines voltage or magnetic potential input at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Value
Potential value.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `NPND,1 ,5.0 ,3 ,1 ,`
This command specifies 5.0 units of voltage or magnetic potential at nodes 1, 2 and 3.

NPPT

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Define by Points

The NPPT command specifies voltage or magnetic potential input at all nodes associated with any of the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Value
Potential value.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example:
```
NPPT, 1, -5.0, 3, 1,
```
This command specifies -5.0 units of voltage or magnetic potential at all nodes associated with keypoints 1, 2 or 3.

NPCR

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Define by Curves

The NPCR command specifies electric or magnetic potential input at all nodes associated with any of the curves in the specified pattern.

(Refer to the NPPT command for syntax and details.)

NPSF

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Define by Surfaces

The NPSF command specifies electric or magnetic potential input at all nodes associated with any of the surfaces in the specified pattern.

(Refer to the NPPT command for syntax and details.)




NPCT

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Define by Contours

The NPCT command specifies voltage or magnetic potential input at all nodes associated with any of the contours in the specified pattern.

(Refer to the NPPT command for syntax and details.)

NPRG

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Define by Regions

The NPRG command specifies voltage or magnetic potential input at all nodes associated with any of the regions in the specified pattern.

(Refer to the NPPT command for syntax and details.)

NPNDEL

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Delete by Nodes

The NPNDEL command deletes previously defined electric or magnetic potential at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `NPNDEL, 1, 3, 1`

This command deletes voltage or magnetic potential input at nodes 1, 2 and 3.

NPPDEL

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Delete by Points

The NPPDEL command deletes previously defined electric or magnetic potential at all nodes associated with any of the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `NPPDEL, 1, 3, 1`

This command deletes voltage or magnetic potential input at all nodes associated with keypoints 1, 2 or 3.

NPCDEL

Geo Panel: **LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Delete by Curves**

The NPCDEL command deletes previously defined electric or magnetic potentials at all nodes associated with any of the curves in the specified pattern.

(Refer to the NPPDEL command for syntax and details.)

NPSDEL

Geo Panel: **LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Delete by Surfaces**

The NPSDEL command deletes previously defined electric or magnetic potentials at all nodes associated with any of the surfaces in the specified pattern.

(Refer to the NPPDEL command for syntax and details.)

NPCTDEL

Geo Panel: **LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Delete by Contours**

The NPCTDEL command deletes previously defined electric or magnetic potential at all nodes associated with any of the contours in the specified pattern.

(Refer to the NPPDEL command for syntax and details.)




NPRDEL

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Delete by Regions

The NPRDEL command deletes previously defined electric or magnetic potential at all nodes associated with any of the regions in the specified pattern.

(Refer to the NPPDEL command for syntax and details.)

NPPLOT

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > Plot

The NPPLOT command plots a predefined symbol at nodes with specified voltage or magnetic potential input.

Entry & Option Description

Beginning node
Beginning node of the pattern.

Ending node
Ending node of the pattern.
(default is the maximum node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `NPPLOT;`

The above command plots a prescribed symbol at all nodes with prescribed voltage or magnetic potentials.

NPLIST

Geo Panel: LoadsBC > E-MAGNETIC > MAG_POTENTIAL > List

The NPLIST command lists currently defined nodal electric or magnetic potential for a specified pattern of nodes.

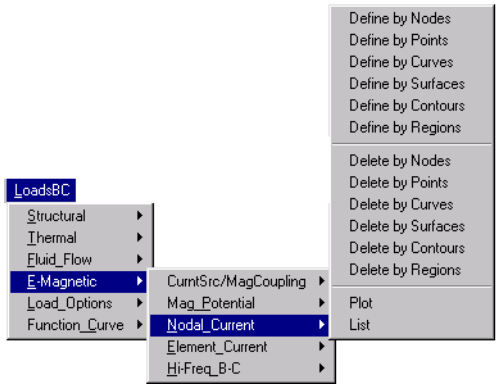
(Refer to the NPPLOT command for syntax.)




▼NODAL_CURRENT Menu

Geo Panel: **LoadsBC > E-MAGNETIC > NODAL_CURRENT**

Figure 7-30 Nodal_Current Menu



NJND

Geo Panel: **LoadsBC > E-MAGNETIC > NODAL_CURRENT > Define by Nodes**

The NJND command defines current input at the nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Value
Current value.

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Note
Nodal currents are not applicable in 3D magnetostatic analysis.

Example: `NJND,1 ,5.0 ,3 ,1 ,`
This command specifies 5.0 units of current at nodes 1, 2 and 3.

NJPT

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > Define by Points

The NJPT command specifies current input at all nodes associated with any of the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.
(default is 1)

Value
Current value.
(default is 0.0)

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Note
Nodal currents are not applicable in 3D magnetostatic analysis.

Example:
```
NJPT, 1, -5.0, 3, 1,
```
This command specifies -5.0 units of current at all nodes associated with keypoints 1, 2 or 3.

NJCR

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > Define by Curves

The NJCR command specifies current input at all nodes associated with any of the curves in the specified pattern.

(Refer to the NJPT command for syntax and details.)

NJSF

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > Define by Surfaces

The NJSF command specifies current input at all nodes associated with any of the surfaces in the specified pattern.

(Refer to the NJPT command for syntax and details.)

NJCT

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > Define by Contours

The NJCT command specifies current input at all nodes associated with any of the contours in the specified pattern.

(Refer to the NJPT command for syntax and details.)

NJRG

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > Define by Regions

The NJRG command specifies current input at all nodes associated with any of the regions in the specified pattern.

(Refer to the NJPT command for syntax and details.)

NJNDEL

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > Delete by Nodes

The NJNDEL command deletes previously defined currents at all nodes in the specified pattern.

Entry & Option Description

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is beginning node)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `NJNDEL, 1, 3, 1`
This command deletes current input at nodes 1, 2 and 3.

NJPDEL

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > Delete by Points

The NJPDEL command deletes previously defined currents at all nodes associated with any of the keypoints in the specified pattern.

Entry & Option Description

Beginning keypoint
Beginning keypoint in the pattern.
(default is 1)

Ending keypoint
Ending keypoint in the pattern.
(default is beginning keypoint)

Increment
Increment between keypoints in the pattern.
(default is 1)

Example: `NJPDEL, 1, 3, 1`

This command deletes current input at all nodes associated with keypoints 1, 2 or 3.

NJCDEL

Geo Panel: **LoadsBC > E-MAGNETIC > NODAL_CURRENT > Deletes by Curves**

The NJCDEL command deletes previously defined currents at all nodes associated with any of the curves in the specified pattern.

(Refer to the NJPDEL command for syntax.)

NJSDEL

Geo Panel: **LoadsBC > E-MAGNETIC > NODAL_CURRENT > Deletes by Surfaces**

The NJSDEL command deletes previously defined currents at all nodes associated with any of the surfaces in the specified pattern.

(Refer to the NJPDEL command for syntax.)

NJCTDEL

Geo Panel: **LoadsBC > E-MAGNETIC > NODAL_CURRENT > Deletes by Contours**

The NJCTDEL command deletes previously defined currents at all nodes associated with any of the contours in the specified pattern.

(Refer to the NJPDEL command for syntax.)




NJRDEL

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > Deletes by Regions

The NJRDEL command deletes previously defined currents at all nodes associated with any of the regions in the specified pattern.

(Refer to the NJPDEL command for syntax.)

NJPLOT

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > Plot

The NJPLOT command plots a predefined symbol at nodes with specified current input.

Entry & Option Description

Beginning node
Beginning node of the pattern.
(default is 1)

Ending node
Ending node of the pattern.
(default is the maximum node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `NJPLOT;`
The above command plots a predefined symbol at nodes with prescribed current input.

NJLIST

Geo Panel: LoadsBC > E-MAGNETIC > NODAL_CURRENT > List

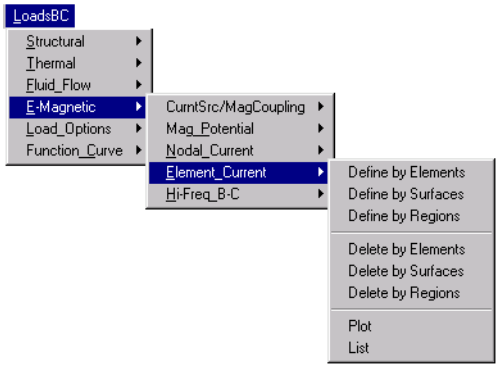
The NJLIST command lists currently defined nodal currents for a specified pattern of nodes.

(Refer to the NJPLOT command for syntax.)

▼ELEMENT_CURRENT Menu

Geo Panel: **LoadsBC > E-MAGNETIC > ELEMENT_CURRENT**

Figure 7-31 Element_Current Menu



JEL

Geo Panel: **LoadsBC > E-MAGNETIC > ELEMENT_CURRENT > Define by Elements**

The JEL command defines current or charge density for all elements in the specified pattern.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Value
Current density.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Note

Current densities for 3D magnetostatic analysis cannot be specified through this command. 3D current densities may be defined through JSDEF command.

Example: `JEL,1 ,5. ,3 ,1 ,`

This command specifies 5. units of current or charge density for elements 1, 2 and 3.

JESF

Geo Panel: **LoadsBC > E-MAGNETIC > ELEMENT_CURRENT > Define by Surfaces**

The JESF command specifies current or charge density for all elements associated with any of the surfaces in the specified pattern.

Entry & Option Description

Beginning surface
Beginning surface in the pattern.
(default is 1)

Value
Current density.

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Note

Current densities for 3D magnetostatic analysis cannot be specified through this command. 3D current densities may be defined through JSDEF command.

Example:
```
JESF, 1, -5.0, 3, 1,
```
This command specifies -5.0 units of current or charge density for all elements associated with surfaces 1, 2 or 3.

JERG

Geo Panel: **LoadsBC > E-MAGNETIC > ELEMENT_CURRENT > Define by Regions**

The JERG command specifies current or charge density for all elements associated with any of the regions in the specified pattern.

(Refer to the JESF command for syntax.)




JEDEL

Geo Panel: LoadsBC > E-MAGNETIC > ELEMENT_CURRENT > Delete by Elements

The JEDEL command deletes previously defined current or charge densities for all elements in the specified pattern.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `JEDEL, 1, 3, 1`
This command deletes current or charge density at elements 1, 2 and 3.

JESDEL

Geo Panel: LoadsBC > E-MAGNETIC > ELEMENT_CURRENT > Delete by Surfaces

The JESDEL command deletes previously defined current or charge densities for all elements associated with any of the surfaces in the specified pattern.

Entry & Option Description

Beginning surface
Beginning surface in the pattern.
(default is 1)

Ending surface
Ending surface in the pattern.
(default is beginning surface)

Increment
Increment between surfaces in the pattern.
(default is 1)

Example: `JESDEL, 1, 3, 1`
This command deletes current or charge density for all elements associated with surfaces 1, 2 or 3.

JERDEL

Geo Panel: LoadsBC > E-MAGNETIC > ELEMENT_CURRENT > Delete by Regions

The JERDEL command deletes previously defined current or charge densities at all elements associated with any of the regions in the specified pattern.

(Refer to the JESDEL command for syntax.)

JEPLOT

Geo Panel: LoadsBC > E-MAGNETIC > ELEMENT_CURRENT > Plot

The JEPLOT command plots a predefined symbol at elements with specified current or charge density.

Entry & Option Description

Beginning element
Beginning element of the pattern.
(default is 1)

Ending element
Ending element of the pattern.
(default is the maximum element number defined)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `JEPLOT;`

The above command plots a predefined symbol at elements with specified current or charge densities.

JELIST

Geo Panel: LoadsBC > E-MAGNETIC > ELEMENT_CURRENT > List

The JELIST command lists defined current or charge densities for a specified pattern of elements.

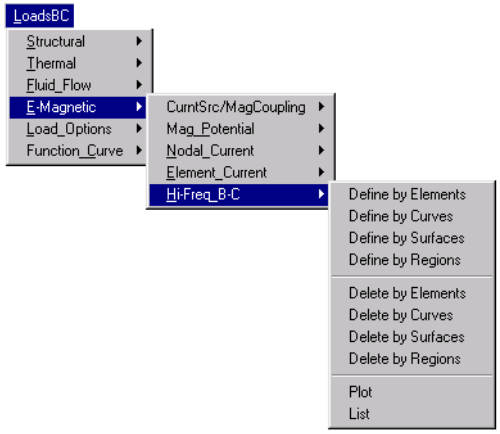
(Refer to the JEPLOT command for syntax.)

▼HI-FREQ B-C_Menu

Geo Panel: LoadsBC > E-MAGNETIC > HI-FREQ_B-C

This menu specifies boundary conditions related to the high-frequency electromagnetics modules.

Figure 7-32 Hi-Freq_B-C Menu



CBEL

Geo Panel: LoadsBC > E-MAGNETIC > HI-FREQ_B-C > Define by Elements

The CBEL command specifies an HF boundary condition for faces of elements in the specified pattern.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Boundary condition type
Boundary condition type.
= fc floating conductor
= gc grounded conductor
= pmc perfect magnetic conductor
= oob open outer boundary
= port port for S-parameters problems
(default is fc)

Conductor number
Conductor number associated with the boundary condition.

Conductivity value
Conductivity of the conductor number.
(default is 5.8e7)

Relative permeability value
Relative permeability of the conductor number.

Face number
Face of the elements on which the HF boundary condition is to be applied.

Ending element
Ending element in the pattern.

Increment
Increment between elements in the pattern.

Note
"Port Number" (1 to 99) is prompted if "Port" is selected for Boundary condition type".

Example:
```
CBEL, 4, fc, 2,,, 5 ,3, 5,,
```
This command defines a floating conductor number 2 on face number 5 of elements 4 and 5 using the default conductivity and permeability (copper).

CBCR

Geo Panel: **LoadsBC > E-MAGNETIC > HI-FREQ_B-C > Define by Curves**

The CBCR command defines an HF boundary condition on a pattern of curves.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Boundary condition
Boundary condition type.
= fc floating conductor
= gc grounded conductor
= pmc perfect magnetic conductor
= oob open outer boundary
= port port for S-parameters problems
(default is fc)

Conductor number
Conductor number associated with the boundary condition.

Conductivity value
Conductivity of the conductor number.

Relative permeability value
Relative permeability of the conductor number.

Ending curve

Ending curve in the pattern.

Increment
Increment in curve numbering.

Example: `CBCR, 2, fc, 1,,, 2,,`
This command defines a floating conductor on curve two of default conductivity and permeability (copper).

CBSF

Geo Panel: **LoadsBC > E-MAGNETIC > HI-FREQ_B-C > Define by Surfaces**

The CBSF command defines a boundary condition on a pattern of surfaces.

(Refer to the CBCR command for syntax and details.)

CBRG

Geo Panel: **LoadsBC > E-MAGNETIC > HI-FREQ_B-C > Define by Regions**

The CBRG command specifies an HF boundary condition on a pattern of regions.

(Refer to the CBCR command for syntax and details.)

CBEDEL

Geo Panel: **LoadsBC > E-MAGNETIC > HI-FREQ_B-C > Delete by Elements**

The CBEDEL command deletes previously defined HF boundary conditions for the specified face for a pattern of elements.

Entry & Option Description

Beginning element
Beginning element in the pattern.

Face number
Face number of the elements for which existing HF boundary condition is to be deleted.

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `CBEDEL, 3, 2, 10, 1`
This command deletes the boundary conditions on face 2 of elements 3 through 10.

CBCDEL

Geo Panel: LoadsBC > E-MAGNETIC > HI-FREQ_B-C > Delete by Curves

The CBCDEL command deletes previously defined HF boundary conditions for elements associated with a pattern of curves.

Entry & Option Description

Beginning curve
Beginning curve in the pattern.

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Example: `CBCDEL, 1, 10, 1`
This command deletes HF boundary conditions for elements associated with curves 1 through 10.

CBSDEL

Geo Panel: LoadsBC > E-MAGNETIC > HI-FREQ_B-C > Delete by Surfaces

The CBSDEL command deletes previously defined HF boundary conditions for elements associated with a pattern of surfaces.

(Refer to the CBCDEL command for syntax and details.)

CBRDEL

Geo Panel: LoadsBC > E-MAGNETIC > HI-FREQ_B-C > Delete by Regions

The CBRDEL command deletes previously defined HF boundary conditions for elements associated with a pattern of regions.

(Refer to the CBCDEL command for syntax and details.)

CBPLOT

Geo Panel: LoadsBC > E-MAGNETIC > HI-FREQ_B-C > Plot

The CBPLOT command plots a predefined symbol at elements with prescribed HF boundary conditions for a pattern of elements. The symbol is shown in the STATUS2 Table.

Entry & Option Description

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is max. element number defined)

Increment
Increment between elements in the pattern.
(default is 1)

Example: `CBPLOT;`

The above command plots a predefined symbol at elements with prescribed HF boundary conditions.

CBLIST

Geo Panel: **LoadsBC > E-MAGNETIC > HI-FREQ_B-C > List**

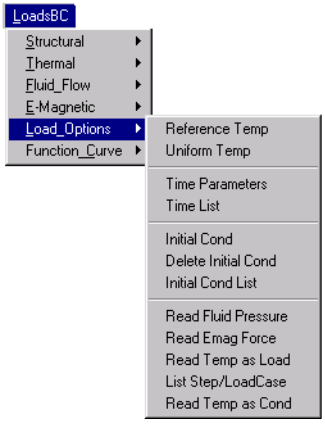
The CBLIST command lists HF element boundary conditions for a pattern of elements.

(Refer to the CBPLOT command for syntax.)

▶ LOAD_OPTIONS Menu

Geo Panel: **LoadsBC > LOAD_ OPTIONS**

Figure 7-33 Load_Options Menu



This menu contains commands to specify initial conditions, reference temperature for structural models, and commands to read loads result-ing from thermal, fluid and electro-magnetic analyses into structural models.

TREF

Geo Panel: **LoadsBC > LOAD_ OPTIONS > Reference Temp)**

The TREF command is used in structural models to define a reference temperature at which no thermal strains exist in the model. This command may also be used to specify reference temperature for natural convection fluid flow problems.

Entry & Option Description

Reference temperature
Reference temperature of the model.
(default is 0.0)

Example: `TREF, 100.0`

This command specifies 100 degrees as the reference temperature for the model. If the model is heated to 200 degrees, then strains will be introduced by the differential temperature (100).

TUNIF

Geo Panel: **LoadsBC > LOAD_OPTIONS > Uniform Temp**

The TUNIF command defines a uniform temperature for all nodes in the model for thermal analysis. The INITIAL command may perform the same function.

Entry & Option Description

Uniform temperature
Uniform temperature assigned to the nodes.
(default is 0.0)

Notes

1. In transient heat transfer analysis problems, the TUNIF command may be used to define the initial temperature of all nodes in the model.

2. In nonlinear steady state heat transfer problems, the TUNIF command defines the initial temperature of the model which is used to define temperature-dependent material properties using the temperature curve. If this command is not issued, material properties at 0.0 temperature are used as starting values.

Example: `TUNIF, 10.0`

For transient heat transfer problems, this command assigns an initial temperature of 10.0 degrees to all the nodes in the model. For nonlinear steady state heat transfer problems, a temperature of 10.0 degrees is used to define the initial material properties using the corresponding temperature curve.

TIMES

Geo Panel: **LoadsBC > LOAD_OPTIONS > Time Parameters**

The TIMES command specifies the starting time, final time and the time increment for nonlinear structural, transient thermal, fluid flow or electromagnetic analyses.

Entry & Option Description

Starting time
Starting solution time.
(default is 0.0 unless the restart option is active in which case the default is the time value at the last successful time step)

Final time
Final solution time.

Time increment
Time increment between two consecutive time steps.

Notes

1. Unit of time should be consistent with other input.
2. The final solution time cannot exceed the largest time in the corresponding curve defined by the CURDEF command.
3. The final solution time must be greater than the starting time.

Example: `TIMES, 0.0, 5.0, 0.1`

This command defines a solution time interval from 0.0 to 5.0 time units with 0.1 increment, i.e. 50 time steps.

TIMELIST

Geo Panel: **LoadsBC > LOAD_OPTIONS > Time List**

The TIMELIST command lists the starting time, ending time, and time increment specified by the TIMES command for nonlinear and transient analyses.

INITIAL

Geo Panel: **LoadsBC > LOAD_OPTIONS > Initial Cond**

The INITIAL command defines initial conditions at nodes in the specified pattern. The valid quantities and their corresponding analyses are listed below.

Entry & Option Description

Initial condition type

Type of initial conditions. Valid types are:

= DISP	displacements (nonlinear dynamic and post-dynamic)
= VEL	velocities (nonlinear dynamic, post-dynamic and fluid flow)
= ACC	accelerations (nonlinear dynamic and post-dynamic)
= TEMP	temperatures (thermal and fluid flow)
= MAGP	voltage or magnetic potentials (electromagnetic)
= PRES	pressure (fluid flow)
= TKE	turbulence kinetic energy (fluid flow)
= EPS	dissipation rate (fluid flow)
= DENS	density
= ENRG	energy
	(default is DISP)

Beginning node

Beginning node in the pattern.
(default is 1)

Ending node

Ending node in the pattern.
(default is highest node number defined)

Increment

Increment between nodes in the pattern.
(default is 1)

X-component

Initial value in the X-direction or the magnitude of a scalar quantity.

Y-component

Initial value in the Y-direction.

Z-component

Initial value in the Z-direction.

Note

For scalar quantities, GEOSTAR prompts for X-component only. Y-component and Z-component are prompted for only in case of vectors.

Example: `INITIAL, DISP, 3, 10, 1, .002, .003, .001`

Initial displacements of .002, .003 and .001 are specified for the X, Y and Z translational components respectively for nodes 3 through 10.

INITDEL

Geo Panel: LoadsBC > LOAD_OPTIONS > Delete Initial Cond

The INITDEL command deletes the specified initial conditions at the specified pattern of nodes.

Entry & Option Description

Initial condition type

Type of initial conditions. Valid types are:

= DISP	displacements
= VEL	velocities
= ACC	accelerations
= TEMP	temperatures
= MAGP	voltage or magnetic potentials
= PRES	pressure
= TKE	turbulence kinetic energy
= EPS	dissipation rate
= DENS	density
= ENRG	energy

(default is DISP)

Beginning node

Beginning node in the pattern.
(default is 1)

Ending node

Ending node in the pattern.
(default is highest node number defined)

Increment

Increment between nodes in the pattern.
(default is 1)

Example: `INITDEL, ACC, 3, 10, 1`

This command deletes the initial accelerations specified at nodes 3 through 10.

INITLIST

Geo Panel: LoadsBC > LOAD_OPTIONS > Initial Cond List

The INITLIST command lists values for the specified type of initial conditions for a pattern of nodes.

Entry & Option Description

Initial condition type

Type of initial conditions. Valid types are:

= DISP	displacements
= VEL	velocities

= ACC accelerations
= TEMP temperatures
= MAGP voltage or magnetic potentials
= PRES pressure
= TKE turbulence kinetic energy
= EPS dissipation rate
= DENS density
= ENRG energy
(default is DISP)

Beginning node
Beginning node in the pattern.
(default is 1)

Ending node
Ending node in the pattern.
(default is highest node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `INITLIST, VEL, 5, 20, 1`
This command lists the specified initial velocities for node 5 through 20.

NPRREAD

Geo Panel: **LoadsBC > LOAD_OPTIONS > Read Fluid Pressure**

The NPRREAD applies the pressure calculated by FLOWSTAR and FLOWPLUS at the given time step on the faces of elements associated with the specified entities.

Entry & Option Description

Time step label
Time step number.
(default is 1)

Entity type
Geometric entity name. Only curves and surfaces can be currently specified.
= Curve curve
= Surface surface
= Region region
(default is curve)

Beginning curve/surface/region
Beginning entity in the pattern.

Ending curve/surface/region
Ending entity in the pattern.
(default is beginning curve/surface/region)

Increment
Increment between entities in the pattern.
(default is 1)

Direction flag
Direction flag.
= Negative negative
= Positive positive (towards element)
(default is positive)

Example: `NPRREAD, 100, CR, 1, 1, 1`

This command applies the pressure calculated by FLOWSTAR or FLOWPLUS at time step 100 to all elements associated with curve 1.

EMFREAD

Geo Panel: **LoadsBC > LOAD_OPTIONS > Read Emag Force**

The EMFREAD command is used for magneto-structural coupling, it provides the structural module with the magnetic nodal forces.

Entry & Option Description

Time step label
Time step at which the magnetic forces are input to the structural analysis.

Example: `EMFREAD, 2`

This command provides the structural analysis module (STAR) with the nodal forces obtained from time step 2 of the magnetic analysis.

TEMPREAD

Geo Panel: **LoadsBC > LOAD_OPTIONS > Read Temp as Load**

The TEMPREAD command assigns temperatures from thermal steady state analysis or transient thermal analysis at a particular time step to the specified load case. Temperatures will be read during analysis for use as thermal loading in structural modules. Note that this command is not needed for using results from transient thermal analysis as thermal loading in NSTAR. All you need to do in that case is to activate thermal loading for NSTAR using the A_NONLINEAR command and start the analysis.

Entry & Option Description

Time step label
Time step number at which nodal temperatures are requested to be assigned. (Use 1 for steady state, use 0 to delete previous assignments.)

Load case number
Load case number.
(default is the active load case)

Notes

1. The special thermal loading flag must be activated in order to consider thermal loading effects. Use A_STATIC for STAR and A_FFESTATIC for FFE STATIC.
2. Temperature profiles may be defined by the TEMPREAD command or through direct nodal temperature specifications for some or all primary load cases.
3. Thermal loading is considered on top of other types of loading such as forces and pressure.
4. If a conflict exists where by some nodal temperatures are directly defined by commands like NTND and the TEMPRERAD command is used for the same load case, then temperatures will be read from the thermal analysis and the directly specified temperature will be ignored. Directly specified temperatures are used in any load case if the TEMPREAD command is not used.
5. Use 0 for the time-step prompt to delete previous assignments.
6. The profile of temperature change is calculated by subtracting the value of the reference temperature defined by the TREF command.

Example 1:
```
R_THERMAL
TEMPREAD, 500, 5,
TEMPREAD, 700, 40,
A_STATIC, T,
R_STATIC
```
The first command runs transient thermal analysis, the second and third commands assign nodal temperatures from time steps 500 and 700 to load cases 5, and 40 respectively. The A_STATIC, T, command activates thermal loading for subsequent linear static analyses using STAR.

Example 2:
```
TEMPREAD, 5, 2,
TEMPREAD 0, 2,
```
The second deletes the assignment made by the first command.

TEMPRDLIST

Geo Panel: **LoadsBC > LOAD_OPTIONS > List Step/Load Case**

The TEMPRDLIST command lists assignments made by the TEMPREAD command. Pairs of time steps and associated load cases are listed on the screen.

TEMPINIT

Geo Panel: **LoadsBC > LOAD_OPTIONS > Read Temp as Cond**

The TEMPINIT command reads nodal temperatures from a steady state or a transient heat transfer analysis at a particular time step and applies them as initial temperature conditions for another thermal analysis run.

Entry & Option Description

Time step label

Time step number at which nodal temperatures are requested to be applied as initial conditions. (Use 1 for steady state.)

Load case number

Load case number.
(default is the active load case)

Example:

```
R_THERMAL          (run transient analysis)
TEMPINIT, 3,
R_THERMAL
```

The first command runs transient thermal analysis, the second applies initial temperatures as obtained at time step 3. The third command runs thermal analysis again with the specified initial temperatures.

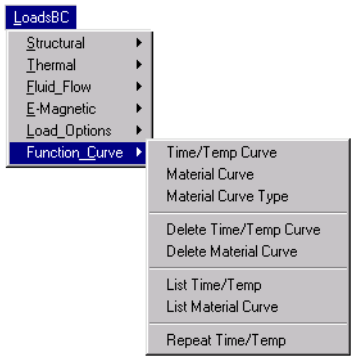



▶ FUNCTION_CURVE Menu

Geo Panel: **LoadsBC > FUNCTION_CURVE**

This menu contains commands related to time, temperature, B-H, asymmetric loading, and material property curves.

Figure 7-34 Function_Curve Menu



CURDEF

Geo Panel: **LoadsBC > FUNCTION_CURVE > Time/Temp Curve**

The CURDEF command may be used to define a time curve, a temperature curve, a curve for asymmetric load coefficients, or B-H curve. Time curves are used by NSTAR in association with loads and boundary conditions. Temperature curves are used by STAR, DSTAR, NSTAR, and HSTAR in association with various material properties. Asymmetric load curves define a curve for the load in the circumferential direction for use with problems that are axisymmetric in all aspects except loading. B-H curves are used by ESTAR to define magnetic permeability. Refer to the MPROP command for details on which material properties may be associated with temperature curves. The ACTSET command may be used to activate a particular time curve, a temperature curve, asymmetric, or a B-H curve. The ACTXYPRE and XYPLOT commands may be used to plot the defined curves. The CURLIST command may be used to list predefined curves. Curve data may be read from an external file. The independent variable values must be input in ascending order.

Entry & Option Description

Curve type

Type of curve.

= TIME	time-dependent curve
= TEMP	temperature-dependent curve
= BHC	electromagnetic B-H curve
= FCOEF	circumferential load coefficients curve for the asymmetric loading option in STAR
= FSIN	Fourier sine coefficients curve for the asymmetric loading option in STAR

= FCOS Fourier cosine coefficients curve for the asymmetric loading option in STAR
(default is TIME)

Curve number
Curve label. Limit is 100 for time and temperature curves, 25 for asymmetric curves, and 100 for B-H curves.
(default is highest label defined + 1)

Starting point
Index of location to start defining the points. Enter 0 to read curve data from an external file.
(default is highest defined location + 1)

External filename for curve info
Name of file containing data. Prompted only if starting point is set to 0. The file must be an ASCII file formatted in one of two ways:

1. The first row lists the number of data points to be read, 0 (for the format type), and the increment value of the independent variable. The dependent variable values are listed one value per row from second row onwards. The independent variable will start from zero.
2. The first row lists the number of data points to be read and 1 (for the format type). The independent and dependent variable values are then listed one pair per row from the second row onwards.

(default is problem_name.xcr)

The following entries are not prompted if data is read from an external file.

Time value for point (i)
Ith time, temperature, asymmetric load coefficient, or magnetic field intensity.

Function value for point (i)
Value of the function at time t(i). For BHC type, this value is the flux density.
(default is 0.0)

Notes

1. A maximum of 5000 points can be defined for a time curve, 200 points for a temperature curve, 1000 for asymmetric load curves, and 25 points for a B-H curve.
2. The command is used to specify nonlinearities. The BHC option can also be used to specify a demagnetization curve for a permanent magnet.
3. The point at the origin of a B-H curve should not be specified.
4. The curve points must be defined in ascending order of the independent variable.
5. For permanent magnets, the demagnetization curve in the second quadrant of a B-H curve is input as if it were shifted to the first quadrant. Hence the first point (Hc ,0) would take the value (0, 0) and the last point (0, Br) would be given the value (Hc, Br). Here Hc is the coercive force and Br is the residual flux density. Furthermore, the user should define the coercivity components




of the permanent magnet using the MPROP command to specify values for the PMAGX, PMAGY and PMAGZ options.

6. The most recently defined curve becomes the active and hence the default curve. For association with other curves, the user should activate the desired curve first through the ACTSET command.
7. Gravity, centrifugal and temperature loadings as well as forces, pressure, and base excitations can be independently associated with time curves.
8. If nodal temperatures are associated with a time curve that has a value of zero at time zero, NSTAR assumes the curve to be prescribing temperatures relative to the reference temperature Tref.

   Trel (n,t) = Curve_value(t)*(Tn-Tref)

   Where:

   Trel (n,t) = relative temperature at node n at time t

   Tn = input temperature at node n

   Tref = reference temperature

   This assumption is made to provide ease of use for certain cases where otherwise each node requires a separate curve to define its temperature.

Example 1:
```
CURDEF, TIME, 1, 1, 0.0, 0.0, 1.0, 10.0, 2.0, 12.0
CURDEF, TIME, 1, 4, 2.5, 15.0,
```
The first command defines 3 points on time-dependent curve number 1, and the second command defines a fourth point.

Example 2:
```
CURDEF, BHC, 2, 1, 1000., 1.0, 3000., 1.5, 4000.,
2.0, 5000., 2.5
```
This command defines 4 points on B-H curve number 2. The field intensities at the 4 points are 1000.0, 3000.0, 4000.0 and 5000.0. The corresponding flux densities are 1.0, 1.5, 2.0 and 2.50.

Example 3:
```
CURDEF, FCOEF, 1, 1, 0, 10, 5, 20, 10, 45,
```
This command defines an asymmetric load in the circumferential direction such that the load is 10, 20, and 45 at angles 0, 5, and 10 degrees, respectively.

MPC

Geo Panel: **LoadsBC > FUNCTION_CURVE > Material Curve**

The MPC command defines a material property set. Points on the strain-stress curve are specified to the material property set. A maximum of 200 stress-strain pairs can be defined for a curve. The assigned material set label cannot exceed 20.

Entry & Option Description

Mat. prop set number

Material property set. (limit is 20)
(default is highest material property set + 1)

Reference temperature
Reference temperature. (Currently not used.)
(default is 0.0)

Starting point
Index of starting point in the curve definition.
(default is highest defined location + 1)

Strain value for point (i)
Strain value at point i.

Stress value for point (i)
Stress value at point i.

Notes

1. Linear interpolation is used to calculate intermediate values.
2. The type of the curve is defined by the MPCTYP command.
3. For plastic curves, linear elastic behavior is assumed between the origin and the first data point on the curve which is considered as the yield point. SIGYLD is ignored if given. Slope of the curve must be decreasing.

Example:
```
MPCTYP, PLASTIC
MPC, 1, 0.0, 0., 0., .001, 10000, .003, 20000
```
The two commands specify three points, (0.0, 0.0), (0.001, 10000.0) and (0.003, 20000.0) on the plastic curve for material set number 1.

MPCTYP

Geo Panel: **LoadsBC > FUNCTION_CURVE > Material Curve Type**

The MPCTYP command is used to set the type of a material property curve to be defined using the MPC command.

Entry & Option Description

Mat. prop curve number
Label of the curve.

MPC type
Type of the material property curve.

= Elastic	elastic
= Plastic	plastic
= MR	Mooney-Rivlin
= OG	Ogden
= VE	viscoelastic
	(default is elastic)

Notes

1. The MPCTYP command must be issued prior to defining any material property curves by the MPC command.
2. The specified option must be consistent with option 5 of the EGROUP command.




Example: `MPCTYP, 2, 1,`

This command states that material property curve number 2 is plastic.

CURDEL

Geo Panel: **LoadsBC > FUNCTION_CURVE > Delete Time/Temp Curve**

The CURDEL command deletes a pattern of curves previously defined by the CURDEF command.

Entry & Option Description

Curve type

Type of curve.

= TIME	time-dependent curve
= TEMP	temperature-dependent curve
= BHC	electromagnetic B-H curve
= FCOEF	circumferential load coefficients curve for the asymmetric loading option in STAR
= FSIN	Fourier sine coefficients curve for the asymmetric loading option in STAR
= FCOS	Fourier cosine coefficients curve for the asymmetric loading option in STAR *(default is TIME)*

Beginning curve

Beginning curve in the pattern.
(default is 1)

Ending curve

Ending curve in the pattern.
(default is beginning curve)

Increment

Increment between curves in the pattern.
(default is 1)

Example: `CURDEL, TIME, 1, 3, 2,`

The above command deletes the previously defined time curves 1 and 3 from the database.

MPCDEL

Geo Panel: **LoadsBC > FUNCTION_CURVE > Delete Material Curve**

The MPCDEL command deletes material property curves associated with a pattern of material property sets.

Entry & Option Description

Beginning curve

Beginning material property set in the pattern.
(default is 1)

Ending curve
Ending material property set in the pattern.
(default is beginning curve)

Increment
Increment between material property sets in the pattern.
(default is 1)

Example: `MPCDEL, 1, 2, 1`

This command deletes all the material property curves defined for material sets 1 and 2. Note that material property sets in the pattern are deleted whether they are curves or regular material property sets.

CURLIST

Geo Panel: **LoadsBC > FUNCTION_CURVE > List Time/Temp**

The CURLIST command lists the data points for curves previously defined by the CURDEF command.

Entry & Option Description

Curve type
Type of curve.

= TIME	time-dependent curve
= TEMP	temperature-dependent curve
= BHC	electromagnetic B-H curve
= FCOEF	circumferential load coefficients curve for the asymmetric loading option in STAR
= FSIN	Fourier sine coefficients curve for the asymmetric loading option in STAR
= FCOS	Fourier cosine coefficients curve for the asymmetric loading option in STAR

(default is TIME)

Beginning curve
Beginning curve in the pattern.
(default is 1)

Ending curve
Ending curve in the pattern.

Increment
Increment between curves in the pattern.
(default is 1)

Example: `CURLIST, , 1, 3, 1,`

The above command lists all the points defined for time curves number 1, 2 and 3.

MPCLIST

Geo Panel: **LoadsBC > FUNCTION_CURVE > List Material Curve**

The MPCLIST command lists nonlinear material property curve information associated with a pattern of material property sets.

Entry & Option Description

Beginning curve
Beginning material property set in the pattern.
(default is 1)

Ending curve
Ending material property set in the pattern.
(default is highest material property set defined)

Increment
Increment between material property sets in the pattern.
(default is 1)

Example: `MPCLIST, 1, 2, 1`
This command lists all the material property curves defined for material sets 1 and 2.

MAKE_CYCLIC

Geo Panel: **LoadsBC > FUNCTION_CURVE > Repeat Time/Temp**

The MAKE_CYCLIC command adds points to a pattern of time, temperature, or B-H curves by repeating the defined pattern for the desired number of cycles.

Entry & Option Description

Curve type
Curve type.

= TIME	time-dependent curve
= TEMP	temperature-dependent curve
= BHC	electromagnetic B-H curve
	(default is TIME)

Beginning curve
Beginning curve in the pattern.
(default is 1)

Ending curve
Ending curve in the pattern.
(default is beginning curve)

Increment
Increment between curves in the pattern.
(default is 1)

Number of cycles to add

Number of cycles to be added.
(default is 1)

Example:

```
CURDEF,TIME, 1, 1, 0, 0, 1, 1, 3, 0,
MAKE_CYCLIC, TIME, 1, 1, 1, 2,
```

The first command defines time curve number 1. The second command adds 4 more points to define two more cycles of the triangular curve defined by the first command. Time curve 1 will then be defined by the following points: (0,0, 1,1 3,0, 4,1, 6,0, 7,1 9,0).




8 Control Menu

CONTROL Menu

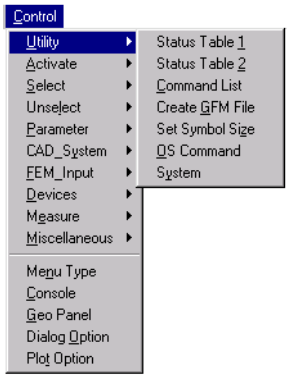
This menu contains general utility commands, commands to control active sets and selection lists and define parametric expressions for input quantities. The menu also provides commands to interact with CAD (IGES and DXF formats) and other finite element packages. The menu includes commands to generate image files in various formats and control printing devices. In addition, a menu to provide some measurements is included.

Figure 8-1 Control Menu

UTILITY Menu

Control > UTILITY

Figure 8-2 Utility Menu

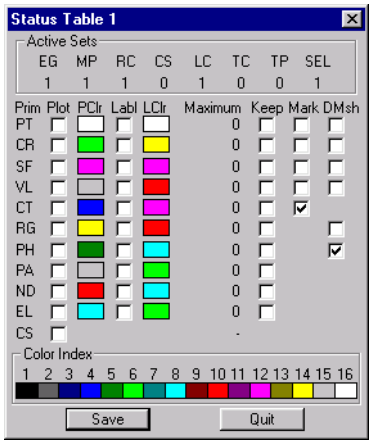


This menu contains general utility commands.

STATUS1

Control > UTILITY > Status Table 1

The STATUS1 command sets options related to subsequent plotting, deleting entities, and default meshing. The command also displays the active sets and the color code. When the command is issued, the following table will be displayed.



The active sets shown in the STATUS1 Table are:

EG	element group
MP	material property set
RC	real constant set
CS	coordinate system
LC	load case
TC	time curve
TP	temperature curve
SEL	selection set (controlled by the STATUS3 Table or the "ACTSET,SEL" command)

The entities considered in the STATUS1 Table are:

PT	keypoints
CR	curves
SF	surfaces

VL	volumes
CT	contours
RG	regions
PH	polyhedra
PA	parts
ND	nodes
EL	elements
CS	coordinate systems

The status information controlled by STATUS1 command for each of the above entties, is listed below:

PLOT	flag (ON/OFF, a checkmark appears when ON)
PCLR	plot color (1 to 16 colors)
LABL	plot label (ON/OFF)
LCLR	label color (1 to 16 colors)
MAXM	maximum number defined
KEEP	flag to keep associated lower entities when higher entities are deleted (ON/OFF, a checkmark appears when ON)
MARK	flag to show direction of entities (ON/OFF, a checkmark appears when ON)
DMSH	flag to activate or deactivate default meshing option (ON/OFF, a checkmark appears when ON)

If the color of an entity is similar to the active background color, that entity will be plotted in the active foreground color.

To set the desired options:

The Settings in the STATUS Table 1 take effect after making the desired settings, clicking Save, and repainting or issuing related commands.

1. Click the **Status1** button in the Geo Panel or click **Control, Utility, Status Table1**.
2. To plot an entity whenever a higher associated entity is plotted, click the corresponding checkbox in the Plot column. A checkmark appears. For example, if you check this flag for Nodes and click Save, the program will plot nodes whenever you plot associated higher geometric entities or elements.
3. To change the color that the program will use in plotting an entity, click the box that corresponds to that entity in the Pclr column until the desired color appears.
4. To instruct GEOSTAR to plot the label of an entity whenever that entity is plotted, click the checkbox that corresponds to that entity in the Labl column. You can select the color of the label by clicking the LClr checkbox until the desired color appears.




5 When you delete an entity, GEOSTAR, by default, deletes all lower associated entities. To instruct GEOSTAR to always keep a lower entity whenever you delete higher entities associated with it, click the corresponding checkbox in the **Keep** column.

6 To show the direction of curves, surfaces, volumes and contours, click the corresponding checkbox in the Mark column.

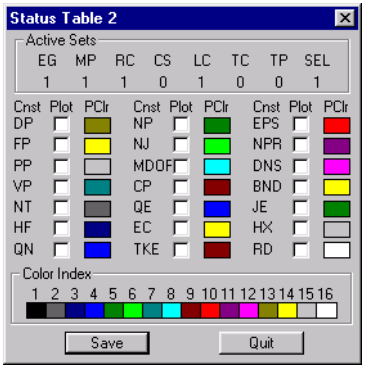
7 To activate default meshing of a particular entity, click the corresponding checkbox in the **DMesh** column.

8 After making all desired changes, click **Save** to activate the settings, or click **Quit** to close the table without making any changes. Pressing the **Escape** key is equivalent to clicking **Quit**. Remember that the settings take effect for subsequent activities only. Repaint to apply the new settings to the active window.

STATUS2

Control > UTILITY > Status Table 2

The STATUS2 command sets options related to subsequent plotting of loads and boundary conditions. The command also displays the active sets and the color code. When the command is issued, the following table will be displayed. The Const is for the type of constraint (Load/BC). The Plot column is to turn on/off the default plotting of the Load/BC (when geometric entities are plotted). The Pclr lets you set the desired color for plotting the Load/BC item.



The active sets shown in the STATUS1 Table are:

EG	element group
MP	material property set
RC	real constant set
CS	coordinate system

LC	load case
TC	time curve
TP	temperature curve

SELselection set (controlled by the STATUS3 Table or the “ACTSET, SEL” command)

Entry & Option Description

DP	displacement B.C.
FP	force
PP	pressure
VP	velocity
NT	nodal temperature
QN	nodal heat generation rate
QE	element heat generation rate
CE	element convection
NP	nodal potentials
NJ	nodal current
MDOF	master degrees of freedom
CP	coupled DOF
TKE	turbulence kinetic energy
EPS	dissipation rate
NDS	nodal pressure
DNS	density
JE	current or charge density
HX	heat flux
RD	radiation
BND	Euler boundary condition

To set the desired options:

The Settings in the STATUS Table 2 take effect after making the desired settings, clicking Save, and repainting or issuing related commands.

1. Click the **Status2** button in the Geo Panel or click **Control, Utility, Status Table 2**.
2. To instruct GEOSTAR to plot the displacement boundary conditions automatically when associated geometry is plotted, click the checkbox for DP in the Plot column.
3. To change the color, click the corresponding color box until the desired color appears.
4. After making all desired changes, click **Save** to activate the settings, or click **Quit** to close the table without making any changes. Pressing the **Escape** key is equivalent to clicking **Quit**. Remember that the settings take effect for subsequent activities only. Repaint to apply the new settings to the active window.




CMDLIST

Control > UTILITY > Command List

The CMDLIST command lists the commands issued on a list screen. Commands are temporarily stored in a buffer which is dumped to the Session file whenever the buffer is filled or the **File, Save Session File** command is issued.

GFORM_OUT

Control > UTILITY > Create GFM File

The GFORM_OUT command operates directly on the model database to generate an ASCII file that can be used to regenerate the model. Geometry is written in a neutral format (refer to commands CRGFORM, SFGFORM, and VLGFORM for details). Information related to the finite element mesh can also be included. The command creates a new file with extension GFM or appends to an existing file in the current directory.

Recreation of a model from the Session file can take a long time due to the fact that it contains a full chronological record of all issued commands. This command ignores the history of the generation and translates the current state of the model. This command is useful in combining independently created parts of a model.

When a selection list is active for an entity, then only members of that entity that are in the selection list will be written to the file. If it is desired to translate the whole model, then selection lists should be first deactivated or initialized.

Entry & Option Description

gfm filename with extension
Name of file to be generated.
(default is ufn.GFM)

gfm flag
Flag to select needed information.

= 0: Geometry	geometric entities only
= 1: FEM	finite element mesh only
= 2: Both	both of the above
	(default is both)

Append to existing file
Appending flag.

= 0: No	overwrite the file
= 1: Yes	append to the same file
	(default is yes)

Entity name
Geometric entity to be written to the file. (prompted only if gfm flag = geometry

or both)

= PT	keypoints
= CR	curves
= SF	surfaces
= VL	volumes
= CT	contours
= PH	polyhedra
= PA	parts
= RG	regions
= AL	all geometric entities *(default is AL)*

Association flag

Association flag.

= 1: Yes	geometric entities will carry their labels in the gfm file
= 0: No	geometric entities will not carry their labels and will only be identified by their equations. (prompted only if gfm flag = both) *(default is yes)*

Node offset

Offset in node numbering. (prompted only if gfm flag = fem or both)
(default is 0)

Element offset

Offset in element numbering. (prompted only if gfm flag = fem or both)
(default is 0)

Coordinate system offset

Offset in coordinate system numbering.
(default is 0)

The following arguments are only prompted if "gmf flag" is set to geometry. Defaults are 0.

Point offset

Offset in keypoint numbering.

Curve offset

Offset in curve numbering.

Surface offset

Offset in surface numbering.

Volume offset

Offset in volume numbering.

Contour offset

Offset in contour numbering.

Region offset

Offset in region numbering.

The following arguments are only prompted if "gmf flag" is set to fem or both. Defaults are 0.

Element group offset
Offset in element group numbering.

Real constant offset
Offset in real constant set numbering.

Element coordinate offset
Offset in element coordinate system numbering.

Load case offset
Offset in load case numbering.

To combine two databases with duplicate labels

1. Open one of the databases.
2. Use the **GFORM** command to write a GFM file. Specify offsets sufficient to avoid duplicate labeling.
3. Open the other database.
4. Load the GFM file generated in step 2 using the **File, Load** command.

Notes

✍ Suppose that the current model has one surface (surface number 1). The GFORM_OUT command was issued with the association flag set to zero and the generated file is read to another database with10 surfaces labeled 1 through 10. In this case, the surface from the gfm file will be labeled 11. The association of geometry with nodes and elements is lost. On the other hand if the association flag is set to 1, then surface 1 from the gfm file will replace surface 1 in the current database.

✍ If a model is reconstructed from a gfm file in which elements are not associated with geometry, then the user must issue the "BOUNDARY,1,0" command before plotting result contours. This command will instruct GEOSTAR to plot the corresponding result on every face of every element. The default is to plot boundary faces only to speed up plotting since other faces will be hidden in the final plot.

Example: `GFORM_OUT, part1.gfm, 2,,,,`
This command creates (or appends to) file "part1.gfm". Both geometry and the finite element mesh are included. No offsets are specified.

SYMBSIZ

Control > UTILITY > Set Symbol Size

The SYMBSIZ command sets the size of the symbols for plotting some loads and boundary conditions, coupling, reaction forces, and shear and moment diagrams. The STATUS2 (or the SETCOLOR) command can be used to set the color of the symbols.

Entry & Option Description

Entity

Name of the symbol. The admissible symbol names are:

= DP	displacement boundary conditions
= FP	applied external forces
= PP	applied pressure
= VP	applied velocity
= CP	coupling
= SM	shear moment diagrams (used by SMPLOT)
= RF	reaction force

Magnification factor 0.1 - 10.0

Relative magnification factor in comparison to the default size. The factor may vary from 0.1 to 10.
(default is 1.0)

Example: `SYMBSIZ, DP, 2.0`

This command doubles the size of the displacement boundary condition for subsequent plots.

OSCOMMAND

Control > UTILITY > OS Command

The OSCOMMAND command provides the user with access to operating system commands while in GEOSTAR. The command may be used in the “GEOFUN” file to perform operating system functions by one-button push.

Entry & Option Description

Command

Operating system command.

Notes

Any operating system command can be issued including invoking an editor, running another program, formatting a drive and executing a batch file as long as enough resources are available in the machine.

✍ It is not recommended to copy GEOSTAR database files using this command. The FILE > SAVE command can be used for this purposes.

Example 1: `OSCOMMAND dir/p`

This command lists the files in the current subdirectory one page at a time in DOS environment.

Example 2: `OSCOMMAND ws`

This command inside GEOSTAR is similar to "ws" from the operating system level. The user may assign a function key to the above command (accomplished by editing the GEOFUN file) to invoke a program called "ws".

SYSTEM

Control > UTILITY > System

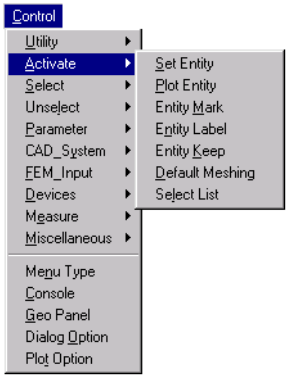
The SYSTEM command open a DOS window in which you can issue any DOS command. The OSCOMMAND is a similar command that lets you issue DOS commands in GEOSTAR.

ACTIVATE Menu

Control > ACTIVATE

This menu contains commands to control the activation of sets in GEOSTAR. A set becomes active once it is defined. All functions in this menu can be conveniently called from the STATUS1, STATUS2, and STATUS3 Tables.

Figure 8-3 Activate Menu



ACTSET

Control > ACTIVATE > Set Entity

The ACTSET command enables you to control the active sets. You can see the active set at the top of the any of the Status Tables. The complete list of admissible types of sets is given below.

Entry & Option Description

Set label

Type of the set. Admissible types are:

EG	element group
MP	material property set
RC	real constant set

LC	load case
TC	time curve
TP	temperature curve
BH	B-H curve (for magnetic analysis)
CS	coordinate system
EC	element coordinate system
LOC	fatigue location
MC	material curve (same as material property set)
SEL	selection list
FCOEF	circumferential load coefficients curve for the asymmetric loading option in STAR
FSIN	Fourier sine coefficients curve for the asymmetric loading option in STAR
FCOS	Fourier cosine coefficients curve for the asymmetric loading option in STAR
ESET	element set
SPT	multi-base motion curve

Set number

Number of the set to be activated.

For CS (Coordinate system):

= 0	global Cartesian
= 1	global cylindrical
= 2	global spherical
= 3 - 5000	local coordinate system

For ECS (Element coordinate system):

= -1	local element coordinate system
= 0	global Cartesian
= N	local coordinate system N

For LOC (Fatigue location):

= 0	activate the all-node option
= N	location N
	(default is the active set)

Notes

1. Local coordinate systems can be Cartesian, cylindrical or spherical.
2. The CURDEF command can be used to create time, temperature, B-H, FCOEF, FSIN, FCOS curves. Time curves for ASTAR should be created by the PD_CURDEF command.

Example: `ACTSET, MP, 2`

This command activates material property set number 2. All elements generated while this set is active will assume material set number 2 unless otherwise specified by the EPROPSET command when generating new elements from existing ones. The STATUS1 command shows the active element attributes in the model.

Example: `ACTSET, LC, 3`

```
FND, 1, FZ, 100, 1,1,
```

```
          ACTSET, LC, 5
FND, 2 FZ, 300, 1,1,
```

The first is applied as part of load case 3. The second load is applied as part of load case 5.

ACTPLOT

Control > ACTIVATE > Plot Entity

The ACTPLOT command enables/disables the plotting of a specified entity whenever higher associated entities are plotted. You can conveniently use the STATUS1 (Control, Utility , Status Table 1) command to control this flag.

Entry & Option Description

Entity

Label of the entity. The admissible labels are:

VL	volumes
SF	surfaces
CR	curves
PT	keypoints
RG	regions
PH	polyhedra
CT	contours
EL	elements
ND	nodes
PA	parts
DP	displacement B.C.
FP	force
PP	pressure
CS	coordinate system
NT	nodal temperature
HF	hydraulic force
QN	nodal heat
NP	nodal potential
NJ	nodal current
VP	velocity
MDOF	master DOF
CP	coupled DOF

Plot flag

Flag for plotting the above entity when higher associated entities are plotted.

= 1: On	do not plot
= 0: Off	plot
	(default is off)

Note

GEOSTAR hierarchies from high to low are:

a. VL > SF > CR > PT

b. PA > PH > SF and RG > CT > CR > PT)

c. Geometric entities >EL >ND

Example 1: `ACTPLOT, CT, 0`

This command specifies that contours will not be plotted automatically when associated regions are plotted. Note that keypoints and curves, which are part of contours, are plotted if the associated flags are active.

Example 2: `ACTPLOT, CR, 1`

This command specifies that associated curves be plotted whenever higher entities (for example surfaces) are plotted.

ACTMARK

Control > ACTIVATE > Entity Mark

The ACTMARK command enables you to plot marks to show the directions of geometric entities. Arrows are used to show the directions of curves and contours, and asterisks are used to show the parametric coordinates of surfaces and volumes. The third parametric curve of a volume is marked by an arrow. You can conveniently use the STATUS1 (Control, Utility , Status Table 1) command to control this flag.

Entry & Option Description

Entity

Label of the entity to be marked. The admissible labels are:

CR	curves
CT	contours
EL	elements
ND	nodes
PT	keypoints
RG	regions
SF	surfaces
VL	volumes

Mark flag

Flag for plotting the marks.

= 0: Off	do not plot
= 1: On	plot
	(default is on)

Notes

1. For surfaces, an asterisk is plotted on the first parametric coordinate. The origin of the coordinate system is at the corner closest to the asterisk.
2. For volumes, an asterisk is plotted to define the surface that contains the first and the second parametric curves. The origin of the coordinate system is at the corner closest to the asterisk and the third parametric curve is identified by an arrow.

Example: `ACTMARK, CR, 1`

This command orders the plotting of arrows on plotted curves to show

their direction.

ACTNUM

Control > ACTIVATE > Entity Label

The ACTNUM command controls the plotting of labels. You can conveniently use the STATUS1 (Control, Utility , Status Table 1) command to control this flag.

Entry & Option Description

Entity

Label of the entity. The admissible labels are:

PT	keypoint
CR	curves
SF	surfaces
CT	contours
RG	regions
VL	volumes
ND	nodes
EL	elements

Number plotting flag

Flag for writing the labels.

= 0: Off	do not write
= 1: On	write
	(default is on)

Example 1: `ACTNUM, EL, 1`

Labels (numbers) of elements will be written when elements are plotted using the EPLOT (Edit, Plot, Elements) command.

ACTKEEP

Control > ACTIVATE > Entity Keep

The ACTKEEP command allows the user to keep a specified entity when higher order entities are deleted. All entities lower in order than the specified entity are also kept.

Entry & Option Description

Entity

Label of the entity to be kept. Entities are ordered from high to low in the direction of the arrows.

VL > SF > CR > PT (EL > ND)

PA > PH > RG > CT > CR > PT (EL > ND)

where:

VL	volumes
SF	surfaces

CR	curves
PT	keypoints
PA	parts
PH	polyhedra
RG	regions
CT	contours
EL	elements
ND	nodes

Keep flag

Keeping flag.

= 0: Off	do not keep
= 1: On	keep
	(default is on)

Note

The three hierarchies are independent, e.g. if nodes and elements are to be kept when volumes are deleted, the ACTKEEP command must be issued to keep elements.

Example 1:
```
ACTKEEP, SF, 1
VLDEL, 1, 10, 1
```
Surfaces, curves and keypoints are kept when volumes are deleted. Note that elements and nodes are also deleted unless the ACTKEEP command was previously issued to keep them.

Example 2:
```
ACTKEEP, EL, 1
RGDEL, 1, 5, 1
```
Elements and nodes are kept when regions are deleted. Contours, curves and keypoints are deleted unless the ACTKEEP command was previously issued to keep them.

ACTDMESH

Control > ACTIVATE > Default Meshing

The ACTDMESH command sets the flag for mesh generation associated with a geometric entity. Whenever a geometric entity is generated from another, the associated mesh (if any) is also generated for the new entity if default meshing is active.

Entry & Option Description

Entity

Label of the entity. The admissible labels are:

CR	curves
SF	surfaces
VL	volumes
RG	regions
PH	polyhedrons
PT	points

Default mesh flag




Flag to activate the default mesh for meshing the type of entity specified in this command, but created subsequently.

= 0: Off	deactivate
= 1: On	activate
	(default is on)

Notes

1. Default meshing can be conveniently activated or deactivated from the STATUS1 Table.
2. Activating default meshing for polyhedra, deactivates default meshing for surfaces if active.

Example: `ACTDMESH, CR, 1,`

This command activates default meshing for curves. All new curves generated from meshed source curves are also meshed. The new mesh of a curve is similar to the mesh of its source, unless otherwise specified by the EPROPSET command.

ACTSEL

Control > ACTIVATE > Select List

The ACTSEL command may be used for one of the following purposes:

1. Turn off a particular selection set for an entity. The active selection set is not changed in this case.
2. Turn on a particular selection set for an entity. The specified selection set becomes active if it exists.
3. Turn on all selection lists for an entity without changing the active selection list.
4. Turn off all selection lists for an entity without changing the active selection list.

The STATUS3 command provides a convenient table for selection lists control. The "ACTSET, SEL" command may also be used to control the active selection list.

Entry & Option Description

Entity

Selection entity. The admissible entities are:

PT	keypoints
CR	curves
SF	surfaces
VL	volumes
PA	parts
PH	polyhedra
RG	regions
CT	contours
ND	nodes
EL	elements

Selection flag

Activate/deactivate flag.

= 0: Off deactivate
= 1: On activates
(default is on)

Selection set number
Selection set number. (0 indicates all selection sets)

Notes

1. Selection lists should be carefully used. The user should deactivate or initialize the selection list as soon as the purpose is served. A lot of confusion could happen whenever the user forgets that he has an active selection list.
2. The INITSEL command can be used to delete a selection list.

Example 1:
```
ACTSEL, EL, 0, 3,
```
The above command turns off selection list #3 for elements. The active selection set is not changed.

Example 2:
```
ACTSEL, ND, 1, 3,
```
The above command turns on selection list #3 for nodes. Selection set #3 becomes active, if it exists for nodes.

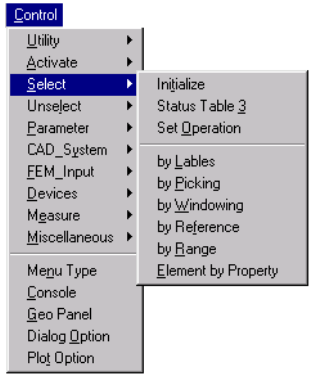
Example 3:
```
ACTSEL, EL, 0, 0,
ACTSEL, VL, 1, 1,
```
The first command turns off all selection lists for elements. The second command turns on all selection lists for volumes. In both cases, the active selection set is not changed.

SELECT Menu

Control > SELECT

This menu contains commands dealing with multiple selection lists. Refer to Chapter 5 of the COSMOS/M User Guide for examples.

Figure 8-4 Select Menu



INITSEL

Control > SELECT > Initialize

The INITSEL command initializes a selection list and gives the user access to all members of the entity. An option to complement the current list is also provided. The command does not change the active selection list.

Entry & Option Description

Entity name
Selection entity. The admissible entities are:

PA	parts
PT	keypoints
PH	polyhedra
CR	curves
SF	surfaces
RG	regions
CT	contours
VL	volumes
ND	nodes
EL	elements
ALL	all entities
	(default is EL)

Initialization flag

Initialization flag.

= 1: Yes	clear the selection list and give access to all members
= -1: Cmplmnt	complement the selection list
	(default is empty)

Selection set number

Selection set number.

= 0	all
= n	set number

Notes

1. A selection list is a filtered list of defined members of the specified entity on which operations like plotting, extrusion, sweeping, meshing, property manipulation, etc., may be performed.
2. The deactivation of the selected list using the INITSEL command results in the permanent loss of the selection list. The ACTSEL command on the other hand can be used to temporarily deactivate a selection list and reclaim it later. The STATUS3 Table provides convenient tools to control multiple selection lists.

Example: `INITSEL, EL, -1, 5,`

The above command redefines a selection set number 5 for elements to be the complement of the existing one. Every element on the old list is not included in the new one, and every element that was not on the old list is included in the new one.

STATUS3

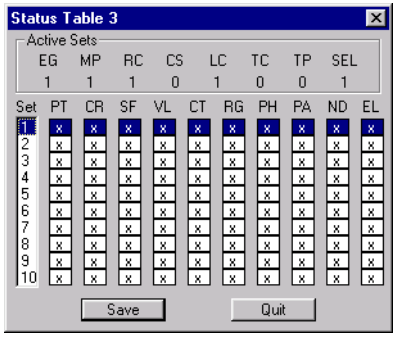
Control > SELECT > Status Table 3

The STATUS3 command displays and controls multiple selection lists. Selection sets control access to defined members of entities. Whenever a selection list is active for an entity, GEOSTAR only recognizes members of that entity that are in the selection list. Ten different selection lists may be defined for any entity in the table, but only one is active at any time. The active selection list is the same for all

the entities at any time. Status of any selection list can be toggled using the mouse. Four possible settings are available:

"X"	none (nothing defined).
"-"	selection list will be initialized when the table is saved.
"ON"	selection list is on.
"OFF"	selection list is off.

Toggling is accomplished by placing the arrow of the mouse on the particular location to be changed and pressing either button. After making the desired changes, the user can save or abandon the changes by moving the mouse cursor to the SAVE or QUIT boxes and clicking the mouse. The "ABORT" icon is equivalent to "SAVE" and the "Escape" key is equivalent to "QUIT". It should be noted that the active selection set is controlled by the first column only and that when a particular set is active, flags for a particular entity can still be toggled as desired. For example, if selection set number 3 is active and was defined for nodes, the user may elect to turn it off, or set it for initialization. The active selection list is changed to the last selection set, the user specifies in selection and un-selection commands. Refer to Chapter 5 in the COSMOS/M User Guide for examples on selection sets.



When this command is issued, the following chart will be displayed.

Entry & Option Description

The entities considered in the STATUS3 Table are:

PT	keypoints
CR	curves
SF	surfaces
VL	volumes
CT	contours
RG	regions
PH	polyhedra
PA	parts
ND	nodes
EL	elements

SELSETOP

Control > SELECT > Set Operation

The SELSETOP command defines or adds new members to an existing selection set for an entity by operating on existing selection lists. The new members of the entity may be defined by set operations like union, intersection, or subtraction.

Entry & Option Description

Target selection set number
Label of the selection set to be created or added to.

Entity
Selection entity. The admissible entities are:

PT	keypoints
PH	polyhedra
PT	points
CR	curves
SF	surfaces
RG	regions
CT	contours
VL	volumes
ND	nodes
EL	elements
	(default is EL)

Set operation
Set operation flag.

= C	complement of a set
= U	union of two existing sets
= I	intersection of two existing sets
= FD	subtraction (set1 - set2)
= BD	subtraction (set2 - set1)
	(default is U)

First source selection set number
Label of existing selection set.

Second source selection set number
Label of existing selection set. (not prompted for if set operation is Complement)

SELINP

Control > SELECT > by Labels

The SELINP command initiates, or adds to an existing selection list for a particular entity by specifying a pattern of members. The specified set number becomes the active set for all entities.

Entry & Option Description

Entity name

Selection entity name. The admissible entities are:

PT	keypoints
CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
ND	nodes
EL	elements
PH	polyhedra
PA	parts
	(default is EL)

Beginning entity
Beginning label in the pattern.

Ending entity
Ending label in the pattern of the set.
(default is beginning entity)

Increment
Increment between the labels.
(default is 1)

Selection set number
Selection set number.

Note
Beginning and ending labels of the pattern can be selected by the mouse.

Example: `SELINP, PT, 1, 3, 1, 4,`
The above command adds keypoints 1, 2, and 3 to selection set number 4 for keypoints. The command initiates selection set number 4 for keypoints if it does not exist. Selection set 4 becomes active for all entities.

SELPIC

Control > SELECT > by Picking

The SELPIC command initiates or adds to the active selection set for a particular entity by successively picking individual members of the entity that are plotted on the screen by the mouse or entering their labels through the keyboard. The command continues prompting for more members until it is stopped by the “ABORT” icon or the “ESC” key. If no selection list is active, the command activates, and operates on selection set number 1.

Entry & Option Description

Entity name for selection set 1
Selection entity name. The admissible entities are:

PT	keypoints

CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
ND	nodes
EL	elements
	(default is EL)

Selection entity (i)
Labels of the items of the entity, picked by the mouse or entered through the keyboard.

Example: `SELPIC, EL, 1, 3, 6,`

The above command adds elements 1, 3, and 6 to the active selection set. The command initiates and activates selection set number 1, if no selection set is active.

SELWIN

Control > SELECT > by Windowing

The SELWIN command initiates, or adds to the active selection list of a specified entity by forming a window surrounding the region of interest. Rectangular, circular or polygon windows can be specified by using the mouse. All plotted members of the specified entity within the window are added to the selection list. The command operates on the active selection list. If no selection list is active, the command will activate, and operate on selection set number 1.

Entry & Option Description

Entity name
Selection entity name. The admissible entities are:

PT	keypoints
CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
PH	polyhedra
PA	parts
ND	nodes
EL	elements
ALL	all entities
	(default is EL)

Window type
Type of window.

= 0: Box	rectangular window specified by two opposite corners using the cross-hairs and the mouse

= 1: Circle — circular window specified by a center and a point on the circumference picked by mouse

= 2: Polygon — polygon window drawn using the mouse in a rubber band fashion. Type “c” to close the polygon, “d” to delete the last segment, or the Escape key to abort.
(default is box)

Selection set number
Selection set number.

Notes

1. Only plotted members can be selected.
2. A member is selected if its center (or label) lies inside the window.
3. If the polygon option is used, the polygon must be closed. The polygon is closed if a selected pixel is within two pixels from the first one. Otherwise the user may type “c” to have GEOSTAR close the polygon by connecting the last and the first pixels. Typing the character “d” deletes the last side of the polygon. The Escape key can be used to abort the command at any time.

SELREF

Control > SELECT > by Reference

The SELREF command initiates or adds to the active selection list of a particular entity. All members of this entity that are associated with a pattern of a reference entity are selected. If no selection list is active, the command activates, and operates on selection set number 1.

Entry & Option Description

Selection entity
Selection entity name. The admissible entities are:

PT	keypoints
PH	polyhedra
PA	parts
CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
ND	nodes
EL	elements

(default is EL)

Reference entity
Reference entity name. The admissible entities are as stated above.

Beginning reference entity
Beginning label in the reference entity pattern.

Ending reference entity

Ending label in reference entity pattern.
(default is beginning reference entity)

Increment
Increment between labels in the reference entity pattern.
(default is 1)

Boundary element flag
Boundary flag.
= 0: No — select members associated with the pattern of reference entity
= 1: Yes — include in addition, all members that have common boundaries with the selected entity
(default is No)

Boundary element flag
Boundary flag.
= 0: No — select members associated with the pattern of reference entity
= 1: Yes — include in addition, all members that have common boundaries with the selected entity
(default is No)

Note
Beginning and ending labels of the pattern can be selected by the mouse.

Example: `SELREF, EL, CR, 1, 3, 1`
The above command adds all elements associated with curves 1, 2, and 3 to the active selection set of elements.

SELRANGE

Control > SELECT > by Range

The SELRANGE command initiates or adds to a selection list for a particular entity, all such members that lie inside a 3D window. The window is defined by specifying the X-, Y-, and Z-coordinate bounds. The specified selection list becomes active.

Entry & Option Description

Entity name
Selection entity name. The admissible entities are:

PT	keypoints
PH	polyhedra
PA	parts
CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
ND	nodes
EL	elements

ALL — all entities
(default is EL)

Coordinate system
Coordinate system to be used in specifying the range.
(default is 0 for global Cartesian)

Coordinate activation flag for x
Coordinate activation flag for bounds in the X-direction.
= 1: Activate — activate the X-direction bounds
= 0: Deactivate — deactivate the X-direction bounds
(default is activate)

Coordinate activation flag for y
Coordinate activation flag for bounds in the Y-direction.
= 1: Activate — activate the Y-direction bounds
= 0: Deactivate — deactivate the Y-direction bounds
(default is activate)

Coordinate activation flag for z
Coordinate activation flag for bounds in the Z-direction.
= 1: Activate — activate the Z-direction bounds
= 0: Deactivate — deactivate the Z-direction bounds
(default is activate)

x-coordinate lower range
Lower bound for X-coordinate.
(default is minimum X-extent)

x-coordinate higher range
Upper bound for x-coordinate.
(default is maximum X-extent)

y-coordinate lower range
Lower bound for Y-coordinate.
(default is minimum Y-extent)

y-coordinate higher range
Upper bound for Y-coordinate.
(default is maximum Y-extent)

z-coordinate lower range
Lower bound for Z-coordinate.
(default is minimum Z-extent)

z-coordinate higher range
Upper bound for Z-coordinate.
(default is maximum Z-extent)

Selection set number
Selection set number.
(default is the active set)

Note
The X-, Y- and Z-bounds are prompted only when the corresponding activation flag is turned on (1). If the activation flag for a coordinate is deactivated, then the

window is infinite in that direction.

Example: `SELRANGE, ND,, 1, 1, 0, 10.0, 40.0, 20.0, 60.0, 3`

The above command adds all the nodes whose X-coordinate is between 10.0 and 40.0, and whose Y-coordinate is between 20.0 and 60.0 to selection set number 3 for nodes. Selection set number 3 becomes active.

ESELPROP

Control > SELECT > Element by Property

The ESELPROP command initiates or adds elements to an element selection list. All elements associated with a pattern of element groups, material property sets, or real constant sets are added to the element selection list. The specified selection set becomes active.

Entry & Option Description

Selection property name

Selection property type. Admissible types are:

EG	element group
MP	material property set
RC	real constant set

(default is EG)

Beginning set

Beginning set number in the pattern.

Ending set

Ending set number in the pattern.
(default is beginning set)

Increment

Increment between the set numbers in the pattern.
(default is 1)

Selection set number

Selection set number.
(default is the active set)

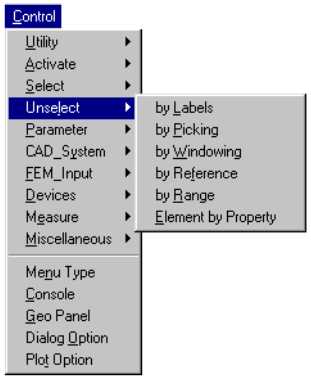
Example: `ESELPROP, RC, 1, 3, 1, 4,`

The above command creates/or adds to element selection set number 4, all elements associated with real constant sets 1, 2, and 3.

UNSELECT Menu

Control > UNSELECT

Figure 8-5 Unselect Menu



This menu includes the following commands to remove entities from s selection list.

UNSELINP

Control > UNSELECT > by Labels

The UNSELINP command removes the specified pattern of members from a selection list of the specified entity. The specified set becomes active. If the specified set selection does not exist, it will be initiated and activated as if it originally existed and contained all members of that entity.

Entry & Option Description

Entity name

Selection entity name. The admissible entities are:

PT	keypoints
PH	polyhedra
PA	parts
CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
ND	nodes
EL	elements

(default is EL)

Beginning entity

Beginning label in the pattern.

Ending entity

Ending label in the pattern of the set.
(default is beginning entity)

Increment

Increment between the labels.
(default is 1)

Selection set number

Set number.
(default is the active set)

Note

Beginning end ending labels of the pattern can be selected by the mouse.

Example: `UNSELINP, PT, 1, 3, 1, 5`

The above command removes keypoints 1 through 3 from the keypoints selection list number 5.

UNSELPIC

Control > UNSELECT > by Picking

The UNSELPIC command removes members from the active selection list for a particular entity by successively picking plotted individual members by the mouse or entering their labels through the keyboard. The command continues prompting for more members until it is stopped by the "ABORT" icon or the "ESC" key. If no selection set is active, the command will operate on selection set number 1 as if it originally existed and contained all members of that entity.

Entry & Option Description

Entity name for selection set 1

Selection entity name. The admissible entities are:

PT	keypoints
PH	polyhedra
PA	parts
CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
ND	nodes
EL	elements

(default is EL)

Unselection entity (i)

Label of the ith member.

Note

Labels can entered by the keyboard or picked by the mouse.

Example: `UNSELPIC, EL, 1, 3, 6, 5`

The above command removes elements 1, 3, 6, and 5 from element selection set number 5.

UNSELWIN

Control > UNSELECT > by Windowing

The UNSELWIN command removes members from the active selection list for a particular entity by forming a window surrounding the region of interest. Rectangular, circular or polygon windows can be specified by using the mouse. All plotted members of the specified entity within the window are removed from the active

selection set. If the specified set selection does not exist, it will be initiated and activated as if it originally existed and contained all members of that entity.

