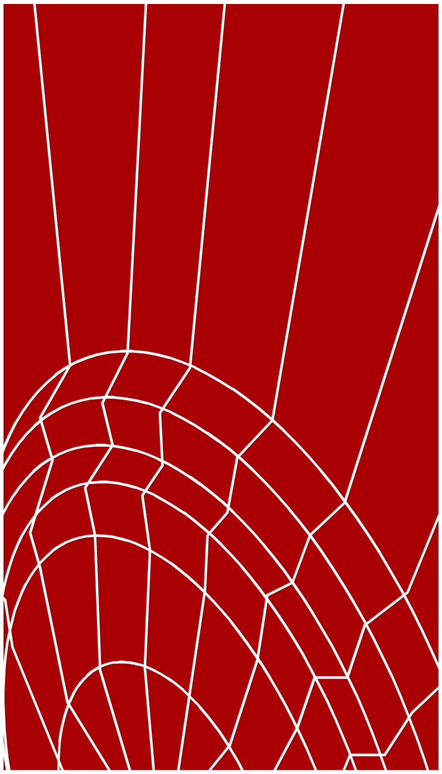
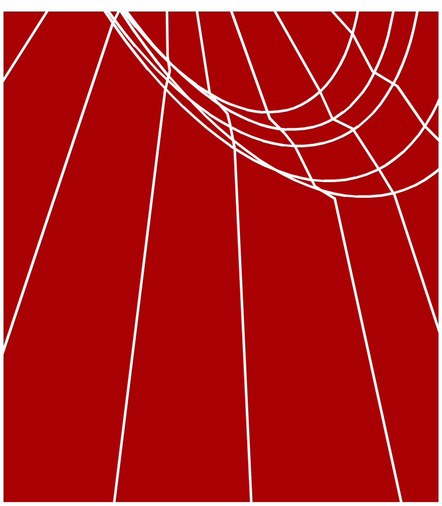


MARC
Analysis
Research
Corporation

Mentat 3.3-MARC K7.3: New Features

User Guide



MARC Analysis Research Corporation
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306 USA

Phone: (650) 329-6800
FAX: (650) 323-5892

Document Title: **Mentat 3.3-MARC K7.3: New Features**
Part Number: UG-3012-01
Revision Date: August, 1998

PROPRIETARY NOTICE

RESTRICTED RIGHTS NOTICE

TRADEMARKS

Contents

Plotting PostScript Image Files

Arrow Settings

Element Extrapolation

4• General Technology

Using N to 1 and N to N Options in Links

Specifying Node Lists and Node Paths

Buckle Solutions Using Lanczos Method

Adaptive Load Stepping

Contents

Design Constraints

Using Design Variables and Constraints

Post-Processing of Sensitivity Results

Post-Processing of Optimization Results

8• Element Technology

New Shell Elements

Contents

New Rebar Elements

9• Fluid Mechanics

Fluid Analysis

Boundary Conditions for Fluid Analysis

Initial Conditions for Fluid Analysis

Material Properties for Fluid Analysis

Loadcases for Fluid Analysis

10• Material Modeling

Using the Narayanaswamy Model

New Approaches in Plasticity Modeling

Mooney-Rivlin and Ogden Formulations

User Subroutines in Hypoelastic Properties

Experimental Data Fitting

Contents

11• Analysis Integration

Using Radiation Viewfactors

Appendixes

A: User Enhancements

List of User Enhancements

B: Fluid Elements

Fluid Element Types

C: MARC Data Reader Support

D: NASTRAN Writer Data Entries

E: Command Line Parameters

Contents

F: Demonstration Problems

Index

1• Introduction

About This User Guide

Supporting Documentation

• Introduction

About This User Guide

What's Covered in This User Guide

This user guide describes the major enhancements in Mentat 3.3-MARC K7.3 and how to use them in nonlinear finite element analysis.

About Mentat 3.3 Enhancements

The Mentat 3.3 enhancements include new meshing capabilities, improvements in the graphics performance and strengthening of the interface to the analysis. The enhancements are covered in the following chapters of this guide:

- *Basic Procedures*—describes how to use new features like the file browser, adaptive plotting, import-export utility and view snapshot (*p. 37*).
- *General Technology*—provides instructions and concepts on the new Links menus for tying and linking data (*p. 73*).
- *Mesh Generation*—describes how to use the new Advancing Front and Delaunay Triangulation meshers for planar and surface geometries (*p. 109*).
- *Analysis Integration*—explains the concepts and procedures to calculate radiation viewfactors for planar, axisymmetric, and 3-D regions (*p. 275*).