Entry & Option Description

Entity name

Selection entity name. The admissible entities are:

PT	keypoints
PH	polyhedra
PA	parts
CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
ND	nodes
EL	elements
ALL	all entities
	(default is EL)

Window type

Type of window.

= 0: Box	rectangular window specified by two opposite corners using the cross-hairs and the mouse
= 1: Circle	circular window specified by a center and a point on the circumference picked by mouse
= 2: Polygon	polygon window drawn using the mouse in a rubber band fashion. Type “c” to close the polygon, “d” to delete the last segment, or the Escape key to abort. *(default is box)*

Selection set number

Set number.
(default is the active set)

Notes

1. Only plotted members can be unselected.
2. A member is selected if its center (or label) lies inside the window.
3. If the polygon option is used, the polygon must be closed. The polygon is closed if a selected pixel is within two pixels from the first one. Otherwise the user may type “c” to have GEOSTAR close the polygon by connecting the last and the first pixels. Typing the character “d” deletes the last side of the polygon. The Escape key can be used to abort the command at any time.

UNSELREF

Control > UNSELECT > by Reference

The UNSELREF command removes members from the selection list of a particular entity. All members of this entity that are associated with a pattern of a reference

entity are removed. If no selection list is active, the command will activate, and operate on selection set number 1.

Entry & Option Description

Selection entity

Selection entity name. The admissible entities are:

PT	keypoints
PH	polyhedra
PA	parts
CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
ND	nodes
EL	elements
	(default is EL)

Reference entity

Reference entity name. The admissible entities are as stated above.

Beginning reference entity

Beginning label in the reference entity pattern.

Ending reference entity

Ending label in reference entity pattern.
(default is beginning reference entity)

Increment

Increment between labels in the reference entity pattern.
(default is 1)

Boundary element flag

Boundary flag.

= 0: No	select members associated with the pattern of reference entity
= 1: Yes	include in addition, all members that have common boundaries with the selected entity

Note

Beginning and ending labels of the pattern can be selected by the mouse.

Example: `UNSELREF, EL, CR 1, 3, 1`

The above command removes all elements associated with curves 1 through 3 from the current element selection list.

UNSELRANGE

Control > UNSELECT > by Range

The UNSELRANGE command removes from a selection list of a particular entity, all such members that lie inside a 3D window. The window is defined by specifying the X-, Y-, and Z-coordinate bounds. The specified selection list becomes active.

Entry & Option Description

Entity name

Selection entity name. The admissible entities are:

PT	keypoints
PH	polyhedra
PA	parts
CR	curves
SF	surfaces
VL	volumes
RG	regions
CT	contours
ND	nodes
EL	elements
ALL	all entities

(default is EL)

Coordinate system

Coordinate system to be used in specifying the range.
(default is 0 for global Cartesian)

Coordinate activation flag for x

Coordinate activation flag for bounds in the X-direction.

= 1: Activate	activate the X-direction bounds
= 0: Deactivate	deactivate the X-direction bounds

(default is activate)

Coordinate activation flag for y

Coordinate activation flag for bounds in the Y-direction.

= 1: Activate	activate the Y-direction bounds
= 0: Deactivate	deactivate the Y-direction bounds

(default is activate)

Coordinate activation flag for z

Coordinate activation flag for bounds in the Z-direction.

= 1: Activate	activate the Z-direction bounds
= 0: Deactivate	deactivate the Z-direction bounds

(default is activate)

x-coordinate lower range

Lower bound for X-coordinate.
(default is minimum X-extent)

x-coordinate higher range

Upper bound for x-coordinate.




(default is maximum X-extent)

y-coordinate lower range
Lower bound for Y-coordinate.
(default is minimum Y-extent)

y-coordinate higher range
Upper bound for Y-coordinate.
(default is maximum Y-extent)

z-coordinate lower range
Lower bound for Z-coordinate.
(default is minimum Z-extent)

z-coordinate higher range
Upper bound for Z-coordinate.
(default is maximum Z-extent)

Selection set number
Selection set number.
(default is the active set)

Note
The X-, Y- and Z-bounds are prompted only when the corresponding activation flag is turned on (1). If the activation flag for a coordinate is deactivated, then the window is infinite in that direction.

UNESELPROP

Control > UNSELECT > Element by Property

The UNESELPROP command removes elements associated with a pattern of specified element groups, material property, or real constant sets from the active selection list. If the specified set selection does not exist, it will be initiated and activated as if it originally existed and contained all members of that entity.

Entry & Option Description

Selection property name
Selection property type. Admissible types are:

EG	element group
MP	material property set
RC	real constant set

(default is EG)

Beginning set
Beginning set number in the pattern.

Ending set
Ending set number in the pattern.
(default is beginning set)

Increment
Increment between the set numbers in the pattern.




(default is 1)

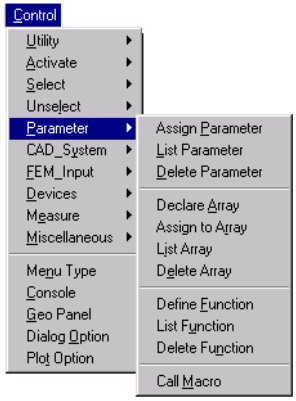
Selection set number
Selection set number.
(default is the active set)

Example: `UNESELPROP, RC, 1, 3, 1, 6`
The above command removes all elements associated with real constant sets 1, 2, and 3 from the selection set number 6.

PARAMETER Menu

Control > PARAMETER

Figure 8-6 Parameter Menu



This menu contains commands to define, list, and delete parametric variables, functions and arrays, in addition to calling macros. Refer to COSMOS/M Command Language appendix in the COSMOS/M User Guide for details.

PARASSIGN

Control > PARAMETER > Assign Parameter

The PARASSIGN command assigns a numeric value for a parametric variable, by specifying its name, type and value. The defined parameters can be subsequently used in response to command prompts, or to define other variables, arrays, functions, or macros (refer to the COSMOS/M Command Language appendix in the COSMOS/M User Guide). Many variables are readily defined by GEOSTAR and may be directly used; a list of all predefined variables is given below.

Warning: The string “C*” is treated to indicate a comment whenever used in expressions. It is advisable to avoid using variable names ending with the character “C”. If such names are used, use parenthesis to separate the “C” and the “*”.

Entry & Option Description

Parameter name
Parametric variable name. The name must start with an alphabetic character, up to 10 alpha-numeric characters may be used. The valid characters are letters of the alphabet and digits 0 through 9.

Data type
Parameter type. The valid types are INT (for integer) and REAL.
(default is REAL)

Parametric real value
Parameter numeric value. An arithmetic expression that uses previously defined

parameters is also acceptable.

Notes

1. Spaces are allowed in arithmetic expressions, only if parenthesis are used (see example 3 below).

2. The admissible arithmetic operations are:
   + addition,
   - subtraction,
   * multiplication,
   / division, and
   ^ power. (e.g. x^2 is x squared)

3. The following is a list of predefined parametric variables:

PTMAX	highest keypoint label in the database
CRMAX	highest curve label in the database
SFMAX	highest surface label in the database
VLMAX	highest volume label in the database
NDMAX	highest node label in the database
ELMAX	highest element label in the database
CTMAX	highest contour label in the database
RGMAX	highest region label in the database
PHMAX	highest polyhedron label in the database
PAMAX	highest part label in the database
CSMAX	highest coordinate system label in the database
EGMAX	highest element group label in the database
MPMAX	highest material property set label in the database
RCMAX	highest real constant set label in the database
EGCUR	active element group label
MPCUR	active material property set
RCCUR	active real constant set
LCCUR	active load
CSCUR	active coordinate system label
SELCUR	active selection list
MAXPR	number of user-defined parameters in the database
MAXARR	number of user-defined arrays in the database
MAXFUNC	number of user-defined functions in the database
CURPOST	current label of active analysis for postprocessing (refer to the ACTPOST command)
ON	fixed, the user may use “on” for “1” (convenient for on/off flags)
OFF	fixed, the user may use “off” for “0” (convenient for on/off flags)
YES	fixed, the user may use “yes” for “1” (convenient for on/off flags)
NO	fixed, the user may use “no” for “0” (convenient for on/off flags)

Example 1: `PARASSIGN, radius, real, 6.5`

This command defines the parameter radius and assigns a numeric

value of 6.5 to it.

Example 2:
```
PARASSIGN, a1,, 3.0
PARASSIGN, a2,, 2.0*(a1 + 4.)+1.2
PT        ,,(a1*2. + a2), a1/2.0, a1+a2-1.0
```
The first command assigns a value of 3.0 to the parameter a1. The second assigns a value of 15.2 for the new parameter a2. The third command specifies the coordinates of a newly created keypoint using arithmetic expressions that include the defined parameters a1 and a2.

Example 3:
```
PARASSIGN, x, real, 12.5
PT        ,, 2.0*x,  x +1.0     5.0 -x
PT        ,, 2.0*x  (x + 1.0)  (5.0  -  x)
```
The first command assigns a value for x. The second command uses x to assign values of 25.0, 12.5 and 1.0 to the x-, y-, and z-coordinates of the newly created keypoint, respectively. The last expression (5.0 -x) is ignored. This is because space between x and 1.0 is used by GEOSTAR like a comma to separate entries. The third command specifies the x-, y-, and z-coordinates of a new keypoint as 25.0, 13.5, and -7.5 respectively.

Example 4:
```
PARASSIGN, dnel, int, (ndmax-elmax)
```
The above command assigns the difference between the highest node label and highest element label in the database to dnel. The value may be listed using the PARLIST command.

PARLIST

Control > PARAMETER > List Parameter

The PARLIST command lists current values of parametric variables defined by the PARASSIGN command. If a name is not given, the command lists the serial numbers, names, types, and values of all defined paramedic variables.

Entry & Option Description

Parameter name
Parametric variable name.
*(default is *, for all variables)*

PARDEL

Control > PARAMETER > Delete Parameter

The PARDEL command deletes a parametric variable from the database. The variable must have been previously defined by the PARASSIGN command. If no name is specified, the command deletes the last parametric variable defined.

Entry & Option Description

Parameter name

Parameter name.

Example: `PARDEL, a1`

This command deletes parameter a1 from the database.

ARRDEF

Control > PARAMETER > Declare Array

The ARRDEF command declares an array by specifying its name, dimensions, and data type. The array name should not be one of the reserved names. One, two, or three dimensional arrays may be defined. Elements of the array may be defined by the ARRASSIGN command. The name of the array must start with an alphabetic character and may have up to 10 alpha-numeric characters. An array must be deleted before it can be redefined.

Entry & Option Description

Array declaration

Array definition. The user must enter the name, followed by an opening bracket, the maximum size of the array in the first direction, and a closing bracket. If a 2D or 3D array is being declared, then the size in the second and third direction should immediately follow between brackets. For example “aaa [10][20][20]” declares “aaa” to be a 3D array of size 10 by 20 by 20.

Data type

Type of data in the array.

= REAL	real (floating) numbers
= INT	integer numbers
	(default is real)

Example: `ARRDEF, S1 [6][6],,,`

The above command declares s1 to be a 6 x 6 matrix.

ARRASSIGN

Control > PARAMETER > Assign to Array

The ARRASSIGN command assigns a value for an element in an array that was already declared by the ARRDEF command. The values in an array may be listed using the ARRLIST command.

Entry & Option Description

Array element

Element of the array. For example S1[1][3] refers to the element in the first row and third column of array S1.

Parametric real value

Value to be assigned to the above element.

ARRLIST

Control > PARAMETER > List Array

The ARRLIST command lists arrays defined by the ARRDEF and ARRASSIGN commands.

Entry & Option Description

Array name
Array name.
*(default is * for all arrays)*

ARRDEL

Control > PARAMETER > Delete Array

The ARRDEL command deletes arrays defined by the ARRDEF and ARRASSIGN commands.

Entry & Option Description

Array name
Array name.

FUNCDEF

Control > PARAMETER > Define Function

The FUNCDEF command defines a function. The function may use a maximum of 20 arguments in its definition. Many functions are readily defined by GEOSTAR and may directly be used. A list of predefined functions is given below. The names of predefined functions are reserved and may not be redefined by the user. A user-defined function must be first deleted before it can be redefined.

Entry & Option Description

Function name
Function name. The name may consist of 1 to 10 alpha-numeric characters, first character must be alphabetic. Cannot use reserved function names or names of functions already defined by the user.

Function type
Function type.

= REAL	real (returns with floating number)
= INT	integer (returns with integer number)
	(default is real)

Function expression
Function definition. The function may be defined by an expression that uses:

1. Constants,




2. Predefined functions (refer to the FUNCDEF command), and
3. Other user-defined functions.

Note

The following is a list of predefined function names and their description:

I. General:

COS(x)	trigonometric cosine of x (x in radians)
SIN(x)	trigonometric sine of x (in radians)
TAN(x)	trigonometric tangent of x (in radians)
ACOS(x)	trigonometric arc cosine of x (returns angle in radians)
ASIN(x)	trigonometric arc sine of x (returns angle in radians)
ATAN(x)	trigonometric arc tangent of x (returns angle in radians)
COSH(x)	hyperbolic cosine of x (x in radians)
SINH(x)	hyperbolic sine of x (x in radians)
TANH(x)	hyperbolic tangent of x (x in radians)
SQRT(x)	square root of x
ABS(x)	absolute value of x
EXP(x)	exponential to x
LOG(x)	natural logarithm (base e)
LOG10(x)	base-10 logarithm
FLOOR(x)	truncates the decimal part of a floating point number (5.99 returns 5.0)
CEIL(x)	returns with the closest higher integer to x (5.01 returns 6.0 and -5.3 returns -5.0)
RAND(x1|x2)	generates a random number between x1 and x2
SIGN(x)	sign of a variable (-1 for negative, +1 for positive, and 0 for 0)
XND(nd|cs)	x-coordinate of node “nd” in coordinate system “cs”
YND(nd|cs)	y-coordinate of node “nd” in coordinate system “cs”
ZND(nd|cs)	z-coordinate of node “nd” in coordinate system “cs”
XPT(pt|cs)	x-coordinate of keypoint “pt” in coordinate system “cs”
YPT(pt|cs)	y-coordinate of keypoint “pt” in coordinate system “cs”
ZPT(pt|cs)	z-coordinate of keypoint “pt” in coordinate system “cs”
NEARND(x|y|z)	nearest node to the location specified by the x-, y- and z-values in the global Cartesian coordinate system
NEARPT(x|y|z)	nearest keypoint to the location specified by the x-, y- and z-values in the global Cartesian coordinate system
NDEL(el|pos)	node in location “pos” of element “el”, NDEL(3|2) returns with the second node in element 3
EXIST(en|label)	returns with true (or 1) if entity type “en” with label “label” exists in the database. Valid entity names are “PT”, “CR”, “SF”, “VL”, “CT”, “RG”, “PH”, “PA”, “ND” and “EL”
LISTSEL(en|label)	returns with true (or 1) if entity of type “en” with label “label” is currently accessible (i.e. the entity is in the current selection list. It will return with true if no selection lists were defined)
ELFACE(el|en|label)	returns with the face number of element “el” which is associated with entity number “label” of type “en”. Valid entries for “en” are “CR”, “SF”, and “RG”
XELF(el|fnum|cs)	returns with the x-coordinate of the center of gravity for face

number “fnum” of element “el” in coordinate system number “cs”. If “0” is given for “fnum”, the function returns with the x-coordinate of the center of gravity of the element

YELF(el|fnum|cs) returns with the y-coordinate of the center of gravity for face number “fnum” of element “el” in coordinate system number “cs”. If “0” is given for “fnum”, the function returns with the y-coordinate of the center of gravity of the element

ZELF(el|fnum|cs) returns with the z-coordinate of the center of gravity for face number “fnum” of element “el” in coordinate system number “cs”. If “0” is given for “fnum”, the function returns with the z-coordinate of the center of gravity of the element

II. Predefined Postprocessing Functions:

These names can not be defined by the FUNCDEF command, they are readily defined and may be used after the results are calculated. All functions return with the absolute maximum, algebraic maximum, or algebraic minimum if “0”, “-1”, or “-2” is entered for a node or element label, respectively.

Refer to the on-line help or the COSMOS/M User Guide for valid entries for different analyses.

Example:

```
FUNCDEF, SS(X|Y), REAL, SQRT((X**2 + Y**2))
PARASSIGN, S1, REAL, 2.0*SS(3.0|4.0)
```

The first command defines SS to be a function of x and y, such that SS is the square root of the sum of the squares of x and y. The second command uses function SS to define S1, S1 will be set equal to 10.0 (2.0* 5.0).

Refer to the *COSMOS/M Command Language* Appendix in the COSMOS/M User Guide for more examples and details.




FUNCLIST

Control > PARAMETER > List Function

The FUNCLIST command lists previously defined user functions. Function number, name, type and associated expression are listed.

Entry & Option Description

Function name
Function name. Any user defined function name may be specified.
(default is "", to list all functions)*

FUNCDEL

Control > PARAMETER > Delete Function

The FUNCDEL command deletes previously defined user functions.

Entry & Option Description

Function name
Function name. Any user defined function name may be specified.

CALLMACRO

Control > PARAMETER > Call Macro

The CALLMACRO command executes a macro. A macro is a series of COSMOS/M command language statements and GEOSTAR commands. The called macro must have been defined in the "GEOMACRO.MAC" file in the COSMOS/M sub-directory, and should appear as a block that starts with "$macro name list of arguments", and ends with "$endm". General programming rules should be followed. Nested macros are allowed. The number of control statements in a macro should not exceed 500. The syntax of valid COSMOS/M command language control statements is as follows:

#LOOP name nmax	(for starting a loop from 1 to nmax in increments of 1)
#LABEL name	(to end the above loop)
#GOTO name	(go to statement #LABEL name)
#IF condition	(if true, following statements are executed until ended by corresponding #ELSEIF, #ELSE, or #ENDIF statement)
#ELSEIF condition	(if true following statements are executed until ended by corresponding #ELSEIF, #ELSE, or #ENDIF statement)
#ELSE	(following statements are executed until ended by corresponding #ENDIF statement)
#ENDIF	(to end a condition if block)

For details and examples, refer to the *COSMOS/M Command Language* Appendix in the COSMOS/M User Guide.

Entry & Option Description

Input macro name
Name of the macro.

Input argument variable (i)
List of arguments.

Notes

1. Numeric constants can not be passed as arguments to the macro, but the desired value may be assigned to a parametric variable using the PARASSIGN command which can then be passed to the macro.
2. Argument values are passed back to GEOSTAR, but other parameters used in the macro are local (just like a subroutine).
3. All predefined parametric variables (like NDMAX), and predefined functions (like SIN(X)), are accessible to macros.

Example 1:
```
Callmacro maxvon lc node value
```
The above command calls macro “maxvon” which is included in file “GEOMACRO.MAC”, as an example to find the maximum nodal von Mises stress and corresponding node.

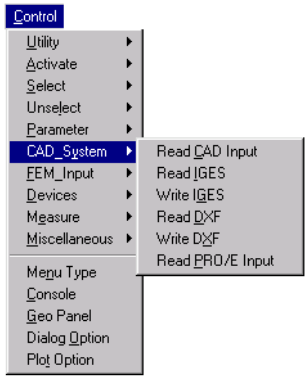
Example 2:
```
PARASSIGN csid INT 1
PARASSIGN fac REAL 10.0
CALLMACRO funcbc csid fac
```
The first command assigns 1 to parametric variable csid. The second command assigns 10.0 to parametric variable fac. The two values are then passed to the predefined “funcbc” which may be used to apply variable forces to nodes in a given coordinate system. (Refer to the COSMOS/M User Guide for details.)




CAD_SYSTEM Menu

Control > CAD_SYSTEM

Figure 8-7 CAD System Menu



This menu contains commands to read or generate IGES and DXF format files to interact with CAD packages. The IGES format is in general preferred over the DXF format for use in finite element model generation because of more compatibility in the representation of entities.

CAD_INP

Control > CAD_SYSTEM > Read CAD Input

The CAD_INP command reads-in an IGES file which contains NURBS-based (Non Uniform Rational B-Splines) trimmed surface geometry for solid- or shell-type models. The command generates equivalent GEOSTAR geometry which may include parts and polyhedra unlike the IGES_INP command which only translates wire-frame geometry.

After reading-in the file to GEOSTAR, you may continue with meshing and other preprocessing as usual.

Entry & Option Description

CAD system

Source CAD system of the IGES file.

= Generic	generic CAD system
= Pro/E 1-17	Pro/ENGINEER Versions 1-17 (PTC)
= BSI-Modeler	MicroStation Modeler from Bentley Systems Inc.
= SolidWorks	SolidWorks
= CV-CADDS5	CADDS5 (ComputerVision)
= Unigraphics	Unigraphics (EDS)
= Pro/E 18+, PT/M 2+	Pro/ENGINEER Version 18 or later and PT/Modeler Version 2 or later (PTC)
= Helix	Helix Design System (MICROCADAM)
= Eureka	Eureka (Cad.Lab)
= I-DEAS	I-DEAS (SDRC)
	(default is Generic)

File name

File name.

IGES_INP

Control > CAD_SYSTEM > Read IGES

The IGES_INP command reads in an IGES file, created by other CAD packages, and translates it into a wireframe within the GEOSTAR environment. GEOSTAR can then complete the model generation and perform the finite element meshing process. A report will be generated in a file with the problem name and extension RPT.

Entry & Option Description

Input file name without extension
Name of the file.

Note
Refer to the *Translators* Appendix in the COSMOS/M User Guide for a list of the entities that are supported by the translator.

Example: `IGES_INP, TEST.IGS`
This command reads in from file TEST.IGS, the geometric data generated by other CAD packages and translates it into GEOSTAR database.

IGES_OUT

Control > CAD_SYSTEM > Write IGES

The IGES_OUT command translates the geometry of the present model to other CAD packages and writes the translation to the specified file.

Entry & Option Description

Output file name
Name of the file.
(default is problem name with extension .IGS)

Note
Refer to the *Translators* Appendix in the COSMOS/M User Guide for a list of the entities that are supported by the translator.

Example: `IGES_OUT, TEST.IGS`
This command creates file TEST.IGS, containing a translation of the geometry of the present model to CAD packages. The file can be used as a CAD file.

DXF_INP

Control > CAD_SYSTEM > Read DXF

The DXF_INP command reads in a DXF file, created by other CAD packages, and translates it into a wireframe within the GEOSTAR environment. GEOSTAR can then complete the model generation and perform the finite element meshing process. A report will be generated in a file with the problem name and extension RPT.




Entry & Option Description

Input file name without extension
Name of the file.

Note
Refer to the *Translators* Appendix in the COSMOS/M User Guide for a list of the entities that are supported by the translator.

Example: `DXF_INP, TEST.DXF`
This command reads in from file TEST.DXF, the geometric data generated by other CAD packages and translates it into GEOSTAR database.

DXF_OUT

Control > CAD_SYSTEM > Write DXF

The DXF_OUT command translates the geometry of the present model to other CAD packages and writes the translation to the specified file in the DXF form.

Entry & Option Description

Output file name
Name of the file.
(default is problem name with extension .DXF)

Note
Refer to the *Translators* Appendix in the COSMOS/M User Guide for a list of the entities that are supported by the translator.

Example: `DXF_OUT, TEST.DXF`
This command creates file TEST.DXF, containing a translation of the geometry of the present model to CAD packages. The file can be used as a CAD file.

PRO_INP

Control > CAD_SYSTEM > Read Pro/E Input

The PRO_INP command reads in Pro/ENGINEER geometry from an ".ntr" file and creates equivalent GEOSTAR geometric entities like regions, polyhedra, and parts. Refer to the *Translators* Appendix in the COSMOS/M User Guide for details.

Entry & Option Description

Input file name without extension
Name of the file.

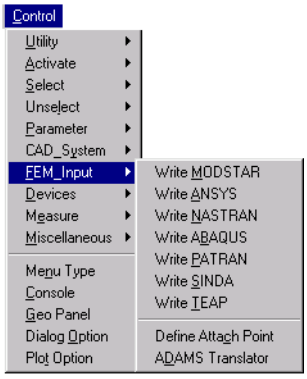
Example: `PRO_INP, TEST.ntr`
This command reads in Pro/ENGINEER geometry from the file TEST.ntr.

FEM_INPUT Menu

Control > FEM_INPUT

This menu includes commands to generate input data files that can be used in ANSYS, NASTRAN, PATRAN, ABAQUS, SINDA, and TEAP.

Figure 8-8 FEM_Input Menu



MODINPUT

Control > FEM_INPUT > Write MODSTAR

The MODINPUT command writes an input file with extension MOD. The file can be used to regenerate the model MODSTAR.

Entry & Option Description

Mod input file name without extension
Name of the file with extension. The recommended extension is “mod”. *(default file name is problem name with extension “.mod”)*

Node offset
Node number offset. The node numbers in the GEOSTAR model will be incremented by this value while writing the MODSTAR input file.

Element offset
Element number offset. The element numbers in the GEOSTAR model will be incremented by this value while writing the MODSTAR input file.

Coordinate system offset
Coordinate system number offset. The coordinate system numbers in the GEOSTAR model will be incremented by this value while writing the MODSTAR input file.

Example: `MODINPUT, TEST.MOD, 100, 50, 10`

This command writes MODSTAR command input file TEST.MOD for the model created in GEOSTAR. The model node numbers, element numbers and coordinate system numbers are incremented by 100, 50, and 10 respectively while writing the MODSTAR input file.

ANSYSINP

Control > FEM_INPUT > Write ANSYS

The ANSYSINP command generates a file with extension ANS that can be used as an input file to regenerate the model in ANSYS. The ANSYS translator is required for this command to work.

Entry & Option Description

Node offset
Node numbering offset.
(default is 0)

Element offset
Element numbering offset.
(default is 0)

Note
Nodes and elements in ANSYS will be numbered by adding the specified offsets to their corresponding labels in GEOSTAR.

NASTRANINP

Control > FEM_INPUT > Write NASTRAN

The NASTRANINP command generates a file with extension DAT that can be used as an input file to regenerate the model in NASTRAN. The NASTRAN translator is required for this command to work.

Entry & Option Description

Node offset
Node numbering offset.
(default is 0)

Element offset
Element numbering offset.
(default is 0)

Output format flag
Output format flag.

= 0: Short and free	short form, free format
= 1: Long and free	long form, free format
= 2: Short and fixed	short form, fixed format
= 3: Long and fixed	long form, fixed format
	(default is Short and Free)

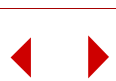
Note
Nodes and elements in NASTRAN will be numbered by adding the specified offsets to their corresponding labels in GEOSTAR.

ABAQUSINP

Control > FEM_INPUT > Write ABAQUS

The ABAQUSINP command generates a file with extension INP that can be used as an input file to regenerate the model in ABAQUS. The ABAQUS translator is required for this command to work.

Entry & Option Description



Node offset
Node numbering offset.
(default is 0)

Element offset
Element numbering offset.
(default is 0)

Note
Nodes and elements in ABAQUS will be numbered by adding the specified offsets to their corresponding labels in GEOSTAR.

PATRANINP

Control > FEM_INPUT > Write PATRAN

The PATRANINP command generates a file with extension NEU that can be used as an input file to regenerate the model in PATRAN. The PATRAN translator is required for this command to work.

Entry & Option Description

Node offset
Node numbering offset.
(default is 0)

Element offset
Element numbering offset.
(default is 0)

Note
Nodes and elements in PATRAN will be numbered by adding the specified offsets to their corresponding labels in GEOSTAR.

SINDAINP

Control > FEM_INPUT > Write SINDA

After all input for a COSMOS/M thermal problem has been entered, the SINDAINP command generates a data input file (with extension SND) for use with the SINDA program. This is useful when it is desirable (or necessary) to present thermal analysis results from SINDA as well as from HSTAR, or TEAP. The generated file is an ASCII file that can be modified with a text editor if additional input is necessary. The results of the SINDA analysis can then be entered into the database for display by GEOSTAR or PLOTSTAR by downloading the SINDA output file and running the translator from the system level (see note below).

The SINDA input file contains the necessary NODE, SOURCE, and CONDUCTOR data sections for SINDA in addition to ARRAY data when variable properties are considered. (The SPV, and SIV data sections are generated automatically.) (This

SINDA input file can also be executed by TEAP from the system prompt if desired.) The SINDA translator is required for this command to work.

Entry & Option Description

Node offset
Node numbering offset.
(default is 0)

Element offset
Element numbering offset.
(default is 0)

Notes

1. The generated file is called problem_name.SND. The user can change the extension of the generated file to .DAT so that it can be read by SINDA.
2. Nodes and elements in SINDA will be numbered by adding the specified offsets to their labels in GEOSTAR.
3. The thermal network that is generated through the SINDAINP command places the nodes in the exact locations as specified in GEOSTAR. The nodes are placed on the corners of the elements rather than at the centroids. This type of network is usually more accurate and eliminates the need for calculating the boundary nodes in the problem. It also enables the use of COSMOS/M graphics to display results calculated by the SINDA program.
4. The translator can also be executed from the system level. Two file names need to be entered in this case. The COSMOS/M problem name is entered without an extension while an extension can be entered can be specified for the SINDA file name.
5. The command line syntax for writing a SINDA input file from the system level is:
   WS3 (COSMOS/M_problem_name 'no extension') (SINDA_input_file)
6. The command line syntax for placing SINDA output in COSMOS/M database from the system level is:
   WS3 (SINDA_output_file_name '.OP extension')
   (COSMOS/M_problem_name 'no extension')

TEAPINP

Control > FEM_INPUT > Write TEAP

The TEAPINP command generates a file that can be used as an input file to regenerate the model in the TEAP program. The TEAP translator is required for this command to work.

NDATTACH

Control > FEM_INPUT > Define Attach Points

The NDATTACH command defines attachment points for use by the R_MDITRANS command. Attachment points are used by the ADAMS program to attach components of a mechanical system to each other.

Entry & Option Description

Beginning node
Beginning node in a pattern to define attachment nodes.

Ending node
Ending node in a pattern to define attachment nodes.

Increment
Increment between nodes in the pattern.
(default is 1)

Attachment flag
Attachment flag.

= 0: No	no attachment
= 1: Yes	attachment
	(default is yes)

Example:
```
NDATTACH, 2, 10, 3,
```
The above command defines nodes 2, 5, and 8 as attachment points.

R_MDITRANS

Control > FEM_INPUT > ADAMS Translator

The R_MDITRANS command runs the 2-way COSMOS/M-ADAMS translator. The command generates a universal file from a GEOSTAR model, or generates a file with extension GED from a universal file which has the fixed name and extension “adams.uni”.

The universal file is used by ADAMS, a leading program for the modeling and analysis of kinematics from Mechanical Dynamics. The utility enables users of ADAMS to include the elastic effects in their studies of the kinematics of mechanisms. On the other hand, forces from ADAMS may be exported to GEOSTAR.

When requested to generate a universal file, the command automatically runs STAR to calculate the stiffness and mass matrices and writes them to the universal file along with other information required by ADAMS. Attachment nodes will be included in the universal file if defined by the NDATTACH command prior to running the translator.




When requested to generate a GED file, the command reads the universal file “adams.uni” and generates “problem-name.ged” which is in the format of GEOSTAR.

Entry & Option Description

Option flag

Translation flag.

= 1: cos>univ — generate a universal file from the current database (extension uni)

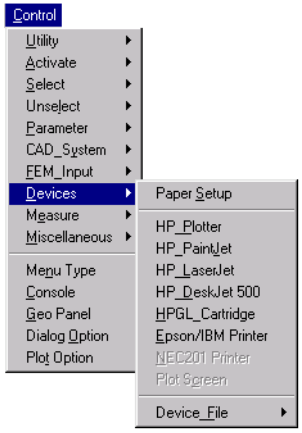
= 2: univ>cos — generate a GEOSTAR (extension GED) input file from the universal file “adams.uni”
(default is cos>univ*)*

DEVICES Menu

Control > DEVICES

It is suggested to use the Windows print utility (File, Print) command instead of commands in this menu.

Figure 8-9 Device Menu



This menu contains commands to directly send screen images to a configured plotter or printer. A menu is provided for the creation, plotting and viewing of image files in various formats. If you have problems generating hardcopies, please check the following:

1. Verify that the device is connected to the correct port.
2. Make sure that GEOSTAR has been correctly configured for the device connected to your computer. The device must be one of the devices specified in the configuration program. If you are using a device that emulates one of the supported devices, make sure that the proper setting of switches has been made for the emulation mode.
3. If you are using HP PAINTJET or HP LASERJET, verify that the DOS file 'MODE.COM' is copied to COSMOS/M directory.
4. Self-test the device.
5. Verify that the port (which the printer is connected to), is active and the cable is operational. This can be done by printing a small file using the DOS PRINT command or the PRINT SCREEN option.

6. If you are plotting from a file (MULTIPRINT, PLT_META or PLT_FILE commands), verify that you are using the appropriate command that corresponds to the file format, and your device supports this format.
7. Note that HPGL files generated for printers can only be output on printers, and HPGL plotter files can only be plotted on plotters.
8. A stand-alone program to generate hardcopies for image files in various formats is supplied with the program. This program, called PLOTFILE, can be called from the operating system level and supports a wide range of devices. The formats supported by this program are PostScript, HPGL and META. The program can be executed by typing PLOTFILE from the operating system level.

PAPER_SETUP

Control > DEVICES > Paper Setup

The PAPER_SETUP command sets the print mode, paper size, margins and logo flag for use with hardcopy devices. User-defined logos previously saved from GEOSTAR images as meta files may be used to replace the standard COSMOS/M logo. A flag to activate the printing of multiple windows is also provided.

Entry & Option Description

Paper size

Paper size.

= A: (34x44)	(34x44)
= B: (22x34)	(22x34)
= C: (17x22)	(17x22)
= D: (11x17)	(11x17)
= E: (8.5x11)	(8.5x11)
	(default is 8.5x11)

Media type

Media type.

= 0: Paper	paper
= 1: Transparency	transparency
	(default is paper)

Orientation

Print mode.

= 0: landscape	landscape
= 1: Portrait	portrait
	(default is 0)

Left margin

Location of left margin. (left end is 0.0, right end is 1.0)
(between 0.0 and 0.9, default is 0.1)

Right margin

Location of right margin. (left end is 0.0, right end is 1.0)
(between left margin + 0.05 and 1.0, default is 0.9)

Bottom margin

Location of bottom margin. (top end is 1.0, bottom end is 0.0)
(between 0.0 and 0.9, default is 0.1)

Top margin
Location of top margin. (top end is 1.0, bottom end is 0.0)
(between bottom margin + 0.05 and 1.0, default is 0.9)

Logo print
Standard logo print flag.

= 1: Yes	print logo
= 0: No	do not print logo
	(default is print logo)

(Logo file)
File name (in the current working directory) to be used for the logo. The user may create this file by making a model to represent the logo in GEOSTAR and then using the METAFILE command to save the desired image.
(default is user logo which contains the COSMOS/M standard logo)

Window print
Multiple window flag. Not applicable to Unix Platforms. You can use the SCREENPLOT command to plot multiple windows.

= 0: Single	use the active window only
= 1: Multiple	use multiple windows
	(default is single)

PLOTTER

Control > DEVICES > HP_Plotter

The PLOTTER command generates a hardcopy for the image in the active window using a configured HP plotter. The command provides an option to generate a file that can be subsequently plotted from the operating system level.

Entry & Option Description

Device
Plotter type.

= 1	HP7475A
= 2	ColorPro
= 3	HP7450
= 4	DraftMaster
	(default is 2)

Resolution
Number of dots per inch. (between 25 and 500 dots/in)
(default is 72)

Communication mode
Communication mode.

= 0	generate a hardcopy only
= 1	generate a file only
= 2	generate both a hardcopy and a file




(default is 0)

(File)
File name if the mode option is set to 1 or 2.
(default is 'problem_name.Aij', where 'i' is a digit that ranges from 0 to 9, and 'j' ranges from 1 to 9 to act as a counter)

Plot option
Plot option.

= 0	line
= 1	fill

Notes

The generated file may be plotted from the system level as follows:

a. For the PC version, use the copy/b command. For example 'copy/b file-name lpt1:' will send the file to a printer connected to the lpt1: port.
b. For Workstations, use the regular printing command.

PAINTJET

Control > DEVICES > HP_PaintJet

The PAINTJET command generates a hardcopy for the image in the active window using a configured HP PAINTJET printer. The command provides an option to generate a file that can be subsequently printed from the operating system level.

Entry & Option Description

Resolution
Number of dots per inch.

= 90	90 dots/in.
= 180	180 dots/in.
	(default is 90)

Mode
Communication mode.

= 0	generate a hardcopy only
= 1	generate a file only
= 2	generate both a hardcopy and a file
	(default is 0)

(File)
File name if the mode option is set to 1 or 2.
(default is 'problem_name.Aij', where 'i' is a digit that ranges from 0 to 9, and 'j' ranges from 1 to 9 to act as a counter)

Note

The generated file may be printed from the system level as follows:

a. For the PC version, use the copy/b command. For example 'copy/b file-name lpt1:' will send the file to a printer connected to the lpt1 port.
b. For Workstations, use the regular printing command.




LASERJET

Control > DEVICES > HP_LaserJet

The LASERJET command generates a hardcopy for the image in the active window using a configured HP LASERJET printer. The command provides an option to generate a file that can be subsequently printed from the operating system level.

Entry & Option Description

Resolution

Number of dots per inch.

= 75	75 dots/in.
= 150	150 dots/in.
= 300	300 dots/in.
	(default is 150)

Mode

Communication mode.

= 0	generate a hardcopy only
= 1	generate a file only
= 2	generate both a hardcopy and a file
	(default is 0)

(File)

File name if the mode option is set to 1 or 2.
(default is 'problem_name.Aij', where 'i' is a digit that ranges from 0 to 9, and 'j' ranges from 1 to 9 to act as a counter)

Note

The generated file may be printed from the system level as follows:

a. For the PC version, use the copy/b command. For example 'copy/b file-name lpt1:' will send the file to a printer connected to the lpt1 port.
b. For Workstations, use the regular printing command.

DESKJET

Control > DEVICES > HP_DeskJet 500

The DESKJET command generates a hardcopy for the image in the active window using a configured HP_DESKJET 500 C printer. The command provides an option to generate a file that can be subsequently printed from the operating system level.

Entry & Option Description

Deskjet type

Deskjet type.

= 1	color
= 2	black and white
	(default is 1)

Resolution

Number of dots per inch.

= 75 75 dots/in.
= 100 100 dots/in.
= 150 150 dots/in.
= 300 300 dots/in.
(default is 150)

Communication mode
Communication mode.
= 0 generate a hardcopy only
= 1 generate a file only
= 2 generate both a hardcopy and a file
(default is 0)

(File)
File name if the mode option is set to 1 or 2.
(default is 'problem_name.Aij', where 'i' is a digit that ranges from `0 to 9, and 'j' ranges from 1 to 9 to act as a counter)

Note

The generated file may be printed from the system level as follows:

a. For the PC version, use the copy/b command. For example 'copy/b file-name lpt1:' will send the file to a printer connected to the lpt1 port.
b. For Workstations, use the regular printing command.

HPGL_CRTRG

Control > DEVICES > HPGL_Cartridge

The HPGL_CRTRG command generates a hardcopy for the image in the active window using a configured printer or plotter with an HPGL/2 cartridge.

Entry & Option Description

Device
Device type.
= 0 printer
= 1 plotter
(default is 0)

Note

HPGL cartridges on plotters may not generate appropriate color-filled or shaded plots. Use the PLOTTER command directly from GEOSTAR instead.

EPSON_IBM

Control > DEVICES > EPSON/IBM Printer

The EPSON_IBM command generates a hardcopy for the image in active window using a configured EPSON, JDL or IBM printer. JDL printers must be configured

in the EPSON mode. The command provides an option to generate a file that can be subsequently printed from the operating system level.

Entry & Option Description

Epson device
Printer type.

= 1	B/W
= 2	color
= 3	JDL750
= 4	JDL850
= 5	IBM color
	(default is 4)

Communication mode
Communication mode.

= 0	generate a hardcopy only
= 1	generate a file only
= 2	generate both a hardcopy and a file
	(default is 0)

(File)
File name if the mode option is set to 1 or 2.
(default is 'problem_name.Aij', where 'i' is a digit that ranges from 0 to 9, and 'j' ranges from 1 to 9 to act as a counter)

Note

The generated file may be printed from the system level as follows:

a. For the PC version, use the copy/b command. For example 'copy/b file-name lpt1:' will send the file to a printer connected to the lpt1 port.
b. For Workstations, use the regular printing command.

NEC201

Control > DEVICES > NEC201 Printer

The NEC201 command generates a hardcopy for the image in active window using a configured NEC201 printer. The command works only for the NEC version.

SCREENPLOT

Control > DEVICES > Plot Screen

The SCREENPLOT command generates a hardcopy from the screen using a configured device. The generated plot is centered on the paper with default margins or as specified by the PAPER_SETUP command.

Entry & Option Description

Pflag
Print flag.

= 0	active window only
= 1	base window (full screen without GEOSTAR frame)
= 2	full screen including GEOSTAR frame
= 3	select desired portion of the screen using crosshairs which will appear on the screen after all input is entered

(default is 0)

Device

Device name.

= 1	plotter
= 2	HP PAINTJET printer
= 3	HP LASERJET printer
= 4	EPSON_IBM printer
= 6	HP DESKJET 500, 500 C, and 550 C printers

(default is 2)

Resolution

Resolution in dots/in.
(default depends on the chosen device)

Notes

If the command does not produce a hardcopy of the screen, please check the following points before seeking help from SRAC:

a. Make sure that GEOSTAR has been correctly configured for the device connected to your computer. The device must be one of the devices specified in the configuration program. If you are using a device that emulates one of the supported devices, make sure that the proper setting of switches has been made for the emulation mode.

b. If you are using HP PAINTJET or HP LASERJET, make sure that the DOS program 'MODE.COM' is copied to GEOSTAR directory. Also make sure that the printer is connected to the correct port.

c. Self-test the device.

d. Verify that the serial or parallel port to which the printer is connected, is active and that nothing is wrong with the cable. This can be done by printing a small file using the DOS PRINT command or the PRINT SCREEN option.

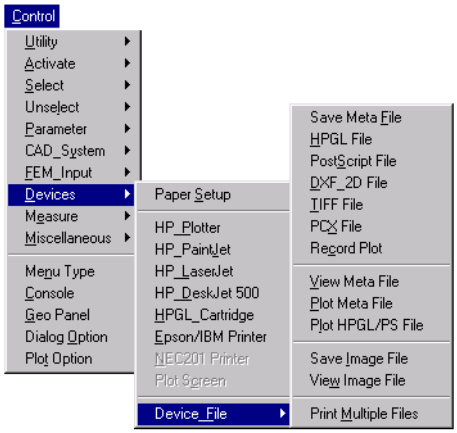



▼DEVICE_FILE Menu

Control > DEVICES > DEVICE_FILE

This menu contains commands to create, view and plot image files in varous formats.

Figure 8-10 Device File Menu



METAFILE

Control > DEVICES > DEVICE_FILE > Save Meta File

The METAFILE command generates a meta file for the image in the active window. A meta file is an ASCII file that can be plotted using the PLT_METAFIL command.

Entry & Option Description

File

Name of file.
(default is ufn.M??, where ?? is a number that is incremented each time a file is generated)

(Over-write)

Overwrite flag. Prompted only if file exists.

= 0	do not overwrite and quit
= 1	overwrite file
	(default is 0)

Note

The Find button lets you browse your disk to search for existing meta files.

HPGLFILE

Control > DEVICES > DEVICE_FILE > HPGL File

The HPGLFILE command generates an HPGL (Hewlett Packard Graphic Language) file for the image in the active window.

Entry & Option Description

File

Name of file.
(default is ufn.G??, where ?? is a number that is incremented each time a file is generated)

Device

Device type.
= 0 printer (generated file can only be dumped to a printer)
= 1 plotter (generated file can only be dumped to a plotter)
(default is 0)

(Over-write)
Overwrite flag. Prompted only if file exists.
= 0 do not overwrite and quit
= 1 overwrite file
(default is 0)

POSTSCRIPT

Control > DEVICES > DEVICE_FILE > PostScript File

The POSTSCRIPT command generates a Postscript file for the image in the active window.

Entry & Option Description

File
Name of file.
(default is ufn.P??, where ?? is a number that is incremented each time a file is generated)

(Over-write)
Overwrite flag. Prompted only if file exists.
= 0 do not overwrite and quit
= 1 overwrite file
(default is 0)

DXF_2D

Control > DEVICES > DEVICE_FILE > DXF_2D File

The DXF_2D command generates a DXF_2D file for the image in the active window.

Entry & Option Description

File
Name of file.
(default is ufn.D??, where ?? is a number that is incremented each time a file is generated)

(Over-write)
Overwrite flag. Prompted only if file exists.
= 0 do not overwrite and quit
= 1 overwrite file
(default is 0)




TIFFILE

Control > DEVICES > DEVICE_FILE > DXF_2D File

Geo Panel: Control > DEVICES > DEVICE_FILE > TIFF File

The TIFFILE command generates a file in the TIFF format for the image in the active window. The generated file may be used in desktop publishing programs.

Entry & Option Description

TIFF file name
Name of file.
(default is ufn.TIF)

(Over-write)
Overwrite flag. Prompted only if file exists.
= 0 do not overwrite and quit
= 1 overwrite file
(default is 0)

PCXFILE

Control > DEVICES > DEVICE_FILE > DXF_2D File

Geo Panel: Control > DEVICES > DEVICE_FILE > PCX File

The PCXFILE command generates a file in the PCX format for the image in the active window. The generated file may be used in desktop publishing programs.

Entry & Option Description

PCX file name
Name of file.
(default is ufn.PCX)

(Over-write)
Overwrite flag. Prompted only if file exists.
= 0 do not overwrite and quit
= 1 overwrite file
(default is 0)

PLOTREC

Control > DEVICES > DEVICE_FILE > DXF_2D File

Geo Panel: Control > DEVICES > DEVICE_FILE > Record Plot

This command is used to record and save the image plotted in the GEOSTAR screen. When PLOTTER is issued in the first time, it begins a plot record session. You may then issue commands like VIEW_META and use PLOTREC again to close and save the plot session to TIFF or PCX file.




Entry & Option Description (for closing the plot session)

Save plot in file format
File type.

= 0	TIFF
= 1	PCX
	(default is 0)

File name
Name of the file to be saved.
(default is problem_name.tif or problem_name.pcx)

VIEW_META

Control > DEVICES > DEVICE_FILE > View Meta File

The VIEW_META command can be used to view a meta file (generated by the METAFILE command) on the screen.

Entry & Option Description

Selection
Name of file to be viewed.

Note
A hardcopy of the viewed file can not be obtained by using commands like LASERJET, PAINTJET, PLOTTER and similar commands that require the existence of database files. The SCREENPLOT command can be used to dump the viewed file to a hardcopy device. The PLT_METAFIL command can also be used to generate a hardcopy of the meta file.

PLT_METAFIL

Control > DEVICES > DEVICE_FILE > Plot Meta File

The PLT_METAFIL command generates a hardcopy from an existing meta file (refer to the METAFILE command).

Entry & Option Description

Device
Hardcopy device.

= 1	HP or GRAPHTEC Plotters
= 2	HP PaintJet
= 3	HP LaserJet
= 4	Epson/IBM
= 5	NEC201 printer (for NEC version)
= 6	HP DESKJET 500 C printer
	(default is 2)

File
Name of file.

(default is problem_name.Mij, where i and j are digits from 0 to 9)

Resolution
Number of dots per inch. (Refer to PLOTTER, PAINTJET, LASERJET and EPSON_IBM commands.)

Communication mode
Communication mode.
= 0 generate a hardcopy only
= 1 generate a file only
= 2 generate both a hardcopy and a file (default is 0)

(File)
File name if the mode option is set to 1 or 2.
(default is 'problem_name.Aij', where 'i' is a digit that ranges from 0 to 9, and 'j' ranges from 1 to 9 to act as a counter)

PLT_FILE

Control > DEVICES > DEVICE_FILE > Plot HPGL/PS File

The PLT_FILE command generates a hardcopy from an existing HPGL or POST-SCRIPT file (refer to the HPGLFILE and POSTSCRIPT commands). A printer or a plotter with an HPGL/2 or POSTSCRIPT cartridge must have been configured.

Entry & Option Description

File-name
Name of file. (The file can be in HPGL or POSTSCRIPT format.)

Device
Device type (used only for HPGL files).
= 0 printer
= 1 plotter
(default is 0)

Notes

1. HPGL files generated for printers can only be dumped to a printer and HPGL files generated for plotters can only be dumped to a plotter.
2. HPGL cartridges on plotters may not generate appropriate color-filled or shaded plots. Use the PLOTTER command directly from GEOSTAR instead.

IMAGESAV

Control > DEVICES > DEVICE_FILE > Save Image File

The IMAGESAV command saves the selected portion of an image on the screen into a file for later restoration.

Entry & Option Description

File name

Name of the file.
(default name is problem name .DIB)

First_corn
First corner of the enclosing box to be picked.

Oppo_corn
Opposite corner of the enclosing box to be picked.

Note
When the command is issued, crosshairs appears on screen. To complete the command, select the first corner point of the enclosing box by positioning the crosshairs at the desired point and picking up by the mouse. Next, move the crosshairs to the opposite corner point to form the box, enclosing the image to be saved, and pick the point to complete the command.

Example: `IMAGESAV,,>,<,`
This command saves the enclosed image in the specified file.

IMAGERES

Control > DEVICES > DEVICE_FILE > View Image File

The IMAGERES command restores a previously saved image onto the screen from a file.

Entry & Option Description

Selection
Name of the file.
(The default file name is "problem name.DIB")

Note
Image restoration is not allowed in window environment.

Example: `IMAGERES,,`
This command restores an image onto the screen from the file "problem name.DIB".

MULTIPRINT

Control > DEVICES > DEVICE_FILE > Print Multiple Files

The MULTIPRINT command prints previously created image files in various formats. Up to 10 files can be printed.

Entry & Option Description

Format
File format.

= 0	file was created by the IMAGESAV command
= 1	file was created by the METAFILE command

= 2 file was created by the HPGLFILE or POSTSCRIPT commands
(default is 0)

(Device)

Hardcopy device.
For format = 0 or 1

= 1 HP or GRAPHTEC plotters
= 2 HP PaintJet
= 3 HP LaserJet
= 4 Epson/IBM
= 5 NEC201 printer
= 6 HP DESKJET 500 C printer
(default is 2)

For format = 2

= 0 printer
= 1 plotter
(default is 0)

(Resolution)

Number of dots per inch. (not prompted if format=2) (Refer to PLOTTER, PAINTJET, LASERJET and EPSON_IBM commands.)

Fname(i)

Name of ith file to be printed. (i =1,2,...,10)

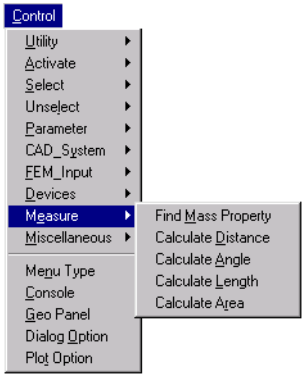
Notes

1. The command will prompt for more file names until a blank name is entered or 10 file names were specified.
2. All files must be of the same format.
3. HPGL files generated for printers can only be dumped to a printer and HPGL files generated for plotters can only be dumped to a plotter.
4. HPGL cartridges on plotters may not generate appropriate color-filled or shaded plots.




MEASURE Menu

Figure 8-11 Measure Menu



Control > MEASURE

This menu contains commands to display mass properties of a group of elements in addition to other measurements like distances, angles, and areas.

MASSPROP

Control > MEASURE > Find Mass Property

The MASSPROP command lists mass properties information for a pattern of elements. The properties listed are: length, area, volume, mass and corresponding centers of gravity, moments of inertia and radii of gyration. Length properties are listed for one dimensional elements only (e.g. BEAM3D, ELBOW, PIPE, etc.), area properties are listed for two dimensional elements only (e.g. SHELL9L, SHELLAX, FLOW2D, MAG2D, etc.), and volume properties are listed for all elements with volume (e.g. SOLID, FLOW3D, BEAM2D, SHELL4, etc.). Mass properties are listed for elements with mass (density must be specified). The command also provides principal moments of inertia and corresponding principal directions with respect to an arbitrary reference Cartesian coordinate system.

Entry & Option Description

Check flag
Check flag.

= 1: Yes	Check for inconsistent data (e.g. undefined material properties and real constants, connectivity, degenerate elements, etc.)
= 0: No	Do not check for inconsistent data *(default is no)*

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is beginning element)

Increment
Increment between elements in the pattern.
(default is 1)

Coordinate system
Reference coordinate system.
(default is 0)

Note

The desired group of elements may be conveniently selected by commands in the CONTROL>SELECT menu before issuing this command.

Example: `MASSPROP, 1, 1, 257, 1, 7`

This command lists the mass properties for elements 1 through 257 with respect to coordinate system number 7. The command checks all elements for inconsistent data.

DISTANCE

Control > MEASURE > Calculate Distance

The DISTANCE command calculates and lists the distance between two keypoints, two nodes, or a keypoint and a node.

Entry & Option Description

Type

Type.

= 0: Keypt-keypt	distance between 2 keypoints
= 1: Keypt-node	distance between a keypoint and a node
= 2: Node-node	distance between 2 nodes
	(default is keypt-keypt)

Keypoint/Node

Label of a keypoint or node.

Keypoint/Node

Label of a keypoint or node.

ANGLE

Control > MEASURE > Calculate Angle

The ANGLE command calculates and lists the angle between three keypoints, three nodes, or two curves. In the case of three keypoints or nodes, the angle measured is at the second keypoint or node. In the case of two curves, the angle between the tangents is calculated.

Entry & Option Description

Type

Type.

= 0: 3 keypts	angle between 3 keypoints
= 1: 3 nodes	angle between 3 nodes
= 2: 2 lines	angle between 2 lines
	(default is 3 keypts)

Keypoint/Node/Curve 1

Label of a keypoint, node, or curve.

Keypoint/Node/Curve 2

Label of a keypoint or node.

Keypoint/Node 3
Label of a keypoint or node. (not prompted for if type = 2 lines)

LENGTH

Control > MEASURE > Calculate Length

The LENGTH command calculates and lists the length of a curve or a contour.

Entry & Option Description

Type
Type.

= 0: Curve	curve
= 1: Contour	contour
	(default is curve)

Curve/Contour
Label of a curve or contour.

AREA

Control > MEASURE > Calculate Area

The AREA command calculates and lists the area defined by four keypoints or nodes, or the area of a surface. For triangular areas, one of the keypoints or nodes may be repeated.

Entry & Option Description

Type
Type.

= 0: 4 keypts	area defined by 4 keypoints
= 1: 4 nodes	area defined by 4 nodes
= 2: 1 Surface (geom)	area of a surface from geometry
= 3: Surfaces (geom)	area of a pattern of surfaces calculated from geometry
= 4: Surfaces (mesh)	area of a pattern of surfaces from undeformed mesh
= 5: Surfaces (deformed)	area of a pattern of surfaces from deformed mesh
= 6: Regions (mesh)	area of a pattern of regions from undeformed mesh
= 7: Regions (deformed)	area of a pattern of regions from deformed mesh
= 8: 1 Polyhedron (mesh)	area of a polyhedron from undeformed mesh
= 9: 1 Polyhedron (deformed)	area of a polyhedron from a deformed mesh
	(default is 4 keypts)

Keypoint 1/Node 1/Surface
Label of a keypoint or node.

The following three arguments are not prompted for type = surface.

Keypoint/Node 2
Label of a keypoint or node.

Keypoint/Node 3

Label of a keypoint or node.

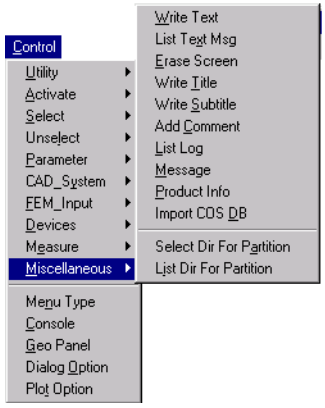
Keypoint/Node 4
Label of a keypoint or node.

MISCELLANEOUS Menu

Control > MISCELLANEOUS

This menu includes commands to perform miscellaneous operations like opening or closing list files, writing comments to the session file, and writing text on the screen.

Figure 8-12 Miscellaneous Menu



WRTEXT

Control > MISCELLANEOUS > Write Text

The WRTEXT command is used to write horizontal or vertical text on the screen. The command recalls an already defined message or defines a new one. A box showing the size of the text appears on the screen. The box can be positioned anywhere in the active window using the mouse. Pressing the left button of the mouse reproduces the text in the specified color at the desired position.

Entry & Option Description

Message number
Message number.

Write/Rewrite
New/old flag.

= 0: New	new message
= 1: Old	old message
	(default is old for an existing message number and new for a new message number)

Print flag
Horizontal/vertical mode.

= 0: H	horizontal text
= 1: V	vertical text
	(default is H)

Scale factor
Parameter to scale size of text font.
(default is 1.0)

Color index
Color in which text is written. Varies between 1 and 16.
(default is 7)

Message
A text string of up to 80 characters.

Notes

1. Images with text can be printed, plotted or saved in a file for later restoration.
2. If the size of the text is too big for the active window then it will be ignored.

Example: `WRTEXT, 1, 0, 2.0,6,COSMOS/M IS GREAT`

This command writes "COSMOS/M IS GREAT" horizontally in the default color index. The message number is 1, and can be recalled afterwards.

WRTEXTLIST

Control > MISCELLANEOUS > List Text Msg

The WRTEXTLIST command lists existing text messages created by the WRTEXT command.

ERASE

Control > MISCELLANEOUS > Erase Screen

The ERASE command erases everything inside a rectangular window specified by selecting two opposite corners using the crosshairs and the mouse.

TITLE

Control > MISCELLANEOUS > Write Title

The TITLE command gives a title to the current problem. The given title will appear in the results and the output file.

Entry & Option Description

Message
A text string up to 80 characters long.

SUBTITLE

Control > MISCELLANEOUS > Write Subtitle

The SUBTITLE command defines a subtitle for the current problem. The given subtitle will appear in the second line of the title heading. The first line is defined by the TITLE command.

Entry & Option Description

Message

A text string up to 80 characters long.

COMMENT

Control > MISCELLANEOUS > Add Comment

The COMMENT command writes a comment to the session file. It may be used repeatedly to add comments to different parts of the session file.

Entry & Option Description

Message
A text string up to 80 characters long.

LISTLOG

Control > MISCELLANEOUS > List Log

The LISTLOG command enables the user to open or close a file to record GEOSTAR response to listing and help commands.

Entry & Option Description

Listlog flag
Open/close file flag.

= 0: Off	close an open file
= 1: On	open a new file
	(default is off if a file is open and on if no file is open)

List filename
Name of file.
(default is problem name.LIS)

List display option flag
Display flag.

= 0: Off	write listings and help to the open file but do not display on the screen
= 1: On	display listing and help on the screen as well as writing them to the file
	(default is on)

Notes

1. If the command is used to open a file while an already open file exists, the command closes the existing file and opens the new file. This allows the user to open several files in a session to record relevant information in separate files. However only one file can be active at any time.
2. If an existing file is opened, the information will be appended to it.
3. GEOSTAR closes the file whenever control is transferred to another module, e.g., if the file A.1 is open when the R_STATIC command is issued, GEOSTAR closes this file and when control is returned to GEOSTAR, this file or another file must be opened before recording is possible.

4. By deactivating the 'list display option flag', it is possible to open listing files and write results to them in batch jobs.

Example:
```
LISTLOG, 1, A.1, 0,
DISLIST,,,,,,
```
The LISTLOG command opens an existing or new file A.1 and writes (or appends) the listing of displacements to it. If file B.1 is already opened by the LISTLOG command, it will be first closed before A.1 is opened.

MESSAGE

Control > MISCELLANEOUS > Message

The MESSAGE command controls the appearance of informative messages on the screen during the problem session.

Entry & Option Description

Message option flag
Flag for writing messages on the screen.

= 0: Off	do not write messages
= 1: On	write messages *(default is on)*

PRODUCT_INF

Control > MISCELLANEOUS > Product Info

The PRODUCT_INF command displays the names of Structural Research modules, translators, and other utilities authorized for use by the user. The user company name, maintenance number and expiration date are also displayed. The same information may be obtained by running the “chk_auth” program from the operating system level.

COSDBASE

Control > MISCELLANEOUS > Import COS DB

The COSDBASE command may be used in conjunction with the COSMOS/M Database Utility to import FEA databases created elsewhere to GEOSTAR. The COSMOS/M Database Utility is a collection of routines that enables you to read and write COSMOS/M database files. Two typical usages of this command are:

a. All preprocessing and analysis are performed outside GEOSTAR and you only want to use GEOSTAR for postprocessing.
b. GEOSTAR is used for meshing, the analysis is performed outside GEOSTAR, and the results are to be postprocessed in GEOSTAR.

The steps for utilizing this command are illustrated below:

1. Use the GET routines of the COSMOS/M Database Utility to read your FEA model information from your application.
2. Use the PUT routines to write the information to COSMOS/M database files.
3. Delete the file <problem-name>.MAS created by the Database Utility.
4. Open the problem in GEOSTAR.
5. Use the COSDBASE command to load the database to GEOSTAR.
6. Continue pre- or postprocessing as desired.

COSMOS/M Database Utility comes with detailed documentation on its use and is available for purchase.

SELDIR

Control > MISCELLANEOUS > Select Dir for Partition

The SELDIR lets you specify several drives and directories to store databases. The utility is particularly useful for large problems where you do not have sufficient disk space in a single disk. The specified information is used by the direct solvers for static, frequency, and buckling analyses only (STAR and DSRAR). For more information about this utility, refer to the “Large File Management” chapter in the Basic System manual.

Entry & Option Description

Partition/Transfer Option

Sets the partition option. The available options are:

= 0: Deactivate	Deactivates this utility (no more prompts).
= 1: Auto	Automatic partitioning in the specified drives.
= 2: Assigned	You will assign the partitioning size of large database files for selected drives manually.
= 3: Transfer	Specify a drive for each large file. *(default is Auto Partition)*

Use Drives (prompted for partitioning options 1 and 2)

Sets the drives to be used. The available options are:

= 0: Fixed	Use the available fixed drives on your local machine only.
= 1: Fixed/Remote	Use all available local and remote drives.
= 2: Fixed/Remote/Removable	Use all available local, remote, and removable drives. *(default is Fixed)*

Block Size

Sets the size of memory blocks. The size is specified in bites.

= 0	Automatic block sizing.
= N	Set block size to N bites.
	(default is 0)

Directory in fixed drive C
Directory in fixed drive D
.
.
.*For partitioning options 1 and 2:*

Sets the directory path in drive C, D, etc. All available fixed drives are listed. Remote disks are also listed if options 1 or 2 are selected for “Use Drives”. Removable disks are listed only if 2 is selected for “Use Drives”. The specified directory will be created if it does not exist. (default directory is “work”. See note 1 below)

For portion/transfer option 3:

Sets a directory to be used for the storage of each of the large files. These files include stiffness matrix file, geometric stiffness/consistent mass file, and the TMP.ORI, ORI.STF, ORI.MAS, and ORI.GST files. (For information about the contents of these files, refer to the “Large File Management” chapter in the Basic System manual.)

Size

Sets the proportional size for each file partition in the specified directory for option 2.
(no default)

Notes

1. If the default “work” directory, or any other specified directory, does not exist for partitioning options 1 and 2, it will be created by the program. The created directories and files will be automatically deleted after the problem is completed unless you requested to save these files through the corresponding analysis option command (Analysis menu).
2. In specifying the desired directories, you should use the proper format for your platform. In Unix platforms for example, “/” instead of “\” is used.
3. If a drive letter is not specified, the default current drive will be used.
4. You may use option 2 to specify a single directory in a drive which is different than the current default drive.
5. Option 1 results in storing segments of the files in various drives proportional to their free disk space. In option 2, you specify the segment size for each drive manually. The size of the segment for each drive may be specified as a percentage, actual size in bites, or by assigning an integer for each drive which indicates the size of the segment relative to other drives. When using option 2, you may specify smaller segments on slower drives to improve the speed of the program.
6. You may use the block size option to reduce the block size and hence save more memory to run other applications simultaneously.

SELDIRLIST

Control > MISCELLANEOUS > List Dir for Partition

This command lists the specifications specified by the SELDIR command (Control > Miscellaneous > Select Dir for Partition) for multiple drive allocation for large problems.

MENUTYPE

Control > Menu Type

The MENUTYPE command is used to select the extended or cryptic command names in the menus.

Entry & Option Description

Menu type (unused option)

Menu mode.

= 0: Pull-down	pull-down (click the mouse to pull a menu, submenu, or a command)
= 1: Drop-down	drop-down (menus, submenus and commands are automatically dropped when highlighted by the mouse. Commands are selected by clicking the mouse.) *(default is pull-down)*

Brief help message (unused option)

Brief help flag. When activated, a one-line help appears at the bottom of the display area.

= 0: Off	deactivate brief help
= 1: On	activate brief help *(default is on)*

Command type

Commands names. Use extended or cryptic names.

= 0: Long form	long form
= 1: Short form	short form *(default is short form)*

Percentage of menu overlap (unused option)

Percentage of menu overlap.

Notes

1. In selecting a command in the drop-down mode, the mouse pointer should not leave the area of the dropped menu or submenu.
2. In the pull-down mode, menus, submenus and commands can be selected by placing the mouse pointer at the desired option and pressing the left button. This procedure can also be used to select items that are higher in the menu tree level even if lower levels had been selected.
3. Full help for the highlighted command can be obtained by pressing the right button of the mouse in both modes.

CONSOLE

Control > Console

The CONSOLE command opens the console window when it is closed. You can also use the Console checkbox in the Geo Panel to show/hide the Console window.

GEO PANEL

Control > Geo Panel

The GEO PANEL command opens the GEO Panel widow when it is closed to give you quick access to frequently used commands.

DLGOPTION

Control > Dialog Option

The DLGOPTION command sets the type of dialog boxes to be displayed and specifies the maximum number of visible parameters (input).

Entry & Option Description

Dialog box type
Dialog box type

= 0	single parameter only
= 1	multiple parameters *(default is 1)*

Maximum number of visible parameters
Maximum number of visible parameters. (3-25) (default is 20)

Notes

1. If "single parameter" is selected for the dialog box type, all prompts will be displayed in the console window and no dialog box will be displayed.
2. If the number of parameters in a command exceeds the maximum number of visible parameters specified, you may scroll in the dialog box to enter the desired parameters.

PLOTOPTION

Control > Plot Option

The PLOTOPTION command sets the fast plot flag on and off for GEOSTAR windows.

Entry & Option Description

fast plot flag

= 0: Off	plot to window and memory at the same time;

= 1: On — plot to window first and dump window image to memory at the end, which is faster than when the flag is off.
(default is 1 if the current flag value is 0, 0 if the current flag value is 1)

Note

1. For a new problem, the fast plot flag is initially off.
2. If the fast plot flag is on, the active GeoStar (main or child) window should be kept totally visible while plotting is going on. Keep Geo Panel, GeoStar Console and windows of other applications outside of the active GeoStar window to avoid their images (the portions covering the active GeoStar window) being dumped to memory and restored to the active GeoStar window later, otherwise you may click the PAINT icon to repaint.
3. For big and complex models (for example, one of over 20000 elements), turn the fast plot flag on to save up to 1/3 of the plotting time.




9 Display Menu

DISPLAY Menu

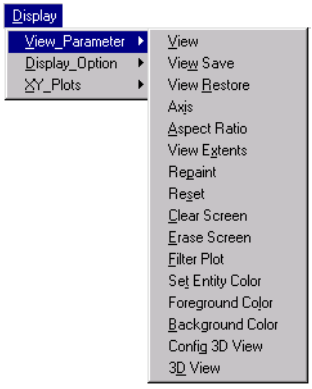
Geo Panel: DISPLAY

This menu contains commands to control view parameters, display parameters, the multiple window environment, and x-y plotting.

Figure 9-1 Display Menu

▶ VIEW_PARAMETER Menu

Figure 9-2 View_Parameter Menu



Geo Panel: **Display > VIEW_PARAMETER**

This menu contains commands to control the viewing parameters.

VIEW

Geo Panel: **Display > VIEW_PARAMETER > View**

The VIEW command defines the view direction relative to which the object is plotted on the screen. The view line is always normal to the screen with the object oriented in the specified direction.

Entry & Option Description

X, Y, Z-Value
The X, Y and Z coordinates of a point along the axis defining the view direction. The view direction is the direction along the vector from the origin to the point specified by the X,Y and Z coordinates.
(defaults are 0.0, 0.0 and 1.0 for X-,Y-, and Z-coordinates respectively)

Coordinate system
Coordinate system.
(default is 0 for global Cartesian)

Example: `VIEW,2,1,1,4`
This command defines view direction along the vector joining the origin (0.,0.,0.) and the point (2.,1.,1). The view is associated with local coordinate system 4 which must be Cartesian.

VIEWSAVE

Geo Panel: **Display > VIEW_PARAMETER > View Save**

The VIEWSAVE command saves the viewing transformation matrix for the current view on the screen in the active window. The saved view may be restored later by the VIEWREST command. Note that the command does not save the image on the screen.

Entry & Option Description

View number
Reference view number. Up to 10 views may be saved.

VIEWREST

Geo Panel: Display > VIEW_PARAMETER > View Restore

The VIEWREST command restores a previously saved viewing transformation matrix and applies it to the current plot in the active window.

Entry & Option Description

View number
Reference view number. Must have been previously saved by the VIEWSAVE command.

AXIS

Geo Panel: Display > VIEW_PARAMETER > Axis

The AXIS command controls the plotting of the axis of the global Cartesian coordinate system on the screen. A color can be specified for axis plotting.

Entry & Option Description

Draw flag
Flag to plot or erase the global Cartesian coordinate axes.
= Plot draw the axes
= No erase the axes
(default is plot)

Axis color
Color of the axis.
(default is 16)

Example: `AXIS, 1, 14`
This command plots the global coordinate system with color 14.

ASPECT

Geo Panel: Display > VIEW_PARAMETER > Aspect Ratio

The ASPECT command defines the screen aspect ratio (y/x) by which the plotting of the model geometry may be distorted either in the vertical or horizontal directions.

Entry & Option Description

Aspect factor y/x
Ratio of the scale in the Y-direction to that in the X-distortion.
(default is 1.0)

Note
This command is useful for better visual evaluation of narrow sections.

Example: `ASPECT, 2.0,`

This command sets the scale in the y-direction to twice that in the x-direction. This scales ratio will be used for all subsequent plots until the ASPECT command is issued again to change the ratio.

EXTENTS

Geo Panel: **Display > VIEW_PARAMETER > View Extents**

The EXTENTS command specifies the range of viewing in space. The range is defined in the global Cartesian coordinate system. Parts of all subsequent plots beyond the specified range are clipped at the borders.

Entry & Option Description

x-value of minimum extent
Minimum value of the X-coordinate.
(default is 0.0)

y-value of minimum extent
Minimum value of the Y-coordinate.
(default is 0.0)

z-value of minimum extent
Minimum value of the Z-coordinate.
(default is 0.0)

x-value of maximum extent
Maximum value of the X-coordinate.
(default is 100.0)

y-value of maximum extent
Maximum value of the Y-coordinate.
(default is 100.0)

z-value of maximum extent
Maximum value of the Z-coordinate.
(default is 100.0)

Notes

1. This command can be used in the beginning to specify the approximate dimensions of the model.
2. For selection of entities in a particular range of coordinates, refer to the SELRANGE command available from CONTROL > SELECT menu.

Example: `EXTENTS, , , , 50.0, 50.0, 50.0`

This command specifies the box size for object clipping as the one with first corner (0.,0.,0.) and the opposite corner (50,50,50). Parts of the object beyond this box are clipped out.

REPAINT

Geo Panel: Display > VIEW_PARAMETER > Repaint

The REPAINT command replots the picture on the screen with the currently active viewing parameters.

RESET

Geo Panel: Display > VIEW_PARAMETER > Reset

The RESET command resets the view, zoom, translation, shrink, scale, boundary and rotation parameters back to the default values.

CLS

Geo Panel: Display > VIEW_PARAMETER > Clear Screen

The CLS command clears the active window and fills it with the specified color. The command can also be executed using the CLS icon, the background in this case assumes the current color in the B_C icon.

Entry & Option Description

Background color
Background color of the cleared screen.
(default is black, 1)

Example: `CLS,1`
This command clears the active screen and fills it with color 1 (black).

FILTER

Geo Panel: Display > VIEW_PARAMETER > Filter Plot

The FILTER command places a specified screen filter on the existing plots. The colors in the plots are then exclusively ored (overlaid and contrasted, XOR) with the filter color.

Entry & Option Description

Screen background color
The filter color.
(default is black, 1)

Example: `FILTER, 13`
This command places a filter of color 13 on the screen. The colors of the already plotted pictures are exclusively ored with the filter color. The “B_C” icon (background color) may also be selected prior to issu-

ing FILTER command for identical results.

SETCOLOR

Geo Panel: **Display > VIEW_PARAMETER > Set Entity Color**

The SETCOLOR command sets the color to be used for subsequent plotting of symbols for entities and other preprocessing information. An option to choose one color for all entities is provided. If the specified color is similar to the active background color, then the foreground color will be used instead. The STATUS1 Table may be alternatively used to set the color for geometric entities in addition to nodes and elements and the STATUS2 Table may alternatively be used to set colors for symbols related to loads and boundary conditions. Refer to the SYMBSIZ command to set the size of the symbols.

Entry & Option Description

Entity

Entity name to set the symbol color for. The admissible entities are:

= PT	keypoints
= CR	curves
= VL	volumes
= CT	contours
= RG	regions
= PH	polyhedra
= PA	parts
= ND	nodes
= EL	elements
= CS	coordinate systems
= DP	displacement boundary conditions
= FP	applied external forces
= PP	applied pressure
= NT	applied nodal temperature
= HF	applied hydraulic flow
= QN	applied nodal heat flow
= NP	applied nodal potential
= NJ	applied nodal current
= VP	applied velocity
= MDOF	master degrees of freedom
= CP	coupling
= SP	super element information
= SM	shear moment diagrams
= EC	applied convection
= HX	applied heat flux

= RD	applied radiation
= QE	applied element heat
= JE	applied element current
= TKE	applied turbulent kinetic energy
= EPS	applied dissipation rate
= NPR	applied nodal pressure (for fluid flow)
= DNS	density boundary conditions
= BND	bonding information
= ALL	all of the above entities

Color
Label of color. Use colors 1 through 16. (Refer to STATUS1 Table.)