About MARC K7.3 Enhancements

The MARC K7.3 enhancements include significant improvements in the functionality, performance, usability and reliability of the program. These enhancements are covered in the following chapters of this guide:

- *General Technology*—explains the procedure for adaptively controlling time steps of nonlinear analysis (*p. 73*).
- *Contact*—describes the contact analysis process and the new friction model to represent perfect stick-slip (*p. 139*).

- *Design Sensitivity and Optimization*—describes how to use design variables including classical variables, homogeneous and composite material properties for optimization problems; explains the procedure for picking and setting constrained responses for a sensitivity analyses (*p. 171*).
- *Element Technology*—describes the new shell and rebar elements (*p. 171*).
- *Fluid Mechanics*—describes the solution of fluid mechanics problems using the Navier-Stokes equations for planar, axisymmetric and 3-D geometries (*p. 217*).
- *Material Modeling*—provides instructions on how to use the new material models, subroutine HYPELA2, and the Experimental Data Fitting feature (*p. 245*).

About the Appendixes

Use the following table to find out about the type of reference information in the appendixes:

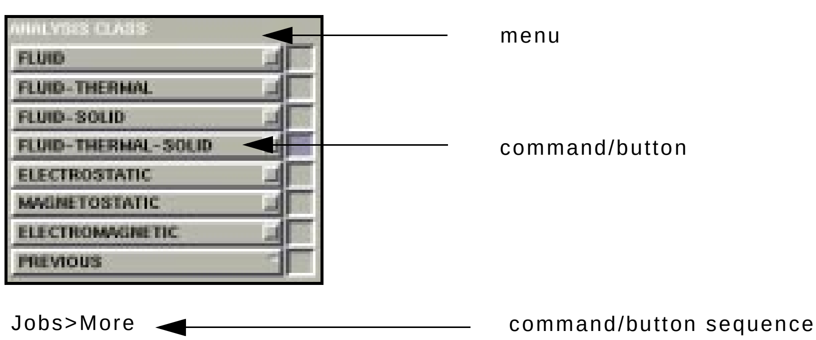
To see...	Refer to Appendix,
lists of new or modified: • parameters • model definitions • history definitions • user subroutines • element types	A (*p. 287*)
types of elements that you can represent in the fluid region in analysis,	B (*p. 293*)
lists of parameters and model-definition options supported by the **MARC** Reader,	C (*p. 297*)
data issues relating to NASTRAN Writer; bulk data entries,	D (*p. 305*)
descriptions of the command line parameters and options in **MENTAT** and **MARC**,	E (*p. 315*)
lists of demonstration problems associated with selected options in **MARC**,	F (*p. 321*)

Conventions Used in This Guide

This user guide uses the following conventions:

Text element in this guide...	Convention used
System-related terms; program names (e.g., Mentat 3.3, MARC K7.3); button names	small caps, bold (e.g., **MENTAT, MARC; LOADCASE**)
Main menus and windows	title case (e.g., Analysis Class)
Left-mouse click	Click; <ML>
Middle mouse click; right mouse click	<MM>;<MR>
Cross-references	italicized text (e.g., *See “Running Mentat 3.3” on page 25*)
Command/ Button sequence in **MENTAT**	Example: Jobs>More>Fluid-Solid>Analysis Options

Figures of menus contain the labels depicting the command/ button sequence (e.g., Jobs>More) in **MENTAT** to display that menu.



Questions or Comments

For questions or comments about this guide, contact:

Technical Publications
MARC Analysis Research Corporation
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306, USA
Tel: 1 650 329 6800
Fax: 1 650 323 5892

E-mail: document@marc.com

Supporting Documentation

In addition to this user guide, both Mentat 3.3 and MARC K7.3 feature a variety of documentation designed for a range of user levels—beginning to advanced. For your convenience, some of these documents are available in both online and print media.

About Printed Documentation for Mentat 3.3

Use the following table to locate additional Mentat 3.3 documents that address your specific needs:

For...	refer to...
an introduction to the basics of MENTAT and tutorials for nonlinear finite element analysis using MENTAT,	*Mentat User's Guide*
a description of the capabilities of MENTAT based on the menu layout,	*Mentat 3.1 Command Reference*

Displaying Online Help in Mentat 3.3

To view online help for a specific command or a group of commands, click <MM> on the relevant button in MENTAT.

click <MM> on a button to display the corresponding online help window

Links>Servo

Printed and Online Documentation for MARC K7.3

Volumes A, B, C, and D contain the documentation for MARC K7.3. These volumes are all available online or in print. To view the volumes online, refer to your CD-documentation instructions.