Example: `SETCOLOR, all, 16,`
This command sets the color for all subsequent plots to be white for all entities. This is useful if black and white printers are used.

FCLR

Geo Panel: **Display > VIEW_PARAMETER > Foreground Color**

The FCLR command sets the foreground color of the screen to the specified color. The command function can also be performed by selecting the desired color through the "F_C" icon with the mouse. Note that the FCLR command makes a difference only when the color of the entity to be plotted is the same as the background color (refer to the STATUS1 command).

Entry & Option Description

Color
Foreground color of the screen.
(default is 13)

Example: `FCLR,14`
This command sets color 14 as the active foreground color.

BCLR

Geo Panel: **Display > VIEW_PARAMETER > Background Color**

The BCLR command sets the background color of the screen to the specified color. The command function can also be performed by selecting the desired color through the "B_C" icon with the mouse.

Entry & Option Description

Color
Background color of the screen.
(default is 1, for black)

Example: `BCLR,1`

This command sets the background of the screen as 1, black.

CONFIGDVIEW

Geo Panel: **Display > VIEW_PARAMETER > Config 3D View**

This command sets the graphical attributes and some functional parameters for Dynamic View. The defaults shown are those when the command is first used. The defaults used for subsequent use of the command will be the values entered by the user.

Entry & Option Description

Track ball rotation mechanism
Track ball rotation mechanism. See note below.
= Off use Coordinate Axis Mode
= On use Trackball Mode
(default is off)

Mouse movement sensitivity (1-100)
Sensitivity of the mouse movement.
(default is 1)

Flip surface normal flag
Flip surface normal flag.
= No do not flip surface normal
= Yes flip surface normal
(default is no)

Plot coordinate system in the model center
Plot coordinate system in the model center.
= No do not plot coordinate system in the model center
= Yes plot coordinate system in the model center
(default is no)

Plot a boundary box around the model
Plot a boundary box around the model.
= No do not plot a boundary box around the model
= Yes plot a boundary box around the model
(default is no)

Backface elimination flag
Eliminate backface.
= Off do not eliminate backface
= On eliminate backface
(default is off)

Notes

1. Trackball mode:
   The left mouse button controls the motion of the virtual trackball in which the model is embedded. The middle mouse button zooms out of the model and the

right mouse button zooms into the model. Translations are not possible in this mode.

2. Coordinate Axis Mode:

   The left, middle and right mouse buttons control the rotation about the X-, Y- and Z-axes respectively. A single press of one of these buttons causes a 10 degree rotation of the model about the corresponding axis. By pressing a button and moving the mouse, continuous rotation will occur with a magnitude to the distance between the two mouse move events (projected onto a horizontal direction). Moving in different directions will cause clockwise or counter-clockwise rotation.

   While holding the shift key, a single press of the left mouse button will move the model one step further away on the screen. Continuous zooming is possible by holding down the mouse button and translating the mouse horizontally on the screen. The use of the middle mouse button with the shift key depressed works in a similar way but moves the object closer. Holding the shift key and pressing the left mouse button permits the user to move the model on the screen. The direction of motion is in the same direction as the mouse motion.

   There are some special keys to be used with dynamic viewing. Some work as toggle keys setting their corresponding functions on or off.

3. Not available for Windows 95.

DVIEW

Geo Panel: **Display > VIEW_PARAMETER > 3D View**

This command provides for the interactive rotation, scaling and translation of models in 3D space. The command can operate in demo or non-demo mode. If the demo mode is selected, the user has no control over the dynamic viewing. If the non-demo mode is selected, the user may use the trackball mode or the coordinate axis mode (refer to the CONFIGDVIEW command for details on using these modes).

Entry & Option Description

Demo display flag

Demo or non-demo mode.
= Off non-demo mode
= On demo mode

Note

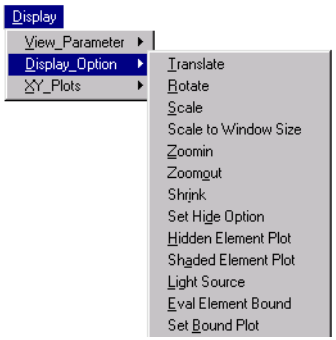
Not available for Windows 95.




▶ DISPLAY_OPTION Menu

Geo Panel: **Display > DISPLAY_OPTION**

Figure 9-3 Display_Option Menu



This menu contains commands to control the display parameters.

TRANSLATE

Geo Panel: **Display > DISPLAY_OPTION > Translate**

The TRANSLATE command allows the user to translate or move the picture from one point to another. The crosshairs appear on the screen once this command is issued. The start and the destination points are picked using the crosshairs to establish the direction and the amount of translation.

Notes

1. The TRANSLATE command is an action command resulting in automatic formation of the translated picture when the command is executed. To complete the command, select the start point by positioning the crosshairs at the desired point using the mouse and then click the mouse or press the “ENTER” key. Next, move the crosshairs to the destination point and click the mouse or press the “ENTER” key again.
2. This command may be repeated for further repositioning of the picture.
3. The “RESET” command replots the object in the original location.

ROTATE

Geo Panel: **Display > DISPLAY_OPTION > Rotate**

The ROTATE command allows the rotation of the object about the X, Y, and Z axes. It sets the view rotation angles, to be applied prior to plotting. Multiple application of the command results in the scalar addition of the angles. The “RESET” command resets the view to its default direction.

Entry & Option Description

X, Y, Z-Rotation

Angles of rotation in degrees to be applied to object about the X, Y, and Z axes. *(default values are 0.0)*

Coordinate system
Coordinate system. (must be Cartesian)

Rescale image
Rescale flag.
= No do not rescale after rotation
= Yes scale after rotation

Example: `ROTATE, 30.0, 0.0, 45.0, 0, 1,`

This command applies a rotation of 30.0 and 45.0 degrees about the X and the Z axes to the object. The subsequent plotting commands will display the object in its rotated state. The global Cartesian coordinate system is used and the image is rescaled.

SCALE

Geo Panel: **Display > DISPLAY_OPTION > Scale**

The SCALE command replots the object after scaling it by the specified factor. The default value of the scale factor is 0.0 which means filling the active window with the object.

Entry & Option Description

Scale factor
Scale factor.
(default is 0.0, which means automatic scaling)

Note

The complete object is considered, including entities that are not plotted. The PSCALE command, on the other hand, considers only the plotted part of the object.

Example: `SCALE, 0.5`

This command scales down the object such that it occupies half the screen size.

PSCALE

Geo Panel: **Display > DISPLAY_OPTION > Scale to Window Size**

The PSCALE command replots the current image by automatically scaling it to fill the active window.

ZOOMIN

Geo Panel: **Display > DISPLAY_OPTION > Zoomin**

The ZOOMIN command allows the user to zoom into the plotted picture with the help of a zoomin window. The zoomin window appears once this command is

issued. The window is specified by selecting two opposite corner points with the use of the mouse and the crosshairs.

Notes

1. The ZOOMIN command is an action command resulting in the automatic formation of the zoomed plot when the command is completed. To complete the command, select the first corner point of the zoomin window by positioning the crosshairs at the desired point and clicking the mouse. Next move the crosshairs to the opposite corner point and click the mouse again.
2. This command may be repeated for further magnification.
3. The "RESET" command resets the plot to the default size.
4. Screens resulting from repeated use of the command can be recovered by the use of the ZOOMOUT command.

ZOOMOUT

Geo Panel: **Display > DISPLAY_OPTION > Zoomout**

The ZOOMOUT command allows the user to recover preceding screens resulting from repeated use of the ZOOMIN command. The last ten screens resulting from the ZOOMIN command are stored. The ZOOMOUT command recovers the stored screens in the reversed order. If the ZOOMIN command was not used or if all screens are recovered, the command repaints the screen with the current plot.

Example: Suppose that the ZOOMIN command was issued 8 times and suppose that the generated screens are numbered accordingly, such that the first zoomin command issued produces screen 1, and so on until the eighth ZOOMIN command issued produces screen 8. Now, if the ZOOMOUT command is issued, screen 8 is recovered and if the command is repeated, screens 7, 6, ... , 1 are recovered consecutively.

SHRINK

Geo Panel: **Display > DISPLAY_OPTION > Shrink**

The SHRINK command specifies the shrinking of elements in future plots. The shrink factor specifies the amount of shrinkage relative to the element size.

Entry & Option Description

Shrink factor

Shrink factor. Ratio by which the elements are shrunk. Must be between 0.0 and 0.5.
(default is 0.15)

Notes

1. This command makes it possible to examine element connectivities.
2. The command may be repeated for further shrinking of the elements.
3. The "RESET" command resets the plot to the default setting (no shrink).
4. A shrink factor of 0.0 results in regular subsequent element plots.

Example: `SHRINK, 0.3`

This command instructs the program to shrink subsequent element plots by a factor of 30%.

HIDE_OPT

Geo Panel: **Display > DISPLAY_OPTION > Set Hide Option**

The HIDE_OPT command sets options for hiding element faces for subsequent element plots.

Entry & Option Description

Hidden face removal algorithm

Algorithm to be used for hidden face removal.

= Depth sort	Depth-sort algorithm
= Z-buffer	Z-buffer algorithm
	(default is depth sort)

Element face plot flag

Element face plot flag.

= All	plot all faces
= N. normal	plot negative-normal faces only
= P. normal	plot positive-normal faces only
	(default is all)

Hidden face removal for hardcopy

Hidden face removal algorithm for hardcopies. Prompted only if "Hidden face removal algorithm" is Z-buffer.

= Depth sort	Depth-sort algorithm
= Z-buffer	Z-buffer algorithm
	(default is depth sort)

Note

The Z-buffer is much more demanding on the machine resources because of higher resolution required. Obtaining a hardcopy may be very slow if high resolution printers are used. The process may be accelerated by choosing the Depth-sort algorithm for the generation of the hardcopy.

HIDDEN

Geo Panel: **Display > DISPLAY_OPTION > Hidden Element Plot**

The HIDDEN command activates or deactivates the plotting of hidden elements faces.

Entry & Option Description

Hidden plot flag

Specifies type of plot.

= Wire frame	wire frame (show all elements)
= Hidden plot	hidden plot (do not show hidden elements)
	(default is hidden plot)

Example: `HIDDEN,1,`

This command activates hidden line element plot. Subsequent EPLOT commands will result in plotting element edges that are not hidden only.

SHADE

Geo Panel: **Display > DISPLAY_OPTION > Shaded Element Plot**

The SHADE command activates or deactivates shading of element plots.

Entry & Option Description

Shaded plot flag

Shading flag.

= Off	non-shaded plots
= Flat	use flat shading
= Gouraud	use Gouraud shading
	(default is flat)

Positive normal side color

Color index to be used for positive normal side of the elements. Refer to Note 4 for valid colors.
(default is 4, red)

Negative normal side color

Color index to be used for the negative normal side of the elements. Refer to Note 4 for valid colors.
(default is 1, blue)

Tolerance intensity for gouraud shading

Gouraud shading intensity. Prompted only if Gouraud shading is selected. (between 0.0 and 1.0)
(default is 0.2)

Notes

1. The color variation of the shaded plot is a function of the view direction and the direction of light source, specified by the LIGHT command.
2. Shading is represented only by the two distinct specified colors for 16-color graphic cards. Shades of the specified colors are used for graphic adapters supporting 256 colors.
3. The shading flag is specifically helpful for shell elements. Adjacent elements shown in different distinct colors indicate that the top and bottom fibers are reversed. Top and bottom stresses will be mixed up for these elements. To fix

this problem, the user needs to redefine the nodal connectivity for some elements and reverse the orientation. The orientation of all elements associated with a surface can be reversed by turning on the default meshing flag for surfaces and using the SFREORIENT command.

4. The following colors are accepted:
   1. Blue 2. Green 3. Cyan 4. Red 5. Magenta 6. Yellow 7. Gray
5. Printing of images with Gouraud shading is very demanding on disk space. Flat shading on the other hand is not as demanding on disk space and hardcopies can be obtained using that option.

Example: `SHADE, 1;`

This command activates shading for all subsequent element plots.

LIGHT

Geo Panel: **Display > DISPLAY_OPTION > Light Source**

The LIGHT command specifies the location of the light source for shaded plots.

Entry & Option Description

XYZ-Coord of light source

Coordinates of the light source relative to view direction system and not in absolute 3D space.
(defaults are 0.0, 0.0, 0.0)

Light option

Light option.

= Image	light source based on image
= Object based	light source based on object
	(default is image)

Note

The user may choose the location of the light source for improved shading effects.

Example: `LIGHT,1,2,2`

This command places the light source at point (1., 2., 2.) relative to the view direction coordinates.

EVAL_BOUND

Geo Panel: **Display > DISPLAY_OPTION > Eval Element Bound**

The EVAL_BOUND command sets flags for evaluating boundary faces and edges. The command is particularly useful when nodes and elements are not associated with geometric entities.

Entry & Option Description

Boundary face evaluation

Flag for boundary face evaluation for 3D elements.

= No	do not evaluate boundary faces
= Yes	evaluate boundary faces for 3D elements (bricks and tetras), a face that is not common to two or more elements is identified as a boundary face. The resulting information is stored and used later to accelerate plotting *(default is no)*

Boundary edge evaluation

Flag for edge evaluation.

= No	do not find edges
= Yes	find and evaluate model edges. An edge is identified if the angle between the normals to two adjacent boundary faces exceeds the threshold angle to be specified below *(default is no)*

Tolerance angle to ignore curvature

Threshold angle in degrees for edge evaluation. *(default is 20)*

Evaluation criterion

Mesh or geometry.

Notes

1. Face evaluation is not needed if all 3D elements are created by meshing of geometric entities. Face evaluation for large 3D models can be time consuming, and the user may elect to plot every face of every element instead.
2. Edge evaluation applies to both 2D and 3D elements for 3D models, and works with both Gouraud and flat shading algorithms (refer to the SHADE command).

BOUNDARY

Geo Panel: **Display > DISPLAY_OPTION > Set Bound Plot**

The BOUNDARY command controls plotting of element boundaries for subsequent shaded and hidden element plots. All element postprocessing plots will be based on options selected in this command.

Entry & Option Description

Boundary plot

Flag to specify element boundary plotting.

= None	do not plot element boundaries
= Face	plot element boundaries

= Edge — find and plot model edges only. The edges are found by comparing the angle between the normals to every two adjacent boundary faces to the specified angle threshold. Refer to the EVAL_BOUND command. If the angle is larger than the tolerance, an edge is identified along the common boundary *(default is face)*

Geometry associated boundary plot flag

Flag used for plotting element faces for 3D elements.

= Yes — plot boundary faces only. A boundary face is a face that is associated with a surface or region

= No — plot all faces of all elements *(default is yes)*

Example: `BOUNDARY, 1, 0,`

This command activates plotting of element boundary lines and instructs the program to plot every face of every element.

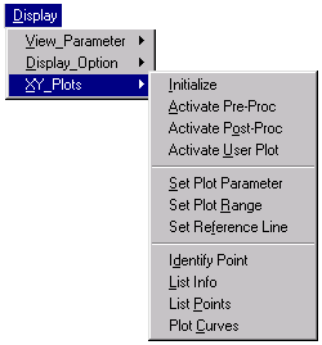



▶ XY_PLOTS Menu

Geo Panel: Display > XY PLOTS

This menu contains commands to activate, setup, and generate xy-plots. The data for xy-plots may be acquired from the preprocessor using the ACTXYPRE command, from the post-processor using the ACTXYPOST command, or from |a user-file using the ACTXYUSR command.

Figure 9-4 XY_Plots Menu



INITXYPLOT

Geo Panel: Display > XY_PLOTS > Initialize

The INITXYPLOT command clears all previously activated xy-plots from the xy-plot buffer. The initialization includes graphs activated using the ACTXYPRE, ACTXYPOST, and the ACTXYUSR commands.

ACTXYPRE

Geo Panel: Display > XY_PLOTS > Activate Pre-Proc

The ACTXYPRE command loads data that was generated by the preprocessor (like time, temperature, post dynamic, and B-H curves). This command will initialize any previously activated graphs by commands ACTXYUSR or ACTXYPOST.

Entry & Option Description

Graph number

Number of the graph to be activated. Up to 6 graphs can be activated simultaneously.
(default is maximum graph number activated + 1)

Curve type

Type of graph to be activated.

= Time/temp/BHC/MPC	time, temperature, B-H, MPC (material property), or asymmetric loading curve
= Postdynamic	postdynamic curves
= Fatigue	fatigue curves
	(default is time/temp/BHC/MPC)

Component

Component to be activated. Valid components depend on “curve type”.

For curve type = time/temp/BHC/MPC:

= TIME	time curve
= TEMP	temperature curve
= BHC	B-H curve for electromagnetics

= MPC	material property curve
= FCOEF	circumferential load coefficients curve for the asymmetric loading option in STAR
= FSIN	Fourier sine coefficients curve for asymmetric loading option in STAR
= FCOS	Fourier cosine coefficients curve for asymmetric loading option in STAR

For curve type = postdynamic:

= TIME	postdynamic time curve
= FREQ	postdynamic frequency curve

For curve type = fatigue:

= PSN	PSN fatigue curve
= SN	SN fatigue curve
= SMT	SMT fatigue curve

Curve number
Curve label.

Graph color
Color to be used for plotting. (1 to 16)
(default is foreground color)

Graph line style
Line style to plot graph.

= Blank	blank
= Solid	solid line
= Broken	dotted line
	(default is solid)

Graph symbol style
Symbol type for plotting at points on the xy graph.

= No symbol	no symbol
= Circle	circle
= Filled circle	filled circle
= Square	square
= Filled square	filled square
= Upward triangle	upward triangle
= Filled upward triangle	filled upward triangle
= Downward triangle	downward triangle
= Filled downward triangle	filled downward triangle
= Star	star
= Filled star	filled star
	(default is circle)

ACTXYPOST

Geo Panel: Display > XY_PLOTS > Activate Post-Proc

The ACTXYPOST command loads information to be used in generating x-y plots related to postprocessing from the database to the plot buffer. The graph information is read from existing files prepared by analysis modules. The prompts depend on the analysis type that is active for postprocessing (see ACTPOST command). Each graph is assigned a number. Up to 6 graphs can be loaded in one plot. This command will initialize any graphs previously activated by ACTXYUSR or ACTXYPRE.

Linear Structural Analysis

Entry & Option Description

Graph number

Graph number. (1 to 6)
(default is highest defined + 1)

y-variable

Name of variable in the y-direction. Note that x-y plots are only available when an adaptive method has been activated by the ADAPTIVE command before running the analysis. The x-axis is fixed to be the iteration number. The first iteration is not stored in the plot buffer and hence the actual iteration numbers are those shown in the graph + 1.

ERROR	global error. Refer to the ACTSTR and ADAPTIVE commands
ENERGY	total strain energy
DOF	number of degrees of freedom
DISP	maximum resultant displacement
STRESS	maximum von Mises stress
	(default is ERROR)

Graph color

Color to be used for plotting. (1 to 16)
(default is foreground color)

Graph line style

Line style to plot graph.

= Blank	blank
= Solid	solid line
= Broken	dotted line
	(default is solid)

Graph symbol type

Symbol type for plotting at points on the x-y graph.

= No symbol	no symbol
= Circle	circle
= Filled circle	filled circle
= Square	square
= Filled square	filled square
= Upward triangle	upward triangle

= Filled upward triangle	filled upward triangle
= Downward triangle	downward triangle
= Filled downward triangle	filled downward triangle
= Star	star
= Filled star	filled star
	(default is circle)

Graph id
Graph identification. Default depends on the y-variable entry.

Nonlinear Structural Analysis NSTAR

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is highest defined + 1)

x-variable
Name of variable in the x-direction. You may plot response versus time steps or a displacement component versus load factor.

TIME	time step number
FREQ	frequency
UX	displacement in the X-direction
UY	displacement in the Y-direction
UZ	displacement in the Z-direction
RX	rotation about the X-axis
RY	rotation about the Y-axis
RZ	rotation about the Z-axis
URES	resultant displacement
RFX	reaction force in the x-direction
RFY	reaction force in the y-direction
RFZ	reaction force in the z-direction
RFRES	resultant reaction force
RMX	reaction moment about the x-direction
RMY	reaction moment about the y-direction
RMZ	reaction moment about the z-direction
RMRES	resultant reaction moment
	(default is TIME)

y-variable
Not prompted if the x-axis is a displacement component in which case the y-axis is fixed to be the load factor.

UX	displacement in the x-direction
UY	displacement in the y-direction
UZ	displacement in the z-direction
RX	rotation about the x-direction
RY	rotation about the y-direction
RZ	rotation about the z-direction
URES	resultant displacement
VX	velocity in the x-direction
VY	velocity in the y-direction

VZ	velocity in the z-direction
WX	angular velocity about x-direction
WY	angular velocity about y-direction
WZ	angular velocity about z-direction
VRES	resultant velocity
AX	acceleration in the x-direction
AY	acceleration in the y-direction
AZ	acceleration in the z-direction
BX	angular acceleration about x-direction
BY	angular acceleration about y-direction
BZ	angular acceleration about z-direction
ARES	resultant acceleration
LFACT	load factor
RFX	reaction force in the x-direction
RFY	reaction force in the y-direction
RFZ	reaction force in the z-direction
RFRES	resultant reaction force
RMX	reaction moment about the x-direction
RMY	reaction moment about the y-direction
RMZ	reaction moment about the z-direction
RMRES	resultant reaction moment
FORCE	force
MOMENT	moment
STRESS	stress for 2D and 3D elements
SX	normal stress in the x-direction
SY	normal stress in the y-direction
SZ	normal stress in the z-direction
TXY	x-y shear stress
TXZ	x-z shear stress
TYZ	y-z shear stress
P1	first principal stress
P2	second principal stress
P3	third principal stress
VON	von Mises stress
	(default is UX)

Node number
Node number.

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

= Blank	blank
= Solid	solid line
= Broken	dotted line
	(default is solid)

Graph symbol type
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-variable entry.

Advanced Dynamic Analysis ASTAR

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is highest defined + 1)

x-variable
May be one of the following.

TIME	time
FREQ	frequency
	(default is TIME)

y-variable
May be one of the following.

UX	displacement in the x-direction
UY	displacement in the y-direction
UZ	displacement in the z-direction
RX	rotation about the x-direction
RY	rotation about the y-direction
RZ	rotation about the z-direction
URES	resultant displacement
VX	velocity in the x-direction
VY	velocity in the y-direction
VZ	velocity in the z-direction
WX	angular velocity about x-direction
WY	angular velocity about y-direction
WZ	angular velocity about z-direction
VRES	resultant velocity
AX	acceleration in the x-direction
AY	acceleration in the y-direction
AZ	acceleration in the z-direction
BX	angular acceleration about x-direction
BY	angular acceleration about y-direction
BZ	angular acceleration about z-direction
ARES	resultant acceleration
RFX	reaction force in the x-direction
RFY	reaction force in the y-direction
RFZ	reaction force in the z-direction
RFRES	resultant reaction force
RMX	reaction moment about the x-direction
RMY	reaction moment about the y-direction
RMZ	reaction moment about the z-direction
RMRES	resultant reaction moment
FORCE	force

STRESS	stress for 2D and 3D elements or force/moment for 1D elements *(default is UX)*

Node/Element number
Node or element number depending on the y-axis entry.

Mag/Phase
Flag to plot magnitude or absolute phase angle of a displacement component versus frequency. Prompted for harmonic analysis only. Phase angles are prompted in harmonic analysis for response graphs (displacements, velocities, and accelerations).

= 0	magnitude
= 1	phase *(default is 0)*

Direction/Component
Component label. Prompted only when the y-variable is set to FORCE, MOMENT or STRESS.

= 1	x-translation (or equivalent)
= 2	y-translation (or equivalent)
= 3	z-translation (or equivalent)
= 4	rotation about the x-axis (or equivalent)
= 5	rotation about the y-axis (or equivalent)
= 6	rotation about the z-axis (or equivalent)

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

= Blank	blank
= Solid	solid line
= Broken	dotted line *(default is solid)*

Graph symbol type
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-variable entry.

Transient Heat Transfer Analysis

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is highest defined + 1)

x-variable
Only time is valid for heat transfer.

TIME time

y-variable
May be one of the following.

TEMP	nodal temperature
GRADX	temperature gradient in the x-direction
GRADY	temperature gradient in the y-direction
GRADZ	temperature gradient in the z-direction
GRADN	resultant temperature gradient
HFLUXX	heat flux in the x-direction
HFLUXY	heat flux in the y-direction
HFLUXZ	heat flux in the z-direction
HFLUXN	resultant heat flux
	(default is TEMP)

Node number
Node number.

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

= Blank	blank
= Solid	solid line
= Broken	dotted line
	(default is solid)

Graph symbol type
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-variable entry.

Transient Fluid Flow Analysis

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is highest defined + 1)

x-variable
Only time is valid for fluid flow.

TIME	time

y-variable
May be one of the following.

VRES	resultant velocity
VX	velocity in the x-direction
VY	velocity in the y-direction
VZ	velocity in the z-direction

STREAM	stream function
PRESS	pressure
TAUXY	tauxy
TAUYZ	tauyz
TAUZX	tauzx
TEMP	temperature
GRADX	temperature gradient in X-direction
GRADY	temperature gradient in Y-direction
GRADZ	temperature gradient in Z-direction
GRADN	temperature gradient in normal direction
TKE	turbulence kinetic energy
EPS	dissipation rate
DENS	fluid density
MACH#	mach number
	(default is VRES)

Node number
Node number.

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

Graph symbol type
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-variable entry.

High-Frequency 2DHFRQ Electromagnetic Analysis

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is highest defined + 1)

Node number
Node number.
(default is 1)

y-variable
The y-axis may be one of the components described below. The x-axis is not prompted for and is fixed to be frequency in GHz.

ALPHA	real part of propagation constant. Non-zero for decaying modes only
BETA	imaginary part of propagation constant
EPSEFF	effective dielectric constant
PHASEV	phase velocity in meters/second
ALPHAC	attenuation constant in dB/m due to conductor losses

ALPHAD	attenuation constant in dB/m due to dielectric losses

The following components are computed only when the number of conductors is non-zero.

ZMI	modal impedance from power-current calculation
LMI	modal inductance from power-current calculation
CMI	modal capacitance from power-current calculation
RMI	modal resistance from power-current calculation
GMI	modal conductance from power-current calculation

The following components are computed only when the number of integration paths is non-zero.

ZMV	modal impedance from power-voltage calculation
LMV	modal inductance from power-voltage calculation
CMV	modal capacitance from power-voltage calculation
RMI	modal resistance from power-voltage calculation
GMV	modal conductance from power-voltage calculation *(default EPSEFF)*

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Grade line style
Line style to plot graph.

Graph symbol style
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-variable entry.

High-Frequency XTALK and 2DXTALK Electromagnetic Analysis

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is highest defined + 1)

Mode number
Mode number.
(default is 1)

y-variable
The following components are plotted versus mode number for 2DXTALK only.

BETA	imaginary part of propagation constant
EPSEFF	effective dielectric constant
PHASEV	phase velocity in meters/second
ALPHAC	attenuation constant in dB/m due to conductor losses
ALPHAD	attenuation constant in dB/m due to dielectric losses
ZM	modal impedance

LM	modal inductance
CM	modal capacitance
RM	modal resistance
GM	modal conductance

The following components are plotted versus time for both XTALK and 2DXTALK.

VTLSNEAR	near end voltages
VLTSFAR	far end voltages
	(default EPSEFF)

Line number
Line number (prompted only when the y-axis is VLTSNEAR or VLTSFAR).

Refer to syntax for adaptive linear static analysis for help on the following prompts.

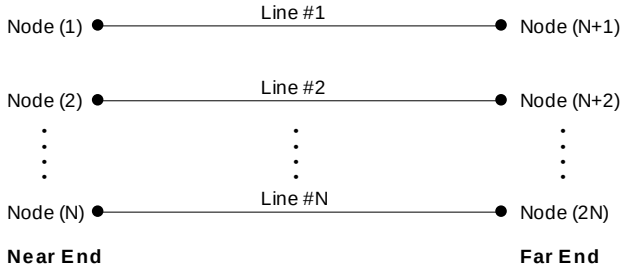
Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

Graph symbol style
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-variable entry.

Note
The following sketch illustrates the near/far end and node numbering convention in 2DXTALK and XTALK. This convention is used in writing the SPICE input deck from 2DXTALK.



High-Frequency CAV3D and CAVAXI Electromagnetic Analysis

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is highest defined + 1)

Mode

Mode number. Prompted only for CAVAXI in which case the x-axis is the harmonic number.
(default is 1)

y-variable
The following components are plotted versus mode number.

RFREQ	resonant frequency in MHz
QTOTAL	total quality factor of the cavity
QCOND	quality factor associated with conductor losses only
QDIELEC	quality factor associated with dielectric losses only

The following components are only computed if an integration path has been defined.

RESIST	equivalent resistance in Ohms
INDUCT	equivalent inductance in Henrys
CAPAC	equivalent capacitance in Farads

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

Graph symbol style
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-variable entry.

High-Frequency S-Parameters Electromagnetic Analysis

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is highest defined + 1)

row-number
Row number for the S-matrix element to be plotted.

column-number
Column number for the S-matrix element to be plotted.

y-variable
The following components are plotted versus frequency plot.

MatrixS_Mag	Magnitude of the S-Matrix element.
MatrixS-Phase	Phase value for the S-Matrix element.
MatrixS_dB	Magnitude in db of the S-Matrix element.
Renorm_Mag	Magnitude of the renormalized S-Matrix element.
Renorm-Phase	Renormalized phase value for the S-Matrix element.
Renorm_dB	Magnitude in db of the renormalized S-Matrix element.
Imped_Mag	Impedance magnitude of the S-Matrix element.

Imped-Phase	Impedance Phase value for the S-Matrix element.
Imped_dB	Magnitude in dB of Impedance for the S-Matrix element.
Admit_Mag	Admittance magnitude of the S-Matrix element.
Admit-Phase	Admittance phase value for the S-Matrix element.
Admit_dB	Magnitude in dB of the admittance for the S-Matrix element.
Deemb_Mag	Magnitude of the Deembedded S-Matrix element.
Deemb-Phase	Phase value for the deembedded S-Matrix element.
Deemb_dB	Magnitude in dB of the deembedded S-Matrix element.




graph color
Color to be used for plotting.

graph line style
Line style to plot graph.

graph symbol style
Symbol type for plotting at points on the x-y graph.

graph id
Graph identification. Default depends on the y-axis entry.

Low-Frequency Electromagnetics ESTAR

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is highest defined + 1)

Entity flag
Entity flag.

= Node	node
= Element	element
	(default is node)

x-axis
May be one of the following.

TIME	time
FREQUENCY	frequency
	(default is TIME)

y-variable
May be one of the following.

MFLUXXR	magnetic flux in the x-direction. Real component
MFLUXXI	magnetic flux in the x-direction. Imaginary component
MFLUXXA	magnetic flux in the x-direction. Amplitude
MFLUXYR	magnetic flux in the y-direction. Real component
MFLUXYI	magnetic flux in the y-direction. Imaginary component
MFLUXYA	magnetic flux in the y-direction. Amplitude
MFLUXZR	magnetic flux in the z-direction. Real component
MFLUXZI	magnetic flux in the z-direction. Imaginary component
MFLUXZA	magnetic flux in the z-direction. Amplitude
MFLUXR	resultant magnetic flux. Real component
MFLUXI	resultant magnetic flux. Imaginary component
MFIELDX	magnetic field intensity in the x-direction
MFIELDY	magnetic field intensity in the y-direction
MFIELDZ	magnetic field intensity in the z-direction
MFIELDR	resultant magnetic field intensity
MFORCEXR	magnetic force in the x-direction. Real component (nodal value only)

MFORCEXI	magnetic force in the x-direction. Imaginary component (nodal value only)
MFORCEXA	magnetic force in the x-direction. Amplitude (nodal value only)
MFORCEYR	magnetic force in the y-direction. Real component (nodal value only)
MFORCEYI	magnetic force in the y-direction. Imaginary component (nodal value only)
MFORCEYA	magnetic force in the y-direction. Amplitude (nodal value only)
MFORCEZR	magnetic force in the z-direction. Real component (nodal value only)
MFORCEZI	magnetic force in the z-direction. Imaginary component (nodal value only)
MFORCEZA	magnetic force in the z-direction. Amplitude (nodal value only)
MFORCER	resultant magnetic force. Real component (nodal value only)
MFORCEI	resultant magnetic force. Imaginary component (nodal value only)
AFORCEX	average force in x-direction (nodal value only)
AFORCEY	average force in y-direction (nodal value only)
AFORCEZ	average force in z-direction (nodal value only)
AFORCE	resultant average force (nodal value only)
EFIELDX	electrical field intensity in the x-direction
EFIELDY	electrical field intensity in the y-direction
EFIELDZ	electrical field intensity in the z-direction
EFIELDR	resultant electrical field intensity
CURDENXR	current density in the x-direction. Real component
CURDENXI	current density in the x-direction. Imaginary component
CURDENXA	current density in the x-direction. Amplitude
CURDENYR	current density in the y-direction. Real component
CURDENYI	current density in the y-direction. Imaginary component
CURDENYA	current density in the y-direction. Amplitude
CURDENZR	current density in the z-direction. Real component
CURDENZI	current density in the z-direction. Imaginary component
CURDENZA	current density in the z-direction. Amplitude
CURDENR	resultant current density. Real component
CURDENI	resultant current density. Imaginary component
NPOTENR	voltage or nodal potential. Real component
NPOTENI	voltage or nodal potential. Imaginary component
NPOTEN	voltage or nodal potential. Amplitude
PLOSSR	real component of power loss (elemental value only)
PLOSSI	imaginary component of power loss (elemental value only)
PLOSSA	amplitude of power loss (elemental value only)




Node Element Number
Node or element number.

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

Graph symbol style
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-variable entry.

Optimization Analysis

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is 1)

y-variable
Name of variable in the y-direction. The x-axis refers to the optimization set number. Valid entries are:

OP_OBJ	objective function
OP_CON	behavior constraints
OP_DVAR	design variables
	(default is OP_DVAR)

Set number
Behavior constraint or design variable set number (not prompted when y-axis = OP_OBJ).
(default is 1)

Type of results
Type of results (prompted only for behavior constraints and objective function).

= 0	results based on the finite element analysis (FEA)
= 1	results based on the approximation technique used during optimization. (Available only for loops exceeding 1 + number of design variables. Values for the converged loop are not available)
= 2	quality factor for approximations used in optimization. The quality factor varies from zero (no match between FEA and approximation) to unity (perfect match). (Available only for design set numbers exceeding 1 + number of design variables. Values for the converged loop are not available) *(default is 0)*




Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

Graph symbol style
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-variable entry.

Sensitivity Analysis Type 0 (All)

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is 1)

Response set number
Response quantity set number. The x-axis refers to the normalized design variables. All design variables are incremented in the range of bounds simulta neously. On the x-axis, a “0” value refers to the lower bound of each design vari-able while “1” refers to the upper bound of each design variable.
(default is 1)

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

Graph symbol style
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the 'response set number' entry.

Sensitivity Analysis Type 1

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is 1)

Design variable set number
Design variable set number.
(default is 1)

Response set number
Response quantity set number.
(default is 1)

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

Graph symbol style
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the 'response set number' entry.

Sensitivity Analysis Type 2 (Offset)

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is 1)

Response set number
Response quantity set number. The x-axis refers to the sensitivity set number.
(default is 1)

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

Graph symbol type
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the y-axis entry.

Sensitivity Analysis Type 3 (Local)

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is 1)

Response set number
Response quantity set number. The x-axis refers to the design variable number.
(default is 1)

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph color
Color to be used for plotting.

Graph line style
Line style to plot graph.

Graph symbol style
Symbol type for plotting at points on the x-y graph.

Graph id
Graph identification. Default depends on the 'response set number' entry.

Sensitivity Analysis Type 4 (Optm)

Entry & Option Description

Graph number
Graph number. (1 to 6)
(default is 1)

x-axis
Type of data for the x-axis. Valid entries are:

DVAR	design variables
LOOP	optimization loops
	(default is DVAR)

Dvar-loop
Design variable set number if x-axis = LOOP. Optimization loop number if x-axis = DVAR.

y-axis
Type of variables in the y-direction.

= 0	behavior constraints
= 1	objective function
	(default is 0)

(y-set)
Constraint set number. (prompted only if y-axis = 0)
(default is 1)

Refer to syntax for adaptive linear static analysis for help on the following prompts.

Graph-color
Color to be used for plotting.

Line-style
Line style to plot graph.

Symbol-type
Symbol type for plotting at points on the x-y graph.

Graph-id
Graph identification. Default depends on the y-axis entry.




ACTXYUSR

Geo Panel: **Display > XY_PLOTS > Activate User Plot**

The ACTXYUSR command loads vectors to be used in time-history like plots from a user-created file. Each graph is assigned a number. Up to 6 graphs can be loaded in one plot. This command will initialize any previously activated graphs by commands ACTXYPRE or ACTXYPOST.

Entry & Option Description

File name containing graph data
Name of the file containing user specified xy data as explained below.

Graph number
Number of the graph in the file to be loaded.

Graph color
Color to be used for plotting. (1 to 16)
(default is foreground color)

Graph line style
Line style to plot graph.

= Solid	solid line
= Broken	dotted line
= Blank	blank
	(default is solid)

Graph symbol style
Symbol type for plotting at points on the xy graph.

= No symbol	no symbol
= Circle	circle
= Filled circle	filled circle
= Square	square
= Filled square	filled square
= Upward triangle	upward triangle
= Filled upward triangle	filled upward triangle
= Downward triangle	downward triangle
= Filled downward triangle	filled downward triangle
= Star	star
= Filled star	filled star
	(default is circle)

Graph id
Graph identification.

Note

The user file is an ASCII file that should be created as follows:

a. The first line must have the number of points in the graph and then the number of graphs.
b. The second line is reserved for titles.

c. The rest lists the values of the independent variable and up to six dependent variables. The first column is for the independent variable, the second column is for the first graph, the third column for the second graph, etc. All values are read in free format.

SETXYPLOT

Geo Panel: **Display > XY_PLOTS > Set Plot Parameter**

The SETXYPLOT command sets flags and parameters for plots to be generated by the XYPLOT command. The user needs to issue this command only when the default setting is not desired.

Entry & Option Description

x logarithmic
Logarithmic scale flag in the x-direction.
= No linear scale
= Yes logarithmic scale
(default is no)

y logarithmic
Logarithmic scale flag in the y-direction.
= No linear scale
= Yes logarithmic scale
(default is no)

Number of x intervals
Number of intervals in the x-direction. Prompted only if 'no' is selected for x logarithmic flag.
(default is 10)

Number of y intervals
Number of intervals in the y-direction. Prompted only if 'no' is selected for y logarithmic flag.
(default is 10)

Show axes
Axis drawing flag.
= No do not draw axes
= x only draw the x-axis only
= y only draw the y-axis only
= Both draw both axes
(default is no)

x axis at
Location of plotting the x-axis.
= Min plot x-axis at minimum y value
= Zero plot x-axis at y = 0.0
= Max plot x-axis at maximum y value
(default is min)

y axis at
Location of plotting the y-axis.

= Min	plot y-axis at minimum x value
= Zero	plot y-axis at y = 0.0
= Max	plot y-axis at maximum x value
	(default is min)

x grid type
Grid type in the x-direction.

= 0	no grid
= 1	solid lines
= 2	dotted lines
	(default is 0)

y grid type
Grid type in the y-direction.

= 0	no grid
= 1	solid lines
= 2	dotted lines
	(default is 0)

Grid fill color
Background color number for xyplots.

Grid color
Color number for grid lines.

Relative width of XY plot
Relative width of xyplot with respect to current window. Should be between 0.0 and 1.0.
(default is 1.0)

Relative height of XY plot
Relative height of xyplot with respect to current window. Should be between 0.0 and 1.0.
(default is 1.0)

Horizontal relative position of XY plot
Relative x-position or xyplot. Should be between 0.0 and 1.0.
(default is 0.0)

Vertical relative position of XY plot
Relative y-position or xyplot. Should be between 0.0 and 1.0.
(default is 0.0)

XYRANGE

Geo Panel: **Display > XY_PLOTS > Set Plot Range**

The XYRANGE command specifies the graph ranges and scale factors.

Entry & Option Description

x scale factor
Scale factor to determine the x-range for the graph.
(default is 1.0 indicating default range)

y scale factor
Scale factor to determine the y-range for the graph.
(default is 1.0 indicating default range)

x minimum
Minimum x value in the graph.
*(default is the original (x-minimum) * (x scale factor))*

x maximum
Maximum x value in the graph.
*(default is the original (x-maximum) * (x scale factor))*

y minimum
Minimum y value in the graph.
*(default is the original (y-minimum) * (y scale factor))*

y maximum
Maximum y value in the graph.
*(default is the original (y-maximum) * (y scale factor))*

XYREFLINE

Geo Panel: **Display > XY_PLOTS > Set Reference Line**

The XYREFLINE command results in the plotting of a reference line parallel to the x- or y-axis.

Entry & Option Description

Reference line
Axis parallel to the reference line to be plotted.

= Y	Y-axis
= X	X-axis
	(default is X)

Value
Distance from the origin to the reference line. (X or Y coordinate of the reference line)

XYIDENTIFY

Geo Panel: **Display > XY_PLOTS > Identify Point**

The XYIDENTIFY command identifies the coordinates of the pixel closest to the tip point of the mouse arrow. The point can be anywhere in the graph range.




XYLIST

Geo Panel: Display > XY_PLOTS > List Info

The XYLIST command lists the available information to produce xy-plots from program-generated files. Title, node, name, and direction (if applicable) are listed for current analysis.

XYPTLIST

Geo Panel: Display > XY_PLOTS > List Points

The XYPTLIST command lists numeric values of a pattern of points for all the activated xy-graphs. Graph number, title and the corresponding values of the dependent and independent variables are listed.

Entry & Option Description

Beginning point
Beginning point to be listed. (default is 1)

Ending point
Ending point in the pattern.
(default is the last point)

Increment
Increment between points in the pattern.
(default is 1)

XYPLOT

Geo Panel: Display > XY_PLOTS > Plot Curves

The XYPLOT command generates a time-history-like plot for the vectors stored in the specified graph. The plot data must have been activated by command ACTXYPRE, ACTXYPOST, or ACTXYUSR.

Entry & Option Description

Plot graph (i)
Number of the graph to be plotted. Up to 6 graphs can be activated and plotted simultaneously.

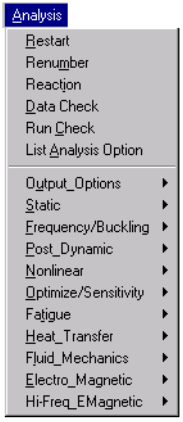
= No	no
= Yes	yes
	(default is maximum graph number activated + yes)




10 Analysis Menu

ANALYSIS Menu

Figure 10-1 Analysis Menu



This menu contains commands that specify analysis options and perform the analysis. In addition to a few general commands located at the top level, the rest of the commands are grouped based on type of analysis.

RESTART

ANALYSIS > Restart

The RESTART command activates or deactivates the restart flag for performing nonlinear (NSTAR) and transient thermal analyses. If the analysis is executed while the restart flag is on, the program will continue the analysis from the last successful solution step. If the restart flag is off, the program starts all over again from step 1. For adaptive static analysis, the ADAPTIVE command has its own restart flag to preserve results from the previous run for convergence plots.

Restart flag

Flag to activate or deactivate the restart option.

= 1: On	activate restart option
= 0: Off	deactivate restart option
	(default is on)

Example: Suppose that an NSTAR problem diverged at solution step number 5.

You can change the solution time increment, or some other parameters, to improve the chances of convergence. If the restart flag is off and you run NSTAR again, the solution will start from step 1 ignoring previous results. If the restart flag is on, the solution will continue from step 5.

RENUMBER

ANALYSIS > Renumber

The RENUMBER command sets the node renumbering flag on/off. When this flag is set on, the analysis program, internally, renumbers the nodes of the model to minimize the bandwidth and profile of the stiffness matrix for efficient solution. The renumbering process is transparent to the user, i.e. node numbers remain unchanged for all user interface. The flag is used by the skyline solver only. The results of renumbering are written to an ASCII file with extension "RNM". The command also sets the memory allocation factor.

Renumber flag

Renumbering flag.

= 1: On	renumber nodes
= 0: Off	do not renumber nodes
	(default is on)

Memory allocation factor

Memory allocation factor.

GE. 1	factor by which memory allocation is increased for renumbering
	(default is 1.0)

Renumbering scheme

Renumbering scheme.

= 0: Hybrid	hybrid
= 1: GPS	GPS
	(default is hybrid)

Notes

- This command only reads and stores the assigned value of the "renumber flag" argument. Actual renumbering is done during the analysis phase.
- When the allocated default memory is not sufficient for the RENUMBER program to perform bandwidth and profile optimization, the program will generate a message to this effect. In this case, repeat the RENUMBER command and specify a factor that increases the allocated memory and repeat the analysis.
- Using the Hybrid scheme (default option) is recommended. It can considerably cut the solution time for many problems.

Example: `RENUMBER,1, 1.0,0`

R_STATIC,

This command sets the flag to optimize the renumbering of nodes before running the analysis to improve solution time.

REACTION

ANALYSIS > Reaction

The REACTION command sets the flag for calculating the reaction forces and moments at all constrained nodes for linear static, nonlinear, and advanced dynamic analyses. Reaction forces are calculated by default for static and nonlinear analyses. For post dynamic analysis, due to the amount of calculations needed for large problems, you must request reaction forces before running the analysis. If the command is issued, the setting is used by all analysis types (on or off for all). After running the analysis, reaction forces will be available in the output file in the Cartesian coordinate system and may be listed on the screen in any desired coordinate system using the DISLIST (Results, List, Displacement/Response/Reaction) command. The ACTDIS and DISPLOT (Results, Plot, Displacement/Response/Reaction) commands can be used for plotting.

Reaction force flag

Reaction force flag.

= 0: Off	do not calculate reactions
= 1: On	calculate reactions
	(default: see command description above)

Example:

```
REACTION, 1,
R_STATIC
```

Reactions will be calculated during linear static analysis.

DATA_CHECK

ANALYSIS > Data Check

The DATA_CHECK command checks that element attributes (element group, material property set, and real constant set) have been defined for each element in the database, if needed. The R_CHECK (Analysis, Run Check) command performs more elaborate checking on the database that includes all checks performed by this command.

R_CHECK

ANALYSIS > Run Check

The R_CHECK command performs a thorough check on the current database and prepares a report on the status of the input in a file named “problem_name.CHK”. It is recommended that this command be always issued before running any analysis.

In addition to checking the existence of the database, nodes, and elements, the command performs the following:

- Checks that there is an element group, a material property set and a real constant set associated with each element whenever needed. A SPRING element for example does not need a material property set and no error is given if none is defined. An error message will result if no material set has been defined for a SHELL4 element.
- Checks element coordinate systems when appropriate.
- Issues a warning message if a non existing node is used to define an element.
- For plane and all shell elements, a warning message is issued in the following cases:
  - If the aspect ratio of an element is larger than a reasonable value depending on the analysis and element types.
  - If an angle in a 3-node element is less than 20 or more than 135 degrees.
  - If an angle in a 4-node element is less than 45 or more than 135 degrees.
- For BEAM3D, BOUND, and all triangular SHELL elements, element connectivity is checked and a warning is issued if the area defined by the 3 nodes is less than 1.0E-15. Orientation of BEAM3D elements is also checked.
- Checks warping and twisting of brick elements like SOLID, MAG3D, and FLOW3D.
- Other checks are also done depending on the analysis type, relevant messages are issued if errors are found.

Analysis type
Analysis type.

= STATIC	linear structural
= NONLINEAR	nonlinear structural
= FREQUENCY	frequency
= BUCKLING	buckling
= THERMAL	thermal
= FLUIDFLOW	fluid flow
= EMAGNETIC	electro-magnetic
	(default is last analysis run or STATIC)

A_LIST

ANALYSIS > List Analysis Option

The A_LIST command lists the active options for the desired analysis type.

Type
Type of analysis.

= STATIC	linear static analysis
= FFESTATIC	FFE linear static analysis
= STRESS	linear stress analysis
= FREQUENCY	frequency analysis
= BUCKLING	buckling analysis
= OPTIMIZATION	optimization analysis
= SENSITIVITY	sensitivity analysis
= FATIGUE	fatigue analysis
= NONLINEAR	nonlinear analysis
= THERMAL	thermal analysis
= FFETHERMAL	FFE thermal analysis
= FLOW	fluid flow analysis
= EMAG	electromagnetic analysis
= HFREQUENCY	high-frequency electromagnetic analysis

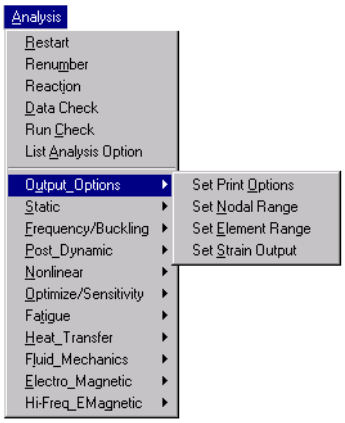
(default is STATIC)




OUTPUT_OPTIONS Menu

Analysis > OUTPUT_OPTIONS

Figure 10-2 Output_Options Menu



This menu contains commands to control information to be written to the output file (.OUT). This ASCII output file can get very large if left uncontrolled for large models. This menu contains commands to select results at specific nodes and elements. It also controls whether to include strain results in the output file. Binary results, accessed by commands in the Results menu, are always stored for all elements and nodes automatically.

PRINT_OPS

Analysis > OUTPUT_OPTIONS > Set Print Options

The PRINT_OPS command controls the printing of displacements, velocities, accelerations, mode shapes, stiffness matrix, and detailed input information in the output file. An option to print the mass, stiffness, and geometric stiffness matrices is also provided. A FORTRAN code to read the generated files is given. For temperature, temperature gradient, and heat flux printing, use the HT_OUTPUT (Analysis, Heat_Transfer, Thermal Output Options) command.

Displacement print flag

Flag for displacement printout.

= 0	no printout
= N	print at every N solution step *(default is 1)*

Velocity print flag

Flag for velocity printout.

= 0	no printout
= N	print at every N solution step *(default is 0)*

Acceleration print flag

Flag for acceleration printout.

= 0	no printout
= N	print at every N solution step *(default is 0)*

Mode shape print flag

Flag for mode shape printout.

= 0: Do not print	no printout
= 1: Mass normalized	print mode shapes normalized with respect to mass

= 2: Unity normalized print mode shapes normalized to unity
(default is 1)

Stiffness matrix print flag

Flag for stiffness matrix and equation number printout.

= 0: No printout	no printout
= 1: Stiffness matrix	print stiffness matrix in the output file
= 2: Stf/Mass/G st matrices	depending on the analysis type, print the stiffness, mass and geometric stiffness matrices in files "ori.stf", "ori.mas" and "ori.gst", respectively. Not available for NSTAR. (See note below)
= 3: Elemental Stf/Mass matrices	print elemental stiffness and mass matrices. See notes 2 and 3 below.

(default is no printout)

Temperature print flag

Flag for temperature printout. Currently not active, use the HT_OUTPUT command.

Temperature gradient print flag

Flag for temperature gradient printout. Currently not active, use the HT_OUTPUT command.

Heat flow print flag

Flag for heat flow printout. Currently not active, use the HT_OUTPUT command.

Input print flag

Flag for detailed input printout.

= 0: No	no printout
= 1: Yes	print nodal coordinates, element connectivities, material and real constant sets, and applied load vector including the effect of surface pressures

(default is no)

Output flag

Overwrite/append flag for analysis output.

= 0: Overwrite	overwrite on previous output
= 1: Append	append to previous output

(default is overwrite)

Notes

- The following code is provided primarily for opening the binary files containing the system stiffness, mass, and geometric stiffness matrices generated by COSMOS/M for external use with other programs. The three files, ORI.STF, ORI.MAS, and ORI.GST, representing the system stiffness, mass and geometric stiffness matrices, are created when the 'stiffness matrix print flag' is set equal to three matrices.

  The program below reads the contents of each file and writes to the screen. You can, however, modify the code or integrate it into your own program as needed. Due to symmetry, only the upper half of the matrix is read column by

column and stored in the “STOR” array. Each column lists the diagonal of the matrix and all terms up to the skyline. All terms above the skyline, in any column, are zeros.

Program

```
C   Program to read the original K,KG,MASS matrices
C   written by STAR with stiffness matrix print flag set
C   to print the three matrices. You will find this
C   code in your COSMOS/M directory in the ORI.f file.
C
C   DEFINITION OF VARIABLE:
C
C   STOR  : Array to store the [K], [KG] or [M] matrices
C   IWRK  : Unit number
C   NEQ   : Total number of equations
C   N     : Equation number
C   IRECL : Record length of each equation
C
    REAL*8 STOR
    DIMENSION STOR (20000)
    LOGICAL FLE
    IWRK = 32
C   Open file to read the original stiffness matrix.
C
          INQUIRE(FILE='ORI.STF'//CHAR(0),EXIST=FLE)
          IF(FLE) THEN
          OPEN(IWRK,FILE='ORI.STF',FORM='UNFORMATTED',
       1  ACCESS='SEQUENTIAL',STATUS='OLD')
C
          READ (IWRK,*) NEQ
          WRITE (*,*) ' STIFFNESS MATRIX, NEQ = ' ,NEQ
          DO 10 I=1,NEQ
          READ (IWRK,*) N,IRECL,(STOR(J),J=1,IRECL)
          WRITE(*,*) ' Equation = ' ,N
          WRITE(*,'(10e12.4)') (STOR(J),J=1,IRECL)
       10 CONTINUE
C
          ENDIF
C   Open file to read the original mass matrix.
C
          INQUIRE(FILE='ORI.MAS'//CHAR(0),EXIST=FLE)
          IF(FLE) THEN
          OPEN(IWRK,FILE='ORI.MAS',FORM='UNFORMATTED',
       1  ACCESS='SEQUENTIAL',STATUS='OLD')
C
          READ (IWRK,*) NEQ
          WRITE (*,*) ' MASS MATRIX, NEQ = ' ,NEQ
          IF(MASS.NE.2) THEN
          WRITE(*,*) ' LUMPED MASS VECTOR, NEQ=', NEQ
          READ(IWRK,*) (STOR(I), I=1,NEQ)
          WRITE(*,'(10e12.4)') (STOR(J), J=1,NEQ)
          ELSE
          WRITE (*,*)' CONSISTENT MASS MATRIX, NEQ=', NEQ
          DO 20 I=1,NEQ
```

```
            READ (IWRK,*) N,IRECL,(STOR(J),J=1,IRECL)
            WRITE(*,*) ' Equation = ' ,N
            WRITE(*,'(10e12.4)') (STOR(J),J=1,IRECL)
         20 CONTINUE
            ENDIF
            ENDIF
C   Open file to read the original geometric stiffness
    matrix.
C
            INQUIRE(FILE='ORI.GST'//CHAR(0),EXIST=FLE)
            IF(FLE) THEN
            OPEN(IWRK,FILE='ORI.GST',FORM='UNFORMATTED',
         1  ACCESS='SEQUENTIAL',STATUS='OLD')
            READ (IWRK,*) NEQ
            WRITE (*,*) ' GEOMETRIC STIFFNESS MATRIX,
             NEQ = ' ,NEQ
            DO 30 I=1,NEQ
            READ (IWRK,*) N,IRECL,(STOR(J),J=1,IRECL)
            WRITE(*,*) ' Equation = ' ,N
            WRITE(*,'(10e12.4)') (STOR(J),J=1,IRECL)
         30 CONTINUE
            ENDIF
            STOP
            END
```

- Reading element Stiffness and Mass Matrices:

  Element stiffness and mass matrices are stored in terms of the global Cartesian system in the problem-name.ESF and problem-name.EMM files. These files contain titles, general control parameters in addition to the matrices themselves. The records in these files are described below:

  Record 1: File_Code, Message (5A1, 75A1)
  - field 1: '08.01' code for stiffness matrix
    '09.01' code for mass matrix
  - field 2: File name

  Record 2: Title (up to 80 characters)

  Record 3: File creation date.

  Record 4: Nelem, NDOF, IMASS-Type (3I10)
  - field 1: Number of elements in the mesh
  - field 2: Number of DOF per node in the element
  - field 3: Mass formulation 0=consistent, 1=lumped

  Records 5 to 13 are reserved for future use

  Record 14: IELEM, NNODE, NMATC, EL-ID, I-connect
  - field 1: Element label
  - field 2: Number of nodes in the element
  - field 3: Number of elements in the upper triangle or number of diagonal terms for mass matrices with lumped option.
  - field 4: ID of element type (see note 3 below)

field 5: Connectivity of the elements (node label list)

Record 15: Element stiffness or mass matrix (upper triangle or diagonal terms of lumped mass option). NMATC values are written. This record contains a string of the rows of the upper triangular matrix sequentially starting from the top row down to the bottom row, one after the other. The order of terms in the first row corresponds to the number of degrees of freedom for the first node and then those for second node and so on.

Records 14 and 15 are repeated to write all elements in the model sequentially in each file.

- EL-ID is listed below for various types of elements:

  1: BEAM2D & BEAM3D
  2: TRUSS2D & TRUSS3D
  3: PIPE
  4: ELBOW
  5: BOUND
  6: SOLID
  7: SHELL3T
  8: PLANE2D
  9: SHELL3
  10: SHELL3L
  11: MASS
  12: SHELL4T
  13: SHELL4L
  14: SHELL4
  18: SPRING
  20: GENSTIFF
  21: TETRA4
  22: TETRA10
  25: CPCNS (coupling point to point)
  26: CPCNS (coupling point to line)
  27: CPCNS (coupling point to surface)
  28: CPDOF
  29: SHELL9
  30: SHELL9L
  31: RBAR
  32: SOLIDL
  33: TETRA4R
  34: SHELL6
  37: BOND

Example: `PRINT_OPS, 1, 0, 1;`

This command instructs the program to print displacements and accelerations every other solution step when the analysis is performed.

Velocities are not to be printed.

PRINT_NDSET

Analysis > OUTPUT_OPTIONS > Set Nodal Range

The PRINT_NDSET command defines groups of nodes for which displacements, velocities, accelerations, and temperatures will be written in the output file (used with STAR, NSTAR, ASTAR, and HSTAR modules only). Up to 10 groups can be specified. Binary results, accessed by commands in the Results menu, are always stored for all nodes automatically.

Number of groups
Number of groups to be specified.

Beginning node of group (i)
Beginning node of group i.

Ending node of group (i)
Ending node of group i. (i=1,2,.....,10)

Example: `PRINT_NDSET, 3, 1, 35, 40, 45, 140, 150`

This command instructs the program to print displacements, velocities and accelerations for nodes 1 through 35, 40 through 45 and 140 through 150 in the output file.

PRINT_ELSET

Analysis > OUTPUT_OPTIONS > Set Element Range

The PRINT_ELSET command defines groups of elements for which stresses will be written in the output file (used with STAR and NSTAR modules only). Up to 10 groups can be specified. Binary results, accessed by commands in the Results menu, are always stored for all elements automatically.

Number of groups
Number of groups to be specified.

Beginning element of group (i)
Beginning element of group i.

Ending element of group (i)
Ending element of group i. (i=1,2,.....,10)

Example: `PRINT_ELSET, 3, 1, 35, 40, 45, 140, 150`

This command instructs the program to print stresses for elements 1 through 35, 40 through 45, and 140 through 150 in the output file when the analysis is performed.

STRAIN_OUT

Analysis > OUTPUT_ OPTIONS > Set Strain Output

The STRAIN_OUT command controls the writing of strains to the output file for nonlinear analysis.

Strain output flag
Main flag for strain output.
= 0: No — do not write strains (No more prompts are issued)
= 1: Yes — write strain components as specified next
(default is no)

Total strain output flag
Flag for total strain output.
= 0: No — no total strain output
= 1: Yes — include total strain output
(default is no)

Thermal strain output flag
Flag for thermal strain output.
= 0: No — no thermal strain output
= 1: Yes — include thermal strain output
(default is no)

Creep strain output flag
Flag for creep strain output.
= 0: No — no creep strain output
= 1: Yes — include creep strain output
(default is no)

Plastic strain output flag
Flag for plastic strain output.
= 0: No — no plastic strain output
= 1: Yes — include plastic strain output
(default is no)

Principal strain output flag
Flag for principal strains.
= 0: No — no principal strains output
= 1: Yes — include principal strains in the output file
(default is no)

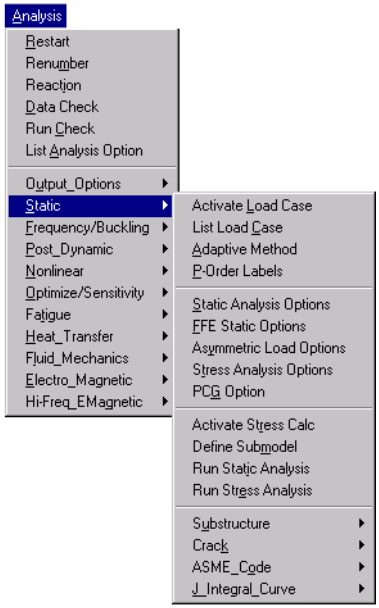
Note
At least one type of strain output should be requested if strain output flag is on.

Example: `STRAIN_OUT,1,1,,,1`
This command requests total and plastic strains output for the nonlinear analysis to be performed.

STATIC Menu

Analysis > STATIC

Figure 10-2 Static Menu



This menu contains commands related to STAR, the linear static structural analysis module. STAR calculates displacements due to various types of loads using linear analysis. Refer to the COSMOS/M Basic System manual for details. The STRESS module is also included in this menu and it calculates strains and stresses based on displacement results from STAR. Menus for the use of the substructuring utility and crack specification are provided.

LCSET

Analysis > STATIC > Activate Load Case

The LCSET command activates or deactivates specified primary load cases for static analysis. The linear static analysis program calculates the response of the system to all active load cases. You need to use this command only if you want to instruct the program to ignore some load cases.

Beginning load case
Beginning load case in the pattern.
(default is 1)

Ending load case
Ending load case in the pattern.
(default is beginning load case)

Increment
Increment between load cases in the pattern.
(default is 1)

Set flag
Running flag.
= 0: Deactivate deactivate load case
= 1: Activate activate load case
(default is deactivate)

Example: `LCSET, 1, 5, 2, 0,`
This command deactivates load cases 1, 3 and 5. These particular load

cases will be ignored by the R_STATIC command.

LCLIST

Analysis > STATIC > List Load Case

The LCLIST command lists the type, number and run flag for all defined load cases.

Notes

- The lctype heading refers to the load case type: P refers to primary and S refers to secondary. Secondary load cases are defined by combinations of other load cases using the LCCOMB (Results, Combine Load Cases)
- In the run heading,1 indicates that the load case is set for running (active). 0 indicates that the load case is not set for running (nonactive).

ADAPTIVE

Analysis > STATIC > Adaptive Method

The ADAPTIVE command specifies parameters for adaptive meshing. The parameters of this command are used by the R_STATIC command to progressively improve the mesh until a desired accuracy level is reached. The improvement is accomplished by either refining the mesh (H-METHOD), or increasing the polynomial order (P-METHOD). The HP-METHOD refines the mesh first and then increases the polynomial order.

Mesh improvement is based on error estimates using stress results. The highest errors are usually associated with elements at locations of stress concentration and elements with high aspect ratios.

Four options are available:

1. The H-METHOD: Elements with high relative error are progressively subdivided until calculated average error becomes less than the specified error level. The error is based on the energy norm. This method is currently available for the 3- and 6-node TRIANG, SHELL3 and SHELL3T elements and the TETRA4, TETRA4R, and TETRA10 elements.
2. The P-METHOD: **(one selected order)** A polynomial order is specified and used to calculate displacements and stresses without any iterations. No new nodes or elements are created. This method is currently available for second order TRIANG, PLANE2D, SHELL9L (9-node option only) and TETRA10 elements. This method is a special case of the adaptive P-METHOD described below.
3. The HP-METHOD: This method combines the H- and the P-methods. The H-METHOD is progressively used until the error criterion is satisfied or the maximum number of loops is reached. After that the P-METHOD is used to

solve the refined mesh using the specified polynomial order. The method is applicable to the TRIANG and TETRA10 elements only.

4. The P-METHOD: **(adaptive)** The polynomial order for each element side is increased selectively based on errors in resultant displacements or errors in von Mises stresses. The local order is increased until convergence is achieved, the maximum number of loops, or the highest polynomial order is reached. This method is currently available for 6-node TRIANG, 8-node PLANE2D, and TETRA10 elements.

Adaptive method

Flag to define the method used in the adaptive meshing.

= 0: Off	deactivate the adaptive utility
= 1: H	H-METHOD
= 2: P	P-METHOD (one specified order or selective adaptive)
= 3: HP	HP-METHOD
	(default is P)

Maximum adaptive loops

Maximum number of loops.

Minimum polynomial order

Minimum order of the polynomial to be used for adaptive P-method.

Error tolerance

Maximum allowable error percentage. The definition of the error depends on the method: For H-method, the error is calculated by averaging the energy norm. For adaptive P-method, the error is calculated for each midside as the relative change in the resultant displacement (or von Mises stresses) between two consecutive runs.

Maximum polynomial order

Maximum order of the polynomial to be used for adaptive P-METHOD. If maximum polynomial order = minimum polynomial order, the specified P order will be used. (Limit is 10 for TRIANG and PLANE2D elements, and 5 for SHELL9L and TETRA10 elements.)
(default is minimum polynomial order)

Restart flag

Errors, number of DOF, displacement, total strain energy and stress results are stored in a file for convergence plots (refer to the ACTXYPOST command for linear static analysis). The restart option gives the user a chance to append the results or initialize the file.

= 1: Yes	append
= 0: No	new start

Criterion flag

Error calculation criteria.

= 0: Disp	displacement
= 1: Von str	von Mises stress
= 2: stn eng	strain energy
	(default is strain energy)

Element's strain energy error %

Element's strain energy error tolerance percentage, used if error calculation criteria for P-ADAPTIVE method is set to 2. If the strain energy convergence criterion is not met, then the p-orders of an element with strain energy error greater than this tolerance will be increased.

(default is 2.0)

Notes

- The following table illustrates the use of different arguments for various options and their default values:

Options	H-METHOD	P-ADAPTIVE	HP-METHOD
Maximum adaptive loops	Max. number of loops (1 to 10). Default is 4.	Max. number of loops (1 to 10). Default is 1. Not used if min. polynomial order = max. polynomial order.	Max. number of h-loops (1 to 10). Default is 4.
Minimum polynomial order	Not used.	Min. polynomial order (1 to 10). Default is 2.	Polynomial order. Default is 3.
Error tolerance	Allowable error percentage. Default is 2.	Allowable error percentage. Default is 2. Not used if min. polynomial order = max. polynomial order.	Allowable error percentage. Default is 2.
Maximum polynomial order	Not used.	Max. polynomial order (1 to 10). Default is min. polynomial order.	Not used.
Restart flag	No for new start, Yes to continue.	No for new start, Yes to continue. Default is 1.	No for new start, Yes to continue. Default is 1.
Criterion flag	Not used.	Error criterion: disp: displacement, von str: von Mises stress Not used if min. polynomial order = max. polynomial order. Default is von str.	Not used.

- The error estimation is based on the principle of stress continuity. The stress field resulting from finite element analysis, is usually discontinuous. Nodal stresses for each element are averaged to smoothen the discontinuous element stresses. The stress error in each element is defined as the difference between the element nodal stress and the average nodal stress corrected using the shape functions. This error is used to calculate the error in the energy norm for each element. The average error is calculated by simple averaging of element errors. Elements with errors higher than the average error are refined (or the order of the polynomial is raised) to improve the mesh. For detailed information on the error estimation procedure, refer to the following papers:
- "*A Simple Error Estimator and Adaptive Procedure for Practical Engineering Analysis*" by O.C. Zienkiewicz and J. Z. Zhu, International Journal for Numerical Methods in Engineering, vol. 24, pp. 337-357 (1987).

- "*An Error Analysis and Mesh Adaptation Method for Shape Design of Structural Components*" by K. H. Chang and K.K. Choi, Computers & Structures, vol. 44, No. 6., pp. 1275-1289 (1992).
- Newly created nodes are automatically pushed to the geometrical boundary, which results in more accurate modeling as the mesh is improved.
- For each loop, the static analysis is repeated and errors are compared with the specified bound. GEOSTAR stops improving the mesh (by refinement for the H-METHOD, or increasing the polynomial order for some element sides for the P-ADAPTIVE method), whenever the error bound is satisfied, the maximum number of loops or the highest polynomial order is reached.
- The "maximum polynomial order" entry for method 2 should be greater than or equal to "minimum polynomial order". The one-run P-METHOD is used if the two orders are equal. Otherwise, the selective P-ADAPTIVE method will be used.
- The convergence error (which may be plotted using the ACTXYPOST and XYPLOT commands), is calculated by averaging energy error norm for all methods. No matter what error criterion is specified for the P-ADAPTIVE method (changes in iterative displacement or von Mises stresses), the convergence error is calculated as the average energy error norm so that meaningful comparisons can be made.
- If the ADAPTIVE procedure is repeated and the 'restart' option is activated, convergence plots will include all results of all previous adaptive runs. The mesh is either refined (H-METHOD), or the polynomial orders are increased (P-ADAPTIVE method) over the final order reached in previous runs before running the analysis.
- Concentrated forces applied at the mid-side nodes are ignored when the P-METHOD is used, an equivalent pressure may be applied. Concentrated forces may be applied to corner nodes only.
- If a pressure loading is used in conjunction with the H_METHOD or the HP-METHOD, then the pressure must be applied normal to the faces of the elements. Pressure applied in any other direction will not work properly with these methods.

PORDERLABS

Analysis > STATIC > P-Order Labels

After running a linear static analysis using the P-adaptive method, you may be interested in knowing the polynomial order used for different sides of elements. The PORDERLABS command activates or deactivates a flag that instructs the

NPLOT (Edit, Plot, Nodes) command to plot the polynomial order used in the P-adaptive analysis instead of regular node labels. If the flag is activated, the NPLOT command plots the character 'P' followed by the polynomial order near the locations of middle nodes. The command must be issued again to deactivate this flag in order for the NPLOT command to resume its regular function of plotting node and their labels. Plotting of node labels must be activated before the NPLOT is issued. Node label plots may be activated from the STATUS1 Table, or using the "ACTNUM, ND, 1" command.

Adaptive p-order plot flag

Flag for plotting polynomial orders.

= 0: Deactivate	deactivate plotting of polynomial orders
= 1: Activate	activate plotting of polynomial orders
	(default is activate if not active, deactivate if active)

Example:

```
PORDERLABS, 1
ACTNUM, ND, 1
NPLOT;
```

The first command activates the adaptive p-order plot flag. The second command activates plotting of node labels. The third command plots polynomial orders used in the last P-adaptive analysis. Orders are plotted at locations of middle nodes in the pattern specified in the NPLOT command.

A_STATIC

Analysis > STATIC > Static Analysis Options

The A_STATIC command sets analysis options for linear static analysis using the STAR module. Note that the A_FFESTATIC command specifies analysis options for linear static analysis using the FFE Static module. The most recently issued command out of the two commands (A_STATIC and A_FFESTATIC) determines whether the R_STATIC command will run STAR or FFE Static. The default is to run STAR.

Loading flag

Special loading flag. Any one character can be assigned. Two or three characters can be assigned in any combination of C, G and T.

= N	do not include special loading
= C	include centrifugal loading
= G	include gravity loading
= T	include thermal loading
= CG	include centrifugal and gravity loadings
= CT	include centrifugal and thermal loadings
= GT	include gravity and thermal loadings
= CGT	include centrifugal, gravity, and thermal loading

(default is N)

Inplane effect

Flag to include in-plane stiffness effects.

= 0: No	no in-plane effects considered
= 1: Yes	in-plane effects considered
	(default is no)

Soft spring

Flag for soft spring addition.

= 0: No	no soft spring added
= 1: Yes	soft spring added
	(default is no)

Soft spring value

Soft spring constant.
(default is 1.0E-06)

Bcs stiffness value

Value of spring constant to impose boundary conditions in local coordinates.
(default is 1.0E10)

Save stiff matrix

Flag to save decomposed stiffness matrix.

= 0: No	do not save
= 1: Yes	save
	(default is no)

Form stiff matrix

Flag to specify stiffness matrix formation.

= 1: Form	form stiffness matrix
= 0: Don't form	use saved decomposed stiffness matrix from a previous run
	(default is form stiffness matrix)

Update coordinates

Flag to update coordinates to deformed configuration.

= 0: No	do not update coordinates
= 1: Yes	update coordinates
	(default is no)

End moment for shell elements

Fixed end moments flag for shell and elbow elements.

= 0: Off	do not apply fixed end moments
= 1: On	apply fixed end moments
	(default is off)

Grid force balance

Grid force balance calculation flag.

= 0: Off	do not calculate grid forces
= 1: On	calculate grid forces
	(default is off)

Inertia-relief

Flag to balance the external forces with inertia forces for accelerating objects.

= 0	do not apply inertia forces
= 1	apply inertia forces to balance external forces
	(default is 0)

Rigid connections flag

This flag controls the continuity between solid and shell elements connected to each other. Solid elements like TETRA4, TETRA10, and SOLID do not consider rotational degrees of freedom (DOF) in their formulation. Rotations in these solid elements can be expressed in terms of the translational DOF. Shell elements on the other hand consider rotational DOF explicitly in their formulation.

Traditionally, you need to introduce some coupling constraints when connecting such incompatible elements. This flag, when set to rigid, will take care of this condition automatically. The default option (hinge) assumes a hinged connection between solid and the shell elements. The model must be stabilized by applying sufficient boundary conditions.