Use the following table to locate additional MARC K7.3 documents that address your specific needs:

For...	refer to...
background information on the capabilities of MARC,	*Volume A: Theory and User Information*
description of elements in the library and the data necessary to use them,	*Volume B: Element Library*
instructions on the file format of the MARC input file,	*Volume C: Program Input*
description of user subroutines,	*Volume D: User Subroutines/ Special Routines*
demonstration problems for the illustration of additional analysis capabilities of MARC	*Volume E: Demonstration Problems*

Periodic Updates for Mentat 3.3 and MARC K7.3 Documentation

For periodic updates on our tutorials and documentation, choose the Tutorials/Docs link in our homepage at the following website:

http://www.marc.com

2• Getting Started

Running Mentat 3.3

Running MARC K7.3

Understanding the Mentat 3.3 Menu System

Dynamic Viewing

• Getting Started

Running Mentat 3.3

Running Mentat 3.3 at the Command Line

After you have installed **MENTAT** on your workstation, you can run **MENTAT** by typing either of the following at the command line:

- mentat
- mentat -(*command line parameter*)

New Command Line Parameters Featured in Mentat 3.3

The new command line parameters in **MENTAT** enable you to utilize the new capabilities at the command line level. Use the following table to match your operation with a command line parameter (for a list of additional command line parameters, *see Appendix E: Command Line Parameters,* *p. 315*);

Operation: To...	use the command line parameter...	Option
use the OpenGL graphic system	-ogl	
create a log file with the name *lfile* (the default filename is *mentat.log*)	-lf	*lfile*
record all input into a procedure file named, *mentat.proc*	-rf	*rfile*
run **MENTAT** in grayscale	-gr	

When you launch MENTAT, it automatically creates a log file and a procedure file. These files are deleted when you launch your next MENTAT session.

Getting Started

Getting Started

Running MARC K7.3

About Shell Scripts

MARC uses shell scripts to run machine-dependent controls or command statements. The shell script submits a job and automatically takes care of all file assignments. You must execute the shell script in the directory where all the input and output files for the MARC job are available.

You should also ensure that every MARC job has a unique name qualifier and that all MARC output files connected to that job use this same qualifier. Use the default MARC FORTRAN units for restart, post, and change state.

> **MARC input filenames should be in the form:**
>
> ***job_name.dat***
>
> **The prefix, *job_name*, is the name qualifier that you can choose. All input files have a *.dat* extension.**

Submitting a Job in MARC

To submit an actual job in MARC, type the following command (the backslash, \, ensures continuation of the command line):

```
run_marc -prog prog_name -jid job_name -rid rid_name -pid pid_name \
-sid sid_name -user user_name -back back_value -prt prtflag \
-ver verify_value -save save_value -vf viewname -def defn
```

The minimum requirement for the command line is:

-jid job_name

For a description of the parameters, including the two new commands, -vf and -def, see Appendix E–*Table of MARC K7.3 Command Line Parameters* (*p. 319*).

Additional Information

Use the following table to find more information:

For...	refer to the Chapter...	in...
installation instructions for Mentat 3.3	*Running Mentat*	Mentat 3.3 Installation Guide
installation instructions for MARC K7.3	*Running MARC*	MARC K7.3 Installation Guide
a description of how to execute MARC K7.3 on your computer	*Chapter 2– Program Initiation*	Vol. A
description of commands and options	*Chapter 2– Program Initiation*	Vol. A
description of commands and options with examples of valid input	*Running MARC*	MARC K7.3 Installation Guide

Understanding the Mentat 3.3 Menu System

About Enhancements in the Mentat 3.3 Menus

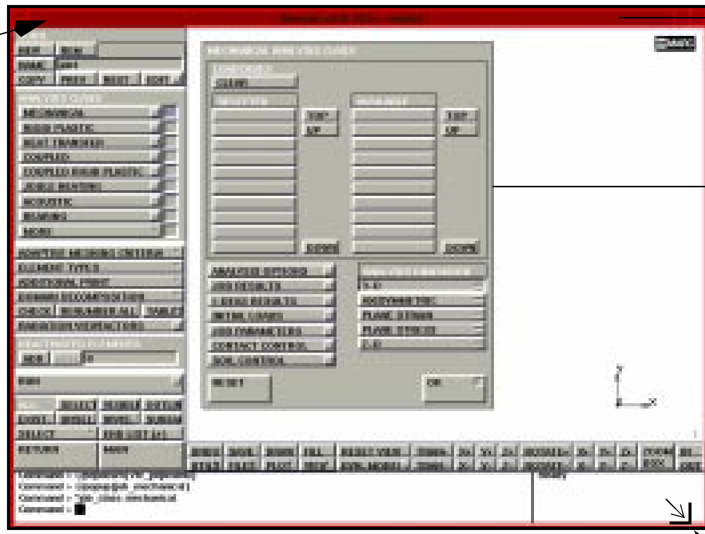
Here are some of the key enhancements to the Mentat 3.3 menu system:

- Menu File is simpler and easier to use; you can customize using the Window Parenting feature (*See "Using the Root Window for Parenting" on page 31*).
- At any time during a MENTAT session, you can resize the selected windows dynamically (*See "Resizing a Mentat 3.3 Window" on page 29*).
- Procedure file, containing a record of all of your commands, is created automatically (*See "New Command Line Parameters Featured in Mentat 3.3" on page 25*); the default filename is mentat.proc.
- Log file, containing diagnostics that MENTAT reports in the scroll area, is created automatically; you can use it in situations such as, checking geometry surfaces, where many messages may appear (*See "New Command Line Parameters Featured in Mentat 3.3" on page 25*); the default filename is mentat.log.
- Menu system is faster as the menu area is selectively redrawn.
- Menu buttons are highlighted as you move the cursor over them.
- You can create stand-alone static menus that are permanent.

Resizing a Mentat 3.3 Window

You can now resize all MENTAT windows (e.g., Main Menu) that are parented by the root window of your system's window manager.

During a MENTAT session, you can resize a window parented by the root window as you would any X-window in your environment. The following figure shows a type of resizing supported by the root window of the window manager in a HP-UX 10.0 environment.

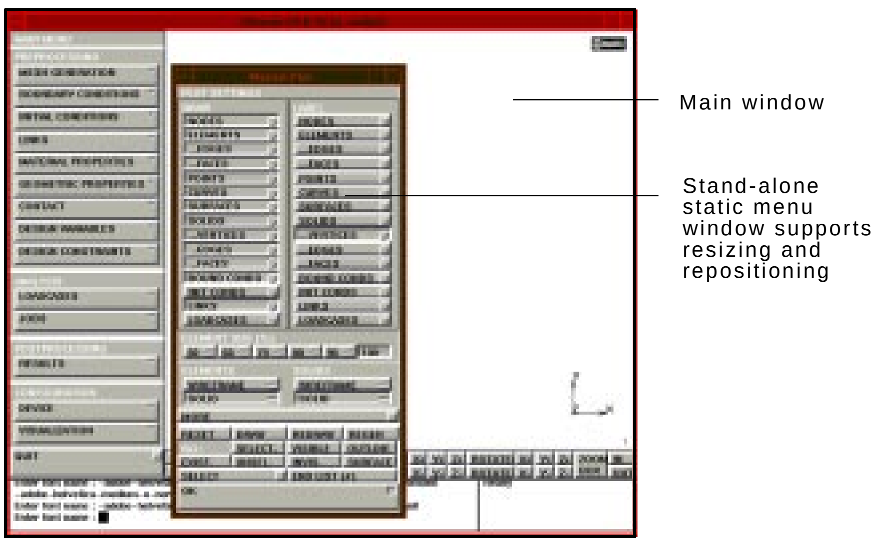


Resizable and Non-Resizable Windows

Setting Stand-Alone Static Menus

You can now set up static menus to be permanent and parented by the root window of your system's window manager. The stand-alone feature enables you to keep the following static menus permanently on display:

- UTILS
- FILE
- PLOT
- VIEW
- SELECT



To set stand-alone static menus in MENTAT, add the following line to the top of the **main.ms** file:

```
#define STATIC_STANDALONE
```

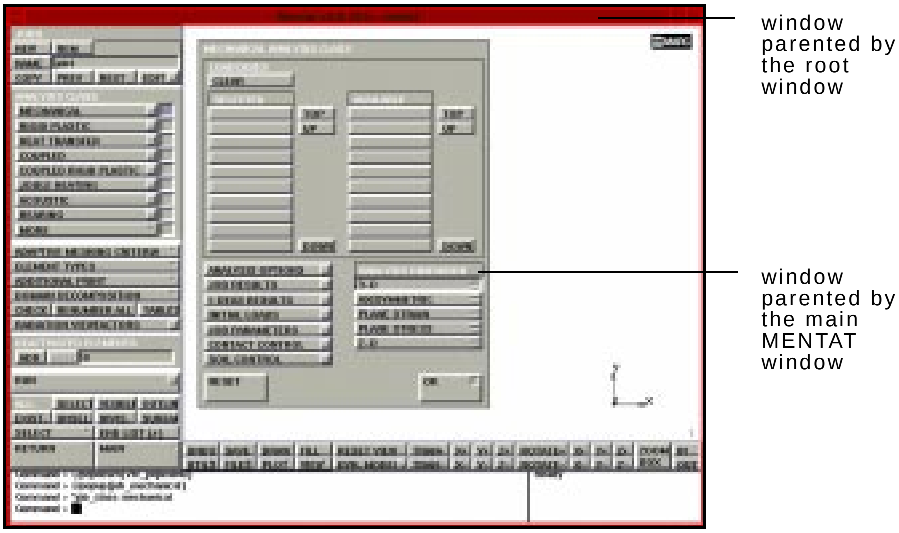
You can also enable the stand-alone feature by starting MENTAT with the following command line option:

```
-df  STATIC_STANDALONE
```

Using the Root Window for Parenting

All MENTAT menus and pop-ups appear in windows. The windows are parented by either the root window of your system's window manager or the main MENTAT window.

Windows that are parented by the root window of the window manager contain the window manager attributes: resizable borders, window manager menu, iconify or maximize buttons, and title.



Parenting of Windows in Mentat 3.3

You can set a window to be parented by the root window system. The window then has the typical attributes (window manager borders and options, iconize and minimize buttons, etc.) of a window that is parented by a root window.

Setting a Window to Be Parented by the Root Window

To set a window to be parented by the root window, remove the parent attribute from the definition of the window in the menu file.

EXAMPLE

```
in_window graphics_window {
#parent mentat                          Comment this line
title “Mentat Graphics Window”          Add this line
origin 32 0 size 96 112
background_color background
border_width 0 border_color border
buffering double
}
```

Dynamic Viewing

About the Dynamic Viewing Feature

The Dynamic Viewing feature enables you to control the movement of the model with respect to the screen using the mouse. The Dynamic Viewing feature in Mentat 3.3 supports more keyboard and mouse options than earlier versions of MENTAT.

Moving an Object or a Model

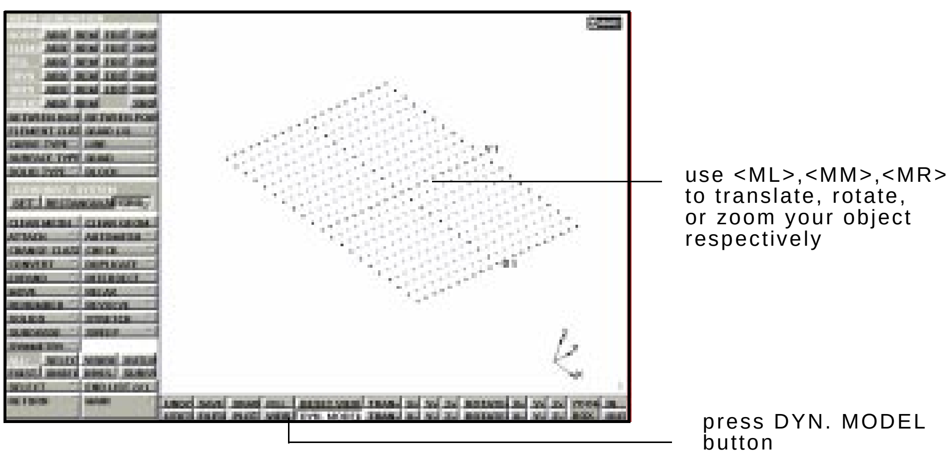
To move a grid object or a model:

Using the DYN. MODEL Button

1. Click DYN.MODEL in the static menu area.
2. Hold down <ML> and move the mouse over your object or model to reposition it.

Using the Alt Key (for UNIX Systems Only)

1. Press Alt.
2. Hold down <ML> and move the cursor over the graphics area.



Moving the Grid Dynamically

Getting Started

About Dynamic Viewing Options

You can activate the dynamic viewing feature by choosing **DYN.MODEL** in the static menu area. After you activate the dynamic viewing feature, these are your viewing options:

- Translate
- Rotate
- Zoom In

Translating a View

To translate a view:

1. Click **DYN.MODEL**.
2. Hold down <ML> and move the cursor in the graphics area.

Rotating a View

To rotate a view (the rotating behavior is similar to that of a space-ball device):

1. Click **DYN.MODEL**.
2. Hold down <MM> and move the cursor in the graphics area.

Zooming a View In

To zoom a view in:

1. Click **DYN.MODEL**.
2. Hold down <MR> and move the cursor in the graphics area.

Additional Information

Use the following table to find more information:

For...	refer to the Chapter...	in...
an introduction to the MENTAT environment	*Mechanics of Mentat*	Mentat User's Guide