= 1: Rigid	Assume rigid connections between all solid and shell elements.
= 0: Hinge	Assume hinged connections between all solid and shell elements.
	(default is hinge)

Solver option

= 0: Sparse	Use the sparse direct solver.
= 1: Skyline	Use the skyline direct solver.
= 2: Iterative(PCG)	Use the PCG iterative solver.
	(default is Sparse)

Solution accuracy

By activating this flag, the analysis program calculates the following quantities for the direct solvers (for each load case)

$$\|[A]\{X\}-\{F\}\| \text{ and } \frac{\|[A]\{X\}-\{F\}\|}{\|\{F\}\|}$$

which correspond to “absolute error solution” and “normalized solution error”, respectively, where:

[A]- assembled stiffness matrix
{F}- assembled load vector
{X}- displacement vector

The above quantities are helpful in detecting wrong solution in cases such as ill-conditioned matrices or structural instabilities.

= 0: Off	Deactivate.
= 1: On	Activate
	(default is Off)

Change to 2nd Order

This flag controls the formulation of first-order elements (i.e. elements without mid-side nodes). When the flag is activated, the formulation of first-order will be changed to second-order formulation based on mid-side nodes along straight edges. Existing second-order elements (i.e. elements with existing mid-side nodes) will be used as is. This command cannot be used to change the formulation of second-order elements to first order.

= 0: off Do not use second-order formulation for first-order elements
= 1: on Use second-order formulation for first-order elements
(Default is off)

Initial Contact Clearance

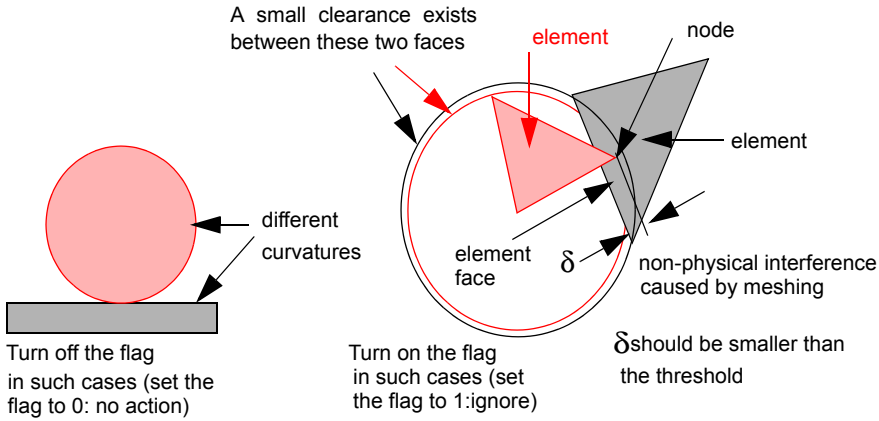
Solving contact problems is an iterative process that starts with an initial deformation guess. Starting with a good initial guess helps the iterative process converge to the correct solution quickly. The status of this option affects the initial guess.

✍ This flag is used with surface to contact only.

When the flag is turned off (0: no action), initial contact between a node and the associated face is assumed if the initial distance is less than or equal to a certain threshold (0.5% of the global element size). If the initial distance exceeds the threshold, no initial contact is assumed.
If the flag is turned on (1: ignore), initial contact (between each node and its associated face) is assumed regardless of the initial distance between them. The initial gaps will be ignored during contact iterations.
This flag should be turned off (default) for most applications. It is specifically designed to take care of a specific situation when, due to tolerances, a small clearance exists between faces that should have been initially in contact.
Due to this clearance, the mesh on the two faces, set for contact, may not be compatible and the elements on both sides may interfere with each other as dramatized in the figure below. The interference in this case is non-physical (introduced by approximating the geometry by the mesh. The flag should be turned on in such cases.

✍ The flag should not be turned on when the two faces set for surface contact have large curvature differences.

= 0: no action do not ignore initial clearance for surface contact
= 1: ignore ignore initial clearance for surface contact
(Default is no action)



Notes

- If you select to save the decomposed stiffness matrix (using the save stiff matrix option), you can instruct the program to read it, instead of recalculating it, for subsequent static, frequency, or buckling. This option can reduce the solution time considerably for large problems.
- The "inplane effect" flag cannot be activated in the presence of GAP elements in the model.
- If the grid force balance flag is activated, the program will calculate the internal nodal forces for each element. The summation of these nodal forces at a node should balance the applied loads at that node. The reaction forces at a node may be calculated as the unbalanced forces resulting from the summation of all internal and applied forces. The grid forces and reactions are printed in the output (problem-name.OUT) file. Use the REACTION command to request reactions before running the analysis and use the DISLIST command to list them after running the analysis.

To run static analysis using STAR:

1 Use the **A_STATIC** command to set the desired options.

2 Click **Analysis, Static, Run Static Analysis**.

To run static analysis using FFE:

1 Use the **A_FFESTATIC** command to set the desired options.

2 Click **Analysis, Static, Run Static Analysis**.

Example:
```
A_STATIC, CGT, 1,,,,,,,
R_STATIC
```
The first command causes the second command to consider centrifugal, gravity and thermal loadings and to include inplane stiffness effects. Centrifugal loading, gravity, and thermal loading are considered in all load cases with specified input for such loadings (e.g. NTND and coefficient of thermal expansion for thermal, ACEL and mass density for gravity, and OMEGA and mass density for centrifugal).

A_FFESTATIC

Analysis > STATIC > FFE Static Options

The A_FFESTATIC command specifies analysis options for linear static analysis using the FFE Static module. Note that the A_STATIC command specifies analysis options for linear static analysis using the STAR module. The most recently issued command out of the two commands (A_STATIC and A_FFESTATIC) determines whether the R_STATIC command will run STAR or FFE Static. The default is to run STAR.

Element order

Order of the element to be used. In spite of the element group name in the database, you may specify through this option whether first (linear) or second (parabolic) elements will be used. As an example, if you define TETRA4 elements and use second order, middle nodes on straight edges will be considered during analysis. On the other hand you may define TETRA10 elements and specify to use first order. TRIANG and PLANE2D elements are treated similarly except that for these elements the same element group names are used for both first and second orders.

= 1: First	use first order for continuum elements
= 2: Second	use second order for continuum elements
	(default is second)

Loading flag

Special loading flag. Any one character can be assigned. Two or three characters can be assigned in any combination of C, G and T.

= N	do not include special loading
= C	include centrifugal loading
= G	include gravity loading
= T	include thermal loading
= CG	include centrifugal and gravity loading
= CT	include centrifugal and thermal loading
= GT	include gravity and thermal loading
= CGT	include centrifugal, gravity, and thermal loading
	(default is N)

Rigid connections flag

This flag controls the continuity between solid and shell or beam elements connected to each other. Solid elements like TETRA4, TETRA10, and SOLID do not have explicit rotational degrees of freedom (DOF). Rotations of solid elements can be expressed in terms of the translational DOF. Beam and shell elements on the other hand have explicit rotational DOF.

Traditionally, you need to introduce some coupling constraints when connecting such incompatible elements to ensure continuity. This flag, when active, takes care of this condition automatically and rigid connections between all such incompatible elements in the model are assumed. When you want to specify hinge connections or you need to compare COSMOS/FFE results to results from traditional finite element systems which assume hinge connections between solid and shell or beam elements, you must turn this flag off before running the analysis.

= 1: Yes	activate rigid connections
= 0: No	deactivate rigid connections *(default is yes)*

A_ASYM

Analysis > STATIC > Asymmetric Load Options

The A_ASYM command specifies asymmetric loading for the linear static analysis of axisymmetric SHELLAX and PLANE2D elements. The command can also be used to specify asymmetric mode calculations for frequency and buckling analyses for SHELLAX and PLANE2D elements. Use the “A_LIST, STATIC” command to check active asymmetric loading options.




Asymmetric loading

Asymmetric loading flag. Activates or deactivates the asymmetric loading option for linear static, frequency, and buckling analyses.

= 0: No	deactivate asymmetric loading
= 1: Yes	activate asymmetric loading
	(default is yes)

Harmonic type

Type of harmonic terms to be used for the asymmetric loading (static analysis only).

= 0: Gen	consider both sine and cosine terms
= 1: Sin	consider sine terms only
= 2: Cos	consider cosine terms only
	(default is gen)

Beginning harmonic number

Beginning harmonic number for static, frequency, and buckling analyses.
(default is 0)

Ending harmonic number

Ending harmonic number for static, frequency, and buckling analyses.
(default is beginning harmonic number)

Starting angle of output

Beginning angle in degrees for printing and plotting the results of the analysis.
(default is 0)

Ending angle of output

Ending angle in degrees for printing and plotting the results of the analysis.
(default is 360)

Increment

Increment angle in degrees for printing and plotting the results of the analysis.
(default is 90)

Notes

- The “harmonic type” entry is used only in the linear static analysis and is not applicable for frequency and buckling analyses.
- For frequency analysis, the “beginning harmonic number” entry specifies the harmonic number for which frequencies and mode shapes are to be calculated. The number of modes is specified in the “A_FREQUENCY” command.

A_STRESS

Analysis > STATIC > Stress Analysis Options

The A_STRESS command sets options for running stress analysis to be performed by the R_STRESS or the R_STATIC commands.

AISC code check

Flag to perform AISC stress check.

= 0: No	no AISC code check.
= 1: Yes	writes beam element nodal forces into a file to be used later by the AISC code check module. The file extension is "AIS". *(default is no)*

Stress print number

Flag for stress printing.

= -1	no stresses will be printed
= 0	stresses printed for all elements
= N	stresses printed for N elements with highest stresses. N is limited to 2000 *(default is -1)*

Princ. stress print

Flag for principal stress printout.

= 0: No	no principal stresses printed
= 1: Yes	principal stresses printed *(default is no)*

Strain print

Flag for strain printout.

= 0: No	no strains printed
= 1: Yes	strains printed *(default is no)*

Layer number (post dynamic xy-plots)

Layer number for stress xy-plot in multilayer shell or solid elements in ASTAR. Limited to 50. Used in ASTAR only.
(default is 1)

Type of failure analysis

Flag for failure analysis of composite shell elements.

= 0: None	do not perform failure analysis
= 1: Tsai-Wu criterion	perform failure analysis using Tsai-Wu criterion
= 2: Hill criterion	perform failure analysis using Hill criterion. Refer to the COSMOS/M Basic System Manual *(default is none)*

Shell face (post dynamic xy-plots)

Flag to specify face of shell and SOLIDL elements for which xy-plots will be generated in ASTAR. Refer to the User Guide, Chapter 4 for definition of top and bottom faces. Used in ASTAR only.

= 0: Top	top face
= 1: Bottom	bottom *(default is top)*

ASME (ASME code check flag)

Flag for ASME code stress check consideration.

= 0: No	do not consider the ASME code check
= 1: Yes	consider the ASME code check *(default is no)*

J-integral calculation flag

= 0: No — do not calculate J-integral
= 1: Yes — Calculate J-integral
(default is No)

Stress printout at Gauss points flag

Flag for stress printout at Gauss points.
= 0: No — do not print stresses at Gauss points
= 1: Yes — print stresses at Gauss points
(default is No)

Note

Stresses at Gauss points are printed in an ASCII file with extension ".GUS". These stresses are only available for the SOLID, PLANE2D, TETRA4, and TETRA10 elements. The results are written consequently for each element, starting with a header row consisting of the following integer entries after the character "E":

E, N, NGUS, ETYP

where N is the element label, NGUS is the number of Gauss points, and ETYP indicates the element type.

ETYP can have the following values:

21, for TETRA4 elements
22, for TETRA10 elements
6, for SOLID elements
8, for PLANE2D elements

The character 'E', at the beginning of the row, indicates that this is a header row. In the following rows, the actual Gauss points stresses are written consequently for each Gauss points in the following form:

G, S1, S2, ..., Sm, Svon, P1, P2, P3

where:

character 'G' corresponds to the rows containing the stress values, S1, S2, ..., Sm corresponds to the stress components, sx, sy, txy, sz, svon, sr1, sr2, q for PLANE2D elements, and sx, sy, sz, txy, txz, tyz, svon, sr1, sr2, sr3 for 3D elements. Svon corresponds to the von Mises element stresses and sr1, sr2 and sr3 correspond to the principal stresses at Gauss points.

'q' is the orientation angle for principal stresses. The stress components are in Global Cartesian coordinates and are available for static analysis as well as the time history stress calculations.

Use the following FORTRAN formats to read and process the information described above:

FORMAT (1x, 'E', 1x, I6, 2(1x, I3)) for specification row, and
FORMAT (1x, 'G', 8(1x, E10.4)) for 2D stress results, and
FORMAT (1x, 'G', 10(1x, E10.4)) for 3D stress results.

Example:
```
A_STRESS,,,1,,
R_STRESS
```
The first command causes the second command to calculate and write principal stresses in the output file.

PCG_Options

Analysis > STATIC > PCG Option

The PCG_OPTION command specifies options for the iterative PCG (Preconditioned Conjugate Gradient) static analysis.

Convergence tolerance
Convergence tolerance for the PCG static analyst
(default is 1.0e-4, minimum input is 1.0e-24)

Mode
Mode for the PCG solver

0: Automatic	automatic
1: General method	general method
2: Tet memory saver	Tetrahedral elements memory saver
3: Tet ultra-memory saver	Tetrahedral elements ultra-memory saver

(default is Automatic)

Out-of-core solution
- Out-of-core solution option for the PCG solver

0: No	use in-core solution option
1: Yes	use out-of-core solution option

(default is 0)

Iteration cut factor
- Iteration cut factor for the PCG solver
(default is 0.0, input less than 1.0e-19 will be ignored and treated as 0.0 for no cut)

Zero pivot
- Zero pivot for the PCG solver
(default is 1.0e-9, minimum is 1.0e-29)

Notes:

- You need to issue this command only if you prefer to change default settings.
- The parameters specified in this command are used in the analysis program only when the program is using the iterative (PCG) solver. To activate the iterative solver, use the A_STATIC (Analysis, Static, Static Analysis Options) command.
- For problems with contact (gap elements), the solver takes a default value of 1.e-6 for the *Convergence Tolerance*. Any specified value smaller than 1.e-6 will be ignored by the solver.

- You can use three modes of solution with the iterative solver:
  - The general mode is used with all types of elements supported by STAR (i.e. Shells, Solids, Tetra, Beams, etc.).
  - Mode 2 (Tet memory saver) and mode 3 (Tet ultra-memory saver) are used to efficiently solve very large ten-noded structural tetrahedral models. However, they can handle a small fraction of other element types as multi-point constraints (up to %30 of total number of elements) efficiently. In mode 2 you gain more than %25 saving on the memory (with a small loss in speed) over the general mode. In mode 3, you gain an additional %25 saving in memory over mode 2.
  - In the default *Automatic* mode (or when this command is not issued), the solver uses mode 2 unless the number of non-tetrahedral (10-noded) elements exceeds %30 of the total elements in the model.
  - *Iteration Cut Factor* (ICF) is used to limit the number of iterations the PCG solver can take. The solver ends iterations if the number of Iterations exceeds: ICF*NUMNOD*nDofPerNode, where, NUMNOD is the number of nodes in the model and NDofPerNode is the maximum number of d.o.f. per node (3 for regular tets and 6 for shells, beams, and tets with rotations).
- *Zero Pivot* is a threshold for detecting zero pivots while factoring the *Preconditioner*. If the pivot value is less than the specified value, the solver considers the problem to be unconstrained and returns with an error message. For problems with contact (gap elements), the program takes a default value of 1.e-12 for the zero pivot value. Any specified value smaller than 1.e-12 will be ignored by the solver for contact.
- Use A_LIST command to list the active PCG_OPTION parameters. The iterative solver option in the A_STATIC should be active in order to list these parameters.




STRESS

Analysis > STATIC > Activate Stress Calc

The STRESS command sets the flag for stress calculations. The flag is used by the R_STATIC command.

Stress flag
Stress calculations flag.
= 1: On calculate stresses
= 0: Off do not calculate stresses
(default is on)

Example:
```
STRESS, 0,
R_STATIC,
```
Stresses will not be calculated by the R_STATIC command since the stress flag is turned off. If this command is not issued, stresses will be calculated by default.

SUBMODEL

Analysis > STATIC > Define SubModel

The SUBMODEL command activates or deactivates the submodeling option. Submodeling is used to improve the results in stress-concentration localities after an initial run with a relatively coarse mesh. When submodeling is activated, elements in the active element selection list will be refined and will be designated as submodel elements. When analysis is performed with submodeling active, only elements in the submodel will be considered. Commands in the CONTROL > SELECTION menu such as SELWIN, SELRANGE and SELREF can be conveniently used prior to this command to select the submodel in the stress-concentration areas. The submodel can consist of several groups of disconnected localities. Submodeling is available only for use with linear static analysis using STAR.

An alternative to submodeling is to refine elements in the selected areas and rerun static analysis. This run can take a long time for large models because the whole model is analyzed again. Submodeling saves you time because it accepts results from the initial run, with coarse mesh, for less important areas. Only the selected areas of the model will be refined and analyzed, a much smaller problem size.




Push boundary node flag

A flag to push new generated nodes to the geometric boundary of the model.
= 0: No do not push nodes to the boundary
= 1: Yes push nodes to the boundary
(default is no)

To use submodeling:

1. Run regular static analysis.
2. Use commands in the *Control, Select* menu to select elements in area(s) of stress concentration. Every selected element must belong to a 3-node TRIANG, SHELL3, SHELL3T, TETRA4 or TETRA4R element group.
3. Click **Analysis, Static, Define Submodel**.
4. From the **Submodel** option drop-down menu, select **1:on**.
5. From the **Push boundary node flag**, select **1:Yes**. You do not need to select this option if no curved boundaries exist in the model or none of the selected elements is on the boundary of the model.
6. Click **OK**.
7. Click **Analysis, Static, Run** to calculate displacements, strains and stresses. The program automatically improves the results for the selected elements (submodel). If the number of selected elements is small, this run should be quick.
8. Visualize the results of the submodel as you do for regular static analysis. To visualize results of the whole model, use the **Status3** table to turn off the active element selection list.

Some new elements may appear outside the boundary of elements in the original selection list since the new submodel elements are automatically added to the original selection list upon completing the analysis for convenient post-processing.

Notes

- Submodeling works with the 3-node TRIANG, SHELL3, SHELL3T, TETRA4 and TETRA4R elements only. Other types of elements can be used outside of the submodel.
- You can deactivate submodeling at any time and run the analysis (R_STATIC) for the whole model.
- You can repeat the submodeling process as desired until sufficient accuracy is achieved.

- You need not be concerned about the nodal forces or element pressures within the submodel. The refinement of the mesh is accomplished by breaking the elements while preserving the original node locations. However, you may need to define nodal temperatures for the new nodes generated by the refinement process.
- You can use submodeling with multiple load cases.

Examples:

Example 1:
```
A_STATIC
R_STATIC
SELWIN          (select elements with stress concentration)
SELWIN          (select elements with stress concentration)
SUBMODEL, 1;
R_STATIC
```
After running the original problem, two separate neighborhoods with high stress concentration are selected by the SELWIN command for use with submodeling. The SUBMODEL command activates submodeling. R_STATIC runs STAR to improve results of the submodel.

Example 2:
```
A_STATIC
R_STATIC
SELREF          (select elements by reference to geometric entities)
SUBMODEL, 1;
R_STATIC
INITSEL;
```
After running the submodel, the element selection list is initialized and postprocessing is performed on the whole model.

R_STATIC

Analysis > STATIC > Run Static Analysis

The R_STATIC command performs linear STATIC analysis. The command runs FFE Static if the A_STATIC command has been issued and was not followed by the A_STATIC command or runs STAR if the A_FFESTATIC command has not been issued or was issued but followed by the A_STATIC command. Upon a successful run, the command automatically calculates strains and stresses in addition to displacements unless the STAR module was used and the STRESS command has been used to turn off stress calculations in which case the R_STRESS command may be used later to calculate stresses.

To run static analysis using FFE Static:

1. Create the model.
2. Optional: click **Analysis, Run Check** to check input data.
3. Click **Analysis, Static, FFE Static Options** to set the desired options.

4 Click **Analysis, Static, Run Static Analysis**.

If the run is not successful, a clear message will be given. Refer to Appendix A of the COSMOS/M FFE Static manual for explaining and fixing the problem. The message is also written to the output file (.OUT).

To run static analysis using STAR:

1 Create the model.

2 Optional: click **Analysis, Run Check** to check input data.

3 Click **Analysis, Static, Static Analysis Options** to set the desired options. If you choose the PCG solver, you can change default settings using the PCG_Options (**Analysis, STATIC, PCG Options**) command.

This step is optional unless you have already clicked the **Analysis, Static, FFE Static Options** command.

4 Click **Analysis, Static, Run Static Analysis**.

If the run is not successful, a clear message will be given. Refer to Appendix D of the Basic System User's Guide Part I for explaining and fixing the problem. The message is also written to the output file (.OUT).

Notes

- The command will calculate displacements and stresses for all load cases set to run. After a successful run, you can use the LCCOMB (**Results, Combine, Load Cases**) command to define secondary load cases.
- If adaptive analysis is active, the analysis will be repeated as specified by the ADAPTIVE (**Analysis, Static, Adaptive Method**) command if STAR is used.

R_STRESS

Analysis > STATIC > Run Stress Analysis

The R_STRESS command performs stress analysis. It calculates stresses for linear problems only, using the "STRESS" module of the COSMOS/M package. Stresses are computed using nodal displacements data, previously generated by the R_STATIC command.

Notes

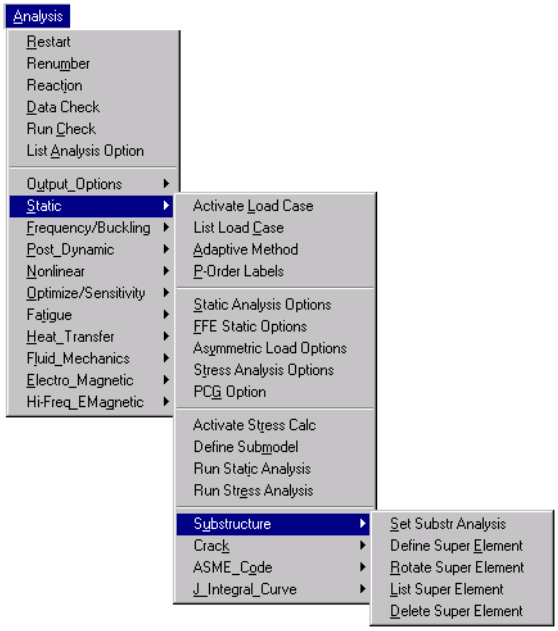
- The R_STRESS command uses flags specified by the A_STRESS command.
- Stresses are computed for all primary load cases that are set to run.




▼SUBSTRUCTURE Menu

Analysis > STATIC > SUBSTRUCTURE

Figure 10-4 Substructure Menu



Substructuring lets you divide a large model into a number of smaller models to improve the efficiency of the solution.

SPSTR

Analysis > STATIC > SUBSTRUCTURE > Set Substr Analysis

The SPSTR command can be used to define the substructure analysis option, create or assemble super elements, and recover displacements and stresses in super elements.

Analysis type

Flag to specify type of analysis.

= 0: No substruct	no substructuring considered
= 1: Create substr super elem	create substructure super element

= 2: Assemble super elem	assemble super elements attached to the main structure only
= 3: Displacement recover	displacement recovery in the substructure *(default is no substruct)*

Super element
Super element number. (not valid for no substruct)

Notes

- The substructuring option involves super element generation, assembly and static analysis runs, and displacement and stress recovery within the super elements.
- For each super element, the condensed stiffness matrix and force vector will be stored in files TMP.S?? and TMP.F??. The question marks refer to the super element number. For example, files TMP.S03 and TMP.F03 are created with the generation of super element number 3.
- Substructures, created by different computers, can be transferred via TMP files to the computer which performs the assembly.
- When several super elements have the same geometry and force vector, only generate one and then copy its TMP files to create the others.
- It is recommended you issue the STRESS command to deactivate stress calculation when forming the condensed stiffness matrix. Otherwise, you should ignore the “no displacement available” message.

Example: `SPSTR, 1, 1`
This command instructs substructure analysis to create super element number 1.

SPELE

Analysis > STATIC > SUBSTRUCTURE > Define Super Element

The SPELE command defines super element connectivity. Up to 20 super nodes can be defined. The command can be repeated to define more super nodes for the same element.

Super element
Super element number. (default is maximum defined + 1)

Starting location in the node set.
Starting location in the node set.
(default is maximum previously defined number of super nodes for the super element + 1)

Number of nodes to be entered
Number of super nodes to be defined by this command. (limit is 20)




Node (i)
Label of the ith node of the super element. (i=1,2,...,20)

Example:
```
SPELE, 1, 1, 5, 10, 12, 14, 15, 20
SPELE, 1, 4, 1, 16
```
The first command defines nodes 10, 12, 14, 15 and 16 to be the first through the fifth super nodes of super element number 1, respectively. The second command modifies the fourth super node to be node 16 instead of node 15.

SPROT

Analysis > STATIC > SUBSTRUCTURE > Rotate Super Element

The SPROT command rotates an existing super element about the global Cartesian coordinate axes by the specified angle(s).

Super element
Super element to be rotated.

x- rotation
Angle of rotation about the x-axis in degrees.

y- rotation
Angle of rotation about the y-axis.

z- rotation
Angle of rotation about the z-axis.
(default for the angles is 0.0)

Rotation flag
Flag to specify degrees of freedom (DOF) associated with super nodes.

= 0: For 3D	for 3D problems. DOF must be: a. UX, UY, UZ, or b. RX, RY, RZ, or c. UX, UY, UZ, RX, RY, RZ.
= 1: For 2D	for 2D problems. DOF must be: a. UX, UY, or b. UX, UZ, or c. UY, UZ. *(default is for 3D)*

Notes

- The DOF at each super node of the super element being rotated should match one of the six options shown above. Other combinations will cause erroneous results.
- This command can be used to create a new super element by rotating an existing one. The TMP files containing the condensed stiffness matrix and force vector of the existing super element must be copied into other files before this transformation is performed to generate the new super element.

Example: `SPROT,2,,,90.0, 1`

This command rotates super element number 2 by 90.0 degrees about the z-axis. (DOF must be: UX, UY or UX, UZ or UY, UZ).

SPLIST

Analysis > STATIC > SUBSTRUCTURE > List Super Element

The SPLIST command lists a pattern of super elements in terms of their associated super nodes.

Beginning super element
Beginning super element in the pattern.
(default is 1)

Ending super element
Ending super element in the pattern.
(default is max. super element defined)

Increment
Increment between super elements in the pattern.
(default is 1)

Example: `SPLIST, 7, 12, 2`

This command lists super nodes associated with super element numbers 7, 9 and 11.

SPDEL

Analysis > STATIC > SUBSTRUCTURE > Delete Super Element

The SPDEL command deletes a pattern of previously defined super elements.

Beginning super element
Beginning super element in the pattern.

Ending super element
Ending super element in the pattern.
(default is beginning super element)

Increment
Increment between super elements in the pattern.
(default is 1)

Example: `SPDEL, 7, 12, 2`

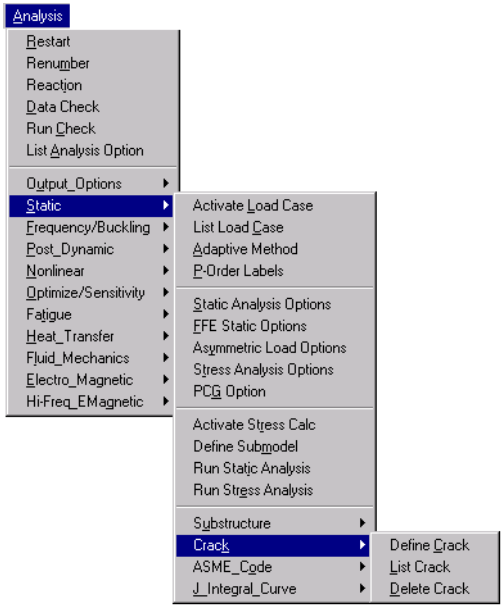
This command deletes super elements 7, 9 and 11.

▼CRACK Menu

Analysis > STATIC > CRACK

This menu contains commands the following commands to define, delete, and list cracks.

Figure 10-5 Crack Menu



CRACK

Analysis > STATIC > CRACK > Define Crack

The CRACK command defines a crack set by specifying the corresponding 3 nodes. For 2D models, cracks may only be used with the plane stress or plane strain options in the PLANE2D element. For 3D models, cracks can only be used with 20-node SOLID elements. The first two nodes must be corner nodes.

Crack number

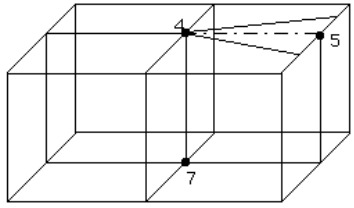
Crack set number. Limit is 15.
(default is the highest set number defined + 1)

Node 1

Node at the crack tip.

Node 2
Node to define the crack direction.

Node 3
Node to define the direction of the ridge of the crack.



Note
Only two nodes are needed for 2D crack.

Example: `CRACK, 2, 4, 5, 7`

This command defines crack set 2, starting at node 4, with the crack the directed along the line connecting node 4 to node 5. The ridge of the crack runs from node 4 to node 7.

CRACKLIST

Analysis > STATIC > CRACK > List Crack

The CRACKLIST command lists a pattern of crack sets and the associated nodes.

Beginning crack
Beginning crack set in the pattern.
(default is 1)

Ending crack
Ending crack set in the pattern.
(default is 15)

Increment
Increment between crack sets in the pattern.
(default is 1)

Example: `CRACKLIST;`

This command lists all defined crack sets.

CRACKDEL

Analysis > STATIC > CRACK > Delete Crack

The CRACKDEL command deletes a pattern of crack sets.

Beginning crack
Beginning crack set in the pattern.
(default is 1)

Ending crack
Ending crack set in the pattern.
(default is beginning crack)

Increment
Increment between crack sets in the pattern.
(default is 1)

Example: `CRACKDEL, 2, 3, 1`

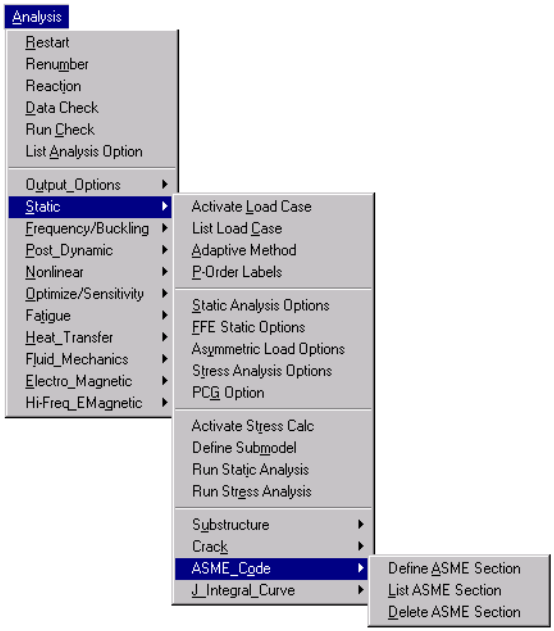
This command deletes crack sets 2 and 3.

▼ASME_CODE Menu

Analysis > STATIC > ASME_CODE

The ASME_Code menu contains commands to define, list and delete ASME section properties.

Figure 10-6 ASME_Code Menu



ASMESECDEF

Analysis > STATIC > ASME_CODE > Define ASME Section

The ASMESECDEF command specifies a section for which the ASME code section III will be used to compute the stress components, membrane and bending stresses, and peak stress intensity. The command can only be used with TRIANG and PLANE2D elements.

ASME section
Section label. Up to 300 sections may be defined.
(default is the highest section defined + 1)

First node
Node label at one end of the section.

Second node
Node label at the other end of the section.

Number of points
Number of integration points along the section.
(default in 25)

Radius of curvature
In-plane average radius of curvature of the inside and outside surfaces of the axi-symmetric section.

0	plane
> 0	axisym
-1	straight

Thickness direction bending stress flag
Thickness direction flag for bending stress flag. (prompted only if the radius of curvature is not 0)

= 1: Yes	consider thickness direction bending stress
= No	ignore
	(default is yes)

Notes

- Nodes can be picked by the mouse if they are plotted.
- A very large radius of curvature (i.e., a flat surface) can be specified by entering a value of "-1" for the curvature radius.
- For a Cartesian formulation, use zero for the radius (refer to the Basic System Manual).
- This option is only available for linear static and time history advanced dynamic analyses.
- All computations related to ASME sections are performed in the stress module. After running STAR, you can add or modify sections as desired and run stress using the R_STRESS command. You do not need to run STAR again.
- The nodes specifying the two ends of the section can be defined anywhere within the model and are not confined to the model mesh (i.e. they may or may not have been used in element connectivity).




ASMESECLIST

Analysis > STATIC > ASME_CODE > List ASME Section

The ASMESECLIST command lists a pattern of ASME sections and their related specifications on the screen.

Beginning ASME section
Beginning section in the pattern.
(default is 1)

Ending ASME section
Ending section in the pattern.
(default is highest defined section)

Increment
Increment between sections in the pattern.
(default is 1)

ASMESECDEL

Analysis > STATIC > ASME_CODE > Delete ASME Section

The ASMESECDEL command deletes a pattern of ASME sections from the database.

Beginning ASME section
Beginning section in the pattern.

Ending ASME section
Ending section in the pattern.
(default is beginning section)

Increment
Increment between sections in the pattern.
(default is 1)

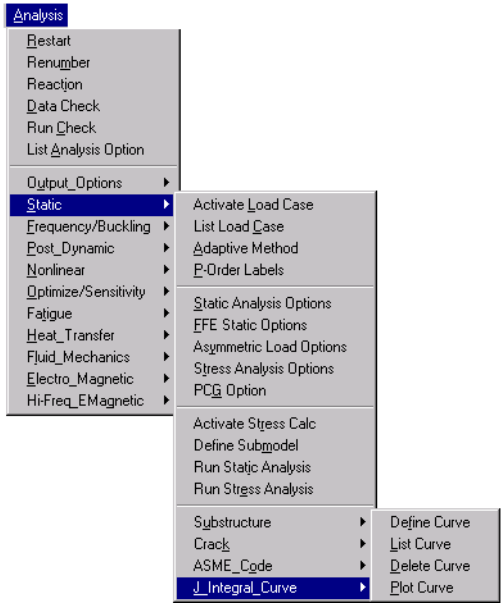



▼J_INTEGRAL CURVE_Menu

Analysis > STATIC > J_Integral_Curve

The J_integral computes the stress intensity factor due to cracks.

Figure 10-6 J_Integral Curve Menu



J_INTCRDEF

Analysis > STATIC > J_Integral_Curve > Define Curve

The J_INTCRDEF command defines a circular J-integral path for use with STAR to solve problems involving fracture mechanics. The J_integral computes the stress intensity factor for linear and parabolic PLANE2D and TRIANG elements. Three nodes are required to define the crack: the node at the tip of the crack, a node to define the radius of the path and its starting point, and a node to define the direction of the crack. Multiple paths can be defined for a crack.

You must activate the J-Integral flag in the A_STRESS command (Analysis > Static > Stress Analysis Options) prior to running stress analysis in order for the program to consider the defined J-Integrals.

J-Integral curve number

J-Integral curve number. Up to 10 curves (paths) can be defined.
(default is the highest curve number defined + 1)

Node at crack tip

Node at the tip of the crack. This node is also used as the center of the circular path.

Starting node of path

Node to define the start of the path. The distance between the crack-node and this node defines the radius of the circular path.

Node defining crack direction

Node to define the direction of the crack.

Path type

Flag to define the type of the path:
= 0: full circle the path is a full circle
= 1: half circle the path is a half circle

Number of integration points

Number of integration points on the path. A total of 3000 points can be used for all paths.
(default is 100)

To calculate stress concentration factors using J-Integrals:

1. Click **Analysis, STATIC, J_Integral_Curve, Define Curve** to define the desired path.
2. You can repeat the command to define more J_Integral paths.
3. For PLANE2D elements, you can define crack elements using the CRACK (**Analysis, Static, Crack, Define Crack**) command to improve over-all accuracy.
4. Click **Analysis, STATIC, Run Static Analysis** to run static analysis.
5. A stress concentration factor is written in the output file for each J_Inegral curve.

Notes

- The starting node should be consistent with the counter clockwise direction of the path and should be located on a crack face for full circular paths. Refer to the Basic System manual for exceptions to this rule when using half circular paths.
- The mesh on and inside the path does not have to be symmetric with respect to the crack axis.
- The number of integration points can be increased to improve the accuracy. Use more integration points when the path crosses areas with high stress gradients.
- Paths should not be defined too close to the tip of the crack to avoid excessive errors.

- The direction node should always lie on one of the crack edges.
- To properly define the rupture due to the crack, coinciding nodes on the two edges of the crack should not be merged.

J_INTCRLIST

Analysis > STATIC > J_Integral_Curve > List Curve

The J_INTCRLIST command lists a pattern of J-integral curves defined by the J_INTCRDEF (Analysis > Static > J_Integral Curve > Define Curve) command.

Beginning set
Beginning J-integral curve in the pattern.
(default is 1)

Ending set
Ending J-integral curve in the pattern.
(default is the highest curve defined)

Increment
Increment between J-integral curves in the pattern.
(default is 1)

J_INTCRDEL

Analysis > STATIC > J_Integral_Curve > Delete Curve

The J_INTCRDEL command deletes a pattern of J-integral curves defined by the J_INTCRDEF (Analysis > Static > J_Integral Curve > Define Curve) command.

Beginning set
Beginning J-integral curve in the pattern.
(default is 1)

Ending set
Ending J-integral curve in the pattern.
(default is the highest curve defined)

Increment
Increment between J-integral curves in the pattern.
(default is 1)

J_INTCRPLOT

Analysis > STATIC > J_Integral_Curve > Plot Curve

The J_INTCRPLOT command plots a pattern of J-integral curves defined by the J_INTCRDEF (Analysis > Static > J_Integral Curve > Define Curve) command.

Beginning set
Beginning J-integral curve in the pattern.
(default is 1)

Ending set
Ending J-integral curve in the pattern.
(default is the highest curve defined)

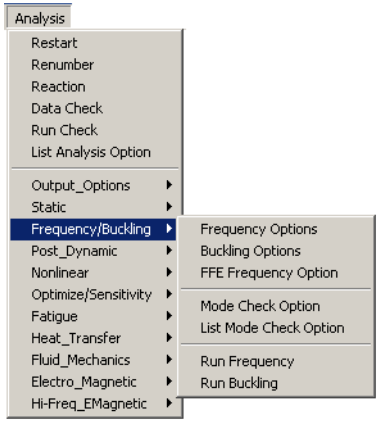
Increment
Increment between J-integral curves in the pattern.
(default is 1)

FREQUENCY/BUCKLING Menu

Analysis > FREQUENCY/BUCKLING

This menu contains commands related to DSTAR, the frequency and buckling analyses module. For frequency analysis, DSTAR calculates the natural frequencies and corresponding mode shapes; for buckling analysis, DSTAR calculates eigenvalues to be multiplied by the specified loads to realize buckling. Refer to the COSMOS/M Basic System User Guide for details.

Figure 10-7 Frequency/Buckling Menu



A_FREQUENCY

Analysis > FREQUENCY/BUCKLING > Frequency Options

The A_FREQUENCY command sets the options for subsequent frequency analysis performed by the R_FREQUENCY (**Analysis, Frequency/Buckling, Run Frequency Analysis**) command.

Number of frequencies

Number of frequencies to be calculated.
(default is 1)

Method

Frequency analysis method.

= S	subspace iteration method
= J	Jacobi method (limited to 50 DOF)
= L	Lanczos method
= C	complex eigenvalue analysis (limited to 50 DOF)
= I	inverse power method (for number of frequencies = 1 only)
= GS	subspace Guyan reduction method
= GJ	Jacobi Guyan reduction method
	(default is S)

Maximum number of iterations

Maximum number of iterations for subspace iterations and inverse power methods.
(default is 16)

Sturm sequence

Flag for Sturm sequence. (see note 1 below)

= No	no Sturm sequence check
= 1: Yes	apply Sturm sequence check
	(default is no)

Shift flag

Flag for eigenvalue shift. (see note 2 below)

= 0: No eig. val. shift	no eigenvalue shift
= 1: Shift by program	program applies eigenvalue shift
= 2: Shift by user	user applies eigenvalue shift
	(default is no eig. val. shift)

Shift value

Value of frequency shift in radians/time. Used only if shift flag = shift by user.
(default is 0.0)

Inplane effect

Flag to include in-plane stiffness effects.

= No	no in-plane effects considered
= 1: Yes	in-plane effects considered
	(default is no)

Tolerance

Convergence tolerance for eigenvalues.
(default is 1.0E-5)

Soft spring addition

Flag for soft spring addition.

= 0: No	no soft spring added
= 1: Yes	soft spring added
	(default is no)

Soft spring value

Soft spring constant.

= 0.0	constant set to 1.0E-6
> 0.0	constant to be used
	(default is 1.0E-6)

Harmonic number

Harmonic number for axisymmetric shell elements.
(default is 0)

Damping constant

Structural damping constant to be used in calculating complex frequencies (option C of frequency analysis method).
(default is 0.0)

Mass flag

Type of mass matrix. Only the lumped mass option is supported by the post-dynamic analysis module (ASTAR), so do not use the consistent mass matrix if you are planning to use ASTAR.

= 0: Lumped mass	use lumped mass
= 1: Consistent mass	use consistent mass
	(default is lump)

Modal acceleration

Modal acceleration flag.

= 0: No	do not prepare files for the modal acceleration option
= 1: Yes	prepare files for the modal acceleration option
	(default is no)

Mass participation factor

Flag for calculation of mass participation factors.

= 1: Yes	calculate
= 0: No	do not calculate
	(default is no)

Save stiffness matrix flag

Flag for saving decomposed stiffness matrix.

= 0: No	do not save
= 1: Yes	save
	(default is no)

Frequency for nonlinear analysis flag

Flag to allow the use of the deformed shape resulting from the nonlinear analysis as the basis of the frequency problem.

= 0: No	do not use deformed shape from NSTAR
= 1: Yes	use deformed shape from NSTAR
	(default is no)

Form stiff matrix flag

Flag for forming the stiffness matrix.

= 0: Form	form stiffness matrix
= 1: Don't form	use saved decomposed stiffness matrix from a previous run
	(default is form)

spin-softening

Flag to include the spin-softening effect.

= 0: No	do not consider the spin-softening effect
= 1: Yes	consider the spin-softening effect
	(default is 0:no)

rigid connections flag

This flag controls the continuity between solid and shell elements connected to each other. Solid elements like TETRA4, TETRA10, and SOLID do not consider rotational degrees of freedom (DOF) in their formulation. Shell elements on the other hand consider rotational DOF explicitly in their formulation.

Traditionally, you need to introduce some coupling constraints when connecting such incompatible elements. This flag, when set to rigid, will take care of this condition automatically. The default option (hinge) assumes a hinged connection between solid and the shell elements in the model must be stabilized by applying sufficient boundary conditions.

= 1: Rigid	assume rigid connections between all solid and shell elements
= 0: Hinge	assume hinged connections between all solid and shell elements
	(default is hinge)

Solver option

= 0: Sparse	Use the sparse direct solver.
= 1: Skyline	Use the skyline direct solver.
	(default is Sparse)

Notes

- When the Sturm Sequence Check is activated, the program checks for lost modes. A message is printed in the output file if lost modes are detected. You may apply a shift or specify more iterations and rerun the analysis.
- The eigenvalue shift can be used in two ways.

  a. To determine the frequencies and mode shapes of structures with rigid body motions.

  b. To determine the frequencies and mode shapes of a system within a certain frequency range.

  If the shift flag is set equal to 1, the program calculates and applies a negative shift. The shift is used to find the frequencies of unstable systems (systems with rigid body modes or zero frequencies).

  If the shift flag is set equal to 2, the user inputs the value of the shift. The value of the shift determines the center of convergence for eigenvalues which will then be ordered according to how close they are to this center. This option must be used when it is desired to find frequencies of the system in a certain range. A positive shift of the square root of [(wa2 + wb2)/2] instructs the program to calculate frequencies in the range of wa to wb. Frequencies outside the range will also be calculated if 'number of frequencies' is greater than the number of frequencies found in the specified range. This option works in conjunction with the Subspace and Lanczos methods.




- You can save the decomposed stiffness matrix (using "save stiffness matrix flag" option) for subsequent runs, for static, frequency or buckling analysis provided that in the subsequent run you request not to form a new stiffness matrix but read the existing one instead. This option could reduce the solution time considerably for large problems.

Example:
```
A_FREQUENCY, 5, S;
R_FREQUENCY
```
This first command instructs the second command to calculate 5 frequencies using the Subspace method.

A_BUCKLING

Analysis > FREQUENCY/BUCKLING > Buckling Options

The A_BUCKLING command sets options for subsequent buckling analysis performed by the R_BUCKLING (**Analysis, Frequency/Buckling, Run Buckling Analysis**) command.

Number of eigenvalues
Number of eigenvalues to be calculated.
(default is 1)

Method
Buckling analysis method.

= I	inverse power method (to calculate only one eigenvalue)
= S	subspace iteration
= J	Jacobi method
= L	Lanczos method
= GJ	Jacobi method with Guyan reduction
= C	complex eigenvalue analysis
	(default is I)

Maximum number of iterations
Maximum number of iterations for buckling when method I or S is used.
(default is 16)

Sturm sequence
Flag for Sturm sequence check.

= 0: No	no Sturm sequence check
= 1: Yes	perform Sturm sequence check
	(default is no)

Shift flag
Flag for eigenvalue shift.

= 0: No eig. val. shift	no eigenvalue shift is applied
= Shift by program	program calculates the eigenvalue shift
= Shift by user	user specifies the eigenvalue shift
	(default is no eig. val. shift)

Shift value

Value of the eigenvalue shift. Used only if shift flag = shift by user.
(default is 0.0)

Tolerance

Convergence tolerance for eigenvalues.
(default is 1.0E-5)

Soft spring addition

Flag for soft spring addition.

= 0: No	no soft spring added
= 1: Yes	soft spring added *(default is no)*

Soft spring value

Soft spring constant.

= 0.0	constant set to 1.0E-6
> 0.0	constant to be used *(default is 1.0E-6)*

Harmonic number

Circumferential harmonic number. Required only when axisymmetric shell elements are used.
(default is 0)

Buckling for Nonlinear analysis flag

Flag to run buckling analysis using saved stiffness matrix saved from an earlier nonlinear run.

= 0: No	do not use information from nonlinear run
= 1: Yes	use information from nonlinear run *(default is 0:no)*

Form stiff matrix flag

Flag for forming the stiffness matrix.

= 1: Form	form stiffness matrix
= 0: Don't form	use saved decomposed stiffness matrix from a previous run *(default is form)*

Rigid connections flag

This flag controls the connection between solid and shell elements in the model. Solid elements like TETRA4, TETRA10, and SOLID do not consider rotational degrees of freedom (DOF) in their formulation. Rotations in these solid elements can be expressed in terms of the translational DOF. Shell elements on the other hand consider rotational DOF explicitly in their formulations.

Traditionally, you need to introduce some coupling constraints when connecting such incompatible elements. This flag, when set to rigid, will take care of this condition automatically. The default option (hinge) assumes a hinged connection between solid and the shell elements in the model must be stabilized by applying sufficient boundary conditions.

= 1: Rigid	assume rigid connections between all solid and shell elements
= 0: Hinge	assume hinged connections between all solid and shell elements *(default is 1:hinge)*

Solver option
= 0: Sparse Use the sparse direct solver.
= 1: Skyline Use the skyline direct solver.
= 2; Iterative (PCG)) Use the iterative PCG solver
(default is 0: Sparse)

Notes

- A shift can be applied to converge to a higher buckling mode. The value of the shift determines the center of convergence for the buckling load factors (eigenvalues). The value of the shift should be close to the desired buckling load factor if approximately known, or else it may converge to adjacent modes.
- The specified shift is applied to the eigenvalues. It works only with the Subspace and Inverse methods. A positive shift of the square root of $[(\omega_a^2 + \omega_b^2)/2]$ instructs the program to calculate eigenvalues in the range of w_a to w_b. Buckling load factors outside the range will also be calculated if the number of requested eigenvalues is greater than the number of eigenvalues in the specified range. Unlike the case with natural frequency calculations where all eigenvalues are positive, it is possible to have both negative and positive buckling load factors in buckling analysis. Usually the option of applying a user-defined shift is not used for buckling load calculation since in most cases only the lowest load is desired. If this option is used, it is important to understand that the sign of the shift determines the center of convergence in either the positive or negative side of the buckling load factor domain.
- You may use an already existing decomposed stiffness matrix saved from an earlier static or frequency run by using 'don't form' for the 'form stiff matrix' flag. Using this option may reduce the solution time by several orders of magnitude for large problems.

Example:
```
A_BUCKLING, 2, S;
R_BUCKLING
```
The first command sets the flags for the second command to use the Subspace method to calculate the lowest two buckling loads. All other defaults are accepted. The actual buckling loads are calculated by multiplying the applied loads by the calculated eigenvalues.

A_FFEFREQ

Analysis > FREQUENCY/BUCKLING > FFE Frequency Option

The A_FFEFREQ command sets analysis options for subsequent frequency analysis using the FFE Dynamics module. Note that the A_FREQUENCY command sets analysis options for frequency analysis using the DSTAR module. The most recently issued command out of the two commands (A_FREQUENCY and

A_FFEFREQ) determines whether the R_FREQUENCY command will run DSTAR or FFE Dynamics. The default is to run DSTAR.

Element order

Order of the element to be used. In spite of the element group name in the database, you may specify through this option whether first (linear) or second (parabolic) elements will be used. As an example you may define TETRA4 elements and specify second order, in which case middle nodes on straight edges will be considered during analysis. On the other hand you may define TETRA10 elements and specify to use first order.

= 1: First	use first order for continuum elements
= 2: Second	use second order for continuum elements
	(default is second)

Number of frequencies

Number of natural frequencies to be calculated. Enter 0 if an unknown number of frequencies is to be calculated in a given range.

= N	calculate N natural frequencies
= 0	calculate all frequencies in the specified range

Lower bound value

Lower bound of the frequency range. This option is currently not used, it is always set to zero.
(default is 0)

Upper bound value

Upper bound of the frequency range. Enter 0 if you specified the number of frequencies to be calculated.

Rigid connections flag

This flag controls the connection between solid and shell or beam elements in the model. Solid elements like TETRA4, TETRA10, and SOLID do not have explicit rotational degrees of freedom (DOF). Rotations of solid elements can be expressed in terms of the translational DOF. Beam and shell elements on the other hand have explicit rotational DOF.

Traditionally, you need to introduce some coupling constraints when connecting such incompatible elements to ensure continuity. This flag, when active, takes care of this condition automatically and rigid connections between all such incompatible elements in the model are assumed. When you want to specify hinge connections or you need to compare COSMOS/FFE results to results from traditional finite element systems which assume hinge connections between solid and shell or beam elements, you must turn this flag off before running the analysis.

= 1: Yes	activate rigid connections
= 0: No	deactivate rigid connections
	(default is yes)

Notes

- Either the number of frequencies or the upper limit must be non-zero.

- The actual number of frequencies calculated will be the number specified + 1 if the specified number is not zero. If the number of frequencies is set to zero, all frequencies in specified range + 1 frequency (outside range) will be calculated.

MODE_CHECK

Analysis > FREQUENCY/BUCKLING >Mode check option

The MODE_CHECK command sets options for modal verification using two sets of modes: primary and secondary. The primary modes are obtained from the current GEOSTAR database. The secondary modes can come from another GEOSTAR database with a different mesh, another FEA program, or can come from experimental data. The primary model may be used to identify the critical positions for the measuring devices for the target mode(s).

The command provides 3 choices to perform: 1) cross-correlation, 2) orthogonality check, and 3) assurance check for the two sets of modes. In each case the corresponding matrix is calculated and printed in the output file by running frequency analysis. The cross-correlation matrix measures the degree of correlation between the primary and secondary modes. The orthogonality matrix provides a measure of the quality of secondary modes obtained from testing (as orthogonality is automatic for FEA models), The modal assurance matrix evaluates the mode independence.

The main quantities used in these calculations are: the primary and secondary mode matrices ($[\Phi_P]$ and $[\Phi_S]$); and the primary and secondary mass matrices ($[M_P]$ and M_S]). For example the matrix multiplication $[\Phi_P]^T[M_P][\Phi_S]$ provides a tool for identifying the secondary modes $[\Phi_S]$ and comparing them with the primary modes $[\Phi_P]$.

✍ This functionality may not be used in models containing elements with different DOF/node. The elements supported by this command are: TRIANG, PLANE2D, and tetrahedral and shell elements.

Check Option

Sets the check option.

= 0: None — no mode check to be performed. When this option is selected, R_Frequency will calculate modes as usual. When another option is selected, the R_FREQUENCY command will only perform the specified mode checking.

= 1: Cross-correlation — Perform cross-correlation check

= 2: Orthogonality — Perform orthogonality check

= 3: Assurance — Perform assurance check

(default is 1: Cross-correlation)

Mass-normalize modes

Flag for normalizing primary and secondary modes with respect to the primary mass matrix.

= 0: Normalize normalize modes with respect to the primary mass matrix

= 1: Do not normalize do not normalize modes

(default is 0:Normalize)

Search option

Sets the criterion for matching the secondary modes with the primary ones. This operation modifies the secondary modes to correspond to the nodal locations of the primary nodes.

= 0: Closest grid use the closest grid point.

= 1: Closest mass use closest mass value

= 2: Interpolation use interpolation to calculate the associated mass value

(default is 0:Closest grid)

Secondary mode type

Source of secondary modes.

= 0: FEA Model another problem run by GEOSTAR

= 1: User-provided files information provided by the user in a certain format.

*(default is 0:*FEA Model*)*

Secondary mode name

Name of the secondary model.

Grid Tolerance

Radius to be used for searching for mass.

(default is 10^{-4})

Mode shape similarity tolerance

Tolerance for deciding mode shape similarity.

(default is 10%)

Mode comparison range

Flag for mode shapes to be compared.

= 0: All Modes another problem run by GEOSTAR

= 1: Range of modes use the range of modes specified by the following entries.

*(default is 0:*FEA Model*)*

The following entries are prompted only if you select 1: Range of modes.

Starting primary mode

Label of starting primary mode.

(default is 1)

Ending primary mode

Label of ending primary mode.

Starting secondary mode

Label of starting secondary mode.

(default is 1)

Ending secondary mode
Label of ending secondary mode.

To check/identify mode shapes:

1. Prepare the secondary data in one of the following ways:
   - Run frequency analysis in GEOSTAR to calculate the desired number of modes.
   - If you are using experimental data or modes resulting from an FEA program other than GEOSTAR, prepare the secondary data in the format specified in the ***External file format for the secondary data*** section below.
2. For the primary model, run frequency analysis to calculate the desired number of modes. Then select **Analysis > FREQUENCY/BUCKLING >Mode check option** to specify the desired options, and run frequency analysis. The current database will be used as the primary set. The results of the mode check will be written to the output file.

When you run frequency analysis while the check option is set to an option other the (0: Off) option, the program performs only the check, it does not calculate the frequencies. To return to the regular mode, issue the MODE_CHECK command again and select the (0: Off) option.

External file format for the secondary data (Test or FEM modes)

Original nodal locations

- Data should be in text (ASCII) format.
- File extension should be **.ND2**. The name of the file is requested by the Mode_Check command.
- First row in the file includes two integers in the following order:
  - Number of nodes, and
  - Maximum node label (not necessarily equal to the number of nodes).
- Second row onwards, each row has 4 entries as follows:
  - First entry is the node label (integer), and
  - Second through fourth entry are the X, Y, and Z coordinates of the node (real numbers)

Element Connectivity

Used only when activating the interpolation flag.

- Data should be in text (ASCII) format.
- File extension should be **.EL2**. The name of the file is requested by the Mode_Check command.
- First row in the file includes 3 integers:
  - Number of elements,
  - Maximum element label (not necessarily equal to the number of elements).
  - Number of nodes per element
- Second row onwards, each row has:
  - Element label,
  - List of node defining the element (nodal connectivity)

Mode Shapes

- Data should be in text (ASCII) format.
- File extension should be **.LC2**. The name of the file is requested by the Mode_Check command.
- First row in the file includes 2 integers:
  - Number of frequencies (modes in the file), and
  - Degrees of freedom per/node. This can be 2 (for PLANE2D), 3 (TETRA4 or TETRA10), or 6 (SHELL4, TETRA4R). You may not mix elements with different number of DOF/node.
- Second row onwards, each row has the following entries (integers):
  - Number of nodes (N) used for this frequency (not necessarily equal for each mode), and
  - Frequency in Hz
- Third row onwards, each row has the following entries (integers):
  - Node, and displacements in DOF-1 through DOF-n (n is the number of DOF/node as specified in the first row).




- After finishing the first mode, the following mode should start at row (N + 3) with the frequency in Hz and the number of nodes used in this mode. Following rows should list the node and the associated displacements (DOF-1 through DOF-n). The number of modes may be different than N.
- The format of the last bullet should be used to list all desired modes.

Mass Information

✍ Used only when the closest mass option is selected.

- Data should be in text (ASCII) format.
- File extension should be **.MS2**. The name of the file is requested by the Mode_Check command.
- First row in the file includes 2 integers:
  - Number of nodes, and
  - Rotation consideration flag. Enter 1 for translation only (no rotation), or 4 for translation and rotation.
- Second row onwards, each row has 2 or 3 entries:
  - Node label (integer), and
  - Mass at node, and Mass moment of inertia if selected in the first row.

R_FREQUENCY

Analysis > FREQUENCY/BUCKLING > Run Frequency

The R_FREQUENCY command performs dynamic analysis. It calculates frequencies and mode shapes using the DSTAR module. The most recently issued command out of the two commands (A_FREQUENCY and A_FFEFREQ) determines whether the R_FREQUENCY command will run DSTAR or FFE Dynamics. The default is to run DSTAR.

✍ This command will not calculate frequencies and mode shapes if the **MODE_CHECK (Analysis, FREQUENCY/BUCKLING, Mode check option)** command was issued and the check option was set to an option other than **None**. In such cases, only the specified mode check option will be performed. Refer to the **MODE_CHECK** command for details. Mode checking is not supported by FFE.

To run frequency analysis using FFE Dynamic:

1 Create your FEA model (geometry, mesh, loads, and boundary conditions).

✍ You do not have to apply displacement constraints. FFE Dynamic can detect rigid body modes.

2 Optional: click **Analysis, Run Check** to check input data.

3 Click **Analysis, Frequency/buckling, FFE Frequency Options** to set the desired options.

4 Click **Analysis, Frequency/buckling, Run Frequency**.

✍ If the run is not successful, a clear message will be given. Refer to Appendix A of the COSMOS/M FFE Dynamic manual for explaining and fixing the problem. The message is also written to the output file (.out).

To run frequency analysis using DSTAR:

1 Create your FEA model (geometry, mesh, loads, and boundary conditions).

2 Optional: click **Analysis, Run Check** to check input data.

3 Click **Analysis, Frequency/buckling, Frequency Options** to set the desired options.

✍ This step is optional unless you have already clicked the **Analysis, Frequency/buckling/ FFE Frequency Options** command.

4 Click **Analysis, Frequency/buckling, Run Frequency**.

✍ If the run is not successful, a clear message will be given. Refer to Appendix D of the Basic System User's Guide Part I for explaining and fixing the problem. The message is also written to the output file (.OUT).

5 To list the calculated frequencies, click **Results, List, Natural Frequency**.

6 To animate mode shapes, click **Results, Plot, Animate**.

7 To use the check mode functionality, refer to the **MODE_CHECK** (**Analysis, FREQUENCY/BUCKLING, Mode check option**) command.

R_BUCKLING

Analysis > FREQUENCY/BUCKLING > Run Buckling

The R_BUCKLING command performs buckling analysis, using the DSTAR program to calculate buckling load factors (eigenvalues). The calculated eigenvalues can be viewed as factors of safety or buckling load factors. Eigenvalues larger than

1.0 indicate safe design. Eigenvalues less than 1.0 indicate failure. Buckling loads can be calculated by multiplying all defined loading by the corresponding eigenvalues. The calculated eigenvalues are listed in the output file and can also be listed by the FREQLIST (**Results List, Natural Frequency**) command.

Negative buckling load factors indicate safe design. Buckling occurs if the directions of all loads are reversed and their magnitudes are multiplied by the absolute value of the corresponding eigenvalue.

To run buckling analysis:

1. Create your FEA model (geometry, mesh, loads, and boundary conditions).
2. Optional: click **Analysis, Run Check** to check input data.
3. Click **Analysis, Frequency/buckling, Buckling Options** to set the desired options.

✍ This step is optional. You need to issue this command if you want the program to calculate more than one mode, or you want to change any of the other defaults.

4. Click **Analysis, Frequency/buckling, Run Frequency**.

✍ If the run is not successful, a clear message will be given. Refer to Appendix D of the Basic System User's Guide Part I for explaining and fixing the problem. The message is also written to the output file (.OUT).

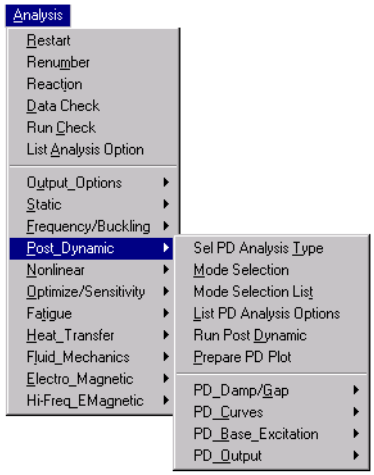
5. To list the calculated buckling load factors (factors of safety), click **Results, List, Natural Frequency**.
6. To animate mode shapes, click **Results, Plot, Animate**.
7. To calculate the critical buckling loading for a particular mode, multiply all loading by the corresponding buckling load factor.

POST_DYNAMIC Menu

Analysis > POST_DYNAMIC

This menu contains commands related to ASTAR, the post dynamics analysis module. ASTAR uses the natural frequencies and mode shapes calculated by DSTAR to evaluate the response of the structure to various types of dynamic loading. The module uses the principle of mode superposition.

Figure 10-8 Post_Dynamic Menu



PD_ATYPE

Analysis > POST_DYNAMIC > Sel PD Analysis Type

The PD_ATYPE command specifies the type of post dynamic analysis to be performed. The command syntax depends on the selected type of analysis. The available type of analysis are:

= 0: Static stress analysis	static stress analysis
= 1: Resp spectra analysis	response spectra analysis
= 2: Modal time history analysis	modal time-history analysis
= 3: Resp spectra generation	response spectra generation
= 4: Random vibration analysis	random vibration
= 5: Harmonic analysis	harmonic analysis
= 6: Direct spectra generation	direct spectrum generation
= 7: Time history generation	time history generation

Response Spectra Analysis

Post dynamic analysis type

= 1: Resp spectra analysis **RESPONSE SPECTRA ANALYSIS**

No. of frequencies

Number of frequencies to be used in the analysis.
(default is 1)

Mode combination method

Mode combination method.

= 0: SRSS	SRSS (Square Root Sum of the Square method)
= 1: CQC	CQC (Complete Quadratic method)
= 2: NRL	NRL (Naval Research Laboratory method)
	(default is SRSS)

Cluster factor

Cluster factor used for the SRSS method only.
(default is 0.0)

Units of existing freqs

Flag defining units of the exciting frequency and Ws and We.

= 0: Rad/sec	radians/second
= 1: HZ	cycles/second (Hz)
	(default is rad/sec)

Unused option

Unused option.

Unused option

Unused option.

Modal displacement print

Flag for printout of mode shapes.

= 1: Yes	suppress printing of mode shapes
= 0: No	print mode shapes
	(default is no)

Response printout type

Flag defining type of response printout.

= 0: Each base spectrum	print response due to each base spectrum separately
= 1: Combined (RMS)	print the combined response (R.M.S) for all base response spectra
= 2: Both and plot file	print both of the above options. Also write the R.M.S. of response to the plot file
	(default is both and plot file)

Notes

1. clusf < [T(i-1) - T(i)] / T(i-1) -> add mode response using the SRSS method
   clusf > [T(i-1) - T(i)] / T(i-1) -> add mode response by absolute values
   where:
   T(i-1) and T(i) represent two consecutive natural periods of structure.
2. For the NRL method, the maximum response due to a base excitation is evaluated from:

ABS {R} + SQRT [{SS} - {R**2}]

where:

{SS} = vector of sum of the squares of mode responses.

{R} = maximum response among all mode responses for each node and direction.

The R.M.S. response due to all excitations is evaluated by combining responses (due to different excitations) by the SRSS method.

Modal Time-History Analysis

Post dynamic analysis type

= 2: Modal time history analysisMODAL TIME HISTORY ANALYSIS

Number of frequencies

Number of frequencies to be used in the analysis.
(default is 1)

Number of steps in the solution

Total number of solution steps.
(default is 10)

Starting time

Starting time for solution.
(default is 0.0)

Time step size

Time step size.

Time integration method

Time integration method.

= 0: Newmark	use Newmark method
= 1: Wilson-theta	use Wilson-theta method
	(default is Newmark)

First integration parameter

First integration parameter.

Newmark method:	[delta] (default is 0.5)
Wilson-theta method:	[theta] (default is 1.4)

Second integration parameter

Second integration parameter.

Newmark method:	[alpha] *(default is 0.0625)*
Wilson-theta method:	not used

Type of response printout

Type of response printout.

= 0: Rel displ & rel vel	print relative displ. and relative vel.
= 1: Rel displ & abs vel	print relative displ. and absolute vel.
= 2: Abs displ & abs vel	print absolute displ. and absolute vel.
	(default is rel displ and rel vel)

Load case for initial conditions
Load case for initial conditions. Give load case number.

Dead load step
Load case from static analysis to be superimposed with the dynamic response.
Load case number: between 1 and 100.
(default is 0)

Dead load multi. factor
Factor by which results of the dead-load case will be multiplied before adding to the dynamic response. (not prompted if dead-load step is 0, default is 1.0)

Notes

- Accelerations are always printed in absolute values.
- The 'load case for initial conditions' flag may be used to assign displacement results from a static analysis load case as initial conditions. The INITIAL command may be used to specify initial conditions to nodes.

Response Spectra Generation

Post dynamic analysis type
= 3: Resp. spectra generation RESPONSE SPECTRA GENERATION

Number of frequencies
Number of frequencies to be used in the analysis.
(default is 1)

Starting frequency
Starting frequency for a response spectrum generation (rad/sec).
(default is 0.0)

Ending frequency
Ending frequency for a response spectrum generation (rad/sec).
(default is 0.0)

Freq. scale flag
Flag for response spectrum abscissa scale.

= 0: Logarithmic	logarithmic scale
= 1: Linear	linear scale
	(default is logarithmic)

Number of points to be used
Number of points to define the response spectrum curve.
(default is 1)

Node label
Label of the node at which response spectrum is to be generated.
(default is 1)

Damping ratio
Damping ratio associated with this node.
(default is 0.0)




X translation flag
Flag for generation of response spectrum for translation in the X-direction.
= 0: No generation — do not generate response spectrum
= 1: Generate response — generate response spectrum
(default is no generation)

Y translation flag
Flag for generation of response spectrum for translation in the Y-direction.
= 0: No generation — do not generate response spectrum
= 1: Generate response — generate response spectrum
(default is no generation)

Z translation flag
Flag for generation of response spectrum for translation in the Z-direction.
= 0: No generation — do not generate response spectrum
= 1: Generate response — generate response spectrum
(default is no generation)

X rotation flag
Flag for generation of response spectrum for rotation in the X-direction.
= 0: No generation — do not generate response spectrum
= 1: Generate response — generate response spectrum
(default is no generation)

Y rotation flag
Flag for generation of response spectrum for rotation in the Y-direction.
= 0: No generation — do not generate response spectrum
= 1: Generate response — generate response spectrum
(default is no generation)

Z rotation flag
Flag for generation of response spectrum for rotation in the Z-direction.
= 0: No generation — do not generate response spectrum
= 1: Generate response — generate response spectrum
(default is no generation)

Note

The results of this analysis are written in files with extension listed in the following table. These files are written in a format which can be read as external files by the PD_CURDEF command and could be used as possible input to secondary systems.

File Extension	Description
.PSV	Pseudo velocity
.PSA	Pseudo acceleration
.RLD	Relative displacement
.RLV	Relative velocity
.ABA	Absolute acceleration

(The frequency unit is Rad/Sec)

Random Vibration

Post dynamic analysis type

= 4: Random vibration analysisRANDOM VIBRATION ANALYSIS

Number of frequencies

Number of frequencies to be used in the analysis.
(default is 1)

Units of exciting freqs

Flag defining units of the exciting frequency and Ws and We.

= 0: Rad/sec	radians/second
= 1: Hz	cycles/second (Hz)
	(default is rad/sec)

Starting frequency

Lower limit of exciting frequency to be considered in the analysis.
(default is 1.0E-11)

Ending frequency

Upper limit of exciting frequency to be considered in the analysis. Must be greater than Ws.
(default is 1.0E+10)

Correlation flag

Correlation flag.

= 0: Fully correlated	fully correlated
= 1: Fully uncorrelated	fully uncorrelated
= 2: Partial	partially uncorrelated
	(default is fully correlated)

Analysis method flag

Method of analysis.

= 0: Standard	standard method
= 1: Approximate	approximate method
	(default is standard)

Number of frequency points

Number of frequency points to be selected between any two adjacent natural frequencies. (See notes below)
(default is 1)

Gauss Integration order

Gauss Integration order to be used in integration of response power spectral densities.

= 2: 2-put Gaussian	two point Gaussian integration
= 3: 3-put Gaussian	three point Gaussian integration
	(default is 2-pt Gaussian)

Biasing parameter

Biasing parameter used to define the location of frequency points to be selected. (See notes below)
(default is 0.0)

Cross-mode cut off ratio

Cross-mode cut off ratio defines a limit on the ratio of any two mode frequencies (Wi/Wj, i>j). If Wi/Wj is greater than this ratio, the cross-mode effects between the two modes will not be considered.
(default is 1.0E+10)

PSD stress computation flag

Calculation of PSD of stresses for every solution step.

= 0: No	do not compute PSD of stresses for each step
= 1: Yes	calculate PSD of stresses for each step (default is 0)
	(default is no)

The following two options are prompted only if correlation flag is set to Partial:

Inside radius for partial correlation

Set the inner radius for partial correlation
(default is 0)

Outside radius for partial correlation

Set the outer radius for partial correlation
(default is 0)

Notes

- The random excitations must be stationary, Gaussian, with a mean value of zero, defined by one-sided power spectral density curves.
- The correlation flag is only used when base excitations in different directions or forces at more than one node are prescribed. If the excitations are correlated, the cross-spectral density terms will also be included in the analysis.
- The standard method performs a classical random vibration analysis. The approximate method assumes that power spectral densities around each mode are flat and that cross-mode effects are negligible.
- The last four entries are used only if the standard method is selected.
- The number of frequency points and bias are used to select frequency points at which the P.S.D. of the response is evaluated. They serve as steps in the numerical integration process in the frequency domain. These parameters must be selected to minimize the integration error and the computational effort.
- The default values for the number of frequency points and bias are based on the modal critical damping ratio:

Modal Damping Ratio zeta	Default for nfpts	Default for bias
zeta < 0.01	21	11.
0.01 < zeta < 0.1	21 - 4.34 1n (zeta/0.01)	11. - 3.47 1n (zeta/0.01)
zeta > 0.1	11	3.

- If the PSD stress computation flag is activated, RMS stresses will be stored in step “n + 1" location where 'n' is the number of solution steps. PSD stresses are stored in the corresponding location for each step.
- Deformation and displacement contour plots for random vibration analysis results in plotting the absolute value of the deformation.

Steady State Harmonic Analysis

Post dynamic analysis type

= 5: Harmonic analysis **HARMONIC ANALYSIS**

Number of frequencies

Number of frequencies to be used in the analysis.
(default is 1)

Units of exciting freqs

Flag defining units of the exciting frequency and starting frequency and ending frequency.

= 0: Rad/sec	radians/second
= 1: Hz	cycles/second (Hz)
	(default is rad/sec)

Starting frequency

Lower limit of exciting frequency to be considered in the analysis.
(default is 1.0E-10)

Ending frequency

Upper limit of exciting frequency to be considered in the analysis. Must be greater than starting frequency.
(default is 1.0E+10)

Number of output freqs

Number of output frequencies to be selected in the range of exciting frequencies to be considered.
(default is 1)

Frequency scale flag

Flag for locating frequency points.

= 0: Logarithmic	logarithmic interpolation
= 1: Linear	linear interpolation
	(default is logarithmic)

Type of response printout

Type of response printout.

= 0: Rel displ & rel vel	write relative displ. and relative vel. both in the output and plot files
= 1: Rel displ & abs vel	write relative displ. but absolute vel. both in the output and plot files
= 2: Abs displ & abs vel	write absolute displ. and absolute vel. both in the output and plot files
	(default is rel displ and rel vel)

Notes

- Accelerations are always printed in absolute values.
- Deformation and displacement contour plots for harmonic analysis result in plotting the absolute value of the deformation.

Direct Spectrum Generation

Post dynamic analysis type
= 6: Direct spectra generation DIRECT SPECTRUM GENERATION

Starting frequency
Starting frequency for a response spectrum generation (Hz).

Ending frequency
Ending frequency for a response spectrum generation (Hz).

Number of points
Number of points to define the response spectrum curve (number of points must be greater than 2).

Gravity units flag
Gravity unit flag.

= 0: MKS	input time history is not in terms of gravity
= 1: FPS	input time history is in terms of gravity (if = MKS, the next prompt is not applicable) *(default is MKS)*

Gravity value
Magnitude of the gravitational constant, (g).

Damping ratio
Damping ratio.
(default is 0)

Step size
Maximum time increment used in the central difference time integration method.
(default is 0.01)

Curve label
Time history input curve label.
(default is 1)




Note

- 1.The results of this analysis are written in files with extension listed in the following table.

File Extension	Description
.PSV	Pseudo velocity
.PSA	Pseudo acceleration
.RLD	Relative displacement
.RLV	Relative velocity
.ABA	Absolute acceleration

- These files are written in a format which can be read by the PD_CURDEF command and can be used for:

  a. Plotting of the results:

  Use PD_CURTYPE, PD_CURDEF (with “external_file” option), and PD_BASE to read files into the database and then use ACTXYPRE and XYPLOT to plot the spectrum. (Note that the directions in PD_BASE may not be applicable here.)

  b.Spectra input to other problems:

- Use PD_CURTYPE, PD_CURDEF, and PD_BASE to read the files into the database. Use the “external_file” option in the PD_CURDEF command

Time History Generation

Post dynamic analysis type
= 7: Time history generation TIME HISTORY GENERATION

Maximum iteration
Maximum number of iterations.