3• Basic Procedures

File Browser

Import-Export Utilities

Adaptive Plotting

View Snapshot

PostScript Image Files

Arrow Settings

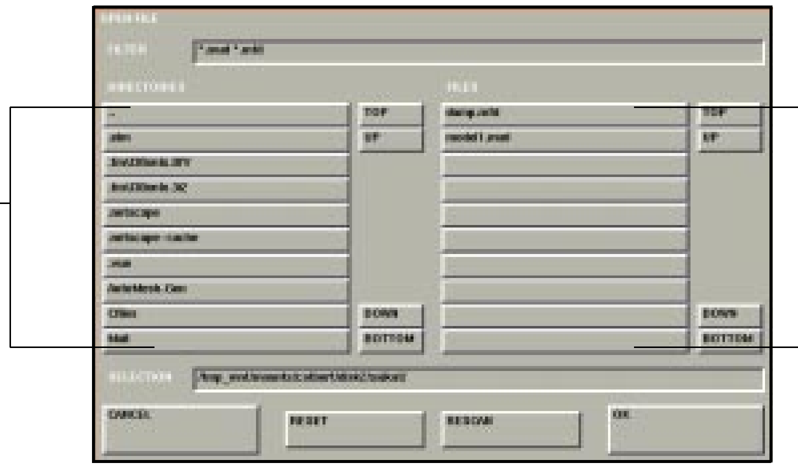
Element Extrapolation

- # Basic Procedures

File Browser

About the File Browser

Use the File Browser to select a file for all input/output (I/O) operations in MENTAT.



File>Open

Selecting a File

To select a file:

1. In the DIRECTORIES area, choose the directory which contains the file.
2. In the FILES area, locate the file.
3. Click the specific file.
4. Click OK.

Basic Procedures

Selecting a File with a Name and Location

To select a file when you know the name and location of the directory and the file:

1. In the **SELECTION** field, type in the name and location (pathname) of the directory and file.
2. Press Enter.
3. Click **OK**.

Buttons in the File Browser

To use a button in the File Browser for a specific purpose, refer to the following table:

To...	choose...
reset the directory back to the default directory	**RESET**
add or delete file extensions	**FILTER**
display any changes to files or directories since you last opened the file browser	**RESCAN**
display all the directories or files in the list above a specific directory or file	**UP**
display all the directories or files in the list below a specific directory or file	**DOWN**
move to the top of the directory or files list	**TOP**
move to the bottom of the directory or files list	**BOTTOM**
quit the I/O operation	**CANCEL**

If you specify a filename that already exists, the program prompts you for permission to overwrite the existing file.

Basic Procedures

Import-Export Utility

About the Import Feature

Use the Import feature to translate the following data types into MENTAT:

- ACIS
- DXF
- I-DEAS
- IGES
- NASTRAN
- PATRAN
- VDAFS
- MARC Input

Use the Import menu in MENTAT to translate the data types.

Files>File I/O>Import

About the Export Feature

Use the Export feature to create the following data types in Mentat:

- ACIS
- IGES
- FIDAP
- NASTRAN

Use the Export menu in **MENTAT** to create the above data types.

Files>File I/O>Export

Importing an ACIS File

To import an ACIS file:

1. Choose Files>Import.
2. Choose **BINARY** or **TEXT.**

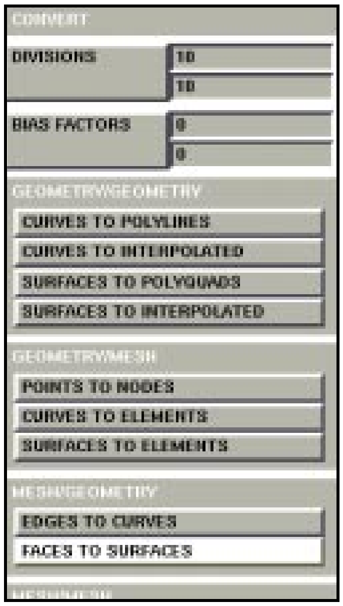
> **ACIS binary files have .sab extension. ACIS text files have a .sat extension.**

3. Click **ACIS.**
4. Using the file browser, choose the ACIS file.
5. Click **OK.**

The ACIS file that you read in comprises of a number of solids, each consisting of solids, faces, edges, and vertices.

Modifying the ACIS File

You can modify the ACIS file using the geometric operators given in the Mesh Generations>Solids menus in MENTAT. To generate a finite element mesh of these solids, you should first convert the ACIS faces into NURB surfaces using the Mesh Generation>Convert option.



Mesh Generation>Convert

Importing a DXF File

To import a DXF file:

1. Choose Files>Import.
2. Click **OPTIONS** next to the **DXF** button and use the following table to set your DXF options:

To...	Choose...
get a color from the input file	**COLOR(S)**
get a particular layer in the input file	**LAYER(S)**
specify a tolerance for the input model	**TOLERANCE**
create a summary report of the input file that includes the following information: • DXF version number • DXF entity summary • Mentat entity summary	**REPORT**

3. Click **DXF**.
4. Using the file browser, choose the AutoCad file.
5. Click **OK**.

The DXF file is read in and converted to surfaces, curves, and geometric points.

Importing an I-DEAS File

To import an I-DEAS file:

1. Choose Files>Import.
2. Click **I-DEAS**.
3. Using the file browser, choose the I-DEAS file.
4. Click **OK**.

The I-DEAS universal file is read in and the finite element model will be updated to contain the model, boundary conditions and material properties from the SDRC model.

Importing an IGES File

To import an IGES file:

1. Choose Files>Import.
2. Click **OPTIONS** and use the following table to set your IGES options:

To...	Choose...
• turn on validation of IGES entities • correct invalid entities so that they can be processed • check explicitly-defined semantics in the IGES specification	**VALIDATE (ON)**
include model space curves from the IGES files in the Mentat model	**REAL SP CRV (ON)**
get a color from the input file	**COLOR(S)**
get a particular level in the input file	**LEVEL (S)**
specify a tolerance for the input model	**TOLERANCE**
create a summary report of the input file that includes the following information: • IGES version number • IGES entity summary • Mentat entity summary	**REPORT**

3. Click **IGES**.
4. Using the file browser, choose the IGES file.
5. Click **OK**.

The IGES file is read in and the geometric entities are converted to surfaces, curves, and points. Finite element entities are converted to equivalent MENTAT element types.

Importing a NASTRAN or a PATRAN File

To import a NASTRAN or a PATRAN file:

1. Choose Files>Import.
2. Choose NASTRAN or PATRAN.
3. Using the file browser, choose a NASTRAN or a PATRAN file.
4. Click OK.

The finite element data is merged into the MENTAT model.

Importing a VDAFS File

To import a VDAFS file:

1. Choose Files>Import.
2. Click **OPTIONS** and use the following table to set your VDAFS options:

To...	Choose...
specify a tolerance for the input model	**TOLERANCE**
create a summary report of the input file that includes the following information: • VDA version number • VDA entity summary • Mentat entity summary	**REPORT FILE**

3. Click **VDAFS**.
4. Choose the VDAFS file using the file browser.
5. Click **OK**.

The VDAFS file is read in and the geometric entities are converted to **MENTAT** trimmed and untrimmed surfaces, curves, and points.

Basic Procedures

Exporting an ACIS File

To export an ACIS file:

1. Choose Files>Export.
2. Choose **TEXT** or **BINARY.** The default setting is **TEXT**.
3. Click **ACIS**.
4. Using the file browser, choose the ACIS file.
5. Click **OK**.

An ACIS file is written out. The file consists of vertices, solids, faces, and edges. This file can be read by other ACIS-based CAD systems or by **MENTAT**.

Exporting an IGES, a FIDAP, or a NASTRAN File

To export an IGES, a FIDAP, or a NASTRAN file:

1. Choose Files>Export.
2. Click the relevant data type (e.g., IGES).
3. Using the file browser, choose the specific file.
4. Click **OK**.

When you export an IGES file, a file containing geometric entities, surfaces, curves, points and finite element entities is created. The file is based on the IGES standard, version 5.x.

When you export using the NASTRAN option, the data file export writes bulk data entries (*See "D: NASTRAN Writer Data Entries" on page 305*) from the current Mentat database. The data apply to structural analyses only.

Importing a MARC Data File

You can read in and merge existing **MARC** files with the current model in **MENTAT**. A stand-alone program, based on **MARC**, reads the parameter and model definition data only and prevents the inclusion of bad incrementation data in the model. For a list of stored data types, see Appendix C: MARC Data Reader Support (*p. 297*).

To import a **MARC** data file:

1. Choose FIles>Read.

Files

2. Choose the relevant file in the file browser.
3. Click **OK**.

Basic Procedures

Adaptive Plotting

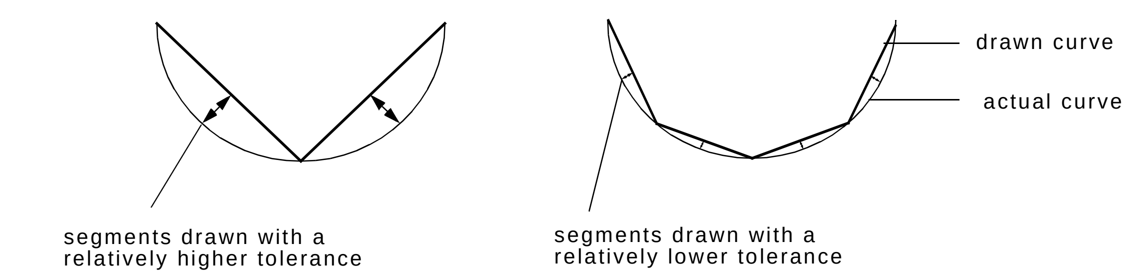
About Adaptive Plotting

The adaptive plotting feature automatically changes the number of divisions used to represent curves or surfaces based on the curvature. The feature uses more divisions in areas of high curvature which enables you to use a minimum number of graphics resources.