Maximum acceleration
Maximum allowable value of acceleration in the time history curve to be generated.

Time duration
Duration of the time history record to be generated.
(default is 20)

Tolerance
Tolerance allowed between the target and calculated spectrums.
(default is 0.05)

Number of curves
Number of desired time history curves. Maximum is two.
(default is 1)

Correction flag

Baseline correction flag.

= 0: Off	uncorrected time history record
= 1: On	corrected time history record
	(default is off)

Curve print interval

Print interval for each time history curve.
(default is 1)

Print time step increment

Time step increment for each time history record to be printed.

= 0	increment is assigned by the program
	(default is 0.)

Plot time step increment

Time step increment for each time history record.

= 0	increment is assigned by the program
	(default is 0.)

Damping ratio

Damping ratio.
(default is 0.)

Step size

Maximum time increment used in the central difference time integration method.
(default is 0.01)

Input time curve label

Time history input profile curve label.

= 0	El-Centro earthquake envelope
	(default is 0)

Target spectra curve label

Acceleration target spectrum curve label (defined as frequency dependent acceleration base excitation).
(default is 1)

Notes

1. The time history input curve must be defined as a positive time-dependent curve applied as acceleration base excitation.
2. The acceleration target spectrum curve must be defined as a frequency-dependent curve applied as acceleration base excitation with the unit of frequency in Hertz.
3. If the acceleration target spectrum curve is in terms of gravity (g's), then the generated records will also be in terms of (g).
4. The profile curve determines the approximate envelope of the time history curve. The horizontal axis represents time and the vertical axis shows acceleration in (g's). This acceleration envelope is always positive and independent of the units in which the spectrum is given, the values of this envelope are in terms of (g). For example, in the case of the El-Centro earthquake, the maximum value defined in the profile curve cannot exceed 0.30.




5. The spectrum tolerance is defined as the maximum relative distance between the target spectrum and the calculated spectrum for all frequencies.

   TOL = MAX{ABS(TS(j) - CS(j)) / TS(j)}

   where:

   j = Frequency number
   TS(j) = Target spectrum at frequency number j
   CS(j) = Computed spectrum at frequency number j

6. Maximum of two time history curves can be requested. Once the first T-H curve (Curve #1) is generated, the second T-H curve (Curve #2) can be generated with a relatively low correlation with the first T-H curve, based on shifting the frequencies of the first T-H curve to the adjoining mid-frequencies.
7. Each T-H record can be modified such that at the end of each record the values of displacement and velocity will be equal to zero. This procedure is called the base line correction method and the corrected Time History (T-H) curves #3 and #4 correspond to curves #1 and #2 respectively.
8. The iterative scheme is only used to generate the first T-H curve. Once the first T-H record (Curve #1) is generated, for the final iteration, the second and corrected T-H curves (Curves #2, #3, and #4) can be generated.
9. During the execution of the program, the results will continue to be printed with the requested print interval until either the number of iterations reaches the limit value or the spectrum convergence is achieved. In which case the final spectrum will be written in the plot file for further postprocessings. The curves available for plotting are summarized in the following table:

   These files are written in a format which can be read as external files by the PD_CURDEF command and can be used for:

File Extension	Description
.CR1	1st T-H Curve
.CR2	2nd T-H Curve
.CR3	Corrected 1st T-H Curve
.CR4	Corrected 2nd T-H Curve

   a. Plotting of the results:

      Use PD_CURTYPE, PD_CURDEF (with “external_file” option), and PD_BASE to read files into the database and then use ACTXYPRE and XYPLOT to plot the spectrum. (Note that the directions in PD_BASE may not be applicable here.)

   b. Spectra input to other problems:

      Use PD_CURTYPE, PD_CURDEF (with “external_file” option) to read the files into the database, and PD_BASE to read the files of the above table into the database.

Static Stress Analysis

Post dynamic analysis type

= 0: Static stress analysis **STATIC STRESS ANALYSIS**

Note

Choosing this option causes subsequent stress calculations to be based on linear static analysis.

PD_MODESEL

Analysis > POST_DYNAMIC > Mode Selection

The PD_MODESEL command can be used to select specific ranges of modes to be included in calculating the response. The command works with all types of advanced dynamic analyses and can be used to filter out modes that are not pertinent to the problem in order to accelerate the solution or to find the contribution of a desired set of modes separately. Once modes are selected using this command, only the selected modes will be used in the analysis regardless of the number of frequencies specified in the PD_ATYPE (Analysis > Post Dynamic > Set PD Analysis_Type) command. The number of frequencies specified in the PD_ATYPE command must be equal to or larger than the number of modes selected by this command.

Activate mode selection

Flag to activate/deactivate mode selection.

= 0: No	deactivate mode selection
= 1: Yes	activate mode selection *(default is 1)*

Beginning mode for set (i)

Beginning mode shape number for set i.

Ending mode for set (i)

Ending mode shape number for set i. i=1,2,....,10

Note

If mode selection is deactivated, the number of modes specified by the PD_ATYPE (Analysis > Post Dynamic > Set PD Analysis_Type) command will be used as usual.

PD_MODELIS

Analysis > POST_DYNAMIC > Mode Selection List

The PD_MODELIS command lists the sets of modes specified by the PD_MODESEL (Analysis > Post Dynamic > Mode Selection) command. If this command lists empty sets, then the number of modes specified in the PD_ATYPE (Analysis > Post Dynamic > Set PD Analysis_Type) command will be used in the analysis.

PD_ALIST

Analysis > POST_DYNAMIC > List PD Analysis Options

The PD_ALIST command lists the active options for the specified type of analysis.

Post dynamic analysis type to list

Type of post dynamic analysis.

= 1	Response Spectra Analysis
= 2	Modal Time-History Analysis
= 3	Response Spectra Generation
= 4	Random Vibration
= 5	Harmonic Analysis
= 6	Direct Spectrum Generation
= 7	Time History Generation
	(default is 2 or last analysis type defined)

R_DYNAMIC

Analysis > POST_DYNAMIC > Run Post Dynamic

The R_DYNAMIC command performs post dynamic analysis, using the "POST" program. You must run frequency analysis to calculate the natural frequencies and mode shapes must be available when this command is issued.

PD_PREPARE

Analysis > POST_DYNAMIC > Prepare PD Plot

The PD_PREPARE command can be issued after performing post dynamic analysis to prepare files for the listing of extreme response values as well as the relative response between nodes.

Option

Flag to indicate the type of files to be prepared.

= 1: Max min	prepares a file for listing the extreme response values specified by the PD_MAXMIN command
= 2: Relresp	evaluates the relative response for the nodes previously assigned by the PD_RELRESP command (generated relative response is written to output file) *(default is max min)*

Step

Time (or frequency) interval for relative response calculation. (Prompted only for modal time history and harmonic analysis.)

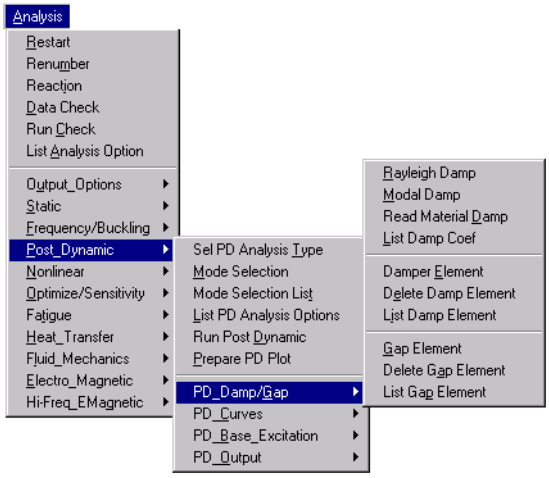



▼PD_DAMP/GAP Menu

Analysis > POST_DYNAMIC > PD_DAMP/GAP

This menu contains commands for damping and gap specification.

Figure 10-9 PD Damp/Gap Menu



PD_RDAMP

Analysis > POST_DYNAMIC > PD_DAMP/GAP > Rayleigh Damp

The PD_RDAMP command defines the parameters for Rayleigh (proportional) damping.

Mass matrix coefficient
Coefficient of Mass used in Rayleigh damping (alpha).
(default is 0.0)

Stiffness matrix coeff
Coefficient of Stiffness used for Rayleigh damping (beta).
(default is 0.0)

Note
Damping Ratio = (alpha). [M] + (beta). [K].

PD_MDAMP

Analysis > POST_DYNAMIC > PD_DAMP/GAP > Modal Damp

The PD_MDAMP command can be used to define modal damping (the ratio of damping to the critical damping) for the model. Different damping values can be assigned to different sets of modes. Up to 8 sets of modes and damping ratios can be defined.

Mode set number
Set number for modes with a common modal damping ratio.
(default is the highest set defined + 1)

First mode
First mode in this set.

Last mode
Last mode in this set.
(default is first mode)

Damping ratio
Critical damping ratio for this set.
(default is 0.0)

PD_DAMPREAD

Analysis > POST_DYNAMIC > PD_DAMP/GAP > Read Material Damp

The PD_DAMPREAD command activates the calculation of damping ratios based on specified Composite Material Damping using the DAMP material property. The command, when issued, overrides information specified by the PD_MDAMP command.

To use this command properly:

1. Click **PropSets, Material Property**. The **MPROP** dialog box opens.
2. In the **Material property set** field, enter the desired label.
3. From the **Material Property Name** drop-down menu, select **DAMP**.
4. Click **Continue**.
5. In the **Property value** field, enter the desired value and click **OK**.
6. Click **Analysis, Buckling/Frequency, Frequency Options** and to set the desired options.
7. Click **Analysis, Buckling/Frequency, Run Frequency** to calculate frequencies and mode shapes.

8 Click **Analysis, Post_Dynamic, PD_Damp/Gap, Read Material Damping** to read the calculated damping ratios.

9 Click **Analysis, Post_Dynamic, PD_Damp/Gap, List Damp Coeff** to list modal damping coefficients calculated by the program.

10 To modify any of the calculated damping ratios, click **Analysis, Post_Dynamic, PD_Damp/Gap, Modal Damp.**

11 Click **Analysis, Post_Dynamic, Run Post Dynamic**.

PD_DAMPLIST

Analysis > POST_DYNAMIC > PD_DAMP/GAP > List Damp Coef

The PD_DAMPLIST command lists the available Rayleigh and modal damping information.

PD_CDAMP

Analysis > POST_DYNAMIC > PD_DAMP/GAP > Damper Element

The PD_CDAMP command defines concentrated dampers for a dynamic modal time-history analysis.

Concentrated damper label
Label of this concentrated damper.
(default is the highest number of defined dampers + 1)

Node label for one end of damper
Node label at one end of damper.

Node label for other end of damper
Node label at the other end of damper if = 0; damper is connected to ground.

X comp of concentrated damper
Value of concentrated damper in X-direction.
(default is 0.0)

Y comp of concentrated damper
Value of concentrated damper in Y-direction.
(default is 0.0)

Z comp of concentrated damper
Value of concentrated damper in Z-direction.
(default is 0.0)

PD_CDDEL

Analysis > POST_DYNAMIC > PD_DAMP/GAP > Delete Damp Element

The PD_CDDEL command deletes a pattern of concentrated dampers.

Beginning damper
Beginning damper of the pattern.

Ending damper
Ending damper in the pattern.
(default is beginning damper)

Increment
Increment between dampers in the pattern.
(default is 1)

PD_CDLIST

Analysis > POST_DYNAMIC > PD_DAMP/GAP > List Damp Element

The PD_CDLIST command lists a pattern of concentrated dampers.

Beginning damper
Beginning damper of the pattern.
(default is 1)

Ending damper
Ending damper in the pattern.
(default is highest concentrated damper defined)

Increment
Increment between dampers in the pattern.
(default is 1)

PD_GAP

Analysis > POST_DYNAMIC > PD_DAMP/GAP > Gap Element

The PD_GAP command defines gap elements for use in modal time-history dynamic analysis. These gaps are ignored for all other types of analyses.

Gap element label
Label assigned to this gap element.
(default is the highest gap element defined +1)

Node label for one end of gap
Node label defining one end of the gap.




Node label for other end of gap
Node label defining the other end of the gap.

Allowable gap opening
Allowable relative displacement between the two nodes with no gap resistance.

> 0.0	gap resists compression
= 0.0	gap is originally closed & resists compression
< 0.0	gap resists tension

Gap-stiffness
Gap-stiffness used to evaluate gap resistant force.
(default is 1.E7)

Coeff of friction
Coefficient of friction used to define the magnitude of friction force to be applied in the direction normal to the gap resisting force.
(default is 0.0)

Type of gap element
Defines type of gap element. Prompted only if coeff of friction > 0.0

= 0: Regular	regular gap element
= 1: Special	once the gap is closed, the evaluated gap force remains constant during the rest of the analysis. This option is useful when friction is caused by a constant normal force. *(default is reg)*

PD_GAPDEL

Analysis > POST_DYNAMIC > PD_DAMP/GAP > Delete Gap Element

The PD_GAPDEL command deletes a pattern of gap elements defined for a post dynamic analysis. using the PD_GAP (Analysis, POST_DYNAMIC, PD_DAMP/GAP, Gap Element) command.

Beginning gap element
Beginning gap element in the pattern.

Ending gap element
Ending gap element in the pattern.
(default is beginning gap element)

Increment
Increment between gap elements in the pattern.
(default is 1)




PD_GAPLIST

Analysis > POST_DYNAMIC > PD_DAMP/GAP > List Gap Element

The PD_GAPLIST command lists a pattern of gap elements defined for a post dynamic analysis using the PD_GAP (Analysis, POST_DYNAMIC, PD_DAMP/GAP, Gap Element) command.

Beginning gap element
Beginning gap element in the pattern.

Ending gap element
Ending gap element in the pattern.
(default is beginning gap element)

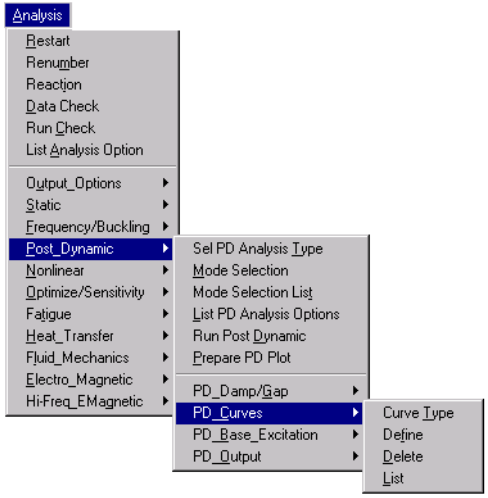
Increment
Increment between gap elements in the pattern.
(default is 1)

▼PD CURVES Menu

Analysis > POST_DYNAMIC > PD_CURVES

This menu contains commands to define, delete, and list curves used in the advanced dynamic analysis.

Figure 10-9 PD Curves Menu



PD_CURTYP

Analysis > POST_DYNAMIC > PD_CURVES > Curve Type

The PD_CURTYP command defines the type of a curve to be used in a post dynamic analysis.

Curve label
Curve label. (limit is 1000)
(default is the highest post dynamics curve label defined + 1)

Type of the curve
Type of the curve.

= 0: Time	time dependent curve (defined by points)
= 1: Frq	frequency dependent curve
= 2: Harmonic	time dependent curve (harmonic)
	(default is time, or last defined value)

Excitation type
Excitation type.

= 0: Force/pressure	concentrated forces or pressure
= 1: Uniform base	uniform base excitation
= 2: Multi base	multi-base excitation
	(default is force/pressure, or last defined value)

PD_CURDEF

Analysis > POST_DYNAMIC > PD_CURVES > Define

The PD_CURDEF command defines a curve to be used in a postdynamic analysis. The syntax and prompts depend on the type of curve specified by the PD_CURTYP (Analysis, Post_Dynamic, PD_Curves, Curve Type) command. The curve data can be input interactively, or read from an external file. Commands in the Display, XY_Plots menu can be used to plot and list the curve. The independent variable values (time/frequency) must be given in ascending order.

Time- or frequency-dependent curve defined by points

Curve label
Curve label. (limit is 1000)
(default is the highest post dynamics curve label defined + 1)

Starting point number (0: external file)
Starting location index to define data points. Enter 0 to read curve data from an external file. Spaces, commas, and tabs in the file are treated as delimiters.
(default is the highest point number defined for the curve + 1)

External file name for curve info

Name of file containing data. Prompted only if starting point number is set to 0. The file must be an ASCII file formatted in one of two ways:

a. The first row lists the number of data points to be read, 0 (for the format type), and the increment value of the independent variable. The dependent variable values are listed one value per row from second row onwards. The independent variable will start from zero.

b. The first row lists the number of data points to be read and 1 (for the format type). The independent and dependent variable values are then listed one pair per row from the second row onwards.

(default is problem_name.xcr)

The following entries are not prompted if data are read from an external file.

Time at point (i)

Time at the ith point for time-dependent curves or frequency at the ith point for frequency-dependent curves.

Curve value at point (i)

Value of the curve at the ith point. (pairs are prompted for i=st to i=st+10)

Notes

1. The starting point number argument allows for modifying existing curve points. Use the default value to define additional points on an existing curve.
2. Up to 5000 points can be defined on a curve.
3. Up to 10 pairs can be defined each time the command is issued. The command terminates if two equal consecutive time values are specified or if a blank is entered for the time or frequency value.

Time-dependent harmonic curve

Curve label

Curve label.
(default is the highest post dynamics curve label defined + 1)

Starting time for harmonic curve

Starting time for harmonic curve.
(default is 0.0)

Ending time for harmonic curve

Ending time for harmonic curve.

Amplitude of sine function

Amplitude of sine function.
(default is 0.0)

Frequency of sine function

Frequency of sine function. (radian/second)
(default is 0.0)

Phase angle of sine function
Phase angle of sine function. (radian)
(default is 0.0)

Amplitude of cosine function
Amplitude of cosine function.
(default is 0.0)

Frequency of cosine function
Frequency of cosine function. (radian/second)
(default is 0.0)

Phase angle of cosine function
Phase angle of cosine function. (radian)
(default is 0.0)

Constant of exponential function
Constant of exponential function.
(default is 0.0)

PD_CURDEL

Analysis > POST_DYNAMIC > PD_CURVES > Delete

The PD_CURDEL command deletes a pattern of curves used in a post dynamic analysis that was defined by the PD_CURDEF (Analysis, POST_DYNAMIC, PD_CURVES, Define) command.

First curve label
Beginning post dynamic curve in the pattern.

Last curve label
Ending post dynamic curve in the pattern.
(default is first curve label)

Increment
Increment between post dynamic curves in the pattern.
(default is 1)

PD_CURLIST

Analysis > POST_DYNAMIC > PD_CURVES > List

The PD_CURLIST command lists a post dynamic curve defined by the PD_CURDEF (Analysis, POST_DYNAMIC, PD_CURVES, Define) command. For a harmonic curve, the command lists the constants defining the harmonic function.

Curve label
Curve label.
(default is the highest post dynamics curve label defined)

First point




First point on the curve to be listed. (not prompted for a harmonic curve)
(default is 1)

Last point

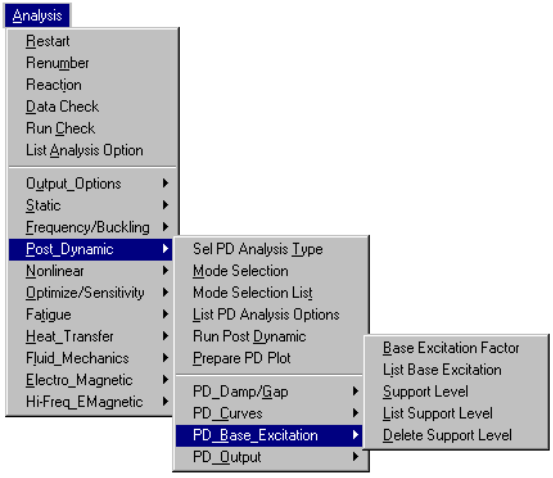
Last point on the curve to be listed. (not prompted for a harmonic curve)
(default is the highest point defined)

▼PD_BASE_EXCITATION Menu

Analysis > POST_DYNAMIC > PD_BASE_EXCITATION

This menu contains commands to specify, list, and delete uniform and multi-base motions.

Figure 10-11 PD Base Excitation Menu



PD_BASEFAC

Analysis > POST_DYNAMIC > PD_BASE_EXCITATION > Base Excitation Factor

The PD_BASEFAC command defines base excitations. All input is interpreted in the active coordinate system.

Curve label

Label of time or frequency curve associated with the base motion.
(default is 1)

Base excitation type
Type of uniform or multi-base excitation.

= 0: Velocity	velocity
= 1: Acceleration	acceleration
= 2: Displacement	displacement
	(default is velocity, or last defined value)

The following three arguments are only valid for uniform base excitation.

Base_cur multiplier X comp
Base curve multiplier for translation in the X-direction.
(default is 0.0)

Base_cur multiplier Y comp
Base curve multiplier for translation in the Y-direction.
(default is 0.0)

Base_cur multiplier Z comp
Base curve multiplier for translation in the Z-direction.
(default is 0.0)

Phase angle (degrees)
Phase angle of the base excitation in degrees. Used only in harmonic analysis for all types of base motion.
(default is 0.0)

The following arguments are only valid for uniform rotational base excitation.

Base_cur multiplier RX comp
Base curve multiplier for rotation in the X-direction.
(default is 0.0)

Base_cur multiplier RY comp
Base curve multiplier for rotation in the Y-direction.
(default is 0.0)

Base_cur multiplier RZ comp
Base curve multiplier for rotation in the Z-direction.
(default is 0.0)

X-Coordinate of center of rotation
X-coordinate of the center of rotation in the active coordinate system.
(default is 0.0)

Y-Coordinate of center of rotation
Y-coordinate of the center of rotation in the active coordinate system.
(default is 0.0)

Z-Coordinate of center of rotation
Z-coordinate of the center of rotation in the active coordinate system.
(default is 0.0)




Notes

1. The form of the harmonic excitation is:

   Sine/Cosine ($\omega t + \phi$) where ω is the frequency, t is the time, and ϕ is the phase in radians ($\phi = 2\pi/360$).
2. The specified base motion is considered in the active coordinate system which can be global or local but must be Cartesian.

PD_BSLIST

Analysis > POST_DYNAMIC > PD_BASE_EXCITATION > List Base Excitation

The PD_BSLIST command lists base excitations defined by the PD_BASEFAC (Analysis, POST_DYNAMIC, PD_BASE_EXCITATION, Base Excitation Factor) command.

PD_SPPRT

Analysis > POST_DYNAMIC > PD_BASE_EXCITATION > Support Level

The PD_SPPRT command applies a base excitation to a pattern of nodes. The excitation in the specified direction is defined by the active post dynamic curve specified by the PD_CURDEF command and a multiplier. The command can be repeatedly used to define multiple base motions for other patterns of nodes. The type of base excitation (displacement, velocity, or acceleration) is specified by the PD_BASEFAC (Analysis, POST_DYNAMIC, PD_BASE_EXCITATION, Base Excitation Factor) command.

Support level number

Support level, a label assigned to the motion to be defined. Maximum number allowed is 1000.
(default is the highest support defined + 1)

Beginning node

Beginning node in the pattern.

Direction

Input direction of excitation.

= UX	global X direction
= UY	global Y direction
= UZ	global Z direction
= RX	rotation about global X direction
= RY	rotation about global Y direction
= RZ	rotation about global Z direction

Curve multiplication factor

Multiplier to the active curve to scale the magnitude of displacement.




Ending node

Ending node in the pattern.
(default is beginning node)

Increment

Increment between nodes in the pattern.
(default is 1)

Notes

1. The type of excitation (i.e. displacement, velocity or acceleration) should be specified by the PD_BASEFAC (Analysis, Post_Dynamic, PD_Base_Excitation Factor) command.
2. The specified base motion is considered in the active coordinate system which may be global or local but must be Cartesian.
3. Base motion assignments and the type of dynamic analysis to be performed must be specified prior to running the frequency program to calculate frequencies and mode shapes.
4. A number of Support Levels may be associated with one curve.
5. Only one base excitation is considered for a given node and input component. Uniform base excitation however may be added on top of the defined motion using the PD_BASEFAC (Analysis, POST_DYNAMIC, PD_BASE_EXCITATION, Base Excitation Factor) command.
6. For Response Spectra Analysis, all base nodes associated with a multi-base motion curve, should be excited in the same direction.
7. For Random Vibration Analysis, all the base nodes associated with a multi-base motion curve, must have the same curve multiplier.
8. All the input for the support levels must be given prior to running DSTAR, any modifications after that are ignored by ASTAR, unless DSTAR is run again.

Example:

```
ACTSET,TC,85,
PD_BASE,1;
PD_SPPRT,3,1,2,1.5,5,1
PD_SPPRT,12,11,3,1.2,31,10
```

The first command activates curve 85. The second command specifies the excitation type as acceleration. The third command defines support level 3 with base excitation for nodes 1 through 5 in the global Y direction. The magnitude of the motion is determined by multiplying curve 85 by 1.5. The fourth command defines support level 12 with base excitation for nodes 11, 21 and 31 in the global Z direction. The magnitude of the motion is determined by multiplying curve 85 by 1.2.




PD_SPPRTLIS

Analysis > POST_DYNAMIC > PD_BASE_EXCITATION > List Support Level

The PD_SPPRTLIS command lists a pattern of base motions previously defined by the PD_SPPRT (Analysis, POST_DYNAMIC, PD_BASE_EXCITATION, Support Level) command.

Beginning support
Beginning support level in the pattern.

Ending support
Ending support level in the pattern.
(default is the highest defined support level)

Increment
Increment between support levels.
(default is 1)

Example: `PD_SPPRTLIS, 1, 5, 1`

This command lists all information defined by the PD_SPPRT command for support levels 1 through 5.

PD_SPPRTDEL

Analysis > POST_DYNAMIC > PD_BASE_EXCITATION > Delete Support Level

The PD_SPPRTDEL command deletes a pattern of base motions previously defined by the PD_SPPRT (Analysis, POST_DYNAMIC, PD_BASE_EXCITATION, Support Level) command.

Beginning support
Beginning support level in the pattern.

Ending support
Ending support level in the pattern.
(default is the beginning support)

Increment
Increment between support levels.
(default is 1)

Example: `PD_SPPRTDEL, 1, 5, 2`

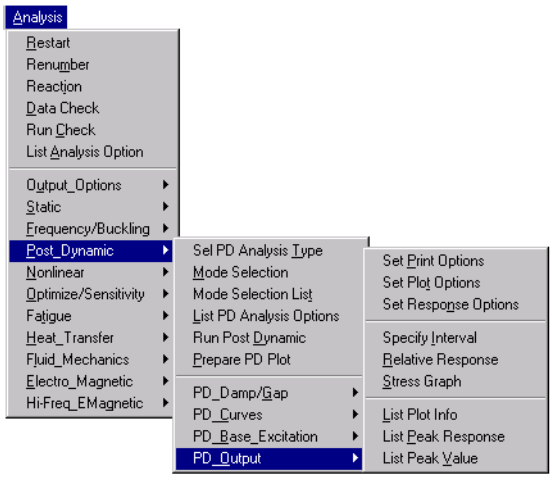
This command deletes base excitation assignments made by the command PD_SPPRT for levels 1, 3 and 5.

▼PD OUTPUT Menu

Analysis > POST_DYNAMIC > PD_OUTPUT

This menu contains commands to specify output options for advanced dynamics analysis.

Figure 10-12 PD Output Menu



PD_PRINT

Analysis > POST_DYNAMIC > PD_OUTPUT > Set Print Options

The PD_PRINT command defines response printout options in the output file.

Displacement print
Flag for displacement printout.

= 1: On	print displacements
= 0: Off	do not print displacements *(default is off)*

Velocity print
Flag for velocity printout.

= 1: On	print velocity
= 0: Off	do not print velocity *(default is off)*

Acceleration print
Flag for acceleration printout.

= 1: On	print acceleration
= 0: Off	do not print acceleration *(default is off)*

Phase angle print

Phase angle printout.

= 0: Off	suppress phase angle printout
= 1: On	prepare phase angle printout used for harmonic (frequency response) analysis only *(default is off)*

Missing mass correction print

Flag for missing mass correction factor printout.

= 0: Off	no print
= 1: On	print correction factors applied to acceleration (see note below) *(default is off)*

Response print interval

Response print interval.
(default is 1)

The next three arguments are prompted for time history, harmonic and Random Vibration analyses only.

Beginning step for stress print

Beginning solution step for stress printout.
(default is 1)

Ending step for stress print

Ending solution step for stress printout.
(default is 1)

Increment

Increment between solution steps.
(default is 1)

Notes

1. For random vibration analysis, these flags control printouts of response power spectral densities. The R.M.S. displacements, velocities, and accelerations are printed by default.
2. Response and stress printout options are only applicable to modal time-history, harmonic and P.S.D. of Random Vibration analyses. For random and response spectra analyses, the R.M.S. stresses are available by default through the R_STRESS (Analysis, Static, Run Stress Analysis) command.
3. The missing mass correction factor accounts for the contribution of higher frequency modes not considered in the analysis. This factor is printed in the output file, and should be used to correct the acceleration and stress responses by multiplying the corresponding values by this factor.
4. Use the PD_PLTLIST (Analysis, POST_DYNAMIC, PD_OUTPUT, List Plot Info) command to list all requested plots.

PD_PLOT

Analysis > POST_DYNAMIC > PD_OUTPUT > Set Plot Options

The PD_PLOT command is a preprocessing command that must be issued before running ASTAR to select the solution steps at which the nodal response is to be stored in the database. The nodal response includes deformation, velocity, and acceleration. If stresses are calculated, nodal results will also be available at the specified steps. Five patterns of solution steps may be specified.

The PD_NRESP (Analysis, POST_DYNAMIC, PD_OUTPUT, Set Response Options) command can be used to select the nodes for which the response is desired at all solution steps.

Set (i) beginning step
Starting solution step for the ith pattern.

Set (i) ending step
Ending solution step for the ith pattern.

Set (i) step increment
Increment between solution steps for the ith pattern.
(default is 1)

Note

Use the PD_PLTLIST (Analysis, POST_DYNAMIC, PD_OUTPUT, List Plot Info) command to list all requested plot and output information including information requested by this command.

PD_NRESP

Analysis > POST_DYNAMIC > PD_OUTPUT > Set Response Options

The PD_NRESP command is a preprocessing command that must be issued before running ASTAR to request storing all response information for the selected nodes. The response at the selected nodes is stored at all steps in the database for later xy-plotting. A maximum of 50 nodes can be selected. The command continues to prompt for more nodes until location 50 is filled, or a null entry for the node number is entered.

Starting location
Starting location.
(default is highest location defined +1)

Node start
Node number at the “start” location.
(default is the previously defined node, if any)

Node start + (i)
Node number at the “start+1” location.
(default is the previously defined node, if any)

Notes

1. The command may be used to change the node number stored in a given

location.

2. Use the PD_PLTLIST (Analysis, POST_DYNAMIC, PD_OUTPUT, List Plot Info) command to list nodal response requests.

PD_PLTINT

Analysis > POST_DYNAMIC > PD_OUTPUT > Specify Interval

The PD_PLTINT command specifies the pattern of solution steps for the graphing of results requested by the PD_NRESP (Analysis, POST_DYNAMIC, PD_OUTPUT, Set Response Options) and PD_SXYSET (Analysis, POST_DYNAMIC, PD_OUTPUT, Stress Graph) commands. If this command is not issued, the xy-plot files will contain all solution steps.

Beginning step
Beginning solution step in the pattern.
(default is 1)

Ending step
Ending solution step in the pattern.
(default is 1)

Step increment
Increment between solution steps in the pattern.
(default is 1)

Note
The PD_PLTINT command can be used before or after running the analysis. If it is used before running the analysis, the plot file will only contain data for the specified steps. You can use this command after running the analysis to select the desired steps for plotting.

PD_RELRESP

Analysis > POST_DYNAMIC > PD_OUTPUT > Relative Response

The PD_RELRESP requests the calculation of Relative Response (displacement, velocity, and acceleration) at the specified nodes. A reference node is specified for each node at which the Relative Response is requested.

Node (i)
Ith node number at which response is desired. (i=1, 2,.....,10)

Reference node (i)
Reference node used to calculate the relative response of node i. (i=1, 2,....,10)

Notes
1. Response of “node (i)” is calculated relative to “reference node (i)”.
2. Up to 10 sets can be specified by this command.

3. Repeating this command overwrites all previous assignments.
4. A value of zero for the reference node results in evaluating the response relative to the uniform base.
5. To obtain the Relative Response for the assigned nodes, issue the PD_PREPARE command afterward.

Example: `PD_RELRESP, 3, 5, 15, 0, 3, 2`

Evaluate the response of node 3 relative to node 5, response of node 15 relative to the Uniform Base, and response of node 3 relative to node 2.

PD_SXYSET

Analysis > POST_DYNAMIC > PD_OUTPUT > Stress Graph

The PD_SXYSET command requests the ASTAR module to generate a file that records a stress component for a specified element versus time or frequency depending on the analysis type. Forces and moments can also be requested for 1D elements such as beams and pipes. The command must be issued prior to performing stress analysis. You can use this command to delete a request for a stress component.

Graph number

Graph number. (limit is 10)
(default is the highest graph number defined + 1)

Element label

Element label. Enter zero (0) to delete a request for a stress component.

Direction flag

Stress, force or moment component. (valid entries are 1 through 6)

= 1: Fr	normal force
= 2: Vs	shear in the s-direction
= 3: Vt	shear in the t-direction (for beam, pipe, and elbow elements)
= 4: Tr	torsion
= 5: Ms	moment in the s-direction
= 6: Mt	moment in the t-direction

Force/Moment location for plot (i)

Location of force or moment defined by local node number. (Refer to notes below.) (Used for 1D elements only.)

= Element end (i) element end (i)
= Element end (j) element end (j)
= Element center element center

Notes

1. Direction is an integer between 1 & 6 defining the direction of a stress, force or moment component. Stresses are considered at the center of elements. Values from 1 to 3 refer to normal stresses in x, y, and z directions, and from 4 to 6 refer to shear stresses tau(xy), tau(xz), and tau(yz), respectively. For 1D elements, forces or moments can be requested. Components are specified by their direction as follows:

    1. Fr = Normal force
    2. Vs = Shear in the s-direction
    3. Vt = Shear in the t-direction (for beam, pipe and elbow elements)
    4. Tr = Torsion element
    5. Ms = Moment in the s-direction
    6. Mt = Moment in the t-direction

    The location of forces or moments for 1D elements is specified by the local node number (i.e., the order of the node in the element definition obtained by the ELIST command).

    = 1 Element end i
    = 2 Element end j
    = 3 Element center (for elbow elements only)

    For each stress run, for one element and one direction, a plot can be requested at one location only.

2. Plots of different components of stress can be requested for the same element. For stress plots at different elements, the element labels must be input in ascending order.
3. Forces or moments can only be requested at one node for a specified element.
4. To delete a previously requested graph, enter “0” for “element label”. To properly identify graph numbers, use the PD_PLTLIST (Analysis, POST_DYNAMIC, PD_OUTPUT, List Plot Info) command first.

PD_PLTLIST

Analysis > POST_DYNAMIC > PD_OUTPUT > List Plot Info

The PD_PLTLIST command lists all information related to post dynamic plots.

PD_MAXMIN

Analysis > POST_DYNAMIC > PD_OUTPUT > List Peak Response

The PD_MAXMIN command lists the highest response values of a set of nodes in a given time (or frequency) interval. A maximum of 10 nodes from a set of 300 nodes can be listed each time. This command must be issued after running postdynamic analysis.

Note that by limiting the number of nodes in the set to the number of nodes to be listed, the maximum response attained by each node with respect to time (or frequency) and its time (or frequency) of occurrence will be listed (1 to 10 nodes at a time). On the other hand, by limiting the time (or frequency) range, nodal response at a particular step of solution can be obtained. The PD_PREPARE (Analysis, POST_DYNAMIC, Prepare PD Plot) command must be issued later to prepare files for listing extreme values.

Type

Flag defining type of maximums.

= Dis	displacement
= Vel	velocity
= Acc	acceleration

(default is dis)

Maximum direction

Flag defining direction of maximums.

= X-trans	X-translation
= Y-trans	Y-translation
= Z-trans	Z-translation
= X-rot	X-rotation
= Y-rot	Y-rotation
= Z-rot	Z-rotation

(default is X-trans)

Number of maximums

Total number of nodes with maximum response values to be listed. (limit is 10 nodes)
(default is 10)

Starting node label

Beginning node of the pattern of nodes to be searched.
(default is 1)

Ending node label

Ending node of the pattern of nodes to be searched. Extremes will be searched among a pattern of nodes starting from 'starting node label' and ending at 'ending node label'.
(default is starting node label + 299)

Starting time/frequency
Starting value of time or frequency-range to be searched.
(default is lowest value available in database)

Ending time/frequency
Ending value of time or frequency-range to be searched. Default is to search the whole range available.
(default is highest value available in database)

Notes

1. In time-history analysis, response is a function of time.
2. In random vibrations, power spectral densities are based on the exciting frequency.
3. In frequency response, the amplitude of harmonic motion is based on the exciting frequency.
4. In the case of random vibration or frequency response, the frequency range limits are assumed to have the same units as the excitation curves.
5. The listed extremes are always relative to the uniform base motion (if applicable) regardless of the type of response requested in the PD_ATYPE (Analysis, Post Dynamic, Sel PD Analysis Type) command.

PD_MAXLIST

Analysis > POST_DYNAMIC > PD_OUTPUT > List Peak Value

The PD_MAXLIST command lists the extreme values of the latest data that was obtained by the PD_MAXMIN (Analysis, POST_DYNAMIC, PD OUTPUT, List Peak Resp) and PD_PREPARE (Analysis, POST_DYNAMIC, Prepare PD Plot) commands.

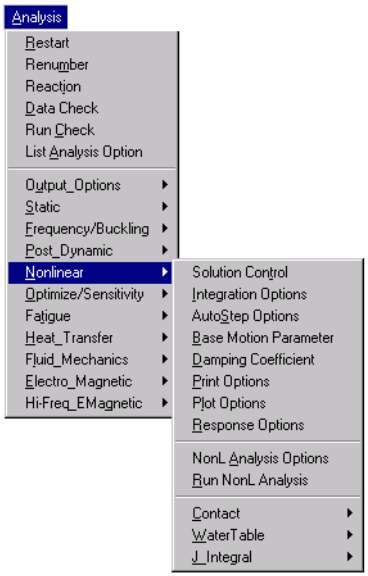



NONLINEAR Menu

Analysis > NONLINEAR

This menu contains commands related to NSTAR, the nonlinear structural analysis module. NSTAR calculates the structural response for nonlinear static and dynamic problems. Refer to the COSMOS/M Advanced Modules User Guide, Part 1 for details.

Figure 10-13 Nonlinear Menu



NL_CONTROL

Analysis > NONLINEAR > Solution Control

The NL_CONTROL command specifies the numerical procedure to be used in nonlinear analysis. The command sets the control technique and the iteration scheme to be used in the subsequent analysis.

Control technique

Control technique flag.

= 0: Force	Force control (static and dynamic analyses)
= 1: Disp	Displacement control (static analysis)
= 2: Arc len	Riks Arc-length control (static analysis)
	(default is force)

Iterative technique

Iterative method flag.

= 0: MNR	MNR iteration (static and dynamic analyses)
= 1: NR	NR iterations (static and dynamic analyses)
= 2: BFGS	BFGS iterations (static analysis without gaps, with Force control only)
	(default is MNR)

Line search flag

Line search flag. Used in the BFGS method only.

= 0: No	do not use line search
= 1: Yes	use line search
	(default is no)

Search tolerance

Search tolerance, used in the BFGS method only.
(default is 0.5)

Node

The node number associated with the degree of freedom to be controlled during the solution process. (for Displacement control only)

Displacement label

Displacement component. Indicates the direction of the controlled DOF (for Displacement control only). Valid entries are:

= UX	displacement in the Global X-direction
= UY	displacement in the Global Y-direction
= UZ	displacement in the Global Z-direction
= RX	rotation about the Global X-direction
= RY	rotation about the Global Y-direction
= RZ	rotation about the Global Z-direction
	(default is UY)

Maximum load parameter

The maximum load-pattern multiplier at which the analysis is to be terminated if exceeded (approximate value). (for Riks Arc-Length control only)
(default is 1.0E+8)

Maximum displacement/rotation

The maximum value of any DOF at which the analysis is to be terminated if exceeded (approximate value). (for Riks Arc-Length control only)
(default is 1.0E-3)

Maximum number of arc steps

The maximum number of arc steps at which analysis is to be terminated if exceeded. (for Riks Arc-Length control only)
(default is 50)

Average number of iterations per step

The desired average number of iterations for each arc step used in adjusting the length of the arc steps. (for Riks Arc-Length control only)
(default is 5)

Initial load parameter

The initial trial load multiplier used to calculate the first arc step. (for Riks Arc-Length control only)
(default is 1.0)

Unloading check

This flag sets the criterion for determining the state of unloading during the solution process. (for Riks Arc-Length control only)

= 0: Determinant	use the sign of the determinant of the system stiffness matrix (negative indicates unloading)
= 1: Energy	use the sign of the incremental work (negative indicates unloading)
= 2: Either	use the sign of the determinant and the incremental work (unloading if any of these signs is negative)
= 3: Both	use the sign of the determinant and the incremental work (unloading if both signs are negative) *(default is det)*

Adjustment coefficient

This coefficient is used in adjusting the automatic calculation of the arc-step length increment during the analysis. (for Riks Arc-Length control only)
(default is 0.5)

The NR, MNR and BFGS iterative methods are effective only if equilibrium iterations are performed.

2. The user must define either “maximum load parameter” or “maximum displacement/rotation” for the arc-length technique. Defaults should not be used for both of them. Otherwise, an error message will be issued by the program “Max load parameter or displ/rotation must be defined”.

Example 1:
```
Let nonlinear static analysis be active (A_NONLINEAR command).
NL_CONTROL, 0, 2, 1, 0.5,
```
This command activates the Force control technique with BFGS iterations including the line search option with a search tolerance of 0.5.

Example 2:
```
NL_CONTROL, 1, 1, 5, UY,
```
This command activates the Displacement control technique with NR iterations. The controlled DOF is the Y displacement of node 5.

Example 3:
```
NL_CONTROL, 2, 0, 100, 30, 50, 5, 10, 0, 0.5
```
This command activates the Riks Arc-Length control technique with MNR iterations. The Arc-Length control parameters are set to:

Max. load parameter = 100.0
Max. displacement/rotation = 30.0
Max. number of arc steps = 50
Desired average number of iterations/step = 5

Initial load parameter = 10.0
Unloading check flag = 0
Arc-Length step adjustment coefficient = 0.5

To select a control method:

1 Click **Analysis, Nonlinear, NonL Analysis** Options.

2 From the **Analysis** option menu, select **Static** or **Dynamic**.

3 Set other entries as desired and click **OK**.

4 Click **Analysis, Nonlinear, Solution Control**.

For dynamic analysis, only the **Force Control** method is available.

5 Select the desired control method and the iteration scheme.

6 Click **OK**. When you run analysis, the specified settings will be used if supported.

NL_INTGR

Analysis > NONLINEAR > Integration Options

The NL_INTGR command sets the direct time integration parameters for use in subsequent nonlinear dynamic analysis.

Integration flag

Direct time integration method.

= 0: Newmark	Newmark-Beta Method
= 1: W-Theta	Wilson-Theta Method
= 2: Central difference	Modified Central Difference *(default is Newmark-Beta)*

Gamma/Theta value

First integration parameter.
Theta used in the Wilson-Theta Method.
(default is 1.4)

Gamma used in the Newmark Method.
(default is 0.5)

Beta value

Second integration parameter, Beta used in the Newmark Method, only.
(default is 0.25)

(Number of sub-steps) (Modified Central Difference method only)

Number of sub-steps (isub) used with the Central Difference method. The actual time step used is calculated by dividing the time increment specified in the TIMES command (LoadsBC > LOAD OPTIONS > Time Parameter) by the

number of sub-steps. Since the Central Difference method is conditionally stable, the program calculates the critical time step. If the actual time calculated using isub is larger than the calculated critical step, the program will issue a message and stop.

(Tolerance)
Tolerance for convergence used with the Central Difference method.

Example: `NL_INTGR, 0, 0.5, 0.26`
This command specifies the Newmark-Beta method for direct time integration with Beta as 0.26, and Gamma as 0.5.

NL_AUTOSTEP

Analysis > NONLINEAR > AutoStep Options

The NL_AUTOSTEP command is used to activate the automatic stepping option to be used in nonlinear structural analysis (NSTAR). This option works with both static and dynamic analyses and supports all control techniques.

Automatic step
Automatic stepping flag.

= 0: Off	do not use automatic stepping
= 1: On	use automatic stepping *(default is off)*

If "On" is selected; the following prompts are used:

Minimum step increment
(default is 1.0E-8)

Maximum step increment
(Default is the final time defined in the TIMES command for the Force and Displacement Control methods. For the Arc-Length control method, default is equal to the maximum displacement or rotation defined in the Arc-length input, in the NL_CONTROL command.)

Maximum number of step adjustments
(default is 5)

Note

The program stops the analysis if:

a. The number of step-size adjustments for any step exceeds the maximum number of step adjustments.
b. The step increment required for convergence becomes smaller than 'minimum step increment'.

✍ In any case, the user may make changes (tolerance, step size, iteration method, etc.) and use the RESTART option to continue.

Example: `NL_AUTO, 1, 0.1, 0.5, 10`

This command activates the adaptive automatic stepping option in nonlinear structural analysis and defines a minimum step increment of 0.1, and a maximum step increment of 0.5. The allowable number of step-size adjustments is set to 10.

NL_BASE

Analysis > NONLINEAR > Base Motion Parameter

The NL_BASE command is used to activate the uniform base motion acceleration option to be used in nonlinear dynamic structural analysis using NSTAR. Actual values of acceleration are specified by a multiplier and associated time curve.

Base motion acceleration flag

Base motion acceleration flag.

= 0: Off	base motion not considered
= 1: On	base motion considered
	(default is off)

Base accel multiplier in X-direction

Base acceleration multiplier in the global X-direction.
(default is 1.0)

Base accel multiplier in Y-direction

Base acceleration multiplier in the global Y-direction.
(default is 1.0)

Base accel multiplier in Z-direction

Base acceleration multiplier in the global Z-direction.
(default is 1.0)

Associated time curve in X-direction

Time curve associated with acceleration in the global X-direction.
(default is the active time curve)

Associated time curve in Y-direction

Time curve associated with acceleration in the global Y-direction.
(default is the active time curve)

Associated time curve in Z-direction

Time curve associated with acceleration in the global Z-direction.
(default is the active time curve)

Example: `NL_BASE, 1, 1, -2, 3, 1, 5, 8,`

This command activates the base motion option for subsequent nonlinear dynamic analysis. The value of the base acceleration at any time is the value read or interpolated from the associated curve multiplied by the corresponding multiplier. In this example, the time-history of

acceleration in the X-direction is 1.0 multiplied by time curve 1, acceleration in the Y-direction is -2.0 multiplied by time curve 5, and acceleration in the Z-direction is 3.0 multiplied by time curve 8.

NL_RDAMP

Analysis > NONLINEAR > Damping Coefficient

The NL_RDAMP command is used to activate the proportional Rayleigh damping option to be used in subsequent nonlinear dynamic analysis. The actual damping matrix is calculated by multiplying the mass and stiffness matrices by the specified coefficients and adding the resulting matrices.

Rayleigh damping

Rayleigh damping flag.

= 0: Off	no proportional damping
= 1: On	consider proportional damping
	(default is on)

Damping coefficient for stiffness

Damping coefficient associated with the stiffness matrix.
(default is 0.01)

Damping coefficient for mass matrix

Damping coefficient associated with the mass matrix.
(default is 0.0)

Note

In this option, the damping matrix [C] of the system is assumed to take the form:

[C] = alpha * [K] + beta * [M]

where:

[K] and [M] are the system stiffness and mass matrices respectively.

Example: `NL_RDAMP, 1, 0.01, 0.1,`

This command activates the Rayleigh damping option during nonlinear dynamic analysis and defines a value of 0.01 to be associated with [K] and a value of 0.1 to be associated with [M] to define [C].

NL_PRINT

Analysis > NONLINEAR > Print Options

The NL_PRINT command controls the information that NSTAR writes in the output file (problem-name.out). This command has no effect on the information for plotting and listing within GEOSTAR. Use this command to limit the size of the output file for large problems or those with many time steps.

Displacement print flag
Flag for displacement printout.
= 0 no printout
= N print at every N solution steps
(default is 1)

Velocity print flag
Flag for velocity printout.
= 0: Off no printout
= 1: On print at solution steps specified by “displacement print flag”
(default is off)

Acceleration flag
Flag for acceleration printout.
= 0: Off no printout
= 1: On print at solution steps specified by “displacement print flag”
(default is off)

Stiffness matrix print flag
Flag for stiffness matrix and equation number printout.
= 0: No printout no printout
= 1: Stiffness matrixprint stiffness matrix in the output file
= 2: Three matrices stiff, mass & geo stiff
(default is no printout)

Input print flag
Flag for detailed input printout.
= 0: Off no printout
= 1: On print nodal coordinates, element connectivity, material and real constant sets, prescribed displacements, and applied loads including the effect of surface pressures
(default is off)

Output print flag
Overwrite/append flag for analysis output.
= 0: Overwrite overwrite previous output unless restart option is on
= 1: Append append to previous output
(default is overwrite)

Concrete model parameter
Concrete model parameter.
= 0: Damage factordamage factor
= 1: Yield factor yield factor
(default is damage factor)

Example:
```
NL_PRINT, 2, 0, 1;
```
This command instructs the program to print displacements and accelerations every other solution step in the output file (problem-name.out). Velocities are not printed. All other defaults are accepted.

NL_PLOT

Analysis > NONLINEAR > Plot Options

The NL_PLOT command sets the solution steps at which the response at all nodes is to be stored in the database for plotting and listing purposes. The response includes displacements and stresses in the case of static analysis, and displacements, stresses, velocities, and accelerations in the case of dynamic analysis. Five patterns of solution steps can be specified.

The NL_NRESP command can be used to specify nodes for which the response is desired at all solution steps for subsequent xy-plotting.

The NL_PLOT command must be issued before running the analysis.

Set (i) beginning step
Starting solution step for the ith pattern.

Set (i) ending step
Ending solution step for the ith pattern.
(default is the set (i) beginning step)

Set (i) step increment
Increment between solution steps for the ith pattern.
(default is 1)

NL_NRESP

Analysis > NONLINEAR > Response Options

The NL_NRESP command instructs the program to store response information at all solution steps for the specified nodes. The information can be used during post-processing to generate response graphs. The response include displacements, velocities, accelerations, reaction forces, and load factors. Load factors are only available for the displacement and arc-length control methods. A maximum of 50 nodes can be selected. The command continues to prompt for more nodes until location 50 is filled, or a null entry for the node number is entered.

The NL_NRESP command must be issued before running the analysis.

Starting location
Starting location.
(default is highest location defined + 1)

Node start
Node number at the “start” location.
(default is the previously defined node, if any)

Node start+1
Node number at the "start+1" location.
(default is the previously defined node, if any)

Note
The command may be used to change the node number stored in a given location.

A_NONLINEAR

Analysis > NONLINEAR > NonL Analysis Options

The A_NONLINEAR command sets the options to be used in performing a subsequent nonlinear analysis through the R_NONLINEAR (Analysis, Nonlinear, Run NonL Analysis) command.

Analysis option
Nonlinear analysis option.

= S	static analysis
= D	dynamic analysis
	(default is S)

Steps between reforming stiffness
Number of solution steps between reforming the stiffness matrix.
(default is 1)

Steps between equilibrium iterations
Number of solution steps between equilibrium iterations.
(default is 1)

Maximum equilibrium iterations
Maximum number of equilibrium iterations at each step.
(default is 20)

Convergence tolerance
Relative displacement tolerance used for equilibrium convergence.
(default is 0.001)

Save information flag
Save information for use by frequency or buckling analysis.

= 0: None	do not save
= 1: Frequency	save information for frequency analysis
= 2: Buckling	save information for buckling analysis
	(default is none)

Loading flag
Special loading flag. Any one character can be assigned. Two or three characters can be assigned in any combination of C (centrifugal), G (gravity) and T (thermal).

= N	do not include special loading
= C	include centrifugal loading
= G	include gravity loading




= T	include thermal loading
= CG	include centrifugal and gravity loading
= GT	include gravity and thermal loading
= CGT	include centrifugal, gravity, and thermal loading
	(default is N)

Displacement dependent load flag

Flag for deformation-dependent loading.

= 0: No	direction and area considered for pressure loading are constant (conservative loading)
= 1: Yes	direction and area change based on the deformed element coordinates (non conservative loading)
	(default is no)

End moment flag for shell elements

Flag for end moment calculation for shell elements under pressure loading.

= 0: Off	do not apply fixed end moments
= 1: On	apply fixed end moments
	(default is off)

Local constraint penalty stiffness value

Local constraint penalty stiffness value to be used in the calculation of the penalty matrix.
(default is 1.0E10)

Local constraint tolerance

Constraint tolerance for local prescribed displacements.
(default is 0.001)

Creep/Plasticity strain increment tolerance

Creep/Plasticity strain increment tolerance (CETOL).
(default is 0.01)

Geometry update flag

Geometry update flag.

= 0: No	do not update geometry
= 1: Yes	update geometry
	(default is no)

Singularity factor

Stiffness singularity elimination factor.

= 1	normal solution
< 1	manipulate stiffness to reduce singularity. See note below.
	(between 0 and 1 inclusive, default is 1)

Apply reaction forces to released Does

Flag to activate or deactivate the application of reaction forces to released Does. Refer to the Numerical Procedures chapter of the NSTAR module in the Advanced Modules manual for more details.

= 0: No	deactivate
= 1: Yes	activate
	(default is no)

Solver option

= 0: Sparse	Use the sparse direct solver.
= 1: Skyline	Use the skyline direct solver. *(default is Sparse)*

Notes

1. The specifications of this command are ignored by NSTAR if no nonlinearities are specified in any of the element groups in the model.
2. Special loadings are considered on top of mechanical loads (forces and pressure) when the relevant flag is activated.
3. The "Geometry update" flag should only be used when it is desired to use the deformed shape of the structure at the final solution step as the original shape (with no strains or stresses) for subsequent analyses.
4. The "Singularity factor" entry can be used to improve convergence of highly nonlinear problems. Acceptable values are: 0.0, 0.5, and 1.0 (default = 1.0). Values less than one (zero for maximum effect) activate a technique which may eliminate the singularity of the structure stiffness due to geometric nonlinearities.

   This technique is useful for problems which involve large deformations and large strains and the nonlinear run stops at some level of loading with "Stiffness Singularities..." message.

   It is recommended to use a factor of less than unity only after the value of unity has been tried, i.e., the normal solution has failed.

   It must be noted that this method is not always effective. In many situations it may lead to convergence difficulties instead of improvements. If the change of factor along with a restart does not help, the solution must be repeated from the beginning.

   It is helpful to realize that this technique can help remove global instability in exchange for some local error which in most cases can be ignored. A lower convergence tolerance can be used to assure accuracy.

   It must also be noted that for cases which involve a sudden out-of-plane buckling, an initial imperfection is required.

Example:

```
A_NONLINEAR, S, 1, 5, 25,,, CG, 0, 0,,,,,,
```

This command specifies nonlinear static analysis with stiffness reformation at every time step, and equilibrium iterations at every fifth time step. The maximum number of equilibrium iterations is set to 25. Default displacement tolerance is used. Gravity and centrifugal loadings are included. Fixed end moments for shell elements under pressure are not considered.




R_NONLINEAR

Analysis > NONLINEAR > Run NonL Analysis

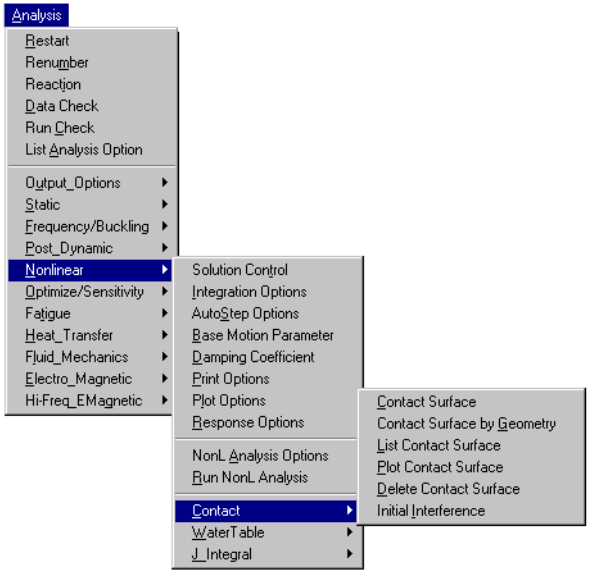
The R_NONLINEAR command performs nonlinear structural analysis using the NSTAR module. Use the A_NONLINEAR (Analysis, NonL Analysis Options) command to activate the desired options before selecting this command. Otherwise, all default options will be used.

▼CONTACT Menu

Analysis > NONLINEAR > CONTACT

Contact can be defined using 2-node GAP elements, or 1-node GAP elements. When using 1-node GAP elements, you can define node-to-line or node-to-surface contact. This menu contains commands related to defining contact lines or surfaces for use with 1-node gap elements.

Figure 10-14 Contact Menu



NL_GS

Analysis > NONLINEAR > CONTACT > Contact Surface

The NL_GS command defines a contact sub-line or sub-surface. A maximum of 3 nodes per line and 9 nodes per surface can be specified. The maximum number of nodes for each contact line or surface is specified in the active GAP element group. The command can be repeatedly used until the combination of sub-surfaces/lines in one group form a continuous or closed area of possible contact.

Use the NL_GSAUTO (**Analysis > NONLINEAR > CONTACT > Contact Surface by Geometry**) command to generate contact surfaces automatically.

Contact surface
Contact line or surface label.
(default is highest defined + 1)

Node (i)
Ith node of line or surface being defined. (i = 1, 2,...., 9)

Notes

1. Each sub-surface is assigned a positive and a negative side. The gap nodes may not exist in the negative side of the surface after solution. Note that graphical display of the positive direction is not shown if the gaps are generated by the NL_GS command. Arrows point towards the positive directions when the NL_GSAUTO command is used.
2. The positive side of the contact line or surface is defined as follows:
   a. For 2D analysis let r be the vector starting at node 1 and ending at node 2. The positive side is in the direction of the vector resulting from the cross product (z x r), where z is a vector in the positive global Z-direction.
   b. For 3D analysis a contact surface is defined by 4 to 9 nodes. Let r and s be defined as shown in the figure below, then the cross product (r x s) points to the positive side of the surface.
   c. In a 3D analysis, contact lines may be specified in (or parallel to) one of the three global planes, XY, YZ, XZ. For lines parallel to the XZ plane, the direction of the positive side is defined by (y x r), for lines parallel to YZ plane the direction in YZ plane by (x x r).
3. For a Gap element group which is to represent node to line or node to surface contact, the gap distance is not required to be input. Contact is assumed to be rigid, and therefore, the only real constant required is the coefficient of friction.
4. The active element group is assigned to all contact sub-lines and sub-surfaces generated. A GAP element group must be active while contact lines or surfaces are defined.
5. Sub-surfaces (or lines) in one group form an overall surface (or line) which may be curved. Thus, each surface (or line) connectivity must be defined such

that its positive side is in agreement with that of the overall surface (or line).

6. The gap elements in the group (each defined by one node) represent nodes which may come in contact with any of the surfaces (or lines) defined for that group.

Example: Consider a contact surface (Q) which is formed by sides of solid or shell elements (or simply defined by fixed nodes). Also, consider nodes i, j, and k to be nodes (may belong to another surface) that may come into contact with surface (Q).

1. Define a1-node GAP element group representing node to surface contact (using the proper options). For this case, each sub-surface is defined by 4 nodes. Activate this element group and the corresponding real constant set.
2. Define 3 gap elements at each of the nodes, i, j and k.
3. Define 8 sub-surfaces such as the surface connecting nodes 1, 6, 7, and 2. Together, these sub-surfaces will form the surface (Q).
4. Nodes i, j and k are in the positive side of the contact surface (based on nodal connectivity) and are forbidden to cross to the negative side during deformation.

Input:

```
EGROUP, 3, GAP, 1, , , 2, 4
EL, 101, PT, 0, 1, i
EL, 102, PT, 0, 1, j
EL, 103, PT, 0, 1, k
NL_GS, 1, 1, 6, 7, 2
NL_GS, 2, 6, 11, 12, 7
NL_GS, 3, 2, 7, 8, 3
NL_GS, 4, 7, 12, 13, 8
NL_GS, 5, 3, 8, 9, 4
NL_GS, 6, 8, 13, 14, 9
NL_GS, 7, 4, 9, 10, 5
NL_GS, 8, 9, 14, 15, 10
```

(Nodes i, j and k must remain in the upper side of contact surface Q, unless they deform beyond the boundaries of the surface. Nodal connectivity used in the NL_GS command specify the upper side as the positive side of the surface Q.)

NL_GSAUTO

Analysis > NONLINEAR > CONTACT > Contact Surface by Geometry

The NL_GSAUTO command generates 1-node gap elements and associated contact lines or surfaces. The user specifies a primary contact entity (curve, surface, or region) and a pattern of target entities (curves, surfaces, or regions). All the geometric entities must have been meshed before issuing this command in order to generate the contact information. The 1-node gap elements are created at every node

associated with the primary contact geometric entity and the gap lines/surfaces are created at every element edge/face on the target entities. The active element group must be the 1-node contact GAP. A flag for the orientation of the contact surfaces is provided and must be accurately specified. The NL_GSLIST command may be used to list the defined contact lines/surfaces.

Source contact entity type
Primary geometric entity type.
= 0: Curve curve
= 1: Surface surface
= 2: Region region
(default is curve)

Source contact entity
Label of primary contact entity.

Target contact entity
Target geometric entity type.
= 0: Curve curve
= 1: Surface surface
= 2: Region region
(default is curve)

Beginning entity
Beginning entity label in the pattern of contact target entities.

Ending entity
Ending entity label in the pattern of contact target entities.

Increment
Increment between entity labels in the pattern of contact target entities.

Face flag (shell only)
Flag to determine the face of shell elements to be used for the target surface.
= 0: Bottom use bottom face (face 6) as the contacting (target) surface
= 1: Top use top face (face 5) as the contacting (target) surface
(default is Bottom)

Note
The NL_GSAUTO command may not be used in cases where the target is a curve meshed with beam elements.

NL_GSLIST

Analysis > NONLINEAR > CONTACT > List Contact Surface

The NL_GSLIST command lists a pattern of contact lines or surfaces. Node connectivity and the associated gap element group are listed for each line/surface.

Beginning contact line or surface
Beginning contact line or surface in the pattern.
(default is 1)

Ending contact line or surface
Ending contact line or surface in the pattern.
(default is the highest contact entity defined)

Increment
Increment between contact lines/surfaces in the pattern.
(default is 1)

Example: `NL_GSLIST;`
This command lists all contact lines and surfaces defined in the model.

NL_GSPLOT

Analysis > NONLINEAR > CONTACT > Plot Contact Surface

The NL_GSPLOT command plots a pattern of contact lines and/or surfaces on the screen. The arrow plotted at the center of the line/surface points towards the positive side. The positive side is the side in which the one-node gap elements are allowed to locate.

Beginning contact line or surface
Beginning contact line or surface.
(default is 1)

Ending contact line or surface
Ending contact line or surface in the pattern.
(default is the highest contact line/surface defined)

Increment
Increment between contact lines/surfaces in the pattern.
(default is 1)

Example: `NL_GSPLOT;`
This command plots all contact lines or surfaces defined in the model.

NL_GSDEL

Analysis > NONLINEAR > CONTACT > Delete Contact Surface

The NL_GSDEL command deletes a pattern of contact lines or surfaces from the database.

Beginning contact line or surface
Beginning contact line or surface in the pattern.

Ending contact line or surface
Ending contact line or surface in the pattern.
(default is beginning contact entity)

Increment
Increment between contact lines/surfaces in the pattern.
(default is 1)

Example: `NL_GSDEL, 10, 20, 2`
This command deletes contact lines/surfaces 10, 12, 14, 16, 18, and 20.

NL_GSINTERF

Analysis > NONLINEAR > CONTACT > Initial Interference

The NL_GSINTERF activates or deactivates performing incremental solution for initial interference in contact problems.

Initial interference check flag
Flag to activate or deactivate incremental solution for initial interference.
= 0: No deactivate
= 1: Yes activate
(default is no)

Note
This flag can be used for problem where:

a. Contact surfaces/lines are used to model an initial interference between two bodies.
b. Besides contact, material/geometric nonlinearities develop due to the initial interference.
c. A step-by-step solution is sought where structures are gradually brought back to equilibrium to give enough time to large displacements and other nonlinearities to develop.
d. External forces do not exist or remain constant throughout the solution.
e. Total number of solution steps is determined by the TIMES command.
f. The higher the nonlinear behavior (Geometric, and/or material), the more steps are required to bring the structures to equilibrium.

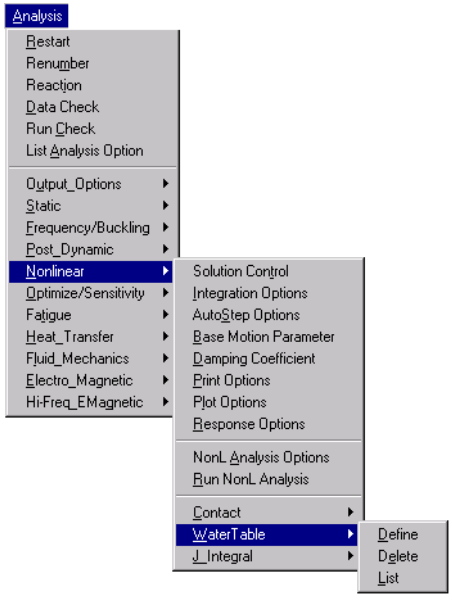



▼WATERTABLE Menu

Analysis > NONLINEAR > WATERTABLE

This menu contains commands to specify, list and delete Water-Motion-Tables (WMT). WMT are used in conjunction with the IMPIPE element.

Figure 10-15 WaterTable Menu



WMTDEF

Analysis > NONLINEAR > WATERTABLE > Define

The WMTDEF command defines the Water-Motion-Table associated with IMPIPE elements for hydrostatic and/or hydrodynamic analysis.

Water motion table number
Reference number for the current Water_Motion_Table (WMT).
(default is highest defined + 1, limit is 20)

Associated material property set
Associated material set number for this WMT.

Set number
Subset number for this WMT. (<= 7, default is last defined)

Starting location of WMT constants for subset

Starting location to enter constants for the specified subset. (>= 1 and <= maximum number of constants for this subset)

No of constants to be entered

Number of constants to be defined for this subset.

WMT (i)

Value of ith constant. (see the following notes)

Notes

1. The number of constants for each subset is defined as follows:

Subset Number	Number of Constants
1	3
2	24
3	12
4	12
5	12
6	8
7	80

constant_1, 2, ... : Constants input for this subset. For each subset, the constants are defined as follows:

1. Subset_number = 1:
   Constant_1 = Water depth to sea bed.
   Constant_2 = Water density.
   Constant_3 = Wave direction angle.
   Total 3 constants.

2. Subset_number = 2:
   Constant_1 = Elevation_1.
   Constant_2 = Current velocity_1.
   Constant_3 = Current direction angle_1.
   .....
   .....
   Repeat input up to eight (8) sets, total 24 constants.

3. Subset_number = 3:
   Constant_1 = Reynolds number_1.
   .....
   .....
   Repeat input up to twelve (12) sets, total 12 constants.

4. Subset_number = 4:
   Constant_1 = Coefficient of normal drag_1.
   .....
   .....
   Repeat input up to twelve (12) sets, total 12 constants.

5. Subset_number = 5:
   Constant_1 = Coefficient of tangential drag_1.

.....
.....
Repeat input up to twelve (12) sets, total 12 constants.

6. Subset_number = 6:
   Constant_1 = Temperature_1 at Elevation_1.
   .....
   .....
   Repeat input up to eight (8) sets, total 8 constants.

7. Subset_number = 7:
   Constant_1 = Wave amplitude_1.
   Constant_2 = Wave period_1.
   Constant_3 = Phase shift_1.
   Constant_4 = Wave length_1.
   .....
   .....
   Repeat input up to (twenty) 20 sets, total 80 constants.

2. The origin of the global coordinate system must be on the water surface and the z-axis must point outward from the water surface.
3. The material property set must be defined before WMT is defined.
4. The maximum number of constants that can be input each time the command is issued is twelve (12). The command can be repeated as needed to specify more constants, the default value for the starting location will change accordingly when the command is used to add constants to an already defined subset and WMT.
5. Acceleration of gravity (z-component of ACEL) need to be input in the static or dynamic analysis. For the dynamic analysis, it must remain constant throughout the loading period.
6. If water depth = 0, then the structure is assumed to be in the air and the rest of the constants will be ignored in the analysis.
7. Elevation input must be in an ascending order starting from the sea bed (= -water depth) and ending at the water surface (= 0); if elevation_1 = 0, then a uniform current is assumed.
8. The z-component of the current velocity is assumed as zero.
9. If Reynolds number_1 = 0, then the drag coefficients are assumed constant (read from the RCONST command), otherwise, they are computed during the analysis.
10. If Reynolds number_1 > 0, then all twelve (12) values need to be input in an ascending order.
11. If Reynolds number dependency of drag coefficients is assumed, then the viscosity (VISC) must be input.
12. Subset_4 is related to subset_3; if coefficient of normal drag_1 = 0, then the coefficient of normal drag is assumed constant.
13. If coefficient of normal drag_1 > 0, then all twelve (12) coefficients of normal drag need to be input.

14. Subset_5 is related to subset_3; if coefficient of tangential drag_1 = 0, then the coefficient of tangential drag is assumed constant.
15. If coefficient of tangential drag > 0, then all twelve (12) coefficients of tangential drag need to be input.
16. Subset_6 is related to subset_2; if all input temperatures are zero, then no description of the temperature variation with z-coordinate is assumed.
17. The temperature defined through the WMTDEF command will overwrite the regular temperature input (NTND, etc.), and these temperatures are used to compute the temperature_dependent_properties.
18. If wave amplitude_1 = 0, then no wave information is assumed in the analysis.
19. 0 <= Wave length < 1000*(water depth); if wave length = 0, then it will be calculated in the program automatically.
20. Please refer to Chapter 4 of the COSMOS/M User Guide for details on the IMPIPE element.

WMTDEL

Analysis > NONLINEAR > WATERTABLE > Delete

The WMTDEL command deletes a pattern of Water-Motion-Tables (WMT) previously defined by the WMTDEF command.

Beginning water motion table
Beginning WMT in the pattern.
(default is 1)

Ending water motion table
Ending WMT in the pattern.
(default is highest WMT label defined)

Increment
Increment between WMT's in the pattern.
(default is 1)

WMTLIST

Analysis > NONLINEAR > WATERTABLE > List

The WMTLIST command lists a pattern of Water-Motion-Tables (WMT) previously defined by the WMTDEF command.

Beginning water motion table
Beginning WMT in the pattern.
(default is 1)

Ending water motion table
Ending WMT in the pattern.
(default is highest WMT label defined)

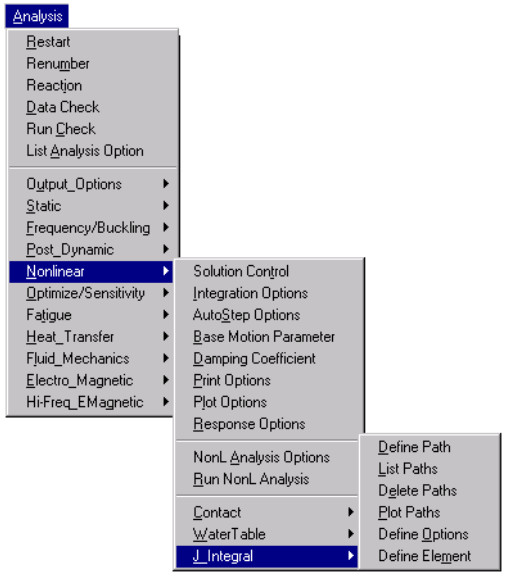
Increment
Increment between WMT's in the pattern.
(default is 1)

▼J_INTEGRAL Menu

Analysis > NONLINEAR > J_INTEGRAL

This menu contains commands to define, list, and delete J_integral paths around a crack tip for use in NSTAR.

Figure 10-16 J Integral Menu



J_INTDEF

Analysis > NONLINEAR > J_INTEGRAL > Define Path

The J_INTDEF command defines a J_integral path around a crack tip. Either half or the full crack may be modeled.

J integral set number
J_integral set number. Each set represents a J_path around a crack tip.
(default is current set)

Starting segment
Starting segment number in the set.
(default is last defined segment + 1)

First node for the segment
Node at the start of the segment. (Required for segment number 1 only. For other segments, the 'end node for the segment' of the previous segment is automatically used as the start node for the new one.)

End node for the segment
Node at the end of the segment. Each segment is defined by an ending node and associated global element number.

Associated element for the segment
Associated global element.

Notes

1. Up to 10 J paths can be defined in an analysis. Different J paths can be defined around different (or same) crack tips.
2. A maximum of 300 segments can be defined on a J path.
3. A maximum of 10 segments can be input each time this command is issued.
4. An intermediate segment can be modified by proper selection of the starting segment. However, new segments can only be added to the last defined segment and intermediate segments cannot be deleted. In these situations, the entire J set must be deleted and then redefined.
5. Special care is needed in selection of the proper node at the start and end of a path where coincident nodes may exist on the crack free surfaces.
6. In case of higher order elements, only the corner nodes need to be input by this command. Mid-side nodes will be found and included by the program.
7. Nodes and elements may by picked by the mouse if the PICK icon is highlighted.

J_INTLIST

Analysis > NONLINEAR > J INTEGRAL > List Path

The J_INTLIST command lists a pattern of J integral paths defined by the J_INTDEF command.

Beginning set
Beginning J integral set in the pattern.

Ending set
Ending J integral set in the pattern.
(default is highest set defined)

Increment
Increment between J sets in the pattern.
(default is 1)

Element list flag
Flag for listing elements inside the J path.
= 0: No do not list elements inside the J path
= 1: Yes list elements inside the J path
(default is no)

J_INTDEL

Analysis > NONLINEAR > J_INTEGRAL > Delete Path

The J_INTDEL command deletes a pattern of J_integral paths defined by the J_INTDEF command.

Beginning set
Beginning J integral set in the pattern.

Ending set
Ending J integral set in the pattern.
(default is beginning set)

Increment
Increment between J sets in the pattern.
(default is 1)

Delete inside element only
Flag for deleting the elements inside the J path only. Used for axisymmetric and thermal analyses only.
= 0: No delete the J sets completely
= 1: Yes delete the inner elements only
(default is no)

J_INTPLOT

Analysis > NONLINEAR > J_INTEGRAL > Plot Path

The J_INTPLOT command plots a pattern of J integral paths defined by the J_INTDEF command.

Beginning set
Beginning J integral set in the pattern.

Ending set
Ending J integral set in the pattern.
(default is highest set defined)

Increment
Increment between J paths in the pattern.
(default is 1)

Element plot flag
Flag for plotting elements inside the J path.
= 0: No — do not plot elements inside the J path
= 1: Yes — plot elements inside the J path
(default is no)

J_INTINF

Analysis > NONLINEAR > J_INTEGRAL > Define Option

The J_INTINF command specifies detailed options and/or constants for use with a J integral path.

J-integral set number
J-integral set number.

Crack model range
Type of crack modeling.
= 0: 0-2PI — entire crack area is modeled (0 to 2 Pi range)
= 1: 0-PI — only half of the crack area is modeled due to symmetry (0 to Pi range)
(default is 0-2PI)

J-integral output type
Type of J output.
= 0: Total J — evaluate and print total J only
= 1: Ji and Jii — evaluate and print the J_integral for modes I and II(available for crack_model = 0 only)

Angle between crack dir. and global x
Angle between the crack axis and the global X-axis in degrees.
(default is 0)

Angle for output of j
Angle between the J_integral evaluation axis and the crack axis in degrees.
(default is 0)

Crack tip radius (axisymmetric)
Crack tip radius. (required for axisymmetric analysis only)
(default is 1.0)

J_INTELEM

Analysis > NONLINEAR > J_INTEGRAL > Define Element

The J_INTELEM command defines a pattern of elements inside a J path for axi-symmetric and/or thermal analyses only.

J-integral set number
J set number for which elements are to be defined.
(default is the current set)

Beginning element
Beginning element in the pattern.
(default is 1)

Ending element
Ending element in the pattern.
(default is highest global element number defined)

Increment
Increment between elements in the pattern.
(default is 1)

Notes

1. A maximum of 700 elements can be defined inside a J path.
2. Elements on the J path are automatically included. However, the program performs an internal checking to avoid errors from repetition of elements.
3. Selection commands such as SELWIN and SELREF may be conveniently used to select the elements inside a J path. Default options may then be used for the pattern information in this command.



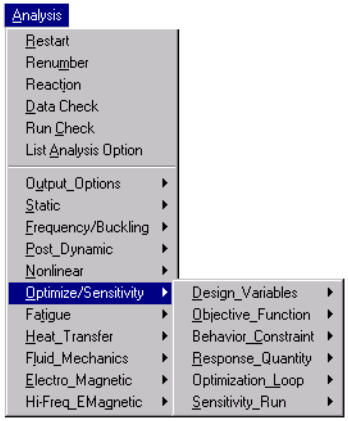

OPTIMIZE/SENSITIVITY Menu

Analysis > OPTIMIZE/SENSITIVITY

This menu contains commands related to the optimization and sensitivity analysis module OPTSTAR. Postprocessing for OPTSTAR is accessible depending on the active analysis type for postprocessing: 0 = Linear Static, 2 = Frequency or Buckling, and 3 = Thermal. (Refer to the ACTPOST command.).

✍ For optimization problems, you must create the geometry inside GEOSTAR.

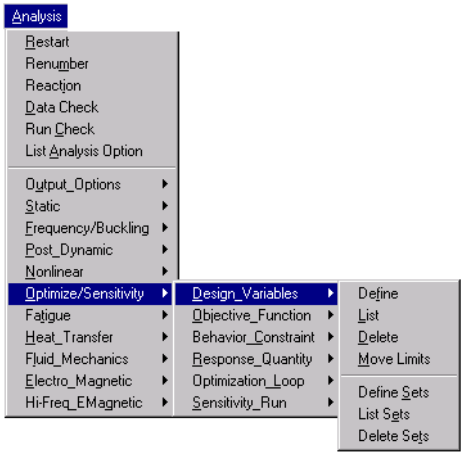
Figure 10-17 Optimization/Sensitivity Me



▼DESIGN_VARIABLES Menu

Analysis > OPTIMIZE/SENSITIVITY > DESIGN VARIABLES

Figure 10-18 Design Variables Menu



DVARDEF

Analysis > OPTIMIZE/SENSITIVITY > DESIGN_VARIABLES > Define

The DVARDEF command defines the type, name, bounds and the tolerance for design variable sets. The design variable can be either of sizing or shape type. The design variable name can be one of the predefined sizing options or any other parameter declared by the PARASSIGN (Control, Parameter, Assign Parameters) command. Prompts of the command (like. lower and upper bounds) can be specified in terms of parameters or parametric expressions. Values will change dynamically from one loop to another.

Design variable set number

Design variable set number.
(default is highest set defined + 1)

Design variable type

Design variable type.

= 0: Size	sizing
= 1: Shape	shape
	(default is shape)

Design variable parametric name

Design variable parametric name as defined by the PARASSIGN command.

Design variable lower bound
Value of the design variable's lower bound (side constraint).

Design variable upper bound
Value of the design variable's upper bound (side constraint).

The following flags are not needed for sensitivity study. Use the default values unless an optimization analysis is also planned.

Design variable convergence tolerance for optimization
Convergence value for the design variable.
(default = 0.01(upper bound - lower bound))*

Pre-opt process: 0=random nonzero=perturb_ratio
Pre-optimization selection of the design variable value (see restriction in note below).

= 0	randomly selected
Otherwise	perturbation ratio *(default is 0)*

Sizing type
Design variable sizing type. Prompted for sizing optimization only.

For linear static analysis:

= T_AREA	area of a truss cross-section (TRUSS2D, TRUSS3D)
= B_WIDTH	width of a rectangular beam cross-section (BEAM2D, BEAM3D)
= B_HEIGHT	height of a rectangular beam cross-section (BEAM2D, BEAM3D)
= C_THICK	thickness of a continuum element (TRIANG, PLANE2D)
= S_THICK	thickness of a shell element (SHELLAX, SHELL3, SHELL4, SHELL9, SHELL3T, SHELL4T, and SHELL6)
= P_THICK	thickness of thin pipe elements (PIPE)
= P_RADIUS	radius of thin pipe elements (PIPE)

For other sizing options or analysis types:

= GENERAL	general sizing parameter *(default is GENERAL)*

(size_elem)
Design variable element type:
In case of using B_WIDTH or B_HEIGHT:

= 2d	2D analysis
= 3d	3D analysis *(default is 2d)*

In case of using S_THICK:

= Bending	bending dominant applications
= Membrane	membrane dominant applications *(default is membrane)*

Notes

1. All design variables should be either randomly selected or perturbed. Mixed definitions are not allowed.
2. In case of sensitivty analysis, the values specified by the SN_SETDEF may violate the lower and upper bounds defined by the DVARDEF command.

DVARLIST

Analysis > OPTIMIZE/SENSITIVITY > DESIGN_VARIABLES > List

The DVARLIST command lists a pattern of design variable sets previously defined by the DVARDEF (**Analysis > OPTIMIZE/SENSITIVITY > DESIGN_VARIABLES > Define**) command. The command lists the set number, design variable type, parametric name, current value, lower and upper bounds, convergence tolerance and pre-optimization perturbation ratio. It should be noted that a zero perturbation ratio indicates randomly selected values in the pre-optimization stage.

Beginning set
Beginning set of design variables.
(default is 1)

Ending set
Ending set of design variables.
(default is the highest set number defined)

Increment
Increment between sets of design variables.
(default is 1)

DVARDEL

Analysis > OPTIMIZE/SENSITIVITY > DESIGN_VARIABLES > Delete

The DVARDEL command deletes a pattern of design variable sets previously defined by the DVARDEF **(Analysis > OPTIMIZE/SENSITIVITY > DESIGN_VARIABLES > Define)** command.

Beginning set
Beginning set of design variables in the pattern.

Ending set
Ending set of design variables in the pattern.
(default is beginning set)

Increment
Increment between sets of design variables in the pattern.
(default is 1)

OP_DVMOVE

Analysis > OPTIMIZE/SENSITIVITY > DESIGN_VARIABLES > Move Limits

The OP_DVMOVE command specifies move limits to control the change in design variables. Small move limits may result in higher computational cost but give higher quality approximations. Prompts of the command may be specified in terms of parameters or parametric expressions. Values will change dynamically from one loop to another.

Initial move limit
Initial value of the move limit.
(default is 0.5)

Lower bound of move limit
Lower bound of the move limit.
(default is initial move limit)

Upper bound of move limit
Upper bound of the move limit.
(default is initial move limit)

Move limit multiplier
Move limit multiplier.
(default is 1.0)

Note
Move limits at subsequent optimization iteration (loop) are determined by the following formula:

move at iteration (n) = min [ubd, max (lbd, x)]

where: x = mult * move at iteration (n-1)

SN_SETDEF

Analysis > OPTIMIZE/SENSITIVITY > DESIGN_VARIABLES > Define Sets

The SN_SETDEF command prepares the design variable sets required for the off-set sensitivity study. Only one set is required for a local sensitivity study.

General Command Syntax

Sensitivity set number
Set number.
(default is highest set defined + 1)

Input type

Method of specifying design variable values.
= Perturb by a perturbation ratio with respect to the initial value
= Value by actual value
(default is perturb)

Parameters

The parameters depend on the input type. Detailed description of the syntax for each type is given below.

Command Syntax for Perturbation Ratio

Ratio_dv1

Perturbation ratio for design variable #1 with respect to its initial value.

Ratio_dv2

Perturbation ratio for design variable #2 with respect to its initial value.

Ratio_dvN

Perturbation ratio for design variable #N with respect to its initial value.

Command Syntax for Value Option

Value_dv1

Value of design variable #1.

Value_dv2

Value of design variable #2.

Value_dvN

Value of design variable #N.

Note

The values specified by the SN_SETDEF command may violate the lower and upper bounds defined by the DVARDEF command.

SN_SETLIST

Analysis > OPTIMIZE/SENSITIVITY > DESIGN_VARIABLES > List Sets

The SN_SETLIST command lists, for each design variable, its label, name, and value (or perturbation ratio) specified for the sensitivity sets defined by the SN_SETDEF command.

Beginning sensitivity

Beginning sensitivity set number.
(default is 1)

Ending sensitivity
Ending sensitivity set number.
(default is the highest set label defined)

Increment
Increment between sensitivity sets.
(default is 1)

SN_SETDEL

Analysis > OPTIMIZE/SENSITIVITY > DESIGN_VARIABLES > Delete Sets

The SN_SETDEL command deletes sensitivity sets previously defined by the SN_SETDEF command.

Beginning sensitivity
Beginning sensitivity set number.

Ending sensitivity
Ending sensitivity set number.
(default is beginning sensitivity)

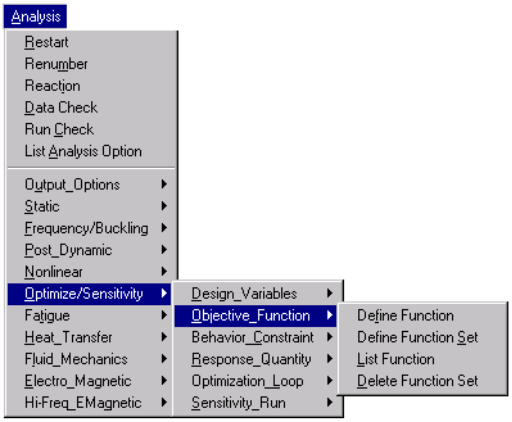
Increment
Increment between sensitivity sets.
(default is 1)




▼OBJECTIVE_FUNCTION Menu

Analysis > OPTIMIZE/SENSITIVITY > OBJECTIVE_FUNCTION

Figure 10-19 Objective Function Menu



OP_OBJDEF

Analysis > OPTIMIZE/SENSITIVITY > OBJECTIVE_FUNCTION > Define Function

The OP_OBJDEF command specifies the type and related parameters for nodal, elemental, global, and user-defined objective functions. Global objective functions may be frequency or linearized buckling load factor. Prompts of the command (e.g. tolerance) may be specified in terms of parameters or parametric expressions. Values will change dynamically from one loop to another.

General Command Syntax

Type

Objective function type.

= User	user-defined quantity
= Nodal	nodal quantity
= Elem	elemental quantity
= Freq	frequency
= Buckling	linearized buckling load factor
	(default is elemental)

Objective

Flag to minimize or maximize the objective function.

= Minimize	minimize the objective function
= Maximize	maximize the objective function
	(default is minimize)

Parameters

The parameters depend on the objective function type. Detailed description of the syntax for each type of objective functions is given below.

Command Syntax for User-Defined Functions

User-defined obj. function parametric name

Parametric name of the user-defined objective function as declared by a PARASSIGN command.

Objective function convergence tolerance (ratio)

Convergence tolerance for objective function. The input value is a ratio of the objective function initial value.
(default = 0.01)

Objective function approximation type

Type of the objective function approximation.

= -3: cubic only	only cubic terms
= -2: quadratic only	only quadratic terms
= -1: linear only	only linear terms
= 0: automatic	automatic determination of the polynomial type
= 1: 0: Linr+Quad	start with linear and add quadratic terms
= 2: 0: Linr+Quad+Cross	start with linear and add quadratic and quadratic cross terms
= 3: 0: Linr+Quad+Cross + cubic	start with linear and add quadratic, quadratic cross terms and cubic terms *(default is 1)*

Command Syntax for Nodal Objective Function

Nodal objective function name

Name of the nodal objective function

Component name

Displacement components.

= UX	translation in the X-direction
= UY	translation in the Y-direction
= UZ	translation in the Z-direction
= RX	rotation about the X-direction
= RY	rotation about the Y-direction
= RZ	rotation about the Z-direction
= URES	magnitude of resultant displacement

Stress components.

= SX	normal stress in the X-direction
= SY	normal stress in the Y-direction
= SZ	normal stress in the Z-direction
= TXY	X-Y shear stress
= TXZ	X-Z shear stress

= TYZ	Y-Z shear stress
= P1	first principal stress
= P2	second principal stress
= P3	third principal stress
= VON	von Mises stress
= INT	stress intensity

Velocity components.

= VX	velocity in the X-direction
= VY	velocity in the Y-direction
= VZ	velocity in the Z-direction
= WX	angular velocity about X-direction
= WY	angular velocity about Y-direction
= WZ	angular velocity about Z-direction
= VRES	resultant velocity

Acceleration components.

= AX	acceleration in the X-direction
= AY	acceleration in the Y-direction
= AZ	acceleration in the Z-direction
= BX	angular acceleration about X-direction
= BY	angular acceleration about Y-direction
= BZ	angular acceleration about Z-direction
= ARES	resultant acceleration

Reaction forces.

= RFX	reaction force in the X-direction
= RFY	reaction force in the Y-direction
= RFZ	reaction force in the Z-direction
= RFRES	resultant reaction force
= RMX	reaction moment about the X-direction
= RMY	reaction moment about the Y-direction
= RMZ	reaction moment about the Z-direction
= RMRES	resultant reaction moment

Temperature.

= TEMP	temperature

Temperature gradient components.

= GRADX	gradient in the X-direction
= GRADY	gradient in the Y-direction
= GRADZ	gradient in the Z-direction
= GRADN	normal gradient

Heat flux components.

= HFLUXX	heat flux in the X-direction
= HFLUXY	heat flux in the Y-direction
= HFLUXZ	heat flux in the Z-direction
= HFLUXN	magnitude of resultant heat flux

Fatigue.

= FTG	fatigue usage factor *(default is VON)*

analysis type

Analysis type from which the postprocessing function is obtained. This option is useful when the same type of postprocessing functions appear in more than one analysis. For example, von Mises stress VON appears in both linear static and nonlinear analyses.

= 0	linear static
= 1	nonlinear
= 3	thermal
= 7	fatigue
= 8	post dynamic
	(default is 0)

layer number

Layer number on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.
(default is 1)

face flag

Face on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.

= 0	top face
= 1	bottom face
= 2	membrane stresses
= 3	bending stresses
	(default is 0)

Criterion flag

Criterion flag for the objective function.

= 0: MaxAbs	maximum absolute value of nodal quantities
= -1: AlgMax	algebraic minimum of nodal quantities
= -2: AlgMin	algebraic maximum of nodal quantities
= -3: MaxNum	maximum numerical value of nodal quantities (similar to option 0 above but with the original sign retained) *(default is MaxAbs)*

Load case/time step: 0 = all

Primary load case or time step number. (enter “0” to consider all load cases or time steps)
(default is 1)

Objective function convergence tolerance (ratio)

Convergence tolerance for objective function. The input value is a ratio of the objective function initial value.
(default = 0.01)

Constraint approximation type

Type of the constraint approximation.

= -3: Cubic only	only cubic terms
= -2: Quadratic only	only quadratic terms
= -1: Linear only	only linear terms
= 0: Automatic	automatic determination of polynomial type
= 1: 0: Nouad	start with linear and add quadratic terms
= 2: 0: Nouad + cross	start with linear and add quadratic and quadratic cross terms
= 3: 0: Nouad + cross + cubic	start with linear and add quadratic, quadratic cross terms and cubic terms

(default is 0: Nouad)

Reference keypoint

Label of a reference keypoint in case of using the relative displacement as an objective function. This entry is prompted only in case of choosing UX, UY, UZ, or URES. Enter "0" for a no reference node use.
(default is 0)

Command Syntax for Elemental Objective Function

Type = elemental

Objective function type. Equals elemental for elemental quantity.

Objective

Flag to minimize or maximize the objective function.

= 0: Minimize	minimize the objective function
= 1: Maximize	maximize the objective function

(default is minimize)

Elemental objective function name

Stress components.

= SX	normal stress in the X-direction
= SY	normal stress in the Y-direction
= SZ	normal stress in the Z-direction
= TXY	X-Y shear stress
= TXZ	X-Z shear stress
= TYZ	Y-Z shear stress
= P1	first principal stress
= P2	second principal stress
= P3	third principal stress
= VON	von Mises stress
= INT	stress intensity
= FIND	failure index
= MFIND	maximum failure index from the layer specified below to the maximum layer number
= ILTXZ	interlaminar shear (X-Z plane)
= ILTYZ	interlaminar shear (Y-Z plane)
= MILTXZ	max interlaminar shear (X-Z plane)
= MILTYZ	max interlaminar shear (Y-Z plane)
= ERROR	error in stress

Strain components.

= EPSX	normal strain in the X-direction
= EPSY	normal strain in the Y-direction
= EPSZ	normal strain in the Z-direction
= GMXY	X-Y shear strain
= GMYZ	Y-Z shear strain
= GMZX	Z-X shear strain
= ESTRN	effective strain
= SED	strain energy density

Weight or volume.

= VOLUME	volume
= WEIGHT	weight
	(default is VOLUME)

analysis type

Analysis type from which the postprocessing function is obtained. This option is useful when the same type of postprocessing functions appear in more than one analysis. For example, von Mises stress VON appears in both linear static and nonlinear analyses.

= 0	linear static
= 1	nonlinear
= 3	thermal
= 8	post dynamic
	(default is 0)

layer number

Layer number on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.
(default is 1)

face flag

Face on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.

= 0	top face
= 1	bottom face
= 2	membrane stresses
= 3	bending stresses
	(default is 0)

Criterion flag

Criterion flag for the objective function. This flag is not used for VOLUME and WEIGHT options.

= 0: MaxAbs	maximum absolute value of nodal quantities
= -1: AlgMax	algebraic minimum of nodal quantities
= -2: AlgMin	algebraic maximum of nodal quantities
= -3: MaxNum	maximum numerical value of nodal quantities (similar to option 0 above but with the original sign retained)
	(default is MaxAbs)

(Load case/time step: 0 = all)
Primary load case or time step number. (enter "0" to consider all load cases or time steps)
(default is 1)

Objective function convergence tolerance (ratio)
Convergence tolerance for objective function. The input value is a ratio of the objective function initial value.
(default = 0.01)

Constraint approximation type
Type of the constraint approximation.

= -3: Cubic only	only cubic terms
= -2: Quadratic only	only quadratic terms
= -1: Linear only	only linear terms
= 0: Automatic	automatic determination of polynomial type
= 1: 0: Nouad	start with linear and add quadratic terms
= 2: 0: Nouad + cross	start with linear and add quadratic and quadratic cross terms
= 3: 0: Nouad + cross + cubic	start with linear and add quadratic, quadratic cross terms and cubic terms

(default is 0: Nouad)

Command Syntax for Frequency Objective Function

Type = frequency
Objective function type. Equals frequency for frequency.

Objective
Flag to minimize or maximize the objective function.

= 0: Minimize	minimize the objective function
= 1: Maximize	maximize the objective function

(default is minimize)

Mode shape number
Mode shape number.
(default is 1)

Objective function convergence tolerance (ratio)
Convergence tolerance for objective function. The input value is a ratio of the objective function initial value.
(default = 0.01)

Constraint approximation type
Type of the constraint approximation.

= -3: Cubic only	only cubic terms
= -2: Quadratic only	only quadratic terms
= -1: Linear only	only linear terms
= 0: Automatic	automatic determination of polynomial type
= 1: 0: Nouad	start with linear and add quadratic terms
= 2: 0: Nouad + cross	start with linear and add quadratic and qua-

	dratic cross terms
= 3: 0: Nouad + cross + cubic	start with linear and add quadratic, quadratic cross terms and cubic terms *(default is 0: Nouad)*

Command Syntax for Linearized Buckling Load Factor Objective Function

Type = buckling

Objective function type. Equals buckling for linearized buckling load factor.

Objective

Flag to minimize or maximize the objective function.

= 0: Minimize	minimize the objective function
= 1: Maximize	maximize the objective function *(default is minimize)*

Objective function convergence tolerance (ratio)

Convergence tolerance for objective function. The input value is a ratio of the objective function initial value.
(default = 0.01)

Constraint approximation type

Type of the constraint approximation.

= -3: Cubic only	only cubic terms
= -2: Quadratic only	only quadratic terms
= -1: Linear only	only linear terms
= 0: Automatic	automatic determination of polynomial type
= 1: Linear + quad	start with linear and add quadratic terms
= 2: Linear + quad + cross	start with linear and add quadratic and quadratic cross terms
= 3: Linear + quad + cross + cubic	start with linear and add quadratic, quadratic cross terms and cubic terms *(default is linear + quad)*

OP_OBJSET

Analysis > OPTIMIZE/SENSITIVITY > OBJECTIVE_FUNCTION > Define Function Set

The OP_OBJSET command specifies different sets for the objective function. Each set can define different parts of the model with same objective function type and name defined by the OP_OBJDEF **(Analysis > OPTIMIZE/SENSITIVITY > OBJECTIVE_FUNCTION > Define Function)** command. There is no need to use this command if the objective function refers to the whole model. The command is most useful for volume and weight objective functions. It cannot be used with frequency and linearized buckling functions. Prompts of the command (e.g. weight factor) can be specified in terms of parameters or parametric expressions. Values will change dynamically from one loop to another.

Objective function set number

Objective function set number.
(default is highest set number defined + 1)

Entity type associated with selected set

Entity type with which the objective function set is associated. Note that mass elements may only be associated with keypoints (PT), 1D elements may be associated with curves (CR), or contours (CT), 2D elements may be associated with surfaces (SF), or regions (RG), and 3D elements may only be associated with volumes (VL). Use ND or EL to specify a pattern of nodes or elements respectively.

= PT	keypoint
= CR	curve
= CT	contour
= SF	surface
= RG	region
= VL	volume
= ND	nodes
= EL	elements
= PH	polyhedra
= PA	parts
	(default is SF)

Beginning entity

Label of the beginning entity. When using the nodes/elements option with shape optimization, it is recommended to set “beginning entity=1”, unless the user is certain that nodes/elements labels will not change during optimization loops.

Ending entity

Label of the ending entity. When using the nodes/elements option with shape optimization, it is recommended to set “ending entity=NDMAX/ELMAX: the highest nodes/elements label”, unless the user is certain that nodes/elements labels will not change during optimization loops.
(default is the highest entity number defined)

Increment

Increment between entity labels. When using the nodes/elements option with shape optimization, it is recommended to set “increment= 1", unless the user is certain that nodes/elements labels will not change during optimization loops.
(default is 1)

Objective function weight factor

Weight factor for the objective function set.
(default is 1)




OP_OBJLIST

Analysis > OPTIMIZE/SENSITIVITY > OBJECTIVE_FUNCTION > List Function

The OP_OBJLIST command lists all the information defined by the OP_OBJDEF (**Analysis > OPTIMIZE/SENSITIVITY > OBJECTIVE_FUNCTION > Define Function**) command: type, component name, criterion, tolerance, approximation type, reference keypoint (in case of defining a relative displacement function) and the current value of the objective function. In addition, the command lists all the information defined by the OP_OBJSET (**Analysis > OPTIMIZE/SENSITIVITY > OBJECTIVE_FUNCTION > Define Function Set**) command like set number, entity type with which sets are associated and corresponding pattern information (beginning, ending and increment), and weight factor.

Beginning set
Beginning set number.
(default is 1)

Ending set
Ending set number.
(default is the highest set number defined)

Increment
Increment.
(default is 1)

OP_OBJDEL

Analysis > OPTIMIZE/SENSITIVITY > OBJECTIVE_FUNCTION > Delete Function Set

The OP_OBJDEL command deletes the objective function sets previously defined by the OP_OBJSET (**Analysis > OPTIMIZE/SENSITIVITY > OBJECTIVE_FUNCTION > Define Function Set**) command.

Beginning set
Beginning set number.

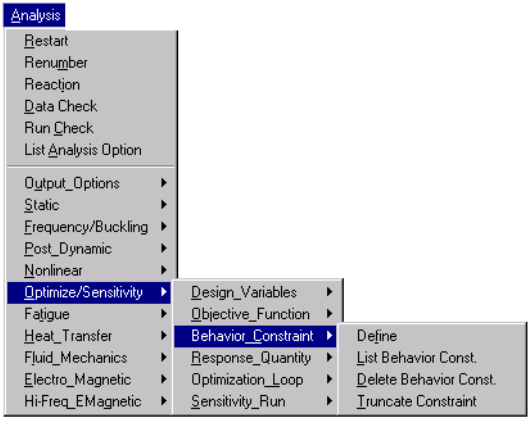
Ending set
Ending set number.
(default is beginning set)

Increment
Increment.
(default is 1)

▼BEHAVIOR_CONSTRAINT Menu

Analysis > OPTIMIZE/SENSITIVITY > BEHAVIOR CONSTRAINT

Figure 10-20 Behavior Constraint Menu



OP_CONDEF

Analysis > OPTIMIZE/SENSITIVITY > BEHAVIOR_CONSTRAINT > Define

The OP_CONDEF command defines nodal, elemental, global (frequency and linearized buckling load factors), and user-defined constraints. Nodes and elements may be specified directly or through the association of geometric entities. A reference keypoint is needed in the case of using a relative displacement constraint. Prompts of the command (e.g. lower and upper bounds) may be specified in terms of parameters or parametric expressions. Values will change dynamically from one loop to another.

General Command Syntax

Constraint set number

Constraint set number.
(default is highest constraint set defined + 1)

Constraint type

Constraint type.

= -1: User	user-defined
= 0: Nodal	nodal constraint
= 1: Elem	elemental constraint

= 2: Freq	frequency
= 3: Buckling	linearized buckling load factor
	(default is elem)

Parameters

The parameters depend on the constraint type. Detailed description of the syntax for each type of constraint is given below.

Command Syntax for User-Defined Constraints

Constraint set number

Constraint set number.
(default is highest constraint set defined + 1)

Constraint type = user

Constraint type. Equals user for user-defined constraint.

User-defined constraint parametric name

Parametric name of the user-defined constraint as declared by a PARASSIGN (Control, Parameter, Assign Parameter) command.

Constraint lower bound

Value of the constraint lower bound.

Constraint upper bound

Value of the constraint upper bound.

Constraint feasibility tolerance

Tolerance for the constraint value.
(default = 0.01(upper bound - lower bound))*

Constraint approximation type

Type of the constraint approximation.

= -3: Cubic only	only cubic terms
= -2: Quadratic only	only quadratic terms
= -1: Linear only	only linear terms
= 0: Automatic	automatic determination of polynomial type
= 1: Linear + quad	start with linear and add quadratic terms
= 2: Linear + quad + cross	start with linear and add quadratic and quadratic cross terms
= 3: Linear + quad + cross + cubic	start with linear and add quadratic, quadratic cross terms and cubic terms
	(default is linear + quad)

Command Syntax for Nodal Constraints

Constraint set number

Constraint set number.
(default is highest constraint set defined + 1)

Constraint type = nodal

Constraint type. Equals nodal for nodal constraint.
(default is highest constraint set defined + 1)

Nodal constraint name

Displacement components.

= UX	translation in the X-direction
= UY	translation in the Y-direction
= UZ	translation in the Z-direction
= RX	rotation about the X-direction
= RY	rotation about the Y-direction
= RZ	rotation about the Z-direction
= URES	magnitude of resultant displacement

Stress components.

= SX	normal stress in the X-direction
= SY	normal stress in the Y-direction
= SZ	normal stress in the Z-direction
= TXY	X-Y shear stress
= TXZ	X-Z shear stress
= TYZ	Y-Z shear stress
= P1	first principal stress
= P2	second principal stress
= P3	third principal stress
= VON	von Mises stress
= INT	stress intensity

Velocity components.

= VX	velocity in the X-direction
= VY	velocity in the Y-direction
= VZ	velocity in the Z-direction
= WX	angular velocity about X-direction
= WY	angular velocity about Y-direction
= WZ	angular velocity about Z-direction
= VRES	resultant velocity

Acceleration components.

= AX	acceleration in the X-direction
= AY	acceleration in the Y-direction
= AZ	acceleration in the Z-direction
= BX	angular acceleration about X-direction
= BY	angular acceleration about Y-direction
= BZ	angular acceleration about Z-direction
= ARES	resultant acceleration

Reaction forces.

= RFX	reaction force in the X-direction
= RFY	reaction force in the Y-direction
= RFZ	reaction force in the Z-direction
= RFRES	resultant reaction force
= RMX	reaction moment about the X-direction
= RMY	reaction moment about the Y-direction
= RMZ	reaction moment about the Z-direction
= RMRES	resultant reaction moment

Temperature.

= TEMP temperature

Temperature gradient components.

= GRADX gradient in the X-direction
= GRADY gradient in the Y-direction
= GRADZ gradient in the Z-direction
= GRADN normal gradient

Heat flux components.

= HFLUXX heat flux in the X-direction
= HFLUXY heat flux in the Y-direction
= HFLUXZ heat flux in the Z-direction
= HFLUXN magnitude of resultant heat flux
(default is VON)

Fatigue.

= FTG fatigue usage factor
(default is 0)

analysis type

Analysis type from which the postprocessing function is obtained. This option is useful when the same type of postprocessing functions appear in more than one analysis. For example, von Mises stress VON appears in both linear static and nonlinear analyses.

= 0 linear static
= 1 nonlinear
= 3 thermal
= 7 fatigue
= 8 post dynamic
(default is 0)

layer number

Layer number on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.
(default is 1)

face flag

Face on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.

= 0 top face
= 1 bottom face
= 2 membrane stresses
= 3 bending stresses
(default is 0)

Criterion flag

Criterion flag for the objective function.

= 0: MaxAbs maximum absolute value of nodal quantities
= -1: AlgMax algebraic minimum of nodal quantities

= -2: AlgMin	algebraic maximum of nodal quantities
= -3: MaxNum	maximum numerical value of nodal quantities (similar to option 0 above but with the original sign retained)
	(default is MaxAbs)

Load case/time step: 0 = all

Primary load case or time step number. (enter "0" to consider all load cases or time steps)
(default is 1)

Entity type associated with the constraint

Entity type with which nodal constraints are associated.

= PT	keypoint
= CR	curve
= CT	contour
= SF	surface
= RG	region
= VL	volume
= ND	nodes
= EL	elements
= PH	polyhedra
= PA	parts
	(default is SF)

Beginning entity

Label of the beginning entity. When using the nodes/elements option with shape optimization, it is recommended to set "beginning entity = 1", unless the user is certain that nodes/elements labels will not change during optimization loops.

Ending entity

Label of the ending entity. When using the nodes/elements option with shape optimization, it is recommended to set "ending entity=NDMAX/ELMAX: the highest nodes/elements label", unless the user is certain that node labels will not change during optimization loops.
(default is the highest entity label defined)

Increment

Increment between entity labels. When using the nodes/elements option with shape optimization, it is recommended to set "increment=1", unless the user is certain that nodes/elements labels will not change during optimization loops.
(default is 1)

Constraint lower bound

Value of the constraint lower bound.

Constraint upper bound

Value of the constraint upper bound.

Constraint feasibility tolerance

Tolerance for the constraint value.
(default = 0.01(upper bound - lower bound))*

Constraint approximation type

Type of the constraint approximation.

= -3: Cubic only	only cubic terms
= -2: Quadratic only	only quadratic terms
= -1: Linear only	only linear terms
= 0: Automatic	automatic determination of polynomial type
= 1: Linear + quad	start with linear and add quadratic terms
= 2: Linear + quad + cross	start with linear and add quadratic and quadratic cross terms
= 3: Linear + quad + cross + cubic	start with linear and add quadratic, quadratic cross terms and cubic terms

(default is linear + quad)

Reference keypoint

Label of a reference keypoint in case of using a relative displacement as a constraint. This entry is prompted only in case of choosing UX, UY, UZ, or URES. Enter "0" if no reference keypoint is used.
(default is 0)

Command Syntax for Elemental Constraints

Constraint set number

Constraint set number.
(default is highest constraint set defined + 1)

Constraint type = elemental

Constraint type. Equals elem for elemental constraint.

Elemental constraint name

Stress components.

= SX	normal stress in the X-direction
= SY	normal stress in the Y-direction
= SZ	normal stress in the Z-direction
= TXY	X-Y shear stress
= TXZ	X-Z shear stress
= TYZ	Y-Z shear stress
= P1	first principal stress
= P2	second principal stress
= P3	third principal stress
= VON	von Mises stress
= INT	stress intensity
= FIND	failure index
= MFIND	maximum failure index from the layer specified below to the maximum layer number
= ILTXZ	interlaminar shear (X-Z plane)
= ILTYZ	interlaminar shear (Y-Z plane)
= MILTXZ	max interlaminar shear (X-Z plane)
= MILTYZ	max interlaminar shear (Y-Z plane)
= ERROR	error in stress




Strain components.

= EPSX	normal strain in the X-direction
= EPSY	normal strain in the Y-direction
= EPSZ	normal strain in the Z-direction
= GMXY	X-Y shear strain
= GMYZ	Y-Z shear strain
= GMZX	Z-X shear strain
= ESTRN	effective strain
= SED	strain energy density

Weight or volume.

= VOLUME	volume
= WEIGHT	weight
	(default is VON)

analysis type

Analysis type from which the postprocessing function is obtained. This option is useful when the same type of postprocessing functions appear in more than one analysis. For example, von Mises stress VON appears in both linear static and nonlinear analyses.

= 0	linear static
= 1	nonlinear
= 3	thermal
= 8	post dynamic
	(default is 0)

layer number

Layer number on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.
(default is 1)

face flag

Face on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.

= 0	top face
= 1	bottom face
= 2	membrane stresses
= 3	bending stresses
	(default is 0)

Criterion flag

Criterion flag for the objective function.

= 0: MaxAbs	maximum absolute value of nodal quantities
= -1: AlgMax	algebraic minimum of nodal quantities
= -2: AlgMin	algebraic maximum of nodal quantities
= -3: MaxNum	maximum numerical value of nodal quantities (similar to option 0 above but with the original sign retained)
	(default is MaxAbs)

Load case/time step: 0 = all

Primary load case or time step number. (enter "0" to consider all load cases or time steps)
(default is 1)

Entity type associated with the constraint

Entity type with which these elemental constraints are associated. Note that mass elements may only be associated with keypoints (PT), 1D elements may be associated with curves (CR) or contours (CT), 2D elements may be associated with surfaces (SF) or regions (RG), and 3D
elements may only be associated with volumes (VL). Use EL to specify a pattern of elements.

= PT	keypoint
= CR	curve
= CT	contour
= SF	surface
= RG	region
= VL	volume
= ND	nodes
= EL	elements
= PH	polyhedra
= PA	parts
	(default is SF)

Beginning entity

Label of the beginning entity. When using the nodes/elements option with shape optimization, it is recommended to set "beginning entity=1", unless the user is certain that nodes/elements labels will not change during optimization loops.

Ending entity

Label of the ending entity. When using the nodes/elements option with shape optimization, it is recommended to set "ending entity=NDMAX/ELMAX: the highest nodes/elements labels", unless the user is certain that nodes/elements labels will not change during optimization loops.
(default is the highest entity number defined)

Increment

Increment between entity labels. When using the nodes/elements option with shape optimization, it is recommended to set "increment=1", unless the user is certain that nodes/elements labels will not change during optimization loops.
(default is 1)

Constraint lower bound

Value of the constraint lower bound.

Constraint upper lower bound

Value of the constraint upper bound.

Constraint feasibility tolerance

Tolerance for the constraint value.
(default = 0.01(upper bound - lower bound))*

Constraint approximation type

Type of the constraint approximation.

= -3: Cubic only — only cubic terms
= -2: Quadratic only — only quadratic terms
= -1: Linear only — only linear terms
= 0: Automatic — automatic determination of polynomial type
= 1: Linear + quad — start with linear and add quadratic terms
= 2: Linear + quad + cross — start with linear and add quadratic and quadratic cross terms
= 3: Linear + quad + cross + cubic — start with linear and add quadratic, quadratic cross terms and cubic terms
(default is linear + quad)

Command Syntax for Frequency Constraints

Constraint set number
Constraint set number.
(default is highest constraint set defined + 1)

Constraint type = frequency
Constraint type. Equals frequency for frequency constraint.

Mode shape number
Mode shape number.

Constraint lower bound
Value of the constraint lower bound.

Constraint upper bound
Value of the constraint upper bound.

Constraint feasibility tolerance
Tolerance for the constraint value.
(default = 0.01(upper bound - lower bound))*

Constraint approximation type
Type of the constraint approximation.
= -3: Cubic only — only cubic terms
= -2: Quadratic only — only quadratic terms
= -1: Linear only — only linear terms
= 0: Automatic — automatic determination of polynomial type
= 1: Linear + quad — start with linear and add quadratic terms
= 2: Linear + quad + cross — start with linear and add quadratic and quadratic cross terms
= 3: Linear + quad + cross + cubic — start with linear and add quadratic, quadratic cross terms and cubic terms
(default is linear + quad)

Command Syntax for Linearized Buckling Load Factor Constraints

Constraint set number
Constraint set number.
(default is constraint set defined + 1)

Constraint type = buckling
Constraint type. Equals buckling for linearized buckling load factor constraint.

Constraint lower bound
Value of the constraint lower bound.

Constraint upper bound
Value of the constraint upper bound.

Constraint feasibility tolerance
Tolerance for the constraint value.
(default = 0.01(upper bound - lower bound))*

Constraint approximation type
Type of the constraint approximation.

= -3: Cubic only	only cubic terms
= -2: Quadratic only	only quadratic terms
= -1: Linear only	only linear terms
= 0: Automatic	automatic determination of polynomial type
= 1: Linear + quad	start with linear and add quadratic terms
= 2: Linear + quad + cross	start with linear and add quadratic and quadratic cross terms
= 3: Linear + quad + cross + cubic	start with linear and add quadratic, quadratic cross terms and cubic terms

(default is linear + quad)

OP_CONLIST

Analysis > OPTIMIZE/SENSITIVITY > BEHAVIOR_CONSTRAINT > List Behavior Const.

The OP_CONLIST command lists constraint sets defined previously by the OP_CONDEF command. The command lists constraint sets, types, names, bounds, tolerances, approximation and criterion flags, entity types with which constraints are associated, and corresponding pattern information (beginning and ending entity labels, and increment), reference keypoint (in case of a relative displacement constraint), the current value of the constraint, and the considered load case or time step. The command also indicates whether a constraint is feasible by writing F (for feasible), or I (for infeasible).

Beginning set
Beginning constraint set number.
(default is 1)

Ending set
Ending constraint set number.
(default is the highest set number defined)

Increment
Increment between constraint sets.
(default is 1)

OP_CONDEL

Analysis > OPTIMIZE/SENSITIVITY > BEHAVIOR_CONSTRAINT > Delete Behavior Const.

The OP_CONDEL command deletes constraint sets previously defined by the OP_CONDEF (**Analysis, OPTIMIZE/SENSITIVITY, BEHAVIOR_CONSTRAINT, Define**) command.

Beginning constraint
Beginning constraint set number.

Ending constraint
Ending constraint set number.
(default is beginning constraint)

Increment
Increment between constraint sets.
(default is 1)

OP_CONTRIM

Analysis > OPTIMIZE/SENSITIVITY > BEHAVIOR_CONSTRAINT > Truncate Constraint

The OP_CONTRIM command specifies truncation (trim) factors for the unviolated constraints. Truncation factors are used to identify critical and potentially critical constraints for every optimization iteration. If the normalized value of a particular constraint is below the truncation factor value, this constraint is included in the list of critical constraints. Only the constraints in the critical constraint list are considered during a particular iteration. All other constraints are ignored by the optimizer for that stage. Prompts of the command may be specified in terms of parameters or parametric expressions. Values will change dynamically from one loop to another.

Minimum number of constraints retained at any stage
Minimum number of constraint sets to be retained at any stage. If, at any stage, the number of violated and potentially critical constraint sets determined by the truncation (trim) factors is less than the number specified by this parameter, more sets are added to the list of critical constraints until 'minimum number of constraints retained at any stage' is reached.
(default is 1)

Maximum number of constraints retained at any stage
Maximum number of constraint sets to be retained at any stage. If, at any stage, the number of critical constraint sets determined by the truncation (trim) factor is more than the number specified by this parameter, some sets (which are least critical in the list) are ignored. All violated VOLUME and WEIGHT constraints are retained even if their number exceeds the 'Maximum number of constraints retained at any stage'.
(default is maximum number of constraints defined)

Type of constraints to be trimmed

Type of constraints to which truncation (trim) factors are to be specified. If a WEIGHT or VOLUME constraint is defined, it is always classified as potentially critical.

= ALL	all types of defined constraints
= BUCKLING	linearized buckling load factor
= DISP	displacements
= FREQ	natural frequencies
= HFLUX	heat flux
= STRAIN	strains (see notes below)
= STRESS	stresses (see notes below)
= TEMP	temperatures
= TGRAD	temperature gradients
= USER	user-defined constraints
= VEL	velocity
= ACCEL	acceleration
= REACTION	reaction force

(default is ALL)

Initial value of truncation factor

Initial value for truncation (trim) factor.
(default is 0.5)

Lower bound of truncation factor

Lower bound of the truncation (trim) factor.
(default is 0.2)

Upper bound of truncation factor

Upper bound of the truncation (trim) factor.
(default is 0.8)

Truncation factor multiplier

Truncation (trim) factor multiplier.
(default is 0.8)

Notes

1. Truncation (trim) factors at subsequent design optimization iterations loops are determined by the following formula:

   Truncation factor at iteration (n) = min [ubd, max (lbd, x)]

   where: x = mult * Truncation factor at iteration (n-1)

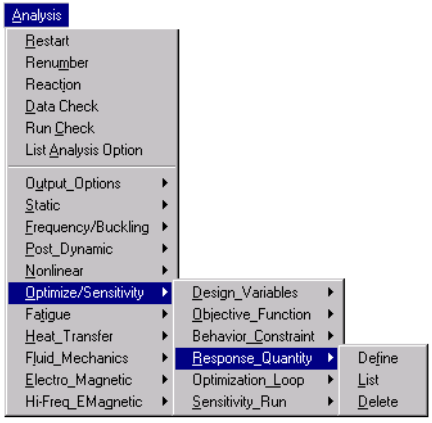
2. Higher values of truncation (trim) factors result in larger number of retained constraints.
3. The strain energy density and error in stresses components are always classified as potentially critical.




▼RESPONSE_QUANTITY Menu

Analysis > OPTIMIZE/SENSITIVITY > RESPONSE_QUANTITY

Figure 10-21 Response Quantity Menu



SN_RESPDEF

Analysis > OPTIMIZE/SENSITIVITY > RESPONSE_QUANTITY > Define

The SN_RESPDEF command defines nodal, elemental, global (frequency and linearized buckling load factor), and user-defined response quantities for sensitivity study. Nodes and elements may be specified directly or through the association of geometric entities. A reference keypoint is needed in case of using a relative displacement quantity.

General Command Syntax

Response set number

Response set number.
(default is highest response set defined + 1)

Type

Response type.

= -1: User	user-defined
= 0: Nodal	nodal quantity
= 1: Elem	elemental quantity
= 2: Freq	frequency
= 3: Buckling	linearized buckling load factor
	(default is nodal)

Parameters

The parameters depend on the response type. Detailed description of the syntax for each type is given below.

Command Syntax for User-Defined Response Quantities

Response set number

Response set number.
(default is highest response set defined + 1)

Type = user defined

Response type. Equals user for user-defined quantity.

Command Syntax for Nodal Response Quantities

Response set number

Response set number.
(default is highest response set defined + 1)

Type = nodal

Response type. Equals nodal for nodal quantity.

Nodal response name

Displacement components.

= UX	translation in the X-direction
= UY	translation in the Y-direction
= UZ	translation in the Z-direction
= RX	rotation about the X-direction
= RY	rotation about the Y-direction
= RZ	rotation about the Z-direction
= URES	magnitude of resultant displacement

Stress components.

= SX	normal stress in the X-direction
= SY	normal stress in the Y-direction
= SZ	normal stress in the Z-direction
= TXY	X-Y shear stress
= TXZ	X-Z shear stress
= TYZ	Y-Z shear stress
= P1	first principal stress
= P2	second principal stress
= P3	third principal stress
= VON	von Mises stress
= INT	stress intensity

Velocity components.

= VX	velocity in the X-direction
= VY	velocity in the Y-direction
= VZ	velocity in the Z-direction

= WX	angular velocity about X-direction
= WY	angular velocity about Y-direction
= WZ	angular velocity about Z-direction
= VRES	resultant velocity

Acceleration components.

= AX	acceleration in the X-direction
= AY	acceleration in the Y-direction
= AZ	acceleration in the Z-direction
= BX	angular acceleration about X-direction
= BY	angular acceleration about Y-direction
= BZ	angular acceleration about Z-direction
= ARES	resultant acceleration

Reaction forces.

= RFX	reaction force in the X-direction
= RFY	reaction force in the Y-direction
= RFZ	reaction force in the Z-direction
= RFRES	resultant reaction force
= RMX	reaction moment about the X-direction
= RMY	reaction moment about the Y-direction
= RMZ	reaction moment about the Z-direction
= RMRES	resultant reaction moment

Temperature.

= TEMP temperature

Temperature gradient components.

= GRADX	gradient in the X-direction
= GRADY	gradient in the Y-direction
= GRADZ	gradient in the Z-direction
= GRADN	normal gradient

Heat flux components.

= HFLUXX	heat flux in the X-direction
= HFLUXY	heat flux in the Y-direction
= HFLUXZ	heat flux in the Z-direction
= HFLUXN	magnitude of resultant heat flux *(default is VON)*

Fatigue.

= FTG	fatigue usage factor *(default is VON)*

analysis type

Analysis type from which the postprocessing function is obtained. This option is useful when the same type of postprocessing functions appear in more than one analysis. For example, von Mises stress VON appears in both linear static and nonlinear analyses.

= 0	linear static
= 1	nonlinear

= 3 thermal
= 7 fatigue
= 8 post dynamic
(default is 0)

layer number

Layer number on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.
(default is 1)

face flag

Face on which the stress or strain component is computed. The entry is prompted only when the stress or strain component is chosen as the comp_name and the option is available only for the composite shell elements.
= 0 top face
= 1 bottom face
= 2 membrane stresses
= 3 bending stresses
(default is 0)

Criterion flag

Criterion flag for the objective function.
= 0: MaxAbs maximum absolute value of nodal quantities
= -1: AlgMax algebraic minimum of nodal quantities
= -2: AlgMin algebraic maximum of nodal quantities
= -3: MaxNum maximum numerical value of nodal quantities (similar to option 0 above but with the original sign retained)
(default is MaxAbs)

Load case/time step: 0 = all

Primary load case or time step number. (enter "0" to consider all load cases or time steps)
(default is 1)

Entity type associated with response set

Entity type with which nodal response quantities are associated.
= PT keypoint
= CR curve
= CT contour
= SF surface
= RG region
= VL volume
= ND nodes
= PH polyhedra
= PA parts
(default is SF)

Beginning entity

Label of the beginning entity. When using the nodes/elements option with shape applications, it is recommended to set "beginning entity=1", unless the user is certain that nodes/elements labels will not change during the sensitivity study.

Ending entity

Label of the ending entity. When using the nodes/elements option with shape applications, it is recommended to set "ending entity=NDMAX/ELMAX: the highest nodes/elements labels", unless the user is certain that nodes/elements labels will not change during optimization loops.
(default is the highest entity label defined)

Increment

Increment between entity labels. When using the nodes/elements option with shape applications, it is recommended to set "increment=1", unless the user is certain that nodes/elements labels will not change during the sensitivity study.
(default is 1)

Reference keypoint

Label of a reference keypoint in case of using a relative displacement response quantity. This entry is prompted only in case of choosing UX, UY, UZ or URES. Enter "0" if no reference keypoint is used.
(default is 0)

Command Syntax for Elemental Response Quantities

Response set number

Response set number.
(default is highest response set defined + 1)

Type = elemental

Response type. Equals elem for elemental quantity.

Element response name

Stress components.

= SX	normal stress in the X-direction
= SY	normal stress in the Y-direction
= SZ	normal stress in the Z-direction
= TXY	X-Y shear stress
= TXZ	X-Z shear stress
= TYZ	Y-Z shear stress
= P1	first principal stress
= P2	second principal stress
= P3	third principal stress
= VON	von Mises stress
= INT	stress intensity
= FIND	failure index
= MFIND	maximum failure index from the layer specified below to the maximum layer number
= ILTXZ	interlaminar shear (X-Z plane)
= ILTYZ	interlaminar shear (Y-Z plane)
= MILTXZ	max interlaminar shear (X-Z plane)
= MILTYZ	max interlaminar shear (Y-Z plane)
= ERROR	error in stress




Strain components.

= EPSX	normal strain in the X-direction
= EPSY	normal strain in the Y-direction
= EPSZ	normal strain in the Z-direction
= GMXY	X-Y shear strain
= GMYZ	Y-Z shear strain
= GMXZ	X-Z shear strain
= ESTRN	effective strain
= SED	strain energy density

Weight or volume.

= VOLUME	volume
= WEIGHT	weight
	(default is VON)

analysis type

Analysis type from which the postprocessing function is obtained. This option is useful when the same type of postprocessing functions appear in more than one analysis. For example, von Mises stress VON appears in both linear static and nonlinear analyses.

= 0	linear static
= 1	nonlinear
= 3	thermal
= 8	post dynamic
	(default is 0)

Criterion flag

Criterion flag for the objective function.

= 0: MaxAbs	maximum absolute value of nodal quantities
= -1: AlgMax	algebraic minimum of nodal quantities
= -2: AlgMin	algebraic maximum of nodal quantities
= -3: MaxNum	maximum numerical value of nodal quantities (similar to option 0 above but with the original sign retained)
	(default is MaxAbs)

Load case/time step: 0 = all

Primary load case or time step number (enter "0" to consider all load cases or time steps). This flag is not used for WEIGHT and VOLUME options.
(default is 1)

Entity type associated with response set

Entity type with which these elemental response quantities are associated. Note that mass elements may only be associated with keypoints (PT), 1D elements may be associated with curves (CR) or contours (CT), 2D elements may be associated with surfaces (SF) or regions (RG), and 3D elements may only be associated with volumes (VL). Use EL to specify a pattern of elements.

= PT	keypoint
= CR	curve
= CT	contour
= SF	surface
= RG	region

= VL	volume
= EL	elements
= ND	noes
= PH	polyhedra
= PA	parts
	(default is SF)

Beginning entity

Label of the beginning entity. When using the nodes/elements option with shape applications, it is recommended to set “beginning entity=1” unless the user is certain that nodes/elements labels will not change during the sensitivity study.

Ending entity

Label of the ending entity. When using the nodes/elements option with shape applications, it is recommended to set “ending entitiy=NDMAX/ELMAX: the highest nodes/elements labels”, unless the user is certain that nodes/elements labels will not change during optimization loops.
(default is the highest entity label defined)

Increment

Increment between entity labels. When using the nodes/elements option with shape applications, it is recommended to set “increment=1”, unless the user is certain that nodes/elements labels will not change during the sensitivity study.
(default is 1)

Command Syntax for Frequency Response Quantities

Response set number

Response set number.
(default is highest response set defined + 1)

Type = frequency

Response type. Equals freq for frequencies.

Mode shape number

Mode shape number.

Command Syntax for Linearized Buckling Load Factor Response Quantities

Response set number

Response set number.
(default is highest response set defined + 1)

Type = buckling

Response type. Equals buckling for linearized buckling load factor.

SN_RESPLIST

Analysis > OPTIMIZE/SENSITIVITY > RESPONSE_QUANTITY > List

The SN_RESPLIST command lists response quantity sets defined previously by the SN_RESPDEF (**Analysis, OPTIMIZE/SENSITIVITY, RESPONSE_QUANTITY, Define**) command. The command lists response quantity sets, types, names, criterion flags, entity types with which response quantities are associated, and corresponding pattern information (beginning and ending entity labels, and increment), reference keypoint (in case of a relative displacement quantity), the current value of the response quantity, and the considered load case or time step.

Beginning response
Beginning response quantity set number.
(default is 1)

Ending response
Ending response quantity set number.
(default is the highest set label defined)

Increment
Increment between response sets.
(default is 1)

SN_RESPDEL

Analysis > OPTIMIZE/SENSITIVITY > RESPONSE_QUANTITY > Delete

The SN_RESPDEL command deletes response sets previously defined by the SN_RESPDEF (**Analysis, OPTIMIZE/SENSITIVITY, RESPONSE_QUANTITY, Define**) command.

Beginning response
Beginning response quantity set number.

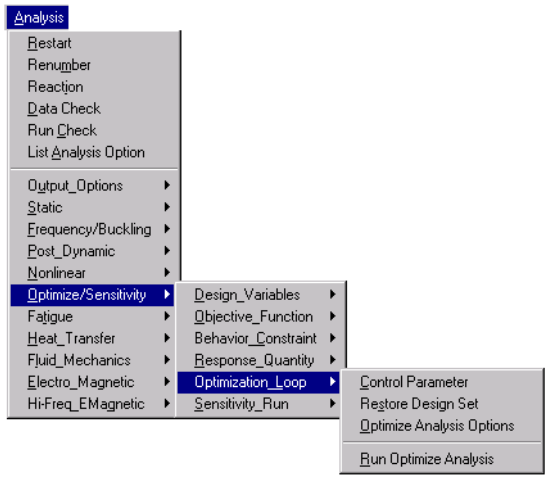
Ending response
Ending response quantity set number.
(default is beginning response)

Increment
Increment between response sets.
(default is 1)

▼OPTIMIZATION LOOP Menu

Analysis > OPTIMIZE/SENSITIVITY > OPTIMIZATION_LOOP

Figure 10-22 Optimization Loop Menu



OP_CONTROL

Analysis > OPTIMIZE/SENSITIVITY > OPTIMIZATION_LOOP > Control Parameter

The OP_CONTROL specifies parameters related to the numerical techniques used in optimization: the modified feasible directions and sequential linear programming methods and the singular value decomposition solver (SVD). Prompts of the command may be specified in terms of parameters or parametric expressions. Values will change dynamically from one loop to another.

Maximum SVD interactions

Maximum number of iterations allowed for the Singular Value Decomposition (SVD) solver.
(default is 30)

Threshold value for SVD solver

Ratio of the maximum singular value to set the threshold for singular values detected by the Singular Value Decomposition (SVD) solver.
(default is 1.0E-06)

Optimization algorithm

Optimization algorithm.

= 0: MFD Modified Feasible Directions
= 1: SLP Sequential Linear Programming (currently not recommended)
(default is MFD)

Maximum iterations for search directions

Maximum number of iterations for search direction of the Modified Feasible Directions optimizer.
(default is 100)

Maximum iterations MFD converge

Number of consecutive iterations in which convergence of the modified feasible directions algorithm should be satisfied.
= 1 comparison is made relative to one previous iteration
= 2 comparison is made relative to two previous iterations
= 3 comparison is made relative to three previous iterations
(default is 2)

Violation value for normalized constraints

Allowed violation value for normalized constraints.
(default is 0.003)

Ratio of active normalized constraints

Ratio for normalized constraints to be active.
(default is -0.03)

Relative change in the objective function

Relative change in the objective function between the modified feasible direction iterations.
(default is 0.001)

Absolute change in the objective function

Absolute change in the objective function between the modified feasible direction iterations.
(default is 0.001)

Maximum iterations for SLP method

Maximum number of iterations for the sequential linear programming method.
(default is 50)

No of consec iterations for SLP convergence

Number of consecutive iterations in which convergence of the sequential linear programming algorithm should be satisfied.
= 1 comparison is made relative to one previous iteration
= 2 comparison is made relative to two previous iterations
= 3 comparison is made relative to three previous iterations
(default is 2)




OP_RESTORE

Analysis > OPTIMIZE/SENSITIVITY > OPTIMIZATION_LOOP > Restore Design Set

The OP_RESTORE restores a given design set for which a solution was obtained during a previous successful optimization loop. The whole database will be reconstructed based on the specified design set and the analysis will be performed.

Optimization stage to restore
Label of the design set to be restored.

A_OPTIMIZE

Analysis > OPTIMIZE/SENSITIVITY > OPTIMIZATION_LOOP > Optimize Analysis Options

The A_OPTIMIZE command specifies the type of analysis to be performed, the optimization loop control parameters, output information of the optimization loop and the restart option.

Maximum number of optimization loops
Maximum number of the design optimization loops.
(default is 10)

Convergence check stages
Number of stages of convergence check for the design optimization loop.

= 1	convergence is achieved if the change in the objective function and design variables compared to the previous loop and the best design so far is less than the tolerance
= 2	convergence is achieved if the change in the objective function and design variables compared to two previous loops and the best design so far is less than the tolerance *(default is 1)*

No of consec. infeasible designs
Number of consecutive infeasible design sets before terminating the optimization loop. This flag is only needed when the initial design set is infeasible.

Output print flag
Flag for output information of the optimization loop written in the output file (.OPT).

= 0: Off	no information will be written
= 1: On	write information of design variables, the objective function and constraints *(default is on)*

Echo option flag

A flag to control the echo of commands in the dialogue area during model regeneration.

= 0: Off	do not echo on screen
= 1: On	echo on screen *(default is on)*

Restart option

Flag to activate or deactivate the restart option.

= 0: Off	deactivate the restart option (fresh start)
= 1: On	activate the restart option *(default is off)*

Type of analysis

Type of analysis performed within the optimization loop.

= NONE	no analysis to be executed
= STATIC	linear static analysis
= FREQUENCY	natural frequencies and mode shapes
= BUCKLING	linearized buckling
= THERMAL	heat transfer
= NONLINEAR	nonlinear structural analysis
= DYNAMIC	post-dynamic analysis
= STRESS	stress analysis
= FATIGUE	fatigue analysis
= FILE	user-defined sequence of analysis modules (“R_” commands) and other GEOSTAR commands stored in a file called GEOFILE.FIL in the local directory *(default is STATIC)*

Note

It should be mentioned that in case of using the FILE option in the A_OPTIMIZE () command, you must execute R_OPTIMIZE command interactively. You cannot include the R_OPTIMIZE (**Analysis, OPTIMIZE/SENSITIVITY, OPTIMIZATION_LOOP, Run Optimize Analysis**) command in an input file.

R_OPTIMIZE

Analysis > OPTIMIZE/SENSITIVITY > OPTIMIZATION_LOOP > Run Optimize Analysis

The R_OPTIMIZE command executes the design optimization process. Details of the analysis parameters may be specified by the A_OPTIMIZE (**Analysis, OPTIMIZE/SENSITIVITY, OPTIMIZATION_LOOP, Optimize Analysis Options**) and OP_CONTROL (**Analysis, OPTIMIZE/SENSITIVITY, OPTIMIZATION_LOOP, Control Parameter**) commands.

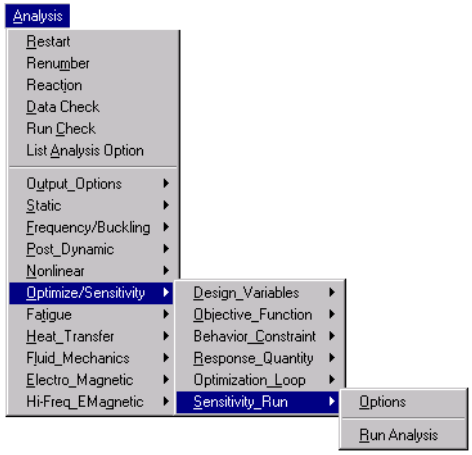
Note

It should be mentioned that in case of using the FILE option in the A_OPTIMIZE command, you must execute R_OPTIMIZE command interactively. You cannot include the R_OPTIMIZE command in an input file.

▼SENSITIVITY_RUN Menu

Analysis > OPTIMIZE/SENSITIVITY > SENSITIVITY_RUN

Figure 10-23 Sensitivity Run Menu



A_SENSITIV

Analysis > OPTIMIZE/SENSITIVITY > SENSITIVITY_RUN > Options

The A_SENSITIV command specifies type of sensitivity study and related parameters and the type of analyses to be performed.

Sensitivity type

Type of sensitivity study.

= 0: All	global - all design variables are incremented simultaneously
= 1: 1 by 1	global - design variables are incremented one at a time
= 2: Offset	offset sensitivity
= 3: Local	local sensitivity

(default is 1 by 1)

Number of increments

Number of increments in case of global sensitivity.
(default is 5)

Set number for local sensitivity

Set number in case of local sensitivity.
(default is 1)

Output print flag

Flag for output information of the sensitivity runs written in the output file (.OPT).

= 0: Off	no information will be written
= 1: On	write information of design variables, and the response quantities *(default is on)*

Echo option flag

A flag to control the echo of commands in the dialogue area during model regeneration.

= 0: Off	do not echo on screen
= 1: On	echo on screen *(default is on)*

Restart option

Flag to activate or deactivate the restart option. Whenever the restart flag is activated, the type of sensitivity as well as entries defined by the SN_RESPDEF, SN_SETDEF, and DVARDEF commands should keep their values identical to that in the preceding run.

= 0: Off	deactivate the restart option (fresh start)
= 1: On	activate the restart option *(default is off)*

Type of analyses

Type of analyses performed for the sensitivity study.

= NONE	no analysis to be executed
= STATIC	linear static analysis
= FREQUENCY	natural frequencies and mode shapes
= BUCKLING	linearized buckling
= THERMAL	heat transfer
= NONLINEAR	nonlinear structural analysis
= DYNAMIC	post-dynamic analysis
= STRESS	stress analysis
= FATIGUE	fatigue analysis
= FILE	user-defined sequence of analysis modules (“R_” commands) and other GEOSTAR commands stored in a file called GEOFILE.FIL in the local directory *(default is STATIC)*

Note

It should be mentioned that in case of using the FILE option in the A_SENSITIV command, you must execute the R_SENSITIV command interactively. You cannot include the R_SENSITIV command in an input file.

R_SENSITIV

Analysis > OPTIMIZE/SENSITIVITY > SENSITIVITY_RUN > Run Analysis

The R_SENSITIV command executes the sensitivity study process. Details of the analysis parameters may be specified by the A_SENSITIV command.

Note

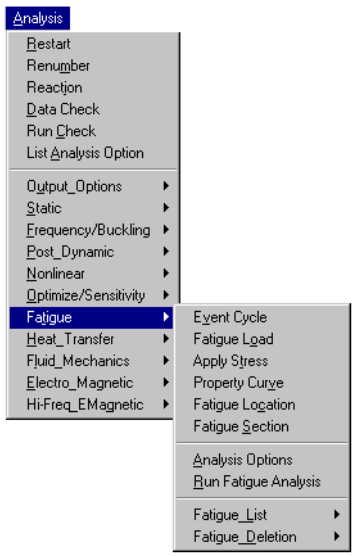
It should be mentioned that in case of using the FILE option in the A_SENSITIV command, you must execute the R_SENSITIV command interactively. You cannot include the R_SENSITIV command in an input file.

FATIGUE Menu

Analysis > FATIGUE

This menu contains commands related to FSTAR, the fatigue analysis module. FSTAR calculates the consumed life of a model due to cyclic loading events. Stress levels can be specified or read from other analyses. Refer to the COSMOS/M Advanced Modules User Guide for details.

Figure 10-24 Fatigue Menu



FT_EVENT

Analysis > FATIGUE > Event Cycle)

The FT_EVENT command specifies the number of cycles for a fatigue event. A maximum of 20 events can be defined.

Reference number for this event
Reference number for this event.
(default is the highest event label defined + 1)

Number of required cycles
Number of cycles for this event.
(default is 1)

Note
Repeat FT_EVENT command to define additional events, or modify the number of cycles for an already defined event.

Example: `FT_EVENT, 3, 500`
This command defines event 3 with a total of 500 cycles.

FT_LOAD

Analysis > FATIGUE > Fatigue Load

The FT_LOAD command defines a fatigue loading.

Reference number
Fatigue loading number.
(default is the highest loading defined + 1)

Associated event
Associated event.

Associated load case
Load case or time step associated with this fatigue loading. Admissible entries are:

= N	a positive integer that refers to the N-th load case (in case of linear analysis) or the time step (in case of nonlinear analysis) of the initial COSMOS/M structural solution
= -1	specifies that the stress conditions for this loading will be provided by the FT_STREAD command, instead of the currently in-memory data
= 0	considers zero stresses for this loading at all locations

Scale factor
Scale factor. (prompted only for load_case > 0)

Notes

1. Multiple fatigue loadings can be associated with the same load case or the time step.
2. Scale factor is a stress multiplier. It applies to stresses which are stored in the database as a result of the COSMOS/M structural solution.
3. At least two fatigue loadings are recommended for an event. They should approximately correspond to the situations where the response of the structure under the cyclic loading, is maximum or minimum (within that event). In situations where the maximum and minimum loadings are not well defined, assign more loadings within that event. Any additional loading not corresponding to the maximum alternating stress for that event is ignored in fatigue calculations.
4. Repeat the command to define additional loading or modify the loading specifications.
5. A particular fatigue loading label cannot be associated with more than one event, i.e., loading labels are unique.

Example 1: `FT_LOAD, 3, 5, 4, 2.`

This command assigns event 5 and load case (or time step) 4 to fatigue loading 3. The stresses from this load case are multiplied by a factor of 2.

Example 2: `FT_LOAD, 2, 3, -1`

This command assigns event 3 to fatigue loading number 2 and specifies that the stress conditions for this loading will be input through the FT_STREAD command.

Example 3: `FT_LOAD, 3, 4, 0`

This command assigns event 4 to loading 3 and specifies zero stress conditions at all locations for this loading.

FT_STREAD

Analysis > FATIGUE > Apply Stress

The FT_STREAD command defines stress conditions directly, instead of being read from an initial COSMOS/M structural solution. These stresses are defined for a particular location and fatigue loading.

Location label
Location label.

Fatigue loading label
Fatigue loading label.

Type
Type.
= 1: Actual stresses — for actual stresses
= 2: Linearized stresses — for linearized stresses
= 3: Nodal temperature — for nodal temperature
(default is actual stresses)

Normal stress in the X-direction
Actual/linearized normal stress in the X-direction.
(default is 0.0)

Normal stress in the Y-direction
Actual/linearized normal stress in the Y-direction.
(default is 0.0)

Normal stress in the Z-direction
Actual/linearized normal stress in the Z-direction.
(default is 0.0)

Shear stress TAU-XY
Actual/linearized shear stress TAU-XY.
(default is 0.0)

Shear stress TAU-XZ
Actual shear stress TAU-XZ. (prompted for if type = actual stresses)
(default is 0.0)

Shear stress TAU-YZ
Actual shear stress TAU-YZ. (prompted for if type = actual stresses)
(default is 0.0)

Temperature
Nodal temperature. (prompted for if type = nodal temperature)
(default is 0.0)

Notes

1. If a value of -1 is assigned for the associated load case argument in the FT_LOAD command, then the corresponding stress conditions must be specified by the FT_STREAD command.
2. This command may be used to redefine stress conditions specified previously, or it may be used to modify previously stored stresses.
3. Stress concentration factors (defined by the FT_LOC command) will apply to the actual stresses defined by this command as well as those read from the load cases (time steps). However the scale factors (defined by the FT_LOAD command) only apply to stress conditions stored as load cases (time steps).
4. Stress conditions defined by this command must be specified in the global Cartesian coordinate system.
5. The linearized stress option is only available for PLANE2D and axisymmetric problems. Only the first four stress components are prompted in such cases.

Example 1: `FT_STREAD, 4, 3, 1, 1., 5., -1., 4., -2., 6.`

This command assigns the specified six components of actual stresses for location 4 and fatigue loading 3.

Example 2: `FT_STREAD, 1, 1, 2, 2., 3., 2., 1.`

This command assigns the specified four in-plane components of linearized stresses for location 1 and fatigue loading 1.

Example 3: `FT_STREAD, 1, 2, 3, 100.0`

This command assigns 100.0 as the nodal temperature for location 1 and fatigue loading 2.

FT_CURDEF

Analysis > FATIGUE > Property Curve

The FT_CURDEF command defines an S-N curve to represent fatigue properties, an Sm_T curve or the elastic-plastic parameters M and N.

Curve/Prop item type number

Type of properties definition.

= 1 - 10	user defines the S-N curve (user will be prompted for ratio)
= 20	user defines Sm-T curve (user will be prompted for curve points)
= 30	for elastic-plastic material property (user will be prompted for M and N values)

(default is 1)

Stress ratio R

Stress ratio (min. stress divided by max. stress).
(default is -1.0)

Value_X1

Number of cycles at point 1 if 'curve/prop item type number' = 1 to 10, or Temperature value at point 1 if 'curve/prop item type number' = 20, or Value of parameter M if 'curve/prop item type number' = 30.

Value_Y1

Alternating_stress at point 1 if 'curve/prop item type number' = 1 to 10, or Design stress at point 1 if 'curve/prop item type number' = 20, or Value of parameter N if 'curve/prop item type number' = 30.

Value_X2

Number of cycles at point 2 if 'curve/prop item type number' = 1 to 20, or Temperature value at point 2 if 'curve/prop item type number' = 20.

Value_Y2

Alternating_stress at point 2 if 'curve/prop item type number' = 1 to 10, or Design stress at point 2 if 'curve/prop item type number' = 20.

Value_X3

Number of cycles at point 3 if 'curve/prop item type number' = 1 to 10, or Temperature value at point 3 if 'curve/prop item type number' = 20.

Value_Y3
Alternating_stress at point 3 if 'curve/prop item type number' = 1 to 10, or Design stress at point 3 if 'curve/prop item type number' = 20.

Value_X4
Number of cycles at point 4 if 'curve/prop item type number' = 1 to 10, or Temperature value at point 4 if 'curve/prop item type number' = 20.

Value_Y4
Alternating_stress at point 4 if 'curve/prop item type number' = 1 to 10, or Design stress at point 4 if 'curve/prop item type number' = 20.

Notes

1. The command can be repeated to define up to 20 points for the S-N curve, and up to 10 points for the Sm-T curve.
2. Since Log-Log interpolation is used in the program for S-N curves, neither cycle nor alternating stress takes a value of zero.
3. Cycles can be integers or real numbers.

Example 1:
```
FT_CURDEF, 1, -1, 3000., 10000., 50000., 5000., 100000., 3000.
```
This command defines an S-N curve with three points. The number of the cycles at the three points are 3000, 50000 and 100000 with corresponding alternating stresses of 10000.0, 5000.0 and 3000.0. The stress ratio of -1.0 corresponds to a fully reversible stress cycling.

Example 2:
```
FT_CURDEF, 20, 300., 20000. 400., 15000., 500., 13000.
```
This command defines an Sm-T curve with three points. The temperatures at these points are 300.0, 400.0, and 500.0 and the corresponding design stresses are 20000.0, 15000.0, and 13000.0.

FT_LOC

Analysis > FATIGUE > Fatigue Location

The FT_LOC command defines a fatigue location by specifying a node label. Stress concentration factors in the global directions can be specified. A maximum of 50 fatigue locations can be defined.

Fatigue location number
Fatigue location number.
(default is highest location defined +1)

Associated node label
Node label.

Stress concentration factor in X-direction
Stress concentration factor in the global X-direction.
(default is 1.0)

Stress concentration factor in Y-direction
Stress concentration factor in the global Y-direction.
(default is 1.0)

Stress concentration factor in Z-direction
Stress concentration factor in the global Z-direction.
(default is 1.0)

Notes

1. A fatigue location is a node in the structure for which fatigue calculations are made.
2. If a node is not pre-defined (i.e., when fatigue calculation is performed with no initial COSMOS/M structural solution), then the node label will be ignored in fatigue calculation.
3. Stress concentration factors apply to the normal components of the stress field (for the corresponding location).
4. Repeat the command to define new locations or modify already existing ones.

Example: `FT_LOC, 2, 7, 1.0, 3.0, 4.0`
This command assigns node 7 to location 2 and defines stress concentration factors of 1.0, 3.0 and 4.0 in the global X-, Y- and Z-directions, respectively.

FT_SEC

Analysis > FATIGUE > Fatigue Section

The FT_SEC command defines a section for simplified Elastic-Plastic formulation.

Section label
Section label.
(default is highest section defined +1)

First location
Location at one end of the section path.

Second location
Location at the other end of the section.

No. of path points along section path
Number of integration points along the path.
(default is 25)

Curv. rad. 0 = pl & 3d, > 0 = axisym, -1 = straight
In-plane average radius of curvature of the inside and outside surfaces of an axisymmetric section (see notes below).

Bending stress flag
Thickness direction bending stresses flag. (Prompted for only if radius of curvature is non-zero.)

= 0: Include	consider thickness direction bending stresses in fatigue analysis
= 1: Ignore	do not consider thickness direction bending stresses in fatigue analysis *(default is ignore)*

Notes

1. A very large radius of curvature (i.e., a flat surface) can be specified by entering a value of "-1" for the curvature.
2. For a Cartesian approximation, consider zero for the curvature. (Refer to the FATIGUE section of the COSMOS/M Advanced Modules User Guide.)
3. The two end locations must be defined at the time this command is issued by using the FT_LOC command.

Example: `FT_SEC, 1, 4, 5, 25, 10, 1`

Defines section 1 with locations 4 and 5 at the two ends. The number of integration points are 25 with a radius of curvature 10.0. Thickness direction bending stresses are ignored.

A_FATIGUE

Analysis > FATIGUE > Analysis Options

The A_FATIGUE command specifies the face and layer number of multi-layered shell or solid elements and activates the type of S-N curve for which fatigue analysis is to be performed.

Face flag for shell element

Face of shell elements.

= 0: T	top face
= 1: B	bottom face *(default is top face)*

Layer number

Layer number. (for multi-layered elements)
(default is 1)

SN curve flag

S-N curve.

= 0: User def.	user defined curve
= 1: Predefined option 1	predefined curve (option 1)
= 2: Predefined option 2	predefined curve (option 2) *(default is user def.)*

Note

The layer option is only applicable for the linear static runs. For nonlinear runs, layer number 1 is always considered.

Example:
```
A_FATIGUE, 1, 3, 1,
ACTSET, LOC, 4
R_FATIGUE
```
Fatigue analysis is performed on bottom face (shell elements only), and layer 3 (layered shell or solid elements). Calculations are performed at location 4. The user defined S-N curve is considered.

R_FATIGUE

Analysis > FATIGUE > Run Fatigue Analysis

The R_FATIGUE command performs fatigue calculations using the FATIGUE module of the COSMOS/M package.

Note

Recommended steps prior to running the FATIGUE module are:

a. Define the required events.

b. Define fatigue loadings and associated parameters.

c. Define fatigue locations and corresponding parameters.
(Not required for the all-nodes calculation option.)

d. Define an S-N curve.
(If no S-N curve is defined, the fatigue calculation will not produce usage factors.) (See the A_FATIGUE command for other options.)

e. Activate a location for fatigue calculation by using the ACTSET command. For the all-nodes option use: ACTSET, LOC, 0.

f. Run the FATIGUE module by executing the R_FATIGUE command.

g. List the fatigue results by using the FT_LIST command.

h. Repeat steps (e), (f) and (g) for other locations if desired.

(For the simplified elastic-plastic formulation, refer to the FSTAR section of the COSMOS/M User Guide Advanced Modules.)

Example 1:
```
ACTSET, LOC, 4
R_FATIGUE
```
Performs fatigue calculation at location 4.

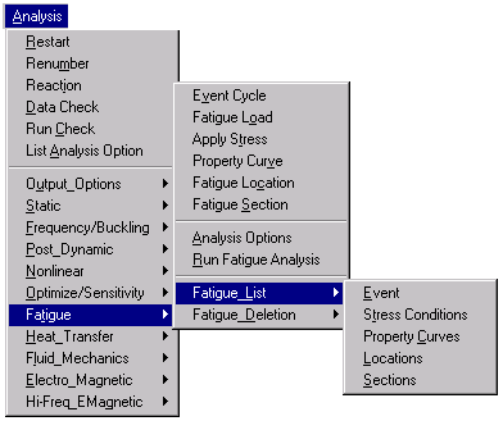
Example 2:
```
ACTSET, LOC, 0
R_FATIGUE
```
Performs fatigue calculations at all nodes.

▼FATIGUE_LIST Menu

Analysis > FATIGUE > FATIGUE_LIST

This menu contains commands for listing of fatigue related specifications.

Figure 10-25 Fatigue List Menu



FT_EVENTLIST

Analysis > FATIGUE > FATIGUE_LIST > Events

The FT_EVENTLIST command lists fatigue events and their specifications.

First event
Beginning event in the pattern.
(default is 1)

Last event
Ending event in the pattern.
(default is the highest event label defined)

Increment
Increment between events in the pattern.
(default is 1)

Note
Events with multiple loadings are sequentially listed.

Example: `FT_EVENTLIST, 4, 7, 1`
Lists event specifications for events 4, 5, 6 and 7 on the screen.

FT_STLIST

Analysis > FATIGUE > FATIGUE_LIST > Stress Conditions

The FT_STLIST command lists the stress conditions stored by the FT_STREAD command for a pattern of fatigue locations and a specified fatigue loading (or all defined fatigue loadings).

Beginning location
Beginning location in the pattern.
(default is 1)

Ending location
Ending location in the pattern.
(default is 50)

Increment
Increment between locations in the pattern.
(default is 1)

Loading label 0 = all, N = N_th loading
Fatigue loading.

= N	for loading N only
= 0	for all loadings *(default is 0)*

Item number
Item number.

= 1: Actual stress	for actual stresses
= 2: Linearized stress	for linearized stresses
= 3: Nodal temperature	for nodal temperatures

Notes

1. When zero is assigned for the loading, stress conditions are listed in groups of loadings. Each group of loading consists of stress conditions for a specified range of locations. These locations are listed in the first column under the heading “Locat” in the output file.
2. The normal stresses are listed under the headings, SX, SY and SZ, the shear stresses are listed under the headings TAU-XY, TAU-XZ and TAU-YZ.

Example 1: `FT_STLIST, 2, 5, 1, 0, 1`

This command lists the actual stresses for locations 2 through 5 for all fatigue loadings.

Example 2: `FT_STLIST, 2, 5, 1, 6, 2`

This command lists linearized stresses for locations 2 through 5 for fatigue loading 6.

FT_CURLIST

Analysis > FATIGUE > FATIGUE_LIST > Property Curves

The FT_CURLIST command lists all defined S-N and Sm-T fatigue property curves

Example: `FT_CURLIST`
This command lists all defined fatigue properties.

FT_LOCLIST

Analysis > FATIGUE > FATIGUE_LIST > Locations

The FT_LOCLIST command lists a pattern of fatigue locations and their corresponding specifications on the screen.

First location to be listed
Beginning location in the pattern.
(default is 1)

Last location to be listed
Ending location in the pattern.
(default is the highest location defined)

Increment
Increment between locations in the pattern.
(default is 1)

Notes

1. This command lists the coordinates of the corresponding nodes under the headings X, Y and Z and the associated coordinate systems under the heading CS. Dash lines under these headings mean that no corresponding nodes are defined in the database (i.e. analysis is not a part of initial COSMOS/M structural solution).
2. Headings SCFX, SCFY and SCFZ correspond to the stress concentration factors in the global X-, Y-, and Z-directions, respectively.

Example: `FT_LOCLIST, 4, 6, 1`
This command lists specifications for fatigue locations 4, 5 and 6.

FT_SECLIST

Analysis > FATIGUE > FATIGUE_LIST > Sections

The FT_SECLIST command lists a pattern of sections and their corresponding specifications on the screen.

First section
Beginning section in the pattern.
(default is 1)

Last section
Ending section in the pattern.
(default is the highest section defined)

Increment
Increment between sections in the pattern.
(default is 1)

Example: `FT_SECLIST, 1, 3, 1`
This command lists section specifications for sections 1, 2 and 3.

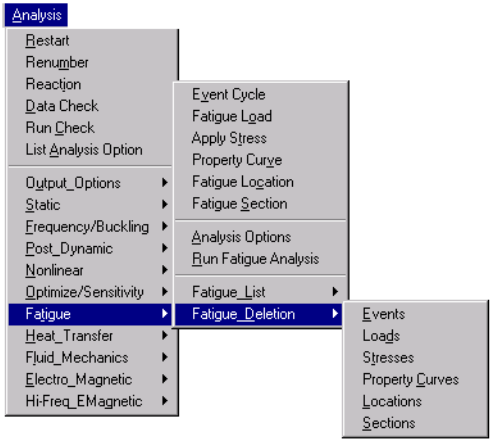



▼FATIGUE DELETION Menu

Analysis > FATIGUE > FATIGUE_DELETION

This menu contains commands related to deletion of fatigue related specifications.

Figure 10-26 FAtigue Deletion Menu



FT_EVENTDEL

Analysis > FATIGUE > FATIGUE_DELETION > Events

The FT_EVENTDEL command deletes a pattern of fatigue events from the database.

First event
Beginning event in the pattern.

Last event
Ending event in the pattern.

Increment
Increment between events in the pattern.
(default is 1)

Note
Use of this command deletes all parameters associated with this event, i.e., cycles and the fatigue loadings assigned to this event. (Assigned fatigue loadings are those specified by the FT_LOAD command.)

Example: `FT_EVENTDEL, 2, 5, 1`
This command deletes events 2, 3, 4, and 5 and their associated param-

eters including associated fatigue loadings.

FT_LOADDEL

Analysis > FATIGUE > FATIGUE_DELETION > Loads

The FT_LOADDEL command deletes a pattern of fatigue loadings from the database.

First loading
Beginning fatigue loading in the pattern.

Last loading
Ending fatigue loading in the pattern.

Increment
Increment between loadings in the pattern.
(default is 1)

Note
This command also disassociates the assigned load cases or time steps.

Example: `FT_LOADDEL, 2, 4, 1`
Deletes fatigue loadings 2, 3 and 4 and associated load case (time step) specifications.

FT_STDEL

Analysis > FATIGUE > FATIGUE_DELETION > Stresses

The FT_STDEL command deletes the stresses from a pattern of fatigue locations previously defined by the FT_STREAD command.

First location to be deleted
Beginning fatigue location in the pattern.

Last location to be deleted
Ending fatigue location in the pattern.

Increment
Increment between locations in the pattern.
(default is 1)

Fatigue loading label
Fatigue loading label.

Type
Type.
= 1: Actual stress for actual stresses

= 2: Linearized stress — for linearized stresses
= 3: Nodal temperature — for nodal temperatures
(default is actual stress)

Note

This command deletes only those stress conditions which were defined by the FT_STREAD command. Any modification in stress conditions stored as load cases (time steps) must be made using the FT_STREAD command.

Example 1: `FT_STDEL, 5, 8, 1, 2, 1`

This command deletes actual stresses stored by using the FT_STREAD command for fatigue loading 2 at locations 5 through 8.

Example 2: `FT_STDEL, 1, 3, 1, 4, 2`

This command deletes linearized stresses stored by using the FT_STREAD command for fatigue loading 4 at locations 1, 2 and 3.

Example 3: `FT_STDEL, 1, 3, 1, 4, 3`

This command deletes nodal temperatures stored by using the FT_STREAD command for fatigue loading 4 at locations 1, 2 and 3.

FT_CURDEL

Analysis > FATIGUE > FATIGUE_DELETION > Property Curves

The FT_CURDEL command deletes fatigue property specifications defined by the FT_CURDEF command.

Curve/Prop item type number

Type of properties definition.
= 1 - 10 — deletes S-N curve specification
= 20 — deletes Sm-T curve specification
= 30 — deletes elastic-plastic parameters M and N
(default is 1)

Example 1: `FT_CURDEL, 1`

This command deletes all previous assignments for the S-N curve, including user-defined and predefined curve.

Example 2: `FT_CURDEL, 2`

This command deletes Sm-T curve.




FT_LOCDEL

Analysis > FATIGUE > FATIGUE_DELETION > Locations

The FT_LOCDEL command deletes a pattern of fatigue locations from the database.

First location
Beginning location in the pattern.

Last location
Ending location in the pattern.

Increment
Increment between locations in the pattern.
(default is 1)

Note
This command deletes all parameters defined by the FT_LOC command for the specified locations (i.e., it deletes the stress factors and disassociates the corresponding nodes).

Example: `FT_LOCDEL, 3, 6, 1`
This command deletes locations 3 through 6 from the database.

FT_SECDEL

Analysis > FATIGUE > FATIGUE DELETION > Sections

The FT_SECDEL command deletes a pattern of sections from the database.

First section
Beginning section in the pattern.

Last section
Ending section in the pattern.

Increment
Increment between sections in the pattern.

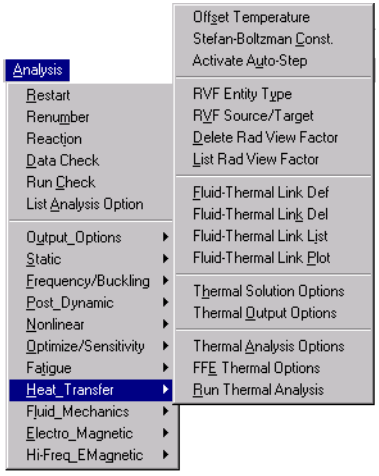
Example: `FT_SECDEL, 1, 3, 1`
This command deletes sections 1, 2 and 3 from the database.

HEAT_TRANSFER Menu

Analysis > HEAT_TRANSFER

This menu contains commands related to HSTAR and FFE Thermal, the heat transfer analysis modules. HSTAR solves linear and nonlinear problems involving conduction, convection and radiation. Refer to the COSMOS/M Advanced Modules User Guide for details. FFE Thermal combines state-of-the-art equation solving techniques and database structures to increase the speed and reduce required resources. (Refer to the FFE Thermal Manual for details.)

Figure 10-27 Heat Transfer Menu



TOFFSET

Analysis > HEAT_TRANSFER > Offset Temperature

The TOFFSET command specifies the temperature offset between absolute zero and zero of temperature system used.

Offset temperature

Difference in degrees between the absolute zero and zero of the temperature scale used.

= 460	for Fahrenheit system
= 273	for Celsius system
= T	offset is T (T is any number)

Note

The temperature is required in absolute scale when used in Radiation Heat Transfer.

Example:
```
TOFFSET, 273,
```
This command specifies the offset for Celsius (centigrade) scale.

SB_CONST

Analysis > HEAT_TRANSFER > Boltzmann Constant

The SB_CONST command specifies the Stefan Boltzmann constant.

Stefan Boltzmann constant
Stefan Boltzmann constant in the current system of units.

Note
The Stefan Boltzmann constant is required for all problems involving radiation.

AUTOSTEP

Analysis > HEAT TRANSFER > Activate Auto-Step

The AUTOSTEP command calculates and uses optimum time increments for transient heat transfer problems involving phase change. Results are printed at real time increments that correspond to the optimal solution time steps.

Reference temperature
Reference temperature for optimal time calculations.
(default is 100)

Time interval for printing
Used to calculate the number of time steps for which results will be printed. Must be larger than the time increment specified in the TIMES command.

Notes
1. The number of optimal steps (steps at which results are printed) is calculated by dividing the total time specified in the TIMES command by 'time interval for printing'.
2. Latent heat is calculated from the specific heat and the active temperature curve. The user should define and activate a temperature curve, define the specific heat and issue the AUTOSTEP command.
3. A temperature curve is always required (refer to verification problem TN04 in the COSMOS/M Advanced Modules User Guide).

Example:
```
ACTSET, MP, 3,
MPROP, 1, C, 1.0,
AUTOSTEP, 100, 5.0,
TIMES, 0, 10, 0.1,
A_THERMAL, T;
```
This example specifies a transient phase change problem with 100

time steps. A reference temperature of 100 degrees is used for calculating the optimum time increment. The results are printed at every 5 units of time.

RVFTYP

Analysis > HEAT_TRANSFER > RVF Entity Type

The RVFTYP command specifies various options associated with the calculations of radiation view factors.

View factor calculation flag

View factor calculation flag.

= 1: Yes	calculate view factors
= 0: No	use the view factors calculated in a previous run
	(default is yes)

Type of geometry

Type of geometry.

= 0: Plane 2D	plane 2D geometry
= 1: 3D	3D geometry
= 2: Axi	Axisymmetric geometry
	(default is 2D)

View factor print

Flag for printing view factors in the output file.

= 0: No	do not print view factors
= 1: Yes	print view factors
	(default is no)

Number of contour divisions

Number of contour divisions. Use 0 to 20, where 0 indicates auto division by the program.
(default is 0)

Number of area divisions

Number of area divisions, maximum is 20.
(default is 5)

System type

Type of system.

= 0: Open	open system
= 1: Closed	closed system
	(default is open)

Environment temperature

Environment temperature.
(default is 0)

Segments for axisym. radiation surface

Number of segments (in the azimuthal direction for an axisymmetric radiation element).
(default is 24)

Segments for axisym. blocking surface
Number of segments (in the azimuthal direction for an axisymmetric blocking element).
(default is 8)

Tolerance for 3D adaptive calculation
Tolerance value for 3D adaptive calculation.
(default is 0.01)

Notes

1. View factors need to be calculated only during the initial run and if there are no changes in the geometry, meshing or view factor definitions for the subsequent runs, view factor calculation flag can be turned off in which case HSTAR will use the existing values (from a previous run) in the database. Note that the view factors are functions of the geometry and the mesh only and are not dependent on material properties, loading, or boundary conditions.
2. Contour and area divisions are used for the line and area integrations for each element face and may be increased to improve accuracy.

Example: `RVFTYP,1,1,1,3,6,0,30.0,24,8`

This command specifies that view factors are to be calculated in a 3D geometry and the calculated view factors will be printed in the output file. The number of contour divisions for each radiation element is equal to 3 and the number of area divisions is set to 6. The radiation surfaces form an open system (they exchange thermal radiation with the ambient atmosphere) and the environment temperature is 30 degrees.

RVFDEF

Analysis > HEAT TRANSFER > RVF Source/Target

The RVFDEF command specifies a radiation set consisting of a source entity and a pattern of target entities. The command may also be used to specify a pattern of blocking entities. Blocking geometric entities stand between the source and target entities and reduce the view factors. The view factors are calculated between each element associated with the source entity and each element associated with the pattern of target entities. If blocking is to be considered, the user should first issue the RVFDEF command to define a new set, specify the source and target entities, and activate the blocking option, then the user should issue the RVFDEF command again to specify a pattern of blocking entities. The type of target and blocking entities should be specified through the geometry option in the RVFTYP command; if the geometry is set to plane 2D, then the target and blocking entities are curves, and if the geometry is set to 3D, then the target and blocking entities are surfaces or regions.

Radiation set
Label of radiation set. Limit is 100 sets.
(default is highest defined + 1)

Source geometric entity type
Type of source entity.
= 0: CR curve
= 1: SF surface
= 2: RG region
(default is CR)

Source entity
Label of source entity.

Type
Flag to specify a pattern of target or blocking entities for the specified source.
= 0: Target target entities will be specified
= 1: Blocker blocking entities will be specified
(default is target)

Target blocking geometric entity type
Type of target or blocking entity.
= 0: CR curve
= 1: SF surface
= 2: RG region
(default is CR)

Beginning entity
Label of beginning target or blocking entity.

Ending entity
Label of ending target or blocking entity.
(default is beginning entity)

Increment
Increment between target or blocking entities in the pattern.
(default is 1)

Blocking flag
Flag to consider blocking.
= 0: No do not consider blocking
= 1: Yes consider blocking
(default is no)

Notes
The RVFTYP command must be issued prior to issuing the RVFDEF command when view factors for 3D or axisymmetric geometry need to be calculated.

Example:
```
RVFDEF,1,1,6,0,1,9,9,1,0,
```
The above commands calculate view factors between all elements of surfaces 6 and 9.




RVFDEL

Analysis > HEAT_TRANSFER > Delete Rad View Factor

The RVFDEL command deletes a pattern of radiation sets from the database previously defined by the RVFDEF command.

Beginning set
Beginning radiation set in the pattern.

Ending set
Ending radiation set in the pattern.
(default is beginning set)

Increment
Increment between radiation sets in the pattern.
(default is 1)

Example: `RVFDEL, 2, 4, 2,`
This command deletes the second and fourth radiation sets.

RVFLIST

Analysis > HEAT_TRANSFER > List Rad View Factor

The RVFLIST command lists previously specified radiation sets.

Beginning set
Beginning radiation set in the pattern.
(default is 1)

Ending set
Ending radiation set in the pattern.
(default is max. label in the database)

Increment
Increment between radiation sets in the pattern.
(default is 1)

Example: `RVFLIST, 2, 4, 2,`
This command lists the second and fourth radiation sets.

FLINKDEF

Analysis > HEAT_TRANSFER > Fluid-Thermal Link Def

The FLINKDEF command defines a set of convection links between a source curve (fluid) and a pattern of target surfaces or regions (thermal solid).

Fluid-thermal link set
Fluid-thermal link set number.
(default is the current maximum defined set number + 1, the limit is 1000)

Source curve
Label of source curve.
(no default for a new set. For an existing set, the default is the current source curve associated with the set)

Target geometric type
Geometric type of targets to be entered.
= 0: Surface
= 1: Region

Beginning Surface(or Region)
Beginning target surface (or region) in the pattern.

Ending Surface(or Region)
Ending target surface(or region) in the pattern.
(default is beginning surface or region)

Increment
Increment between surfaces or regions in the pattern.
(default is 1)

Notes

1. This command can be repeated to define new sets or add more surfaces or regions to a predefined set.
2. Target surfaces/regions in a set are all associated with the same (single) source curve. A target surface/region should not be included in more than one set.
3. A maximum of 50 surfaces or regions can be defined for each fluid-thermal link set.

Example: `FLINKDEF, 1, 1, 0, 2, 2, 1`

This command defines a fluid-thermal link set that associates the source curve number 1 to a target surface number 2.

FLINKDEL

Analysis > HEAT_TRANSFER > Fluid-Thermal Link Del

The FLINKDEL command deletes a pattern of fluid-thermal link sets defined in conjunction with FLUIDT elements.

Beginning set
Beginning fluid-thermal link set in the pattern.

Ending set
Ending fluid-thermal link set in the pattern.
(default is beginning set)

Increment
Increment between sets in the pattern.
(default is 1)

Example: `FLINKDEL, 2, 4, 1`
This command deletes fluid-thermal link sets 2, 3, and 4.

FLINKLIST

Analysis > HEAT_TRANSFER > Fluid-Thermal Link List

The FLINKLIST command lists a pattern of fluid-thermal link sets defined in conjunction with FLUIDT elements.

Beginning set
Beginning fluid-thermal link set in the pattern.

Ending set
Ending fluid-thermal link set in the pattern.
(default is beginning set)

Increment
Increment between sets in the pattern.
(default is 1)

Example: `FLINKLIST, 3, 7, 2`
This command lists fluid-thermal link sets 3, 5, and 7.

FLINKPLOT

Analysis > HEAT_TRANSFER > Fluid-Thermal Link Plot

The FLINKPLOT command plots a pattern of fluid-thermal link sets defined in conjunction with FLUIDT elements.

Beginning set
Beginning fluid-thermal link set in the pattern.

Ending set
Ending fluid-thermal link set in the pattern.
(default is beginning set)

Increment
Increment between sets in the pattern.
(default is 1)

Example: `FLINKPLOT, 3, 3, 1`
This command plots fluid-thermal link set 3.




HT_SOLN

Analysis > HEAT_TRANSFER > Thermal Solution Options

The HT_SOLN command specifies the numerical algorithm to be used in the non-linear thermal analysis.

Solution type

Solution technique for the nonlinear analysis.

= 0: MNR	Modified Newton-Raphson technique
= 1: NR	Newton-Raphson technique
	(default is MNR)

Line search flag

Line search flag.

= 0: No	no line search
= 1: Yes	line search is to be performed
	(default is no)

Search tolerance

Search tolerance.
(default is 0.5)

Notes

1. In the regular Newton-Raphson method, the conductivity matrix is formed and decomposed at each iteration during a particular time step. For this reason the regular Newton-Raphson method has more chances to converge but would in general take more time although less number of iterations is usually needed.
2. In the Modified Newton-Raphson method, the conductivity matrix is formed and decomposed at the user-specified reformation interval. The computed conductivity is then used throughout that interval during the iteration process.
3. The NR and the MNR methods are equivalent if no equilibrium iterations are specified.

HT_OUTPUT

Analysis > HEAT_TRANSFER > Thermal Output Options

The HT_OUTPUT command controls the interval of time steps for printing in the output file and storing results for later plotting.

Print interval

Output print frequency.

= N	results are printed every Nth time step in the output file
	(default is 1 for every time step)

Plot interval

Plotting frequency.

= N plotting information is stored every Nth time step
(default is 1 for every time step)

Gradient/Heat flow calculation

Gradient and heat flux calculation flag.

= 0: No gradient and heat flux are not calculated
= 1: Yes gradient and heat flux are calculated
(default is no)

Example: `HT_OUTPUT, 2, 4, 0`

This command instructs HSTAR to print results every other time step (steps 2, 4, 6, etc.) and to write plotting information for time steps 4, 8, 12, etc. Temperature gradient and heat flux are not to be calculated.

A_THERMAL

Analysis > HEAT TRANSFER > Thermal Analysis Options

The A_THERMAL command specifies the details of the thermal analysis to be performed by HSTAR. The R_THERMAL command will run the analysis. Use the A_FFETHERMAL command to specify options for the COSMOS/FFE Thermal module.

Analysis option

Thermal analysis option.

= S steady state analysis using HSTAR
= T transient analysis using HSTAR

The following options are not needed and were removed from the dialog box. They are however still available in the command line and recognizable by the program for backward compatibility.

= FS steady state analysis using the FFE Thermal module
= FT transient analysis using the FFE Thermal module
= FTN nonlinear transient analysis using the FFE Thermal module
= FSN nonlinear steady analysis using the FFE Thermal module
(default is S)

Convergence tolerance

Convergence tolerance.
(default value is 0.001)

Iteration between reforming stiffness

Number of iterations between reforming the stiffness matrix.
(default is 1)

Time steps between equilibrium iterations

Number of time steps between equilibrium iterations.
(default is 1)

Max. equilibrium iterations
Maximum number of equilibrium iterations at each time step.
(default is 20)

Mass matrix flag
Mass matrix formulation flag.
= 0: Lumped lumped mass matrix
= 1: Consistent consistent mass matrix
(default is lump)

Under relaxation factor
Under relaxation factor.
(default is 1)

Penalty parameter
Penalty parameter.
(default is 1.0E8)

Thermo-electric analysis option
Thermo-electric coupling flag.
= 0: No no coupling is considered
= 1: Yes coupling is considered
(default is no)

Convergence tolerance for thermo-electric iteration
Convergence tolerance for thermo-electric iterations. Prompted only if the thermo-electric analysis option is used.
(default is 0.001)

M. thermo-electric iterations
Number of thermo-electric iterations. Prompted only if the thermo-electric analysis option is used.
(default is 20)

Voltage print flag
Voltage print flag. Prompted only if the thermo-electric analysis option is used.
= 0: No voltage printout is suppressed
= 1: Yes voltage printout is activated
(default is no)

Scaling Factor
Underrelaxation factor used for thermo-electric iterations. Prompted only if the thermo-electric analysis option is used.
(default is 1.0)

Solver option
= 0: Sparse Use the sparse direct solver.
= 1: Skyline Use the skyline direct solver.
= 2: Iterative (PCG) Use the PCG iterative solver.
(default is Sparse)

Change to 2nd Order

This flag controls the formulation of first-order elements (i.e. elements without mid-side nodes). When the flag is activated, the formulation of first-order will be changed to second-order formulation based on mid-side nodes along straight edges. Existing second-order elements (i.e. elements with existing mid-side nodes) will be used as is. This command cannot be used to change the formulation of second-order elements to first order.

= 0: off	Do not use second-order formulation for first-order elements
= 1: on	Use second-order formulation for first-order elements *(Default is off)*

Notes

1. Radiation problems, being of highly nonlinear nature, present convergence problems. The under relaxation factor (Factor) may be used to remedy such situations. The default value of the under relaxation factor is set to 1.0. If a radiation problem does not converge using the default value, you may set it to 0.5 and run the problem again. If the problem still does not converge, rerun the problem twice using values of 0.25 and 0.75. The relative norm of the incremental temperature at each Newton-Raphson iteration is printed in the .RSD file. Inspection of these values generally gives you a clue about how you should go about changing the value of the under relaxation factor to achieve convergence.
2. You should be careful in providing the proper value for Stefan-Boltzmann constant consistent with the units used in the problem and verify that the environment temperature is specified if the radiation system being modeled is also exchanging energy with the ambient atmosphere.
3. If there are some surfaces that are not included in the radiation exchange (through the use of RVFDEF) because they exchange radiation only with ambient atmosphere, verify that the radiation boundary condition (through the use of RECR or RESF) is specified on these surfaces.
4. There is no need to define radiation between a source and target as a two-way radiation as this is automatically done by HSTAR.
5. When solving a steady state problem with multiple unconnected bodies, a proper “temperature anchor” is needed to get physically meaningful solution. A proper temperature anchor could be a temperature boundary condition, convection boundary condition or a radiation boundary condition defined by RECR or RESF.

Example: `A_THERMAL, T, 0.0001, , , , ,`

This command specifies transient thermal analysis and sets the convergence tolerance to 0.0001. All other defaults are accepted.




A_FFETHERMAL

Analysis > HEAT_TRANSFER > FFE Thermal Options

The A_FFETHERMAL command specifies analysis options for heat transfer analysis using the FFE Thermal module. Note that the A_THERMAL command specifies analysis options for heat transfer analysis using the HSTAR module.

Analysis option

Type of analysis to be performed.

= S	steady state
= T	transient
	(default is S)

Element order

Order of the element to be used. Regardless of the element group name in the database, you may specify through this option whether first (linear) or second (parabolic) elements will be used. As an example, if you define TETRA4 elements and use second order, middle nodes on straight edges will be considered during analysis. On the other hand you may define TETRA10 elements and specify to use first order. SOLID elements are treated similarly except that for SOLID elements, the same element group name is used for both first and second orders.

= 1: First	use first order for continuum elements
= 2: Second	use second order for continuum elements
	(default is second)

Convergence tolerance

Convergence tolerance for nonlinear analysis.
(default is 0.001)

Unused option

Unused option.

Formulation flag

Mass-matrix formulation used for transient analysis. It also affects matrix-formulation for convection and radiation.

= 0: Lumped	lumped (compatible with HSTAR; ignored if selected with second-order elements)
= 1: Cons	consistent
	(default is cons)

R_THERMAL

Analysis > HEAT_TRANSFER > Run Thermal Analysis

The R_THERMAL command performs thermal analysis using HSTAR or FFE Thermal. The command runs the FFE Thermal module if the A_FFETHERMAL command has been issued and was not followed by the A_THERMAL command. Use A_FFETHERMAL for the COSMOS/FFE Thermal module and the A_THERMAL command for HSTAR.

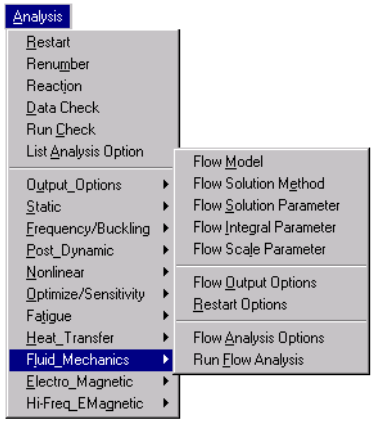



FLUID_MECHANICS Menu

Analysis > FLUID_MECHANICS

This menu contains commands related to FLOWSTAR, the fluid mechanics analysis module. FLOWSTAR solves 2D and 3D steady state and transient fluid flow problems. Thermal effects can also be included. Refer to the COSMOS/M Advanced Modules User Guide for details.

Figure 10-28 Fluid Mechanics Menu



FL_MODEL

Analysis > FLUID_MECHANICS > Flow Model

The FL_MODEL command specifies the type of fluid (Newtonian or non-Newtonian). In the case of a non-Newtonian fluid, the power law model is used.

Model

Fluid model.

= 0: Newtonian	Newtonian fluid
= 1: Non-Newtonian	Non-Newtonian fluid
	(default is Newtonian)

Material set number

Material property set label.
(default is 1)

1st parameter value

Value of first parameter.
(default is 1.0)

1st temperature curve label
Temperature curve associated with first parameter.
(default is 0)

2nd parameter value
Value of second parameter.
(default is 1.0)

2nd temperature curve label
Temperature curve associated with second parameter.
(default is 0)

Example: `FL_MODEL, 1, 3, 1.0, 0, 0.5, 0`

This command specifies that the fluid is non-Newtonian and the values of two parameters are 1.0 and 0.5 respectively. No temperature curves are associated with these parameters.

FL_METHOD

Analysis > FLUID_MECHANICS > Flow Solution Method

The FL_METHOD command specifies the type of the theoretical formulation to be used in fluid flow analysis.

Formulation
Formulation type.

= 0: PFR	penalty function formulation using FLOWSTAR
= 1: VP	velocity-pressure formulation using FLOWSTAR
= 2: SegSol	segregated solver using FLOWPLUS
	(default is SegSol)

Penalty value
Value of the penalty parameter.
(default is 1.0E7)

Method
Solution method for FLOWSTAR. The streamline upwind method is always used in FLOWPLUS.

= 0: Std Galerkin	standard Galerkin method
= 1: Upwind	streamline upwinding method
	(default is std Galerkin)

Example: `FL_METHOD, 0, 1.0E5, 0,`

This command specifies the use of the Penalty function formulation with 1.0E5 as the penalty parameter and the use of the standard Galerkin method in FLOWSTAR.

FL_SOLN

Analysis > FLUID_MECHANICS > Flow Solution Parameter

The FL_SOLN command specifies the parameters related to the solution procedures used in FLOWSTAR. The solution methods that are used to solve the nonlinear equations are the Picard iteration method and the Newton-Raphson method.

Maximum picard iterations
Maximum number of Picard iterations.
(default is 0)

Scale factor for picard iterations
Scaling factor used in Picard iteration.
(default is 1.0)

Maximum N-R iterations
Maximum number of Newton-Raphson iterations.
(default is 15)

Scale factor for N-R iterations
Scaling factor used in N-R iteration.
(default is 1.0)

Iterations between reforming stiffness
Iterations between reforming stiffness matrix in the Newton-Raphson method.
(default is 1)

Convergence tolerance
Convergence tolerance.
(default is 0.01)

Example: `FL_SOLN, 0, 1.0, 10, 1.0, 1, 0.001`

This command specifies that Newton-Raphson method be used for iterative solution and the maximum number of iterations allowed is 10 with a convergence tolerance of 0.001.

FL_INTGR

Analysis > FLUID_MECHANICS > Flow Integral Parameter

The FL_INTGR command specifies various parameters associated with the time integration of transient fluid flow equations.

Number of time steps
Total number of time steps.
(default is 1)

Courant number
Courant number.
(default is 0.8)

Order of time integration
Order of time integration.
= 1 first order time integration
= 2 second order time integration
(default is 1)

Time stepping
Type of time stepping.
= 2 global time stepping
(default is 2)

Time steps between residual calculation
Number of time steps between residual calculation.
(default is 1)

Time steps between delta-t calculation
Number of time steps between Delta -t calculation.
(default is 1)

Example: `FL_INTGR, 2, 2, 10, 15`
This command specifies second order time integration with global time stepping. The number of time steps between residual calculation is 10 and the number of time steps between Delta-t calculation is 15.

FL_SCALES

Analysis > FLUID MECHANICS > Flow Scale Parameter

The FL_SCALES command specifies the reference values used in the non-dimensionalization of the governing equations of the compressible fluid flow analysis.

Reference length
Reference length.
Reference density
Reference density.
Reference velocity
Reference velocity.
Reference pressure
Reference pressure.
Reference temperature
Reference temperature.

Example: `FL_SCALES, 10.0, 1.0, 500.0, 1.0, 200.0`
This command specifies a reference length of 10.0, reference density of 1.0, reference velocity of 500.0, reference pressure of 1.0 and reference temperature of 200.0.

FL_OUTPUT

Analysis > FLUID MECHANICS > Flow Output Options

The FL_OUTPUT command controls the interval at which output is printed or plotted in FLOWSTAR and FLOWPLUS. In FLOWSTAR, interval refers to the

number of time steps and in FLOWPLUS, it refers to the number of sequential solution iterations.

Print interval
Output print flag.
= N output is printed for every N time steps or sequential solution iterations
(default is 1)

Plot interval
Plotting frequency. Note In FLOWPLUS, plotting information is available for the last sequential solution iteration.
= N output plotting can be done for every N time steps
(default is 1)

Stream function calculation flag
Stream function calculation flag.
= 1: Yes stream function is calculated
= 0: No stream function is not calculated
(default is yes)

Summary interval
Summary print flag.
= N summary is printed once every N sequential solution iterations
(default is 10)

Debug interval
Debug print flag.
= N debug information is printed for every N sequential solution iterations
(default is 0)

Example:
```
FL_OUTPUT, 2, 4, 0, 10, 0
```
This command instructs FLOWSTAR to print results every other time step or sequential solution iteration and to write information in plot file every four time steps. Stream function calculation is not done. Summary information is written every 10 sequential solution iterations and debug information is not written.

FL_RESTART

Analysis > FLUID_MECHANICS > Restart Options

The FL_RESTART command determines which type of input data can be read from restart files for the FLOWPLUS module. Use the RESTART command for the FLOWSTAR module.

Geometry restart flag
Geometry restart flag.

= 0: No	analysis model geometry has been modified. Required geometry input will be recreated
= 1: Yes	analysis model geometry has not changed. Restart from previous geometry

Boundary conditions restart flag
Boundary conditions restart flag.

= 0: No	analysis model boundary conditions have been modified. Required boundary conditions input will be recreated
= 1: Yes	boundary conditions have not been modified. Restart from previous boundary conditions

A_FLOW

Analysis > FLUID_MECHANICS > Flow Analysis Options

The A_FLOW command specifies the details of the fluid flow analysis to be performed by FLOWSTAR or FLOWPLUS using the R_FLOW command.

Analysis option
Time dependence flag.

= S	steady state flow
= T	transient fluid flow
	(default is S)

Isothermal flag
Isothermal fluid flow flag.

= 0: Isotherm	isothermal fluid flow
= 1: Non-isotherm	non-isothermal fluid flow
	(default is Isotherm)

Convection flag
Convection flag.

= 0: Forced	forced convection
= 1: Natural	natural convection
	(default is forced)




Compressibility flag

Compressibility flag.

= Incompr	incompressible flow
= Compr	compressible flow *(default is incompr)*

Viscous heat generation

Viscous heat generation flag.

= 0: No	viscous heat generation is not considered
= 1: Yes	viscous heat generation is considered *(default is no)*

State of flow flag

State of flow flag.

= 0: Laminar	laminar fluid flow
= 1: Turb	turbulent fluid flow *(default is laminar)*

Turbulence model

Turbulence model flag.

= 0: K-eps	k-eps model using FLOWPLUS
= 1: RNG	RNG model (Renormalization Group Theory) using FLOWPLUS
= 2: Algebraic	algebraic model using FLOWSTAR *(default is k-eps)*

Example 1: `A_FLOW, S,1,0,0,0,0,0,`

This command instructs FLOWSTAR to do steady, incompressible laminar and non-isothermal fluid flow analysis. Forced convective flow is assumed and viscous heat generation is neglected.

Example 2: `A_FLOW, S,1,0,0,0,1,1,`

This command instructs FLOWPLUS to perform a steady-state, incompressible, turbulent flow analysis including heat transfer effects. FLOWPLUS only uses the values of 'isothermal flag', 'state of flow flag', and 'turbulence model'.

R_FLOW

Analysis > FLUID_MECHANICS > Run Flow Analysis

The R_FLOW command executes the FLOWSTAR or FLOWPLUS program to perform fluid flow analysis.

Note

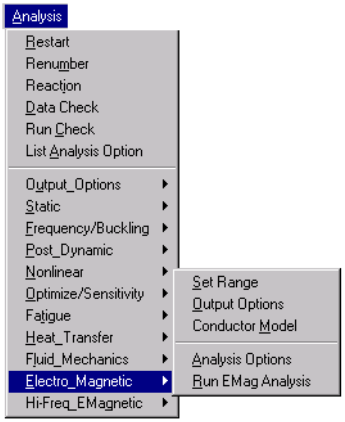
Uses options specified by the A_FLOW command. Default options are used if the A_FLOW command was not issued.

ELECTRO_MAGNETIC Menu

Analysis > ELECTRO_MAGNETIC

This menu contains commands related to ESTAR, the electromagnetic analysis module. ESTAR solves steady and transient 2D and 3D electromagnetics problems. Refer to the COSMOS/M Advanced Modules User Guide for details.

Figure 10-29 Electromagnetic Menu



EM_FRQRANGE

Analysis > ELECTRO_MAGNETIC > Set Range

The EM_FRQRANGE command specifies the frequencies for A. C. Eddy current calculations.

Starting frequency
Starting frequency in Hz for A.C. Eddy current calculations.
(default is 0.0)

Final frequency
Ending frequency in Hz for A.C. Eddy current calculations.
(default is 0.0)

Frequency increment
Frequency increment.
(default is 0.0)

Example: `EM_FRQRANGE, 60, 120, 20,`
This above command specifies that A.C. Eddy current calculations be performed at frequencies 60, 80, 100, and 120 Hz.

EM_OUTPUT

Analysis > ELECTRO_MAGNETIC > Output Options

The EM_OUTPUT command controls the printing intervals of the results and assigns values for the necessary parameters of the postprocessing calculations.

Magnetic field info flag 0 = No N = steps

Electromagnetic field print flag.

= N	print electromagnetic field information at N time step intervals (N is any positive integer) *(default is 1)*

Postprocessing info flag

Postprocessing results print flag.

= 1: Yes	calculates and prints the postprocessing results such as forces, torque, eddy currents, power loss, etc. at N time step intervals (N is a positive integer)
= 0: No	postprocessing results are not obtained *(default is no)*

The following options are prompted only if N is not equal to 0 for the postprocessing info flag.

Force calculation

Flag for the method of force calculation.

= 0: Standard	the Lorentz forces are calculated
= 1: Virtual	the Virtual work method is used *(default is standard)*

Energy calculation flag

Flag for stored electric or magnetic energy calculations. (This option must be activated if inductance is needed.)

= 1: Yes	perform energy calculations
= 0: No	do not perform energy calculations *(default is no)*

Current for inductance calculation

Total current amplitude to be used in inductance calculation. (Available for magnetostatic analysis only.)

= 0.0	inductance is not calculated *(default is 0.0)*

Torque calculation flag

Flag for torque calculation. (Available with the Virtual work option only.)

= 1: Yes	torque is calculated
= 0: No	torque is not calculated *(default is no)*

XYZ-Coordinate value of torque's reference point

X-, Y- and Z-coordinates of the torque's reference point.
(defaults are 0.0)

Notes

1. The types of available electromagnetic field results and postprocessing quantities depend on the type of the analysis performed. Available results for different analysis types are listed below:
   a. Magnetostatics:
      - Flux density B and Field intensity H. (Controlled by the 'magnetic field info flag 0 = No N = steps', and must be obtained at least at one time step.)
      - Magnetic forces, Torque, Stored magnetic energy and coenergy, Input energy, and Inductance. (Controlled by the 'postprocessing info flag'.)

   b. Transient electromagnetics:
      - Flux density B and Field intensity H. (Controlled by the 'magnetic field info flag 0 = No N = steps', and must be obtained at least at one time step.)
      - Magnetic forces, Eddy currents, and Power losses. (Controlled by the 'postprocessing info flag'.)

   c. Electrostatics:
      - Voltages and Electric fields. (Always available.)
      - Stored Electric energy. (Controlled by the 'energy calculation flag'.)

   d. Current flow analysis:
      - Voltages, Electric field and Current density. (Always available.)
      - Power loss and Joule heating. (Controlled by the 'postprocessing info flag'.)

   e. A.C. Eddy current analysis:
      - Flux density B. (Controlled by the 'magnetic field info flag 0 = No N = steps' and must be obtained at least at one frequency.)
      - Magnetic forces, Eddy currents, Power losses. (Controlled by the "postprocessing info flag".)
2. The Lorentz force calculation is only available for 2D and axisymmetric models.
3. Virtual work option is not available for models with 3D current sources.

Example: `EM_OUTPUT, 2, 0,`

This command instructs ESTAR to calculate and print electro-magnetic results every other time step. Postprocessing results are not obtained.

EM_MODEL

Analysis > ELECTRO_MAGNETIC > Conductor Model

The EM_MODEL command specifies whether conductors are stationary or in uniform motion with respect to the magnetic field.

Model type

Motion flag. (currently not available)

= 0: fixed	all conductors are stationary
= 1: moving	all conductors are in uniform motion *(default is stationary)*

Velocity

Velocity of conductors.
(default is 0)

A_MAGNETIC

Analysis > ELECTRO_MAGNETIC > Analysis Options

The A_MAGNETIC command specifies the details of the low-frequency electromagnetic analysis to be performed by ESTAR through the R_MAGNETIC command. Use A_HFRQEM for high-frequency electromagnetics.

Analysis option

Analysis option.

= S	magnetostatic analysis
= T	transient electromagnetic analysis
= E	electrostatic analysis
= C	current flow analysis
= F	frequency domain A.C. analysis
= CAP	capacitance matrix calculations. The matrix is printed in the output file (extension OUT) *(default is S)*

Units

Type of unit system.

= CGS	CGS unit system
= MKS	MKS unit system *(default is MKS)*

Convergence tolerance

Convergence tolerance.
(default value is 0.0001)

Maximum number of iterations

Maximum number of iterations at each time step.
(default value is 15)

Iterations between reforming stiffness

Number of iterations between successive reformations of the stiffness matrix.
(default value is 1)

Thermo-electric coupling flag

Flag for electro-thermal coupling.

= 0: Off	heat densities due to electric power losses are provided for subsequent thermal analysis

= 1: On	no electro-thermal coupling will be considered *(default is no coupling considered)*

Solution type

Equation solver to be used.

= 0: Direct	direct method (Gaussian)
= 1: Iterative	iterative method. Available only if "analysis option" is S, C, or E *(default is direct)*

Number of conductors

Number of conductors for capacitance matrix calculations. Prompted only if "analysis option" is CAP.
(default is 2)

Notes

1. The A_MAGNETIC command is optional. If the command is not issued, default values are used.
2. Electro-thermal coupling is only available for current flow, transient electro-magnetic and time harmonic AC analyses.
3. If the 'thermo-electric coupling flag' is activated, the element heat generation (from ESTAR) are automatically loaded into the database for subsequent thermal analysis.
4. For the iterative solver, a minimum of 400 nodes for 2D problems, and 1000 nodes for 3D problems must exist in the database.

Example: `A_MAGNETIC, T, MKS, 0.001, 10, 2, 0, 0,`

Transient electro-magnetic analysis is specified and the MKS unit system is used. The maximum number of iterations is set to 10. The stiffness matrix is reformed after every two iterations. The convergence tolerance is 0.001. Electro-thermal coupling is not considered. The direct solver will be used.

R_MAGNETIC

Analysis > ELECTRO_MAGNETIC > Run Emag Analysis

The R_MAGNETIC command executes the ESTAR module to perform electro-magnetic analysis.

Note

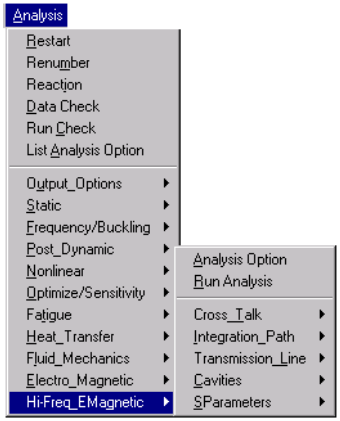
Uses options specified by the A_MAGNETIC command. Default options are used if the A_MAGNETIC command was not issued.

HI-FREQ_EMAGNETIC Menu

Analysis > HI-FREQ_ EMAGNETIC

The menu contains commands and menus related high-frequency electromagnetic analysis.

Figure 10-30 Hi-Freq Emagnetic Men



A_HFRQEM

Analysis > HI-FREQ_ EMAGNETIC > Analysis Option

The A_HFRQEM command defines the high frequency analysis to be run and sets the distance units to be used in the analysis.

Analysis option

= LFRQ	unused
= 2DHFRQ	run the 2D full-wave field solver
= 2DTALK	run the 2D quasi-static field solver to compute RLCG matrices then the time-domain cross-talk simulator to compute cross-talk and distortion
= XTALK	run the time domain cross-talk simulator with pre-computed RLCG matrices
= CAVAXI	run the time axisymmetric cavity field solver
= CAV3D	run the time 3D cavity field solver
= SPARAMETER	calculate s-parameters

(default is 2dhfrq)

Units

Unit for distance measurement to be used.

= Mm	dimensions are in mm
= Cm	dimensions are in cm
= M	dimensions are in m
= In	dimensions are in mils
= Mils	dimensions are in inches
= Um	dimensions are in microns
	(default is mm)

Example 1: `A_HFRQEM, xtalk, 4`

This command sets the high-frequency analysis option to run the cross-talk time domain simulator using pre-computed RLCG matrices with lengths specified in mils.

Example 2 `A_HFRQEM, sparameter, 1`

This command sets the high-frequency analysis option to run COSMOS/HFS 3D to calculate S-parameters. Length is specified in cm.

R_HFRQEM

Analysis > HI-FREQ_EMAGNETIC > Run Analysis

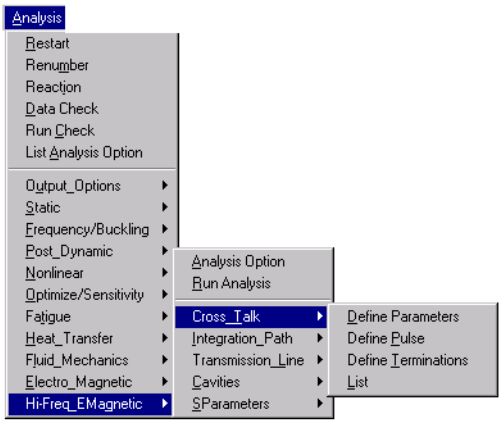
The R_HFRQEM command runs the electromagnetic analysis specified by the A_HFRQEM command.

▼CROSS_TALK Menu

Analysis > HI-FREQ_ EMAGNETIC > CROSS_TALK

This menu contains commands related to cross-talk analysis.

Figure 10-31 CrossTalk Menu



HF_XTKCONF

Analysis > HI-FREQ_EMAGNETIC > CROSS_TALK > Define Parameters

The HF_XTKCONF command defines the configuration parameters for the cross-talk time domain simulator.

Number of lines
Number of transmission lines. (maximum 10)
(default is 2)

Length of lines (m)
Length of the transmission lines in meters.
(default is 0.1)

Time duration (ns)
Duration of the time domain simulation in nano-seconds.
(default is 50)

Example: `HF_XTKCONF, 3, 0.0254, 50`

This command configures the cross-talk simulator to run with 3 lines of length 1" (2.54 cm) and for a duration of 50 nano-seconds.

HF_XTKPULSE

Analysis > HI-FREQ_EMAGNETIC > CROSS_TALK > Define Pulse

The HF_XTKPULSE command defines the excitation pulses on the near-end of chosen lines. If this command is not issued, all lines will be considered quiet (no excitation) by the program. At least one line must have an excitation pulse.

Line excitation flag
Line excitation flag.
= 0: Off quiet line
= 1: On line excited
(default is on)

Line number
Number of the line for which an excitation pulse is placed at the near-end.

Pulse rise time (ns)
Pulse's rise time in nano-seconds.

Pulse width (ns)
Pulse's width in nano-seconds.

Pulse fall time (ns)
Pulse's fall time in nano-seconds.

Pulse magnitude (V)
Pulse's magnitude in volts.

Example: `HF_XTKPULSE, 2, 1., 12, 1., 4.0`
This command defines an excitation pulse at the near end of line number 2 having a rise time of 1 nano-second, a duration of 12 nano-seconds, a fall time of 1 nana-second and a magnitude of 4 volts.

HF_XTKTERM

Analysis > HI-FREQ_EMAGNETIC > CROSS_TALK > Define Terminations

The HF_XTKTERM command defines the terminations at both near- and far-ends of each line. By default all lines are terminated by 50 Ohm resistances at the near ends and 50 Ohm resistances at the far ends. Capacitances can be added at the far-ends using this command.

Line number
Number of the line for which the terminations will be altered from their default values.

Near end resistance (ohms)
Near end resistance.

Far end resistance (ohms)
Far end resistance.

Far end capacitance (pf)
Far end capacitance.

Example: `HF_XTKTERM, 2, 100.0, 25.0, 30`
This command defines the termination for line 2 to be a 100.0 Ohms resistance at the near-end, a 25.0 Ohms resistance at the far-end and a parallel capacitance of 30 pico-Farads at the far end.

HF_XTKLIST

Analysis > HI-FREQ_EMAGNETIC > CROSS_TALK > List

THE HF_XTKLIST command lists pulse excitations and terminations for the specified line.

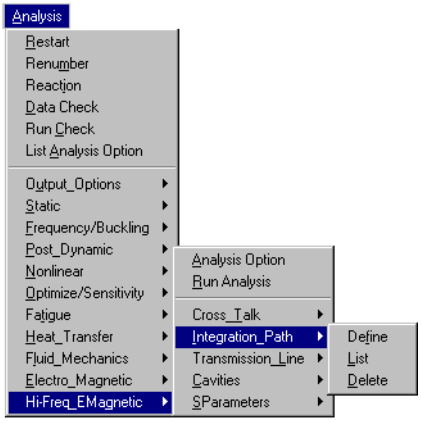
Line number
Line number. (1 through 10)
(default is 1)

▼INTEGRATION_PATH Menu

Analysis > HI-FREQ EMAGNETIC > INTEGRATION_PATH

This menu contains commands related to integration paths for use with transmission lines and cavities.

Figure 10-32 Integration Path Menu



HF_PATH

Analysis > HI-FREQ_EMAGNETIC > INTEGRATION_PATH > Define

The HF_PATH command defines one or more integration paths for the 2D field simulator or for cavity analysis. The integration paths are used in voltage computation based on electric field line integrals.

Path number
Path number. The maximum number of paths is 2.

Path point (i) XYZ- coordinates of point
X, Y, Z coordinate triplets that define the integration path. The minimum number of triplets per path is 2 and the maximum is 13. The list of triplets is terminated by entering a ";" or by repeating the last triplet.

Note
The path X,Y,Z triplets can be picked using the mouse on any specified plane.

Example: `HF_PATH,1,0.0,0.0,0.0,0.0,1.0,1.0,0.0,1.0,2.0;`
This command defines integration path #1 by 3 points (i.e., 2 straight line segments).

HF_PATHLIST

Analysis > HI-FREQ_EMAGNETIC > INTEGRATION_PATH > List

The HF_PATHLIST command lists coordinate triplets of an integration path defined by the HF_PATH.

Path number
Path number. (1 or 2)
(default is 1)

HF_PATHDEL

Analysis > HI-FREQ_EMAGNETIC > INTEGRATION_PATH > Delete

The HF_PATHDEL command deletes an integration path defined by the HF_PATH command.

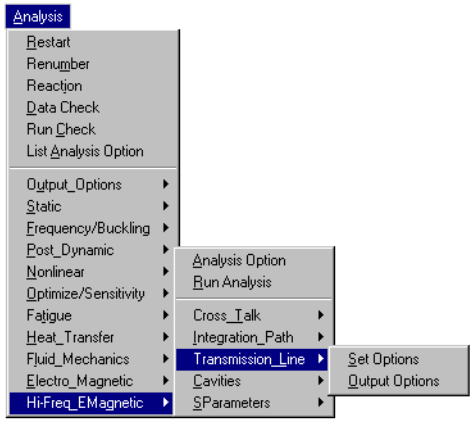
Path number
Path number. (1 or 2)

▼TRANSMISSION_LINE Menu

Analysis > HI-FREQ_EMAGNETIC > TRANSMISSION_LINE

This menu contains commands related to 2D transmission lines.

Figure 10-33 Transmission Path Menu



HF_2DSOLN

Analysis > HI-FREQ_EMAGNETIC > TRANSMISSION_LINE > Set Options

The HF_2DSOLN command defines the solution options for the 2D field solver.

Number of modes
Number of desired modes.

Starting frequency (GHz)
Beginning simulation frequency (in GHz).

Ending frequency (GHz)
Ending simulation frequency (in GHz).

Frequency increment (GHz)
Frequency step (in GHz).

Example: `HF_2DSOLN, 2, 1., 10., 0.5`

This command sets the 2D field solver's solution options to finding the first 2 modes over the frequency range starting at 1GHz and ending at 10GHz in steps of 0.5GHz.

HF_2DOUT

Analysis > HI-FREQ_EMAGNETIC > TRANSMISSION_LINE > Output Options

The HF_2DOUT command sets the output options for the 2D field solver.

Computation flag

Flag to specify which quantities should be computed.

= 1: All	compute modal propagations constants, fields, impedances and generalized RLCG matrices.
= 0: Eigen	compute modal propagation constants and fields only

Output option

Flag to specify the type of output.

= 0: None	no output
= 1: Nodal	output nodal values only
= 2: Elem	output elemental values only
= 3: Both	output both nodal and elemental values

Example: `HF_2DOUT, 1, 2`

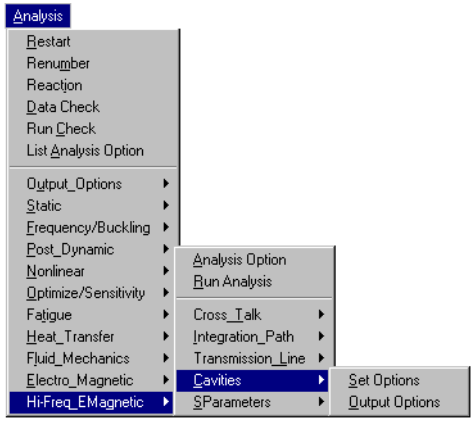
This command sets the 2D field solver's output options to compute modal propagation constants and fields only and write out both nodal and element values of the modal electric and magnetic fields.

▼CAVITIES Menu

Analysis > HI-FREQ_EMAGNETIC > CAVITIES

This menu contains commands related to analysis of cavities.

Figure 10-34 Cavities Menu



HF_CAVSOLN

Analysis > HI-FREQ_EMAGNETIC > CAVITIES > Set Options

The HF_CAVSOLN command defines the solution options for the cavity simulator.

Model flag

Flag to specify model type.

= 0: Axisymmetric	axisymmetric cavities
= 1: 3D	3D cavities
	(default is axisymmetric)

Number of modes

Number of desired modes.

First harmonic

First harmonic (for axisymmetric cavities).

Last harmonic

Last harmonic (for axisymmetric cavities).

Matrix solution method

Solution method to be used (prompted for 3D only).

= 0: Direct	use the direct method
= 1: Iterative	use the iterative method
	(default is direct)

Couple to thermal analysis

Flag to activate or deactivate coupling with thermal analysis.

= 0: No	no coupling with thermal analysis
= 1: Yes	couple with thermal analysis *(default is no)*

Incident power in watts for mode (i)

Specify incident power in watts for each mode.

Example: `HF_CAVSOLN, 0, 2, 1, 3;`

This command sets the cavity simulator's solution options to simulate an axisymmetric cavity and compute its 2 most dominant modes for each of the harmonics: 1, 2, 3.

HF_CAVOUT

Analysis > HI-FREQ_EMAGNETIC > CAVITIES > Output Options

The HF_CAVOUT command sets the output options for the 2D field solver.

Compute quality factor

Flag for cavity quality factor computation.

= 0: No	quality factor is not computed
= 1: Yes	quality factor is computed

Compute rlc equivalent circuit

Flag for equivalent RLC circuit computation.

= 0: No	equivalent RLC circuit is not computed
= 1: Yes	equivalent RLC circuit is computed

Output option

Flag to specify the type of output.

= 0: None	no output
= 1: Nodal	output nodal values only
= 2: Elem	output element values only
= 3: Both	output both nodal and element values

Notes

1. If 'compute quality factor' flag is set to 0, 'compute rlc equivalent circuit' flag is ignored and the RLC equivalent circuit is not computed.
2. If 'compute quality factor' flag is set to 1 and 'compute rlc equivalent circuit' flag is also set to 1, and integration path must be specified for RLC computation.

Example: `HF_CAVOUT, 1, 1, 0`

This command sets the cavity simulator's output options to compute the cavity's quality factor and its equivalent RLC circuit and to output nodal values only of the modal electric and magnetic fields.

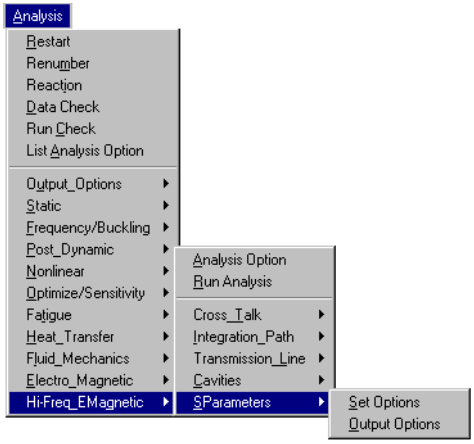



▼SParameters Menu

Analysis > HI-FREQ_EMAGNETIC > SParameters

This menu contains commands related to analysis of scattering parameters.

Figure 10-35 SParameters Menu



HF_SPARSOLN

Analysis > HI-FREQ_EMAGNETIC > SPARAMETERS > Set Options

The HF_SPARSOLN command sets the options for the calculation of S-parameters.

Number of ports
Number of ports defined in this model.

Starting frequency (GHz)
Lowest frequency in GHz at which the S-parameters will be calculated. (default is 1).

Ending frequency (GHz)
Highest frequency in GHz at which the S-parameters will be calculated.

Frequency Increment (GHz)
Increment between frequencies in GHz.

Impedance multiplier
Impedance multiplier. (default is 1).

Matrix Solution Method

= 0: Direct	use the direct solution method.
= 1: Iterative	use the iterative solution method. *(default is 0:Direct)*

Normalize S-matrix Flag

= 0: No	do not no normalize the S-parameters matrix.
= 1: Yes	normalize the S-parameters matrix. *(default is 0:No)*

Port number (i)

Enter the port number i. i = 1, number of Ports entered above.

Number of modes

Number of modes to be considered for the specified port.
(default is 1)

De-embedding Length for Port (i)

Enter the de-embedding length for Port (i).

Impedance method (not prompted if normalization is not requested)

Impedance calculation method to be used. Options are:

= 1: Zpi	power current method.
= 2: Zvi	voltage current method.
= 3: Zpv	power voltage method.

Normalizing Impedance (Real)

(not prompted if normalization is not requested)
Real impedance to be used for normalization.

Normalizing Impedance (Imag.)

(not prompted if normalization is not requested)
Imaginary impedance to be used for normalization.

Path Number

Path number to be used.
(prompted only if normalization is requested by Zpv or Zvi)

Note

Performing a cavity analysis to calculate resonant frequencies prior to running the S-parameters solution helps you determine the frequencies of interest.

HF_SPAROUT

Analysis > HI-FREQ_EMAGNETIC > SPARAMETERS > Output Options

The HF_SPAROUT command sets the output options for the calculation of S-parameters.

Circuit Simulator Flag

Sets the flag for circuit simulation.

= 0: None	none
= 1: Citifile	citifile format

= 2: Compact	compact format
= 3: Touchstone	touchstone format
	(default is None)

Output Option

Sets the flag for writing output results

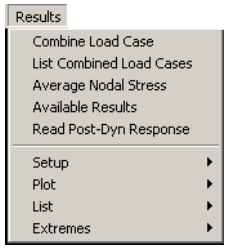
= 0: nodal	write results for nodes only
= 1: elemental	write results for elements only.
= 2: both	write results for nodes and elements.




11 Results Menu

RESULTS Menu

Figure 11-1 Results Menu



RESULTS

This menu provides powerful tools for the visualization of analysis results. Once an analysis has been performed, you can visualize the results in tabular or graphical formats. Tabular results can be directed to a file that can be used as a report. Graphical displays can be saved in several graphical formats. Animations can be saved in AVI files.

LCCOMB

Geo Panel: **RESULTS > Combine Load Cases**

The LCCOMB command is an action command that creates a secondary load case. A secondary load case is a load case defined by scaling and combining other load cases (primary or secondary). A primary load case is a load case for which loading was explicitly defined in the database. Results for a primary load case are obtained by solving the equilibrium equations, while results for a secondary load case are obtained by scaling and superposing the results of load cases used in its definition. The command uses the principle of superposition, and hence it is only valid for linear analysis. The command should only be issued after running static analysis successfully.

The results for secondary load cases are not written to the output file, but the user may obtain a printout using the LISTLOG and the listing commands for the desired results.

New load case number

New secondary load case number. (between 51 and 100, inclusive)

Load case number (i)

Load case number used in the ith term. It can be any previously defined load case, whether primary or secondary. A blank entry will terminate the combination. A maximum of 50 terms can be entered.

Load case factor (i)

Factor associated with load case (i). (i=1,2,...50)
(default is 1.0)

Example: `LCCOMB, 52, 1, 1.5, 51, -1.0, ,`

This command defines load case number 52 by combining load cases 1 and 51 by factors 1.5 and -1.0, respectively. Results for the new load case are calculated by adding results of load case 1 multiplied by 1.5 to the results of load case 51 multiplied by -1.0. Results for load cases 1 and 51 must exist in the database when this command is issued.

LCCOMBLIST

Geo Panel: **RESULTS > List Combined Load Cases**

The LCCOMBLIST command lists the composition (load cases and associated factors) of secondary load cases defined by the LCCOMB (Results, Combine Load Cases) command.

AVERAGE

Geo Panel: **RESULTS > Average Nodal Stress**

The AVERAGE command specifies the procedure for calculating the average nodal principal stresses, von mises stress and the nodal stress intensity. If a node is common to two or more elements, then elements connected to that node will generally generate different stress values at that node. This command gives the user a choice of whether to calculate nodal principal stresses, von Mises stresses and the stress intensity for each element at the common node and then average the resulting values, or first average the primary components (Sx, Sy, Sz, Txy, Txz and Tyz) and then calculate these stresses. This command also specifies whether zero principal stresses will be considered in the calculation of stress intensity.

Post proc. averaging technique

Flag to specify stress-averaging procedure.

= 0: Prev	pre-averaging: The primary components are first averaged and the principal, von Mises, and stress intensity are calculated based on the averages values
= 1:Post	post-averaging: Stress intensity and the von Mises and principal stresses are calculated from element results and are then averaged *(default is 0: Prev)*

Stress intensity calculation criterion

Stress intensity calculation flag to be used when one of the principal stresses is zero in the model.

= 0: P1-P2	consider P1 and P2 only
= 1: P1-P2-P3	consider P1, P2, and P3 *(default is P1-P2-P3)*

Example: `AVERAGE, 0, 1,`

Average Sx, Sy, Sz, Txy, Txz and Tyz are first calculated and the averaged values are used to calculate the von Mises and principal stresses and the stress intensity. All principal stresses are always considered to calculate stress intensity.

RESULTS?

Geo Panel: **RESULTS > Available Results**

The RESULTS? command lists all load cases, mode shapes, or solution steps for which results are available for postprocessing for the active analysis type. The active analysis type for postprocessing may be changed using the ACTPOST command.

READ_PDRESP

Geo Panel: **RESULTS > Read Post-Dyn Response**

The READ_PDRESP command reads displacements and stresses for a particular time/frequency step of postdynamic analysis and stores it as a secondary load case for a linear static analysis. Results from a postdynamic analysis must exist in the database when this command is issued.

Time step number

Time step number for postdynamic analysis.

Load case number to be initialized

Secondary load case number for static analysis. (between 51 and 100, inclusive)

Component flag

Results to be stored.

= 0: Deformations	deformations only
= 1: Stresses	stresses only
= 2: Both	both deformations and stresses *(default is 0: Deformations)*

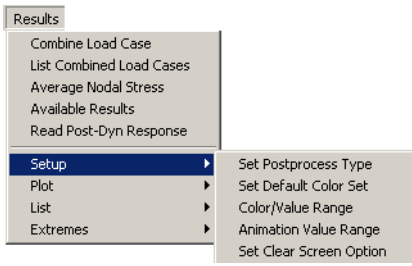



SETUP Menu

Geo Panel: Results > SETUP

This menu includes commands to activate the desired type of analysis and load the desired physical quantity from the database into the plot buffer.

Figure 11-2 Setup Menu



ACTPOST

Geo Panel: Results > SETUP > Set Postprocess Type

The ACTPOST command sets the type of analysis for postprocessing. The command is useful when more than one type of analysis was performed for the current problem.

Analysis type

Analysis type.

= 0: Linear structural	linear structural
= 1: Nonlinear structural	nonlinear structural
= 2: Frequency/Buckling	frequency or buckling
= 3: Thermal	thermal
= 4: Fluid	fluid
= 5: Electromagnetic	electromagnetic
= 6: Optimization	optimization analysis
= 7: Fatigue	fatigue analysis
= 8: Post dynamic	post dynamic analysis
= 9: Sensitivity	sensitivity

Notes

1. The user does not need to issue this command when only one type of analysis is used in the current problem.
2. Frequency and buckling analyses overwrite each other, which means that postprocessing cannot be switched from one of them to other for the same problem.

SETDEFCLRSET

Geo Panel: Results > SETUP > Set Default Color Set

The SETDEFCLRSET command sets the default color set to be used for result visualization. Color sets define the colors to be used in subsequent plots. The SETPLOT command can be used to choose the color set for an individual plot.

Default colorset number

Number of the color set to be used. Sets 1 through 8

SETPLOT

Geo Panel: **Results > SETUP > Color/Value Range**

The SETPLOT command selects a predetermined color set to be used for plotting. The command also specifies the extreme values of interest and the labeling of line contour plots. The active quantity in the plot buffer is plotted or reproduced using the new setting. Users may change the predefined color sets and/or define their own color sets (refer to the *Installation* Appendix).

Colorset number

Color set number.

= 1	6 colors
= 2	8 colors
= 3	10 colors
= 4	12 colors
= 5	40 colors
= 6	64 colors
= 7	76 colors
= 8	32 shades of gray colors, recommended for printing analysis results such as Stress, Displacement or Temperature on black and white devices.

(default is 5 or as set by the SETDEFCLRSET command)

Scale factor

Scale factor to be multiplied by all values for the component to be plotted.
(default is 1.0)

Minimum value

Minimum value to be used in the plot.
(default is the minimum value for the loaded component in the plot buffer)

Maximum value

Maximum value to be used in the plot.
(default is the maximum value for the loaded component in the plot buffer)

Label print count

Approximate distance along the contour lines between labels. Used only for line contour plots.

= 0	do not write contour labels
= X	write contour labels at x intervals

(default is 0)

Chart flag

Draw chart flag.

= 0: No	do not draw chart
= 1: Yes	draw chart

(default is 1)

Horizontal relative position of chart

Relative x-location of chart in the window. Should be between 0.0 and 1.0.

= 0.0	extreme left edge
= 1.0	extreme right edge

Vertical relative position of chart

Relative y-location of chart in the window. Should be between 0.0 and 1.0.

= 0.0 extreme lower edge
= 1.0 extreme upper edge

Chart color

Color to be used in displaying text in charts. Colors 1 through 16 may be used (use the STATUS1 command to view color numbering).

New title flag

Title of the plot (not title of chart) can be changed from default title.

= 0: No use default or existing title
= 1: Yes use new title

Title

If New title is set to 1, then a new title can be input as a sequence of words up to a total of 80 characters. Null string terminate the input

Notes

1. Parameters in this command assume their original default values whenever a new component is activated.
2. The scale factor can be used for conversion of units. As an example, a scale factor of 25.4 may be used to display displacements in millimeters instead of inches.

SETANIMATION

Geo Panel: **Results > SETUP > Animation Value Range**

The SETANIMATION command sets a range of values to be used in animating the active plot. The command provides an option to reset the extreme values to the full range of the plot. To animate the specified range, use the ANIMATE (Results, Plot, Animate) command

Set Flag

Resetting option.

= 1: Set The animation range will be defined by the minimum and maximum values defined below. Use this option to define a specific range of interest or to exclude undesired values.
= 0: Reset Reset to the full range. Use this option to set the animation back to the full range after specifying a different range using this command.
(default is 1: Set)

Minimum value

Minimum value to be used in animating the plot (using the ANIMATE command). Prompted only if 1: Set is selected for the Set Flag.
(default is the minimum value of the loaded component in the plot buffer)

Maximum value

Maximum value to be used in the plot n animating the plot (using the ANIMATE command). Prompted only if *1: Set* is selected for the Set Flag.
(default is the maximum value of the loaded component in the plot buffer)

SETERASE

Geo Panel: Results > SET UP > Set Clear Screen Options

The SETERASE command specifies whether or not to clear the screen (or window) before producing new plots. The active window is cleared by default if the SETERASE command is not issued.

Erase flag

Erasing flag.

= 1: Yes	clear the window before plotting
= 0: No	do not clear the window before plotting

SETLSECPLOT

Geo Panel: Results > SETUP > Set Path Graphs

The SETLSECPLOT command sets flags and parameters for plots to be generated by the LSECPLOT command. The user needs to issue this command only when the default setting is not desired.

Number of X intervals

Number of intervals in the x-direction.

Number of Y intervals

Number of intervals in the y-direction.

Show axes?

Axis drawing flag.

= 0: No	do not draw axes
= 1: X only	draw the x-axis only
= 2: Y only	draw the y-axis only
= 3 Both X and Y	draw both axes
	(default is 3: Both X and Y)

X axis at

Location of plotting the x-axis.

= 0: Min	plot x-axis at minimum y value
= 1: Zero	plot x-axis at y = 0.0
= 2: Max	plot x-axis at maximum y value
	(default is 1)

Y axis at

Location of plotting the y-axis.

= 0: Min	plot y-axis at minimum x value
= 1: Zero	plot y-axis at y = 0.0
= 2: Max	plot y-axis at maximum x value
	(default is 1)

X Grid type

Grid type in the x-direction.

= 0: No grid	no grid
= 1: Solid	solid lines
= 2: Dotted	dotted lines

(default is 2: Dotted)

Y Grid type

Grid type in the y-direction.

= 0: No grid	no grid
= 1: Solid	solid lines
= 2: Dotted	dotted lines

(default is 2: Dotted)

Grid fill color

Background color number for xyplots.

Grid color

Color number for grid lines.

Relative width of XY_plot

Relative width of xyplot with respect to current window. Should be between 0.0 and 1.0.
(default is 0.5 for line sections, 1.0 otherwise)

Relative height of XY_plot

Relative height of xyplot with respect to current window. Should be between 0.0 and 1.0.
(default is 0.5 for line sections, 1.0 otherwise)

Horizontal Relative position of XY_plot

Relative x-position or xyplot. Should be between 0.0 and 1.0.
(default is 0.0)

Vertical Relative position of XY_plot

Relative y-position or xyplot. Should be between 0.0 and 1.0.
(default is 0.0)

Chart color

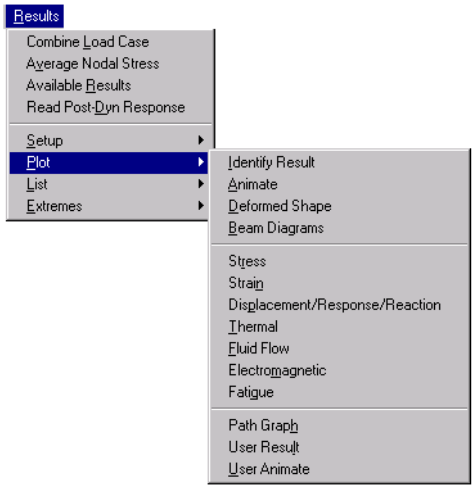
Color to be used in displaying text in charts. Colors 1 through 16 may be used (use STATUS1 command to view color numbering)




PLOT Menu

Geo Panel: **Results > PLOT**

Figure 11-3 Plot Menu



IDRESULT

Geo Panel: **Results > PLOT > Identity Result**

The IDRESULT command shows the value of the plotted quantity near the node (or element) closest to the point picked by the mouse.

ANIMATE

Geo Panel: **Results > PLOT > Animate**

The ANIMATE command results in the animation of the current plot in the active window. If no plot exists on the screen, the deformed shape will be animated for STAR, ASTAR, and NSTAR, and mode shape in the case of DSTAR (buckling or frequency). For other types of analyses, one of the following plots must exist in the active window for the ANIMATE command to work.

1. A filled, line, vector contour plot for any quantity.
2. Section plots for 3D models using the SECPLOT command.
3. An isosurface plot using the ISOPLOT command.

4. An xy-section plot may be added to any of the above plots (refer to the LSECPLOT command).

The animation represents snap shots of plotted data versus time or frequency. If results are only available for one step (linear static analysis, and steady state analyses), results are linearly interpolated for several frames. Animation should be performed in the following sequence for deformed shapes:

a. If no postprocessing plot is in the window, you may directly give the ANIMATE command for deformed/mode shape animation.
b. You may plot some other quantity on the deformed shape (a stress component for example).
c. You may use the ANIMATE command to animate the deformed shape. If a quantity is plotted on top of the deformed shape, then the quantity itself as well as the deformation are linearly interpolated for the load case/mode shape specified in the ANIMATE command.

For animation of other data, use the following sequence:

a. Plot the desired quantity for the active analysis type using commands like STRPLOT, MAGPLOT, FLOWPLOT, ..., etc.
b. You may use the SETPLOT command to change default setting.
c. You may use the LSECPLOT command to produce xy section plots of the plotted data versus distance along the path.
d You may use the SECPLOT to generate a section plot for 3D models.
e. Finally, use the ANIMATE command, all plots present on the active window will be animated simultaneously.

In cases where results are available for several time or frequency steps, you will be prompted for a pattern of steps to be used as animation frames. A maximum of 20 frames can be used for animation.

Beginning Step number
Beginning solution step, load case or mode shape number.

Ending step number
Ending solution step number.(*Not prompted if the active analysis type for post-processing is STAR, DSTAR, linear static, frequency, buckling or other steady state analyses*)

Step increment
Time step increment.(*Not prompted if the active analysis type for postprocessing is STAR, DSTAR, linear static, frequency, buckling or other steady state analyses*)

Animation type
Flag to specify one or two way animation.

= 1: TwoWay	two-way animation. Frames are animated back and forth.
= 0:OneWay	one-way animation. Frames are animated in cyclic order. (with a jump)

(default is 1: TwoWay)

Delay number

Slow_down_factor to reduce speed of animation.

= 0	fast animation, no slow down.
> 0	any number greater than 0 will slow down the animation. The higher the number, the slower the animation.

Number of frames

Number of frames to be animated.

Save and play as AVI

Flag to save and play animation as an AVI file.
= 1: Yes
= 0: No

AVI file name

Destination and name of AVI file. Default is prob-name.AVI.

Number of Iterations

Number of iterations for the AVI file.
(default is 1)

Notes

1. It is recommended to use Microsoft Video 1 for compression when requesting AVI files.
2. A message is issued when the memory requirement for the specified window exceeds the available memory. The user may then define a smaller window for animation.
3. Settings for the xy-plot generated by the LSECPLOT command are controlled by the SETLSECTION command.
4. In animating single step, steady state, or static analysis data, note the following:
   a. If an iso-surface plot or planar section plot exists on the undeformed shape, animation displays the iso-surfaces or planar sections one by one.
   b. If a quantity is plotted on the deformed shape, the deformed shape is animated, but the quantity itself will assume original fixed levels.

Example 1: Animate deformed shape for linear static analysis.

a. Choose display options (SHADE, HIDDEN,..etc).
b. Plot deformation for a load case using the DEFPLOT command.
c. Animate. Several frames are interpolated from the actual values.

Example 2: Animate a stress component on the deformed shape for nonlinear and advanced dynamics (NSTAR and ASTAR) structural analysis.

a. Use the STRPLOT command to plot a component.
b. Use the SETPLOT command to change default color set, extremes, or scale.




c. Use the LSECPLOT command to generate an xy-plot for the stress component along a section.

d. Animate with desired pattern of solution steps. Each frame used in the animation corresponds to a solution step. The section plot is also animated if plotted.

Example 3: Animate temperatures for a transient heat transfer analysis.

a. Use the TEMPPLOT command to plot thermal results.

b. Use the SETPLOT command to change default color set, extremes, or scale.

c. Use the TEMPPLOT command to plot temperatures with desired options.

d. Use the LSECPLOT command to generate an xy-plot for temperatures along a section.

e. Animate with desired pattern of time steps. Each frame used in the animation corresponds to a time step. The section plot is also animated if plotted.

Example 4: This example illustrates the difference between one- and two-way animations.

Suppose that frames 1, 2, ..., 10 are considered for animation. If one-way animation is used, the frames are displayed in the following order:

1, 2, ..., 10, 1, 2, ...10, 1, 2, ...,10 etc.

A jump will generally appear when displaying frame 10 and then frame 1.

If two-way animation is specified, the frames are displayed in the following order:

1, 2,...., 10, 9, 8,....., 1, 2,...,10, 9,...etc.

DEFPLOT

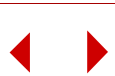
Geo Panel: **Results > PLOT > Deformed Shape**

The DEFPLOT command plots the deformed shape corresponding to a load case, time step, or mode shape, depending on the active type of analysis.

Load case, solution step or mode shape number
Load case, solution step or mode shape number.
(default is 1)

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.



(default is the highest element number defined)

Increment

Increment between elements in the pattern.
(default is 1)

Scale factor

Scale factor for deformation plotting. The given factor is a multiplier of actual (real life) deformation. The default factor [shown between brackets] scales the maximum resultant deformation to 10% of the model extents (largest dimension in the model). A scale factor of 1.0 results in using the same scale factor for both the model and deformation assuming there is no isolated remote nodes, or key-points in the model.

Note

The active type of analysis can be changed using the ACTPOST command.

Example 1: `DEFPLOT,1 , , , , 1, (1 for scale factor)`

This command loads and plots the deformation corresponding to the first load case, solution step, or mode shape for all elements in the selection list using actual scaling.

Example 2: `DEFPLOT, 2;`

Suppose that the maximum model dimension is 100, and the maximum resultant displacement is 0.02. The default scale will be 10% of 100 divided by 0.02 or 500.

Example 3: A scale factor of 2 results in doubling the actual deformation.

SMPLOT

Geo Panel: **Results > PLOT > Beam Diagrams**

The SMPLOT command lets you plot shear forces and bending moment diagrams in the local element coordinate system, for a pattern of elements. The utility is currently available only for primary load cases in STAR and solution steps in NSTAR.

Load case or solution step number

Load case or solution step number. Load case must be primary (1 through 50).
(default is active load case or solution step)

Force label

Component to be plotted. Valid labels are:
= FR: Axial Force
= VS: Shearing Force (S dir)shearing force in local s-direction
= VT: Shearing Force (T dir)shearing force in local t-direction
= TR: Torsional Momenttorsional moment
= MS: Bending Moment (S dir)bending moment about local s-direction
= MT: Bending Moment (T dir)bending moment about local t-direction

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.
(default is highest element number defined)

Increment
Increment between elements in the pattern.

Notes

1. For BEAM2D elements, the r direction goes from the first node to the second, the s direction is in the global X-Y plane and is perpendicular to the r direction. The t direction is normal to the X-Y plane and completes a right hand Cartesian coordinate system.
2. For BEAM3D elements, the r direction goes from the first node to the second, the s direction is in the plane defined by the 3 nodes, perpendicular to the r direction and goes from the first towards the third node. The t direction is normal to the r-s plane and completes a right hand Cartesian coordinate system.
3. Shear and bending moments are computed for all load cases set to run. Positive values are plotted according to the element orientation, and will not necessarily be plotted on the tension side.

Example: `SMPLOT, 1, Mt, 1, , , ,`

This command plots the bending moment diagram about the t-axis of the local element coordinate system for all elements (or elements in the selection list, if any).

Stress Plots

The STRPLOT command plots stresses after a successful static, nonlinear, or post-dynamic analysis run.

You can choose to generate a contour plot, vector plot, Iso plot, or section plot. A vector plot consists of vectors whose direction and magnitude represent that of the component. A contour plot connects points of equal values and can be colored lines or color-filled. Linear interpolation is used to determine the points of equal values or the points at which vectors are drawn. Iso and section plots are used for 3D models only. Default plot parameters are used unless the SETPLOT command is issued.

When selected from the menu tree, the program calls two commands internally: First, the program calls the ACTSTR command to load the desired component, and then it calls a second command to generate the plot. The second command depends on the selected type of plot as follows:

- Contour or vector plot: STRPLOT
- Section plot: SECPLOT




- Iso plot: ISOPLOT

If typed from the console, you need to enter the two commands individually in order.

ACTSTR

Geo Panel: **Results > PLOT > Stress**

Load case or solution step number
Load case or solution step number.
(default is the active load case or solution step)

Component
Stress component.

= SX	normal stress in the x-direction
= SY	normal stress in the y-direction
= SZ	normal stress in the z-direction
= TXY	shear stress in the y-direction, in the plane normal to the x-axis
= TXZ	shear stress in the z-direction, in the plane normal to the x-axis
= TYZ	shear stress in the z-direction, in the plane normal to the y-axis
= P1	normal stress in the first principal direction
= P2	normal stress in the second principal direction
= P3	normal stress in the third principal direction
= VON	von Mises stress
= INT	stress intensity
= ERR	stress error (available only for selected elements in STAR)

The following components are only available in linear static analysis for composite shell elements only:

= MILTXZ	maximum interlaminar shear in x-z plane from the layer specified below to the maximum layer number
= MILTYZ	maximum interlaminar shear in y-z plane from the layer specified below to the maximum layer number
= ILTXZ	interlaminar shear in x-z plane
= ILTYZ	interlaminar shear in y-z plane

The following components are only available in linear static analysis for SHELL3L and SHELL4L elements only:

= FIND	failure index
= MFIND	maximum failure index from the layer specified below to the maximum layer number *(default is VON)*

Stress flag
Stress flag.

=1: Node	nodal stress

= 2: Element element stress
(default is 1: Node)

Layer number

Layer number. (used for layered elements only)
(default is 1)

Face flag (Shell)

Face on which stresses are to be activated.
= 0: Top upper face
= 1: Bottom lower face

The following options are only available for shell elements:

= 2: Membrane membrane stresses
= 3: Bending bending stresses
(default is Top)

Coordinate system

Coordinate system. Not prompted in the following cases:

a. VON, INT, P1, P2, P3 (invariant stresses), or ERROR (scalar).
b. Element stresses (not nodal) are requested.
c. Postdynamic analyses other than time history.
d. Stresses are requested in a coordinate system other than the global Cartesian for one or more of the element groups in the model.
e. SHELLAX elements where stresses are always calculated in the element coordinates.

(default is 0)

Notes

1. The von Mises stress is computed from the basic stress components as follows:

$$VON = \{(^1/_2)[(SX - SY)^2 + (SX - SZ)^2 + (SY - SZ)^2] + 3(TXY^2 + TXZ^2 + TYZ^2)\}^{(1/2)}$$

or equivalently, from the principal stresses as follows:

$$VON = \{(^1/_2)[(P1 - P2)^2 + (P1 - P3)^2 + (P2 - P3)^2]\}^{(1/2)}$$

2. The ERR option is currently available for the TRIANG, TETRA4, TETRA4R, TETRA10, SHELL3 and SHELL4 elements only. The estimator is based on energy error norm and allows good evaluation of local errors. (Refer to International Journal for Numerical Methods in Engineering, vol. 24, 337-357 (1987) "*A Simple Error Estimator and Adaptive Procedure for Practical Engineering Analysis*" by O.C. Zienkiewicz and J. Z. Zhu)
3. Components MILTXZ, MILTYZ, and MFIND are checked for layers starting from the specified layer to the highest layer label. To check all layers, use 1 for the layer entry.
4. The stress intensity (INT) is defined as the difference between the maximum

and minimum principal stresses.

5. The active type of analysis can be changed using the ACTPOST command.
6. Double arrows are used for generating principal stresses where outward pointing arrows (<--->) denote tension, and (---><---) denote compression

STRPLOT (Contour Plot option)

Geo Panel: Results > PLOT > Stress > Contour Plot

Line flag

Type of contour to be generated.
= 0: Fill color-filled contour
= 1: Line colored line contour
(default is 0)

Beginning Element

Beginning element in the pattern.
(default is 1)

Ending Element

Ending element in the pattern.
(default is highest node or element number defined)

Increment

Increment between nodes or elements in the pattern.
(default is 1)

Shape flag

Shape of model to be used for plotting.
= 0: Undeformed shapeundeformed shape
= 1: Deformed shape deformed shape
(default is undeformed shape)

Deformed scale factor

Scale factor for deformation plotting. The given factor is a multiplier of actual deformation. The default factor is a magnification of the factor that scales the maximum deformation to 10% of the model extents (largest dimension in the model). A scale factor of 1.0 plots real-size deformations.

STRPLOT (Vector Plot option)

Geo Panel: Results > PLOT > Stress > Vector Plot

Line flag

Type of the plot to be generated
=2: Vector vector plot

Direction:X, Y, ZComp

x, y, and z coordinates of an arbitrary point to determine (with the origin) the direction of arrows for scalar quantities.

Beginning Node

Beginning node in the pattern.
(default is 1)

Ending Node

Ending node in the pattern.
(default is highest node defined)

Increment

Increment between nodes in the pattern.
(default is 1)

Vector scale factor

Scale factor to be used in determining the length of lines.

=1	maximum value is scaled to 10% of the size of the model
=c	maximum value is scaled to 10% of the size of the model times c (c is any number) *(default is 1)*

Shape flag

Shape of the model to be used for plotting.

= 0:Undeformed	undeformed shape
=1: Deformed	deformed shape *(default is undeformed shape)*

Deformed scale factor

Scale factor for deformation plotting. The given factor is a multiplier of actual deformation. The default factor is a magnification factor that scales the maximum deformation to 10% of the model extents. A scale factor of 1.0 plots real-size deformations.

ISOPLOT

Geo Panel: **Results > PLOT > Stress > Iso Plot**

The ISOPLOT command plots isosurfaces for 3D models only. A similar isolines plot can be obtained for 2D models using the line option in contour plots.

Number of isoplanes to be plotted [1-12]

Number of isoplanes to be plotted.

Specify intensity values for isoplanes

Flag for the choice of isoplanes.

= 0	take default values at 'number of isoplanes' number of intervals of data values
= 1	specify the values for isosurfaces *(default is 0)*

Intensity value for the isoplanes

Specify values for isosurfaces. Prompted only if the 'specify intensity values for isoplanes' flag is 1.

Shape flag




Shape of model to be used for plotting.
= 0: Undeformed undeformed model shape
= 1: Deformed deformed model shape
(default is 0)

Deformed shape scale factor
Scale factor for deformation plotting. The given factor is a multiplier of actual deformation. The default factor is a magnification of the factor that scales the maximum deformation to 10% of the model extents (largest dimension in the model). A scale factor of 1.0 plots real-size deformations.

SECPLOT

Results > PLOT > Stress > Section Plot

The SECPLOT command generates a section plot for 3D models. The section is defined by defining the intersecting planes.

Orientation of section planes
Intersecting plane.
= 0:X plane normal to Cartesian global x-axis
= 1:Y plane normal to Cartesian global y-axis
= 2:Z plane normal to Cartesian global z-axis
= 3: Three nodes plane defined by 3 nodes
(default is x)

Number of section planes [1,12]
Number of sections to be plotted. Prompted only if 'orientation of section planes' is not ‘Three nodes’.

Section plane positions
Locations of the section. Not prompted if 'orientation of section planes' is ‘Three nodes’.
= 0: Defaults take default locations at equal intervals within object extents.
= 1: Yes specify the locations of sections
(default is 0)

Specify positions for sections
Specify locations of sections if specify flag is 1, by entering x, y, or z values for 'orientation of section planes' X, Y, or Z, respectively. Prompted only if specify flag is 1.

Shape flag
Shape of model to be used for plotting.
= 0: Undeformed undeformed model shape
= 1: Deformed deformed model shape
(default is 0)

Deformed scale factor
Scale factor for deformation plotting. The given factor is a multiplier of actual deformation. The default factor is a magnification of the factor that scales the

maximum deformation to 10% of the model extents (largest dimension in the model). A scale factor of 1.0 plots real-size deformations.

Strain Plots

The STNPLOT command plots strains after a successful static, nonlinear, or post-dynamic analysis run.

You can choose to generate a vector or contour plot. A vector plot consists of vectors whose direction and magnitude represent that of the component. A contour plot connects points of equal values and can be colored lines or color-filled. Linear interpolation is used to determine the points of equal values or the points at which vectors are drawn. Default plot parameters are used unless the SETPLOT command is issued.

Internally, the command calls two dialog boxes: ACTSTN to load the desired component, and STNPLOT to generate the plot.

ACTSTN

Results > PLOT > Strain

Load case or solution step number
Load case or solution step number.
(default is active load case or solution step)

Component
Strain component.

= EPSX	normal strain in the x-direction
= EPSY	normal strain in the y-direction
= EPSZ	normal strain in the z-direction
= GMXY	shear strain in the x-y plane
= GMXZ	shear strain in the x-z plane
= GMYZ	shear strain in the y-z plane
= ESTRN	equivalent strain (see note below)
= SED	strain energy density for STAR
= ENERGY	total strain energy for STAR
	(default is ESTRN)

Strain type
Strain type (for nonlinear analysis only).

= Total	total
= Thermal	thermal
= Creep	creep
= Plastic	plastic
	(default is total)

Layer number

Layer number.
(default is 1)

Face flag
Face of element.

= 0: Top	top face (shell elements)
= 1: Bottom	bottom face (shell elements)
= 2: Membrane	membrane stresses (shell elements)
= 3: Bending	bending stresses (shell elements)
	(default is top)

STNPLOT (Contour Plot option)

Results > PLOT > Strain > Contour Plot

Refer to the STRPLOT (Contour Plot option) command description for more details.

Notes

1) The active type of analysis can be changed using the ACTPOST command.

2. The equivalent strain (ESTRN) is calculated from:

$$ESTRN = 2[(\varepsilon_1 + \varepsilon_2)/3]^{(1/2)}$$

where:

$$\varepsilon_1 = 0.5[(EPSX - \varepsilon^*)^2 + (EPSY - \varepsilon^*)^2 + (EPSZ - \varepsilon^*)^2]$$

$$\varepsilon_2 = [(GMXY)^2 + (GMXZ)^2 + (GMYZ)^2]/4$$

$$\varepsilon^* = (EPSX + EPSY + EPSZ)/3$$

STNPLOT (Vector Plot option)

Results > PLOT > Strain > Vector Plot

Refer to the STRPLOT (Vector Plot option) command description for more details.

SECPLOT

Results > PLOT > Strain > Section Plot

Refer to the ISOPLOT command description for more details.

Displacement Plots

The DISPLOT command plots displacements, velocities, accelerations, or reaction forces. You can plot displacement components for static, frequency, buckling, nonlinear, and post-dynamic analysis. For frequency and buckling analyses, the program plots the mode shapes. Velocities and accelerations are plotted for nonlinear

and post-dynamic analyses only. Reaction forces are not calculated for post-dynamic analysis.

You can choose to generate a vector or contour plot. A vector plot consists of vectors whose direction and magnitude represent that of the component. A contour plot connects points of equal values and can be colored lines or color-filled. Linear interpolation is used to determine the points of equal values or the points at which vectors are drawn. Default plot parameters are used unless the SETPLOT command is issued.

✍ Internally, the command calls two dialog boxes: ACTDIS to load the desired component, and DISPLOT to generate the plot

ACTDIS

Results > PLOT > Displacement/Response/Reaction

Load case, solution step or mode shape number
Load case, solution step or mode shape number.
(default is the active load case, mode shape or solution step)

Component
Displacement component.

= UX	displacement in the x-direction
= UY	displacement in the y-direction
= UZ	displacement in the z-direction
= RX	rotation about the x-direction
= RY	rotation about the y-direction
= RZ	rotation about the z-direction
= URES	resultant displacement

The following components are also available for STAR, FFESTAR and NSTAR only:

= RFX	reaction force in the x-direction
= RFY	reaction force in the y-direction
= RFZ	reaction force in the z-direction
= RFRES	resultant reaction force
= RMX	reaction moment about the x-direction
= RMY	reaction moment about the y-direction
= RMZ	reaction moment about the z-direction
= RMRES	resultant reaction moment

The following components are also available for ASTAR and NSTAR only:

= VX	velocity in the x-direction
= VY	velocity in the y-direction
= VZ	velocity in the z-direction
= WX	angular velocity about x-direction

= WY	angular velocity about y-direction
= WZ	angular velocity about z-direction
= VRES	resultant velocity
= AX	acceleration in the x-direction
= AY	acceleration in the y-direction
= AZ	acceleration in the z-direction
= BX	angular acceleration about x-direction
= BY	angular acceleration about y-direction
= BZ	angular acceleration about z-direction
= ARES	resultant acceleration
	(default is URES)

Coordinate system

Coordinate system number. Not prompted for dynamic analysis other than time history.

Notes

1. The active type of analysis can be changed using the ACTPOST command.
2. The interpretation of component depends on the coordinate system type, UX for example refers to the radial component for a cylindrical coordinate system.

DISPLOT (Contour Plot option)

Results > PLOT > Displacement/ Response/Reaction> Contour Plot

Refer to the STRPLOT (Contour Plot option) command description for more details.

DISPLOT (Vector Plot option)

Results > PLOT > Displacement/ Response/ Reaction> Vector Plot

Refer to the STRPLOT (Vector Plot option) command description for more details.

ISOPLOT

Results > PLOT > Displacement/ Response/ Reaction > Iso Plot

Refer to the ISOPLOT command description for more details.

SECPLOT

Results > PLOT > Displacement/ Response/ Reaction > Section Plot

Refer to the SECPLOT command description for more details.

Thermal Plots

The TEMPPLOT command plots thermal results after a successful thermal analysis run.

ACTTEMP

Results > PLOT > Thermal

Time step number
Time step number.
(default is the maximum number of time steps defined)

Component
Component.

= TEMP	nodal temperature
= GRADX	temperature gradient in the X-direction
= GRADY	temperature gradient in the Y-direction
= GRADZ	temperature gradient in the Z-direction
= GRADN	resultant temperature gradient
= HFLUXX	heat flux in the X-direction
= HFLUXY	heat flux in the Y-direction
= HFLUXZ	heat flux in the Z-direction
= HFLUXN	resultant heat flux
	(default is TEMP)

TEMPPLOT (Contour Plot option)

Results > PLOT > Thermal > Contour Plot

Line flag
Type of the contour plot to be generated.

= 0: Fill	color-filled contour
= 1: Line	colored line contour
	(default is 0: Fill)

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.
(default is the highest element number defined)

Increment
Increments between elements in the pattern.
(default is 1)

TEMPPLOT (Vector Plot option)

Results > PLOT > Thermal > Vector Plot

Line flag

Type of the contour plot to be generated.

=2: Vector	vector plot

Direction: X, Y, ZComp

x, y, and z coordinates of an arbitrary point to determine (with the origin) the direction of arrows for scalar quantities.

Beginning Node

Beginning node in the pattern.
(default is 1)

Ending Node

Ending node in the pattern.
(default is highest node defined)

Increment

Increment between nodes in the pattern.
(default is 1)

Vector scale factor

Scale factor to be used in determining the length of lines.

=1	maximum value is scaled to 10% of the size of the model
=c	maximum value is scaled to 10% of the size of the model times c (c is any number) *(default is 1)*

ISOPLOT

Results > PLOT > Thermal > Iso Plot

The ISOPLOT command plots isosurfaces for 3D models only. A similar isolines plot can be obtained for 2D models using the line option in contour plots.

Number of isoplanes to be plotted [1-12]

Number of isoplanes to be plotted.

Specify intensity values for isoplanes

Flag for the choice of isoplanes.

= 0	take default values at 'number of isoplanes' number of intervals of data values
= 1	specify the values for isosurfaces *(default is 0)*

Intensity value for the isoplanes

Specify values for isosurfaces. Prompted only if the 'specify intensity values for isoplanes' flag is 1.

SECPLOT

Results > PLOT > Thermal > Section Plot

The SECPLOT command generates a section plot for 3D models. The section is defined by defining the intersecting planes.

Orientation of section planes

Intersecting plane.

= 0:X	plane normal to Cartesian global x-axis
= 1:Y	plane normal to Cartesian global y-axis
= 2:Z	plane normal to Cartesian global z-axis
= 3: Three nodes	plane defined by 3 nodes
	(default is x)

Number of section planes [1,12]

Number of sections to be plotted. Prompted only if 'orientation of section planes' is not ‘Three nodes’.

Section plane positions

Locations of the section. Not prompted if 'orientation of section planes' is ‘Three nodes’.

= 0: Defaults	take default locations at equal intervals within object extents.
= 1: Yes	specify the locations of sections
	(default is 0)

Specify positions for sections

Specify locations of sections if specify flag is 1, by entering x, y, or z values for 'orientation of section planes' X, Y, or Z, respectively. Prompted only if specify flag is 1.

Fluid Flow Plots

The FLOWPLOT command plots fluid flow results after a successful fluid analysis run.

You can choose to generate a vector or contour plot. A vector plot consists of vectors whose direction and magnitude represent that of the component. A contour plot connects points of equal values and can be colored lines or color-filled. Linear interpolation is used to determine the points of equal values or the points at which vectors are drawn. Default plot parameters are used unless the SETPLOT command is issued.

ACTFLOW

Results > PLOT > Fluid Flow

Time step number

Time step number.
(default is 1)

Component

Component to be loaded to the plot buffer.

= VRES	resultant velocity
= VX	velocity in the X-direction
= VY	velocity in the Y-direction
= VZ	velocity in the Z-direction
= STREAM	stream function
= PRESS	pressure
= TAUXY	shear stress in the X-direction in XZ plane
= TAUYZ	shear stress in the Y-direction in the YX plane
= TAUZX	shear stress in the Z-direction in the ZY plane
= TEMP	temperature
= GRADX	temperature gradient in X-direction
= GRADY	temperature gradient in Y-direction
= GRADZ	temperature gradient in Z-direction
= GRADN	resultant temperature gradient
= TKE	turbulence kinetic energy
= EPS	dissipation rate
= DENS	fluid density
= MACH#	mach number
	(default is VRES)

FLOWPLOT (Contour Plot option)

Results > PLOT > Fluid Flow > Contour Plot

Refer to the TEMPPLOT (Contour Plot option) command description for more details.

FLOWPLOT (Vector Plot option)

Results > PLOT > Fluid Flow > Vector Plot

Line flag

Type of the contour plot to be generated.
=2: Vector vector plot

Beginning Node

Beginning node in the pattern.
(default is 1)

Ending Node

Ending node in the pattern.
(default is highest node defined)

Increment

Increment between nodes in the pattern.

(default is 1)

Vector scale factor

Scale factor to be used in determining the length of lines.

=1	maximum value is scaled to 10% of the size of the model
=c	maximum value is scaled to 10% of the size of the model times c (c is any number) *(default is 1)*

ISOPLOT

Results > PLOT > Fluid Flow > Iso Plot

Refer to the ISOPLOT command description under Thermal Plots for more details.

SECPLOT

Results > PLOT > Fluid Flow > Section Plot

Refer to the SECPLOT command description under Thermal Plots for more details.

Electromagnetics Plots

The MAGPLOT command generates electromagnetic plots after a successful electromagnetic analysis run.

You can choose to generate a vector or contour plot. A vector plot consists of vectors whose direction and magnitude represent that of the component. A contour plot connects points of equal values and can be colored lines or color-filled. Linear interpolation is used to determine the points of equal values or the points at which vectors are drawn. Default plot parameters are used unless the SETPLOT command is issued.

ACTMAG

Geo Panel: **Results > PLOT > Electromagnetic**

Time Domain

Time step number

Time step number.

(default is 1)

Entity flag

Flag to activate results at nodes or centers of elements.

= 1: Node nodes
= 2: Element elements
(default is nodes)

Components

Component.

= MFLUXXR magnetic flux in the X-direction. Real component
= MFLUXXI magnetic flux in the X-direction. Imaginary component
= MFLUXXA magnetic flux in the X-direction. Amplitude
= MFLUXYR magnetic flux in the Y-direction. Real component
= MFLUXYI magnetic flux in the Y-direction. Imaginary component
= MFLUXYA magnetic flux in the Y-direction. Amplitude
= MFLUXZR magnetic flux in the Z-direction. Real component
= MFLUXZI magnetic flux in the Z-direction. Imaginary component
= MFLUXZA magnetic flux in the Z-direction. Amplitude
= MFLUXR resultant magnetic flux. Real component
= MFLUXI resultant magnetic flux. Imaginary component
= MFIELDX magnetic field intensity in the X-direction
= MFIELDY magnetic field intensity in the Y-direction
= MFIELDZ magnetic field intensity in the Z-direction
= MFIELDR resultant magnetic field intensity
= MFORCEXR magnetic force in the X-direction. Real component (nodal value only)
= MFORCEXI magnetic force in the X-direction. Imaginary component (nodal value only)
= MFORCEXA magnetic force in the X-direction. Amplitude (nodal value only)
= MFORCEYR magnetic force in the Y-direction. Real component (nodal value only)
= MFORCEYI magnetic force in the Y-direction. Imaginary component (nodal value only)
= MFORCEYA magnetic force in the Y-direction. Amplitude (nodal value only)
= MFORCEZR magnetic force in the Z-direction. Real component (nodal value only)
= MFORCEZI magnetic force in the Z-direction. Imaginary component (nodal value only)
= MFORCEZA magnetic force in the Z-direction. Amplitude (nodal value only)
= MFORCER resultant magnetic force. Real component (nodal value only)
= MFORCEI resultant magnetic force. Imaginary component (nodal value only)
= AFORCEX average force in X-direction (nodal value only)
= AFORCEY average force in Y-direction (nodal value only)




= AFORCEZ	average force in Z-direction (nodal value only)
= AFORCE	resultant average force (nodal value only)
= EFIELDX	electrical field intensity in the X-direction
= EFIELDY	electrical field intensity in the Y-direction
= EFIELDZ	electrical field intensity in the Z-direction
= EFIELDR	resultant Electrical field intensity
= CURDENXR	current density in the X-direction. Real component
= CURDENXI	current density in the X-direction. Imaginary component
= CURDENXA	current density in the X-direction. Amplitude
= CURDENYR	current density in the Y-direction. Real component
= CURDENYI	current density in the Y-direction. Imaginary component
= CURDENYA	current density in the Y-direction. Amplitude
= CURDENZR	current density in the Z-direction. Real component
= CURDENZI	current density in the Z-direction. Imaginary component
= CURDENZA	current density in the Z-direction. Amplitude
= CURDENR	resultant current density. Real component
= CURDENI	resultant current density. Imaginary component
= NPOTENR	voltage or nodal potential. Real component
= NPOTENI	voltage or nodal potential. Imaginary component
= NPOTEN	voltage or nodal potential. Amplitude
= PLOSSR	real component of power loss (elemental value only)
= PLOSSI	imaginary component of power loss (elemental value only)
= PLOSSA	amplitude of power loss (elemental value only)
= EX_R	Real comp. of electric field intensity in the X-direction.
= EY_R	Real comp. of electric field intensity in the Y-direction.
= EZ_R	Real comp. of electric field intensity in the Z-direction.
= ER_R	Resultant real electric field intensity.
= EX_I	Imaginary comp. of electric field intensity in the X-direction.
= EY_I	Imaginary comp. of electric field intensity in the Y-direction.
= EZ_I	Imaginary comp. of electric field intensity in the Z-direction.
= ER_I	Resultant Imaginary electric field intensity.
= HX_R	Real comp. of magnetic field intensity in the X-direction.
= HY_R	Real comp. of magnetic field intensity in the Y-direction.
= HZ_R	Real comp. of magnetic field intensity in the Z-direction.
= HR_R	Resultant of real magnetic field intensity.
= HX_I	Imaginary comp. of magnetic field intensity in the X-direction.
= HY_I	Imaginary comp. of magnetic field intensity in the Y-direction.
= HZ_I	Imaginary comp. of magnetic field intensity in the Z-direction.
= HR_I	Resultant of real magnetic field intensity.
= JX_R	Real comp. of current or charge density in the X-direction.
= JY_R	Real comp. of current or charge density in the Y-direction.
= JZ_R	Real comp. of current or charge density in the Z-direction.
= JR_R	Resultant of real current or charge density.
= JX_I	Imaginary comp. of current or charge density in X-direction.
= JY_I	Imaginary comp. of current or charge density in Y-direction.

= JZ_I Imaginary comp. of current or charge density in Z-direction.
= JR_I Resultant of real current or charge density.

Frequency Domain

Frequency number

Frequency number (use RESULTS? for corresponding frequency values). Prompted only for 2DHFREQ as "frequency number" and for CAVAXI as "harmonic number".
(default is 1)

Mode number

Mode number.
(default is 1)

Node/Element

Flag to activate results at nodes or centers of elements.
= 1: Node nodes
= 2: Element elements
(default is 1)

Component

Component. Admissible components depend on the performed type of analysis as follows:

For 2DHFRQ and CAV3D analysis:

= EX electric field intensity in the X-direction. Real component
= EY electric field intensity in the Y-direction. Real component
= EZ electric field intensity in the Z-direction. Real component
= ER resultant electric field intensity. Real component
= HX magnetic field intensity in the X-direction. Real component
= HY magnetic field intensity in the Y-direction. Real component
= HZ magnetic field intensity in the Z-direction. Real component
= HR resultant magnetic field intensity. Real component

For 2DXTALK analysis:

= POT electrostatic potential. Real component
= EX electric field intensity in the X-direction. Real component
= EY electric field intensity in the Y-direction. Real component
= ER resultant electric field intensity. Real component

For CAVAXI analysis:

= ERO electric field intensity in the radial direction. Real
= EPHI electric field intensity in the azimuthal (PHI) direction. Real
= EZ electric field intensity in the Z-direction. Real
= ER resultant electric field intensity. Real
= HRO magnetic field intensity in the radial direction. Real
= HPHI magnetic field intensity in the azimuthal (PHI) direction. Real

= HZ magnetic field intensity in the Z-direction. Real
= HR resultant magnetic field intensity. Real

MAGPLOT (Contour Plot option)

Results > PLOT > Electromagnetic > Contour Plot

Refer to the TEMPPLOT (Contour Plot option) command description for more details.

MAGPLOT (Vector Plot option)

Results > PLOT > Electromagnetic > Vector Plot

Refer to the FLOWPLOT (Vector Plot option) command description for more details.

ISOPLOT

Results > PLOT > Electromagnetic > Iso Plot

Refer to the ISOPLOT command description under Thermal Plots for more details.

SECPLOT

Results > PLOT > Electromagnetic > Section Plot

Refer to the SECPLOT command description under Thermal Plots for more details.

Fatigue Plots

The ACTFTG command plots the cumulative fatigue usage factor after a successful fatigue analysis run.

You can choose to generate a vector or contour plot. A vector plot consists of vectors whose direction and magnitude represent that of the component. A contour plot connects points of equal values and can be colored lines or color-filled. Linear interpolation is used to determine the points of equal values or the points at which vectors are drawn. Default plot parameters are used unless the SETPLOT command is issued.

Notes

1. The active type of analysis can be changed using the ACTPOST command.
2. This command is only useful when the all-nodes option is used.

ACTFTG

Results > PLOT > Fatigue

The ACTFTG command plots the cumulative fatigue usage factor after a successful fatigue analysis run.

FTGPLOT (Contour Plot option)

Results > PLOT > Fatigue > Contour Plot

Refer to the TEMPPLOT (Contour Plot option) command description for more details.

FTGPLOT (Vector Plot option)

Results > PLOT > Fatigue > Vector Plot

Refer to the TEMPPLOT (Vector Plot option) command description for more details.

ISOPLOT

Results > PLOT > Fatigue > Iso Plot

Refer to the ISOPLOT command description under Thermal Plots for more details.




SECPLOT

Results > PLOT > Fatigue > Section Plot

Refer to the SECPLOT command description under Thermal Plots for more details

LSECPLOT

Results > PLOT > Path Graph

The LSECPLOT command generates an xy-plot from the current contour plot in the active window. A path is specified by a number of nodes. The x-axis is used for the distance along the specified path, and the y-axis is the value of the plotted data. Two to twenty nodes can be used to define the path, and linear interpolation is used to calculate data along the section. The command continues to prompt for more nodes until a node is selected twice or 20 nodes are specified.

Node1 to define path
Nodes to define the path.

ACTUSRPLOT

Results > PLOT > User Result

The ACTUSRPLOT command loads the specified component from a user file for the purpose of generating a contour plot. The component can be nodal or elemental. The user file must be a text file and must contain the following information:

First line should contain 3 entries: Number of nodes or elements, type flag (0 for nodal, 1 for elemental), and number of components.

Second line should contain component names for all the components specified in the third entry of the first line. Spaces, commas, or tabs separate the component names.

From third line onwards numeric data is listed, first the node or element number and then the values for the components. (See example below.)

The USRPLOT command may then be used to generate the plot.

File name containing plot data
File name containing the data for user component in the above specified format.

Component number
Component number to be loaded.

Example: Suppose the file “disp.plt” is created as follows:

	100	0	3	(this is the first line)
	x-disp	y-disp	z-disp	(titles)
1	0.00120	-.00020	0.00050	(up to 20 columns for each line)
2	0.00130	-.00021	0.00495	
.	.....	.....	.....	
.	.....	.....	.....	
100	0.00160	-.00025	0.0030	

Note that free format is used for each line. The command “ACTUSRPLOT, disp.plt, 1", may be used to load the x-disp. The USRPLOT command may then be used to generate the plot.

USRPLOT

Results > PLOT > User Result > Plot

The USRPLOT command plots a data from a user file. You can plot nodal or elemental quantities. The user file must be a text file formatted as follows:

First line should contain 3 entries: Number of nodes or elements, type flag (0 for nodal, 1 for elemental), and number of components.

Second line should contain component names for all the components specified in the third entry of the first line. Spaces separate the component names.

From third line onwards numeric data is listed, first the node or element number and then the values for the components. (See example below.)

The USRPLOT command may then be used to generate the plot.

Line flag

Type of plot to be generated.
= 0: Fill color-filled contour
= 1: Line colored line contour
= 2: Vector vector plot
(default is 0: Fill)

Direction

x, y and z coordinates of an arbitrary point to determine (with the origin) the direction of arrows. (prompted for scalar quantities only when plot type = 2)

Beginning Node or Element

Beginning node or element in the pattern.
(default is 1)

Ending Node or Element

Ending node or element in the pattern.
(default is highest node or element number defined)

Increment

Increment between nodes or elements in the pattern.
(default is 1)

Scale factor for determining the length of lines

Scale factor to be used in determining the length of lines in vector plots (plot type = 2).

= 0	maximum value is scaled to 10% of the size of the model
= c	maximum value is scaled to 10% of the size of the model times c (c is any number) *(default is 0)*

Shape flag

Shape flag.

= 0: Undeformed	use undeformed shape
= 1: Deformed	use deformed shape *(default is 0: Undeformed)*

Deformed scale factor

Scale factor for deformation plotting. The given factor is a multiplier of actual deformation. The default factor is a magnification of the factor that scales the maximum deformation to 10% of the model extents (largest dimension in the model). A scale factor of 1.0 plots real-size deformation.

Example: Suppose the file "disp.plt" is created as follows:

	100	1	3	(this is the first line)
	x-disp	y-disp	z-disp	(titles)
1	0.00120	-.00020	0.00050	(up to 20 columns for each line)
2	0.00130	-.00021	0.00495	
.	.....	.....	.....	
.	.....	.....	.....	
100	0.00160	-.00025	0.0030	

Note that free formatting is used for each line.

USRANIMATE

Results > PLOT > User Animate

The USRANIMATE command may be used to read frames and display them in order to create an animation effect. Up to 20 frames may be animated. Each frame must have been stored in a meta file, or image file.

Format

Type of file containing the pictures.

= 1: Meta	meta file (generated by the METAFILE command)
= 0: Image	image file (generated by the IMAGSAVE command) *(default is image file)*

Animation type

Flag to specify one or two way animation.

= 1: TwoWay animationtwo-way animation. Frames are animated back and forth.
= 0: OneWay animationone-way animation. Frames are animated in cyclic order (with a jump)
(default is 0: OneWay animation)

Delay number

Slow_down_factor to reduce speed of animation.

= 0	fast animation, no slow down
> 0	greater than 0, slow animation. The higher the number, the slower the animation

File name (i)

Name of ith file. All files must be of the same format. (i=1 to 50)

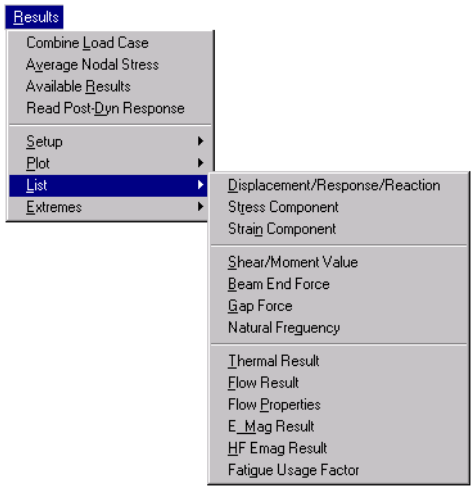



▶ LIST Menu

Results > LIST

This menu contains commands related to listing the results. Use the LISTLOG command to write the list to a file.

Figure 11-4 List Menu



DISLIST

Results > LIST > Displacement/Response/Reaction

The DISLIST command may be used with STAR, DSTAR, NSTAR, and ASTAR to list lists all displacement components for the nodes specified in the pattern in the specified coordinate system.

Load case, mode shape or solution step number
Load case, mode shape or solution step number.
(default is the active load case, mode shape or solution step)

Listing set
Set number.
For linear static analysis (STAR):
= 1: Displacements and rotations
= 2: Reaction forces
= 3: Reaction moments

For nonlinear analysis (NSTAR):

= 1: Displacements and rotations
= 2: Linear and angular velocities
= 3: Linear and angular accelerations
= 4: Reaction forces
= 5: Reaction moments

For frequency and buckling analyses (DSTAR):

= 1: Mode shape

For linear dynamic analysis (ASTAR):

= 1: Displacements and rotations
= 2: Linear and angular velocities
= 3: Linear and angular accelerations
= 4; Reaction forces
= 5; Reaction moments
(default is 1)

Beginning Node
Beginning node in the pattern.
(default is 1)

Ending Node
Ending node in the pattern.
(default is highest node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Coordinate system
Coordinate system. Not prompted for postdynamic analyses other than time history.
(default is 0 for Cartesian global)

Notes

1. The type of analysis can be specified using the ACTPOST command.
2. Reaction forces and moments are available only for STAR and NSTAR.

Example: `DISLIST, 3, 1, 1, 15, 6, 1,`

This command lists displacement components resulting from load case 3, for nodes 1, 7 and 13 in coordinate system 1 (global cylindrical system). UX in this case refers to the radial direction, UY to the tangential direction and UZ is the same as the UZ global Cartesian direction.

STRLIST

Geo Panel: **Results > LIST > Stress Component**

The STRLIST command lists stresses at nodes or element centers for a pattern of nodes or elements. Use the BEAMRESLIST command to list beam stresses. Use SPRRESLIS to list end forces for spring elements.

Load case or time step number
Load case or time step number.
(default is the active load case or time step depending on the active analysis)

Stress flag
Stress flag (for linear static analysis only).
= 1: Node nodal stress
= 2: Element element stress
(default is 1: Node)

List option
Flag for element stresses. (Prompted only if stress flag is 2)
= 0: Center stresses at the center
= 1: Nodal stresses at all nodes of elements (without averaging)
(default is 0: Center)

Layer number
Layer number (for layered elements).
(default is 1)

Face flag
Face on which stresses are to be listed.
= 0: Top upper face (face 5) (used for shell element only)
= 1: Bottom lower face (face 6) (used for shell element only)
= 2: Membrane membrane stresses
= 3: Bending bending stresses

Set number
Set number of components to be listed.
= 1 lists SX, SY, SZ, TXY, TXZ and TYZ
= 2 lists P1, P2, P3, VON, INT and ERR
= 3 lists MILTXZ, MILTYZ, ILTXZ, ILTYZ, FIND and MFIND (elemental results for SHELL3L, SHELL4L and SHELL9L in STAR only, refer to ACSTR or STRMAX help). FIND and MFIND are the failure index and maximum failure index respectively
= 4 lists NX, NY, NXY, MX, MY, any MXY (average element forces and moments) for shell elements in STAR
= 5 lists VX and VY (average element shear forces) for shell elements in STAR
(default is 1)

Beginning node or element
Beginning node or element in the pattern.
(default is 1)

Ending node or element
Ending node or element in the pattern.
(default is highest node or element number defined)

Increment
Increment between nodes or elements in the pattern.
(default is 1)

Coordinate system

Coordinate system. Not prompted in the following cases:

a. VON, INT, P1, P2, P3 (invariant stresses), or ERROR (scalar).

b. Element stresses (not nodal) are requested.

c. Postdynamic analyses other than time history.

d. Stresses are requested in a coordinate system other than the global Cartesian for one or more of the element groups in the model.

e. SHELLAX elements where stresses are always calculated in the element coordinates.

(default is 0)

Notes

1. The type of analysis can be specified by the ACTPOST command.
2. The von Mises stress is computed from the basic stress components as follows:

$$VON = \{(^1/_2)[(SX - SY)^2 + (SX - SZ)^2 + (SY - SZ)^2] + 3(TXY^2 + TXZ^2 + TYZ^2)\}^{(1/2)}$$

equivalently, from the principal stresses as follows:

$$VON = \{(^1/_2)[(P1 - P2)^2 + (P1 - P3)^2 + (P2 - P3)^2]\}^{(1/2)}$$

3. The stresses for 2D and 3D elements are listed either in the Cartesian global coordinate system, or in the ECS (element coordinate system), as specified in the option in the EGROUP command. Stresses for 1D elements are always listed in the local element system. Refer to the BEAMRESLIST for listing results for beam elements.
4. Set 3 and the ERR component in set 2 are only available not available for NSTAR.

Example: **Let linear static analysis be active.**

```
STRLIST , 3, , , 1, 1, 3, 1, 5, 1,
```

This command lists top face nodal stresses resulting from load case 3 of the performed static analysis for nodes 1 through 5. The 'face of element flag' is ignored for elements other than shells.

STNLIST

Results > LIST > Strain Component

The STNLIST command lists strains for a pattern of elements.

Load case or solution step number

Load case or solution step number.

(default is the active load case or solution step)

Layer number
Layer number (for linear analysis only).
(default is 1)

Face flag
Face of element (for linear analysis only).

= 0: Top	top face (shell elements)
= 1: Bottom	bottom face (shell elements)
= 2: Membrane	membrane stresses
= 3: Bending	bending stresses

Set number
Set number of components to be listed (for nonlinear analysis only).

= 1	total strains
= 2	thermal strains. For nonlinear analysis SED and ENERGY for linear static analysis
= 3	creep strains
= 4	plastic strains
= 5	equivalent strains

(default is 1)

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.
(default is highest element number defined)

Increment
Increment between elements in the pattern.
(default is 1)

Note
The type of analysis can be specified by the ACTPOST command.

Example: **Let nonlinear static analysis be active.**

```
STNLIST , 3, 3, 1, 5, 1,
```

This command lists nodal creep strains resulting from solution step 3 of the performed nonlinear static analysis for elements 1 through 5.

SMLIST

Results > LIST > Shear/Moment Value

The SMLIST command lists shearing forces and bending moments for beam elements in linear static analysis.

Load case
Load case or number.

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.
(default is highest element number defined)

Increment
Increment between elements in the pattern.

Notes

1. The command lists shear and moment components at selected locations on the beam. The locations are defined by their parametric coordinates. The number of locations depends on the type of loading on the element. Locations on both sides of a concentrated force are selected to show the variation of shear and moment components. All components are given in the element coordinate system.
2. Refer to the BEAMRESLIST command for explanation of components.

Example: `SMLIST, 1, 1, 5, 1,`

This command lists shearing forces and bending moments for elements 1 through 5.

BEAMRESLIS

Results > LIST > Beam End Force

The BEAMRESLIS command lists nodal forces and stresses in beam elements for STAR and NSTAR results. A sample output is shown below.

Load case or solution step number
Load case or solution step number.
(default is active load case or solution step)

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.
(default is highest element number defined)

Increment
Increment between elements in the pattern.




Example: `BEAMRESLIST, 1, 1, 1, 1,`

Node 1:			Node 2:		
Load Case 1		**Element 1**	**Load Case 1**		**Element 1**
Fr	=	-4.500e+01	Fr	=	4.500e-03
Vs	=	5.700e+01	Vs	=	8.900e+01
Vt	=	0.000e+00	Vt	=	0.000e+00
Tr	=	0.000e+00	Tr	=	0.000e+00
Ms	=	0.000e+00	Ms	=	0.000e+00
Mt	=	-4.819e-05	Mt	=	-3.198e-05
P/A	=	-1.296e+02	P/A	=	1.296e-02
Ms/Ss	=	0.000e+00	Ms/Ss	=	0.000e+00
Mt/St	=	0.000e+00	Mt/St	=	0.000e+00
Smogs	=	-1.296e+02	Smax	=	1.296e-02
Smin	=	-1.296e+02	Smin	=	1.296e-02

where:

Fr	axial force
Vs	shearing force in local s-direction
Vt	shearing force in local t-direction
Tr	torsional moment
Ms	bending moment about local s-direction
Mt	bending moment about local t-direction
P/A	axial stress
Ms/Ss	bending stress due to Ms in the extreme fiber
Mt/St	bending stress due to Mt in the extreme fiber
Smin	minimum stress at the extreme fiber
Smax	maximum stress at the extreme fiber

Refer to Chapter 4 of the COSMOS/M User Guide for definition of s- and t-directions.

Note It should be noted that the Smin and Smax extreme stresses are calculated based on a rectangular cross section and hence are not accurate for non-rectangular sections.

SPRRESLIS

Results > LIST > Spring Force

The SPRRESLIS command lists forces in spring elements. Refer to the online help or the COSMOS/M User's Guide chapter 4 for details.

Load case or solution step number
Load case or solution step number.
(default is active load case or solution step)

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.
(default is highest element number defined)

Increment
Increment between elements in the pattern.

GAPRESLIS

Results > LIST > Gap Force

The GAPRESLIS command lists the normal force, tangential force (friction force), and the direction cosines for the tangential force. A sample output is shown below. Refer to Chapter 2 of the COSMOS/M User Guide for details about gap and contact problems. The command works for NSTAR only.

Load case or solution step number
Load case or solution step number.
(default is active solution step)

Beginning Element
Beginning element in the pattern.
(default is 1)

Ending Element
Ending element in the pattern.
(default is highest element number defined)

Increment
Increment between elements in the pattern.

Notes

1. Non gap elements in the pattern are ignored.
2. Components of the friction force in the global X-, Y-, and Z-directions may be obtained by multiplying Ft by Tlx, Tly, and Tlz, respectively.

Example: **Suppose NSTAR is the active analysis for postprocessing, then the command:**

```
GAPRESLIS, 2, 1, 1, 1,
```

will result in the following screen:

Time Step 2

ELEMENT	Fn	Ft	Tlx	Tly	Tlz
1	10.0	0.20	0.0	0.0	1.0

which means: The normal force in the gap element is 10.0. The friction force is 0.20 in the positive Z-direction.

FREQLIST

Results > LIST > Natural Frequency

The FREQLIST command lists natural frequencies of model resulting from Frequency or Buckling analysis.

TEMPLIST

Results > LIST > Thermal Result

The TEMPLIST command lists nodal temperature, temperature gradients and heat flux for a specified pattern of nodes and a time step.

Time step number
Time step at which results are to be listed.
(default is 1)

Set number
Listing set.
= 1: Temperature and gradient
= 2: Heat flux component/resultant
(default is 1)

Beginning Node
Beginning node label in the pattern.
(default is 1)

Ending Node
Ending node label in the pattern.
(default is the highest node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

Example: `TEMPLIST, 4,, 1, 20, 1 ,`

This command lists the temperature and temperature gradient at nodes 1 through 20 for time step number 4.

FLOWLIST

Results > LIST > Flow Result

The FLOWLIST command lists velocity, pressure, temperature, shear stresses and thermal gradients for a given pattern of nodes and a specified time step.

Time step number
Time step at which results are to be listed.
(default is 1)

Set number
Information to be listed.
= 1: Velocity, pressure and temperature
= 2: Turbulence kinetic energy
= 3: Density and Mach numbers
= 4: Temperature gradients

Beginning Node
Beginning node in the pattern.
(default is 1)

Ending Node
Ending node in the pattern.
(default is the highest node number defined)

Increment
Increment between nodes in the pattern.
(default is 1)

FLOWPROP

Results > LIST > Flow Properties

The FLOWPROP is a postprocessing command that calculates the heat flow, heat transfer coefficient and mass flow rate across a given curve in a flow field.

Time step
Time step number.

Flow property
Heat transfer/flow rate flag.
= 0: Film Coef — heat flow and heat transfer coefficient are calculated
= 1: Mass Flow Rate — mass flow rate is calculated
(default is 0: Film coef)

Entity name
Geometric entity name (for future use).
= CR — curves
= RG — regions
= SF — surfaces
(default is CR)

Beginning Curve

Beginning curve in the pattern.

Ending Curve
Ending curve in the pattern.

Increment
Increment between curves in the pattern.
(default is 1)

Reference temperature
Reference temperature for the calculation of heat transfer coefficient.

Note
Mass flow rate calculation is available only at the inlet and outlet of the flow domain.

Example:
```
FLOWPROP, 100, 0, CR, 1, 1, 1, 50.0
```
This command calculates the total heat flow rate and the heat transfer coefficient across curve 1 at the 100th time step and the reference temperature for the heat transfer coefficient is 50 degrees.

MAGLIST

Results > LIST > E_Mag Result

The MAGLIST command lists results from low- or high-frequency electromagnetic analysis. For low-frequency analysis using ESTAR, the command lists flux densities, field intensities, magnetic forces, magnetic potential (or voltages) and electric field intensities for a given pattern of nodes at a given step. For high-frequency electromagnetic analysis, the command lists electric and magnetic field intensities and magnetic potentials depending on the type of analysis performed.

Time Domain

Time step number
Solution step at which results are to be listed.
(default is 1)

Entity flag
Entity flag.
= 1: Node nodal results
= 2: Element elemental results
(default is 1: Node)

Listing set
Listing set.
= 1: Magnetic flux density
= 2: Magnetic field intensity
= 3: Magnetic force
= 4: Average forces

= 5: Electric field intensity
= 6: Electric current density
(default is 1)

Beginning Node/Element
Beginning node/element label in the pattern.
(default is 1)

Ending Node/Element
Ending node/element label in the pattern.
(default is the highest node/element number defined)

Increment
Increment between nodes/elements in the pattern.
(default is 1)

Frequency Domain

Frequency number
Frequency number (use RESULTS? for corresponding frequency values). Prompted only for 2DHFREQ as “frequency number” and for CAVAXI as “harmonic number”.
(default is 1)

Mode number
Mode number.
(default is 1)

Node/Element
Flag to activate results at nodes or at centers of elements.
= 1: Node nodes
= 2: Element elements
(default is 1)

Set number
Listing set.
= 1: Electric field list electric fields
= 2: Magnetic fieldlist magnetic fields
(default is 1)

Beginning Node/Element
Beginning node/element label in the pattern.
(default is 1)

Ending Node/Element
Ending node/element label in the pattern.
(default is the highest node/element number defined)

Increment
Increment between nodes/elements in the pattern.
(default is 1)

HF_RESLIST

Results > LIST > HF Emag Result

The HF_RESLIST command lists results from the high frequency electromagnetic analysis performed.

FTGLIST

Results > LIST > Fatigue Usage Factor

The FTGLIST command lists the results of fatigue calculations for a specified location.

Location number

Location.

= N	lists the results for location N
= 0	lists the results for the all-nodes option

Notes

1. You may use the EDIT command to review the results of fatigue calculations from fatigue output file with extension ".FTG" (use command PRINT_OPS,,,,,,,,,1 to append all the results).
2. If an S-N curve has not been defined and no usage factor is calculated, then for all possible combinations of stress conditions the alternating stresses are listed in decreasing order (for the all-nodes option no calculation is made in this case).
3. If an S-N curve has been defined, partial and cumulative usage factors are calculated. Partial usage factors and allowable cycles are listed for the alternating stresses, beginning with the highest stress range. Alternating stresses are those calculated from the combination of fatigue loading conditions listed in the first and third columns. Number of event cycles used up by each combination of loading conditions and allowable cycles (interpolated from the S-N curve) are listed as well. The listed partial factors are the ratio of these two cycles.
4. If a simplified elastic-plastic calculation is performed, the results also include the calculated linearized stresses as well as the temperature at the location.

Example 1: `FTGLIST, 5`

Lists the results of fatigue calculations for location 5.

Example 2: `FTGLIST, 0`

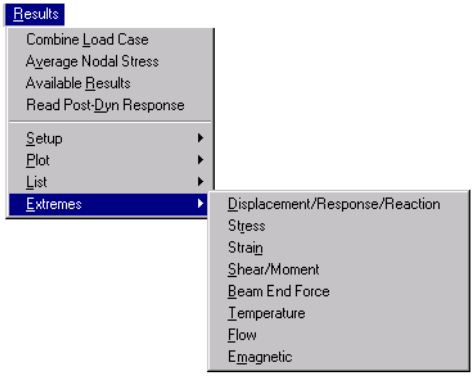
Lists the results of fatigue calculations for the all-nodes option.

▶ EXTREMES Menu

Results > EXTREMES

This menu contains commands related to examining the extreme values. Use the LISTLOG command to write the results to a file.

Figure 11-5 Extremes Menu



DISMAX

Results > EXTREMES >Displacement/Response/Reaction

The DISMAX command lists the extreme values of a specified displacement component for the active type of analysis. Displacement values within a specified percentage of the extreme values are also listed. All listing is done for the active type of analysis.

Load case, mode shape or solution step number

Load case, mode shape or solution step number. Enter 0 to check all load cases, mode shapes, or solution steps.
(default is the active load case, mode shape or solution step)

Component

Displacement component to be listed.

= UX	displacement in the x-direction
= UY	displacement in the y-direction
= UZ	displacement in the z-direction
= RX	rotation about the x-direction
= RY	rotation about the y-direction
= RZ	rotation about the z-direction
= URES	resultant displacement

The following components are also available for STAR, ASTAR, and NSTAR:

= RFX	reaction force in the x-direction
= RFY	reaction force in the y-direction
= RFZ	reaction force in the z-direction
= RFRES	resultant reaction force
= RMX	reaction moment about the x-direction
= RMY	reaction moment about the y-direction
= RMZ	reaction moment about the z-direction
= RMRES	resultant reaction moment

The following components are also available for ASTAR and NSTAR:

= VX	velocity in the x-direction
= VY	velocity in the y-direction
= VZ	velocity in the z-direction
= WX	angular velocity about x-direction
= WY	angular velocity about y-direction
= WZ	angular velocity about z-direction
= VRES	resultant velocity
= AX	acceleration in the x-direction
= AY	acceleration in the y-direction
= AZ	acceleration in the z-direction
= BX	angular acceleration about x-direction
= BY	angular acceleration about y-direction
= BZ	angular acceleration about z-direction
= ARES	resultant acceleration
	(default is URES)

Threshold Percentage

Percentage tolerance in listing. Displacements within the extreme value are also listed.
(default is 5)

Criterion flag

Criterion for the listing.

= 0: Absolute	absolute maximum
= 1: Max	algebraic maximum
= 2: Min	algebraic minimum
	(default is 0)

Sort flag

Sorting flag.

= 0: No	unsorted list (sequential listing)
= 1: Yes	sorted list (displacement value)
	(default is 1: Yes)

Notes

1. The type of analysis can be specified using the ACTPOST command.
2. Reaction forces are only available for STAR and NSTAR.

Example: `DISMAX, 3, UX, 10, 1, 1,`

This command lists the maximum x-displacement. Displacements within 10% of the maximum value are also listed. The listing is done for load case, solution step or mode shape 3. The list is sorted with respect to the maximum algebraic values.

STRMAX

Results > EXTREMES > Stress

The STRMAX command lists the extreme values of a specified stress component for the active type of analysis. Stress values within a specified percentage of the extreme values are also listed. Use BEAMRESMAX for beam elements and SPRRESMAX for spring elements.

Load case or solution step number

Load case or solution step number. Enter 0 to check all load cases or solution steps.
(default is the active load case or solution step)

Component

Stress component.

= SX	normal stress in the x-direction
= SY	normal stress in the y-direction
= SZ	normal stress in the z-direction
= TXY	shear stress in the x-y plane
= TXZ	shear stress in the x-z plane
= TYZ	shear stress in the y-z plane
= P1	normal stress in the principal x-axis direction
= P2	normal stress in the principal y-axis direction
= P3	normal stress in the principal z-axis direction
= VON	von Mises stress
= INT	stress intensity
= ERR	stress error (not available for NSTAR)

The following components are only available in linear static analysis for composite shell elements only:

MILTXZ	maximum interlaminar shear in x-z plane. Layers from the layer specified below, to the maximum layer number are searched. Use 1 for the layer number to search all layers.
MILTYZ	maximum interlaminar shear in y-z plane. Layers from the layer specified below, to the maximum layer number are searched. Use 1 for the layer number to search all layers.
ILTXZ	interlaminar shear in x-z plane
ILTYZ	interlaminar shear in y-z plane
FIND	failure index
MFIND	maximum failure index. Layers from the layer specified below, to the maximum layer number are searched. The search

includes both top and bottom faces. Use 1 for the layer number to search all layers for both top and bottom faces.
(default is VON)

Stress flag

Stress flag.

= 1: Node	nodal stress
= 2: Element	element stress (at element center)

(default is 1: Node)

Layer number

Layer number. (Used for element stresses only)
(default is 1)

Face flag

Face flag for shell elements.

= 0: Top	top face (face 5)
= 1: Bottom	bottom face (face 6)
= 2: Membrane	membrane stresses
= 3: Bending	bending stresses

Threshold percentage

Percentage tolerance in listing. Stresses within the specified percentage from the extreme values are also listed.
(default is 5%)

Criterion flag

Criteria for sorting the list.

= 0: Absolute	absolute maximum.
= 1: Max	algebraic maximum.
= 2: Min	algebraic minimum.

(default is 0)

Sort flag

Sorting flag.

= 0: No	unsorted list (sequential listing)
= 1: Yes	sorted list

(default is 1)

Note

The active type of analysis can be changed using the ACTPOST command.

Example: `STRMAX, 3, SX, 1, 0, 1, 10, 1,,`

This command lists the algebraic maximum SX component for load case, or solution step number 3. Stress values within 10% of the maximum are also listed.




STNMAX

Results > EXTREMES > Strain

The STNMAX command lists the extreme values of a specified strain component for the active type of analysis. Strain values within a specified percentage of the extreme values are also listed.

Load case or solution step number
Load case or solution step number. Enter 0 to check all load cases or solution steps.
(default is the active load case or solution step)

Component
Strain component.

= EPSX	normal strain in the x-direction
= EPSY	normal strain in the y-direction
= EPSZ	normal strain in the z-direction
= GMXY	shear strain in the x-y plane
= GMXZ	shear strain in the x-z plane
= GMYZ	shear strain in the y-z plane
= ESTRN	equivalent shear strain (see note below)
= SED	strain energy density
= ENERGY	total strain energy

(default is ESTRN)

Strain type
Strain type. (for nonlinear analysis only)
= 0: Total
= 1: Thermal
= 2: Creep
= 3: Plastic
(default is 0)

Layer number
Layer number.
(default is 1)

Face flag
Face of element.

= 0: Top	top face (shell elements)
= 1: Bottom	bottom face (shell elements)
= 2: Membrane	membrane stresses
= 3: Bending	bending stresses

Threshold percentage
Percentage tolerance in listing. Stresses within the specified percentage from the extreme values are also listed.
(default is 5%)

Criterion flag
Criteria for sorting the list.

= 0: Absolute	absolute maximum
= 1: Max	algebraic maximum
= 2: Min	algebraic minimum
	(default is 0)

Sort flag
Sorting flag.

= 0: No	unsorted list (sequential listing)
= 1: yes	sorted list
	(default is 1)

Note
The active type of analysis can be changed using the ACTPOST command.

SMMAX

Results > EXTREMES > Shear/Moment

The SMMAX command lists the extreme values of a specified shearing force or bending moment component for beam elements. Unlike the BEAMRESMAX command (which only considers nodal quantities), the SMMAX command evaluates the extreme value along the full length of the beam. The command is applicable for results from STAR. Component values within a specified percentage of the extreme value are also listed. All results are listed in the element coordinate system.

Load case number
Load case number.
(default is the active load case)

Force Label
Beam results component to be listed.

= FR: Axial Force	axial force
= VS: Shearing Force (S dir)	shearing force in local s-direction
= VT: Shearing Force (T dir)	shearing force in local t-direction
= TR: Torsional Moment	torsional moment
= MS: Bending Moment (S dir)	bending moment about local s-direction
= MT: Bending Moment (T dir)	bending moment about local t-direction
	(default is VS)

Threshold percentage
Percentage tolerance in listing.
(default is 5)

Criterion flag
Criterion for the listing.

= 0: Absolute	absolute maximum
= 1: Max	algebraic maximum
= 2: Min	algebraic minimum

(default is 0)

Sort flag

Sort flag.

= 0: No	unsorted list (sequential listing)
= 1: Yes	sorted list (displacement value)

(default is 1: Yes)

BEAMRESMAX

Results > EXTREMES > Beam End Force

The BEAMRESMAX command lists the extreme value of a specified nodal force or stress component for a pattern of beam elements. Component values within a specified percentage of the extreme value are also listed. All results are listed in the element coordinate system. Refer to the BEAMRESLIST and SMMAX, and SMPLOT commands for relevant information.

Load case or solution step number

Load case or solution step number.
(default is the active load case or solution step)

Component

Nodal force or stress component.

= FR: Axial Force	axial force
= VS: Shearing Force (S dir)	shearing force in local s-direction
= VT: SHearing Force (T dir)	shearing force in local t-direction
= TR: Torsional Moment	torsional moment
= MS: Bending Moment (S dir)	bending moment about local s-direction
= MT: Bending Moment (T dir)	bending moment about local t-direction
= SMIN: Minimum Stress	minimum stress at the extreme fiber
= SMAX: Maximum Stress	maximum stress at the extreme fiber
= P/A: Axial Stress	axial stress
= MS/SS: Bending Stress due to MS	bending stress due to Ms in the extreme fiber
= MT/ST: Bending Stress due to MT	bending stress due to Mt in the extreme fiber

(default is Smax)

Threshold percentage

Percentage tolerance in listing.
(default is 5)

Criterion flag

Criterion for the listing.

= 0: Absolute	absolute maximum
= 1: Max	algebraic maximum
= 2: Min	algebraic minimum

(default is 0)

Notes

1. Refer to Chapter 4 of the COSMOS/M User Guide for definition of s- and t-directions.
2. It should be noted that the Smin and Smax extreme stresses are calculated based on a rectangular cross section and hence are not accurate for non-rectangular sections.

SPRRESMAX

Results > EXTREMES > Spring Force

The SPRRESMAX command lists the extreme values of a specified nodal force for a pattern of truss elements. Compo nent values within a specified percentage of the extreme value are also listed. All results are listed in the element coordinate system.

Load case or solution step number

Load case or solution step number.
(default is the active load case or solution step)

Component

Spring force to be searched

= FORCE: Axial Force	axial force
= TORQUE: Torque	shearing force in local s-direction
= SHEAR-S: Shearing Force (S dir)	shearing force in local s-direction
= SHEAR-T: Shearing Force (T dir)	shearing force in local t-direction

(default is FORCE)

Threshold percentage

Percentage tolerance in listing.
(default is 5)

Criterion flag

Criterion for the listing.

= 0: Absolute	absolute maximum
= 1: Max	algebraic maximum
= 2: Min	algebraic minimum

(default is 0)

Note

Refer to Chapter 4 of the COSMOS/M User Guide for definition of s- and t-directions.

TEMPMAX

Results > EXTREMES > Temperature

The TEMPMAX command lists the extreme values of temperature, temperature gradient, or heat flux component. Values within a specified percentage from the extreme values are also listed.

Time step number

Time step number. Enter 0 to check all time steps.
(default is the active time step)

Component

Component.

= TEMP	nodal temperature
= GRADX	temperature gradient in the x-direction
= GRADY	temperature gradient in the y-direction
= GRADZ	temperature gradient in the z-direction
= GRADN	resultant temperature gradient
= HFLUXX	heat flux in the x-direction
= HFLUXY	heat flux in the y-direction
= HFLUXZ	heat flux in the z-direction
= HFLUXN	resultant heat flux.
	(default is TEMP)

Threshold percentage

Percentage tolerance in listing.
(default is 5)

Criterion

Criterion for the listing.

= 0: Absolute	absolute maximum
= 1: Max	algebraic maximum
= 2: Min	algebraic minimum
	(default is 0)

Sort flag

Sorting flag.

= 0: No	unsorted list (sequential listing)
= 1: Yes	sorted list
	(default is 1)

Example: `TEMPMAX, 3, GRADX, 10, 1, 1,`

This command lists the algebraic maximum temperature gradient in the x-direction. Values within 10% of the maximum value are also listed. The list is sorted with respect to the gradient values.

FLOWMAX

Results > EXTREMES > Flow

The FLOWMAX command lists the extreme values of a specified component resulting from a fluid flow analysis. Values within a specified percentage from the extreme values are also listed.

Time step number

Time step number. Enter 0 to check all time steps.

Component

Component to be loaded to the plot buffer.

= VRES	resultant velocity
= VX	velocity in the x-direction
= VY	velocity in the y-direction
= VZ	velocity in the z-direction
= STREAM	stream function
= PRESS	pressure
= TAUXY	shear stress in the X direction in the XZ plane
= TAUYZ	shear stress in the Y direction in the XY plane
= TAUZX	shear stress in the Z direction in the YZ plane
= TEMP	temperature
= GRADX	temperature gradient in X-direction
= GRADY	temperature gradient in Y-direction
= GRADZ	temperature gradient in Z-direction
= GRADN	temperature gradient in normal direction
= TKE	turbulence kinetic energy
= EPS	dissipation rate
= DENS	fluid density
= MACH#	mach number
	(default is VRES)

Threshold percentage

Percentage listing tolerance.
(default is 5)

Criterion flag

Criterion for the listing.

= 0: Absolute	absolute maximum
= 1: Max	algebraic maximum
= 2: Min	algebraic minimum
	(default is 0)

Sort flag

Sorting flag.

= 0: No	unsorted list (sequential listing of nodes)
= 1: Yes	sorted list (Mag. flux values)
	(default is 1)

Example:

```
FLOWMAX, 3, STREAM, 10, 1, 1,
```

This command lists the algebraic maximum stream function values. Values within 10% of the maximum value are also listed. The list is sorted with respect to the algebraic maximum values.

MAGMAX

Results > EXTREMES > Emagnetic

The MAGMAX command lists the extreme values of a specified component resulting from low- or high-frequency electromagnetic analysis. Values within a specified percentage from the extreme values are also listed.

Time Domain

Solution step number
Solution step number. Enter 0 to check all solution steps. (1 to 3000)
(default is the active solution step)

Entity flag
Flag to activate results at nodes or centers of elements.
= 1: Node nodes
= 2: Element elements
(default is 1)

Component
Component. Refer to the ACTMAG command for valid components.

Threshold percentage
Percentage tolerance in listing.
(default is 5)

Criterion flag
Criterion for the listing.
= 0: Absolute absolute maximum
= 1: Max algebraic maximum
= 2: Min algebraic minimum
(default is 0)

Sort flag
Sorting flag.
= 0: No unsorted list (sequential listing of nodes)
= 1: Yes sorted list (Mag. flux values)
(default is 1)

Frequency Domain

Frequency number
Frequency number (use RESULTS? for corresponding frequency values). Prompted only for 2DHFREQ as “frequency number” and for CAVAXI as “harmonic number”.
(default is 1)

Mode number
Mode number.
(default is 1)

Entity flag
Flag to activate results at nodes or centers of elements.
= 1: Node nodes
= 2: Element elements
(default is 1)

Component
Component. Refer to the ACTMAG command for valid components for various

types of analysis.

Threshold percentage

Percentage tolerance in listing.
(default is 5)

Criterion flag

Criterion for the listing.

= 0: Absolute	absolute maximum
= 1: Max	algebraic maximum
= 2: Min	algebraic minimum
	(default is 0)

Sort flag

Sorting flag.

= 0: No	unsorted list (sequential listing)
= 1: Yes	sorted list
	(default is 1)




Index

D

E

F

G

H

I

J

L

M

N

O

P

Q

R

S

T

U

V

W

X

Z