Based on a tolerance that you specify, the feature works identically for curves and surfaces.

Tolerance and Modes of Tolerances

The tolerance in adaptive plotting is the measure of the deviation between the curves that are drawn and the actual curve.



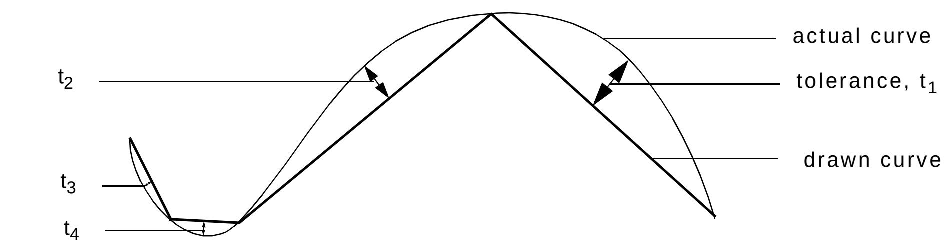
When you change the default tolerance settings in the adaptive plotting feature, you can choose either of the following types of tolerances:

- relative
- absolute

About Relative Tolerance

In relative mode, the tolerances are relative to the length of the curve and longer curves have larger margins of error.

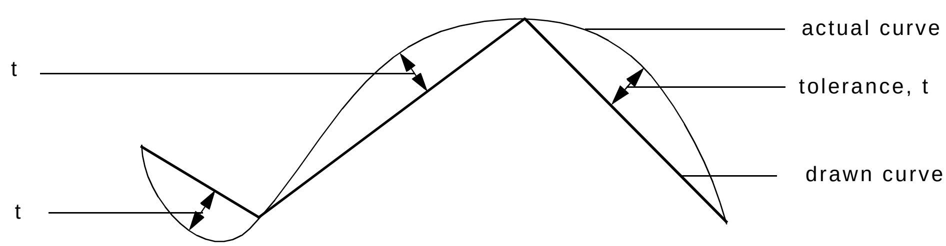
For most cases, use the relative mode of tolerance.



Relative Tolerance

About Absolute Mode of Tolerance

In absolute mode, the same tolerance is applied regardless of the length of the curve. So the margin of error for a small curve would be the same as for a long curve.



Absolute Tolerance

Use absolute tolerance in the following situations, when you:

- wish to spend fewer graphics resources on smaller curves
- are viewing the model from a larger perspective and the smaller curves seem less critical
- know the size of the model
- are able to pick a good tolerance

Safety Features

The **MIN DEPTH** and **MAX DEPTH** features are safety features that control the number of segments generated. The number of segments generated is 2^n where n is the number that you enter in the respective **MIN DEPTH** and **MAX DEPTH** fields. Use the following table to determine the number of segments:

For curves:

When you enter the value...	the lower limit for the number of segments is equal to...	and the upper limit for the number of segments is equal to...
n in **MIN DEPTH**	2^n	2^m
m in **MAX DEPTH**		

For surfaces:

When you enter the value...	the lower limit for the number of patches is equal to...	and the upper limit for the number of patches is equal to...
n in **MIN DEPTH**	$(2^n)^2$	$(2^m)^2$
m in **MAX DEPTH**		

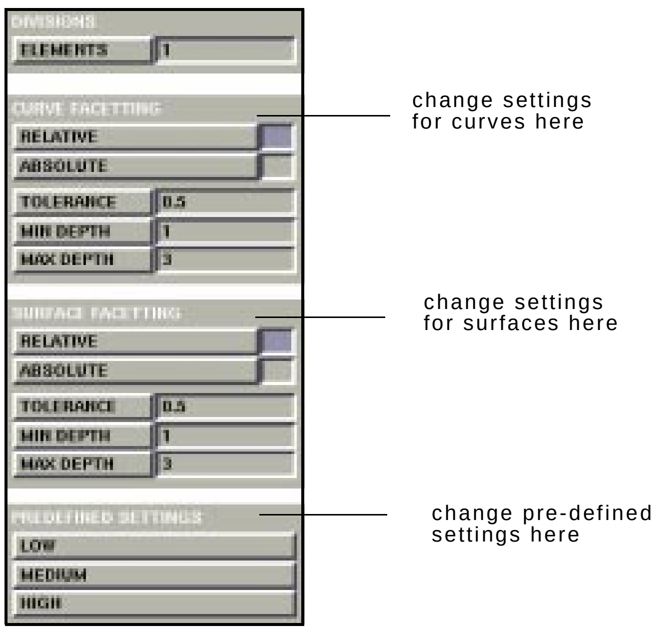
Default Settings

The following table contains the default settings for curves and surfaces:

Field	Curves	Surfaces
TOLERANCE	0.1	0.25
MIN DEPTH	3	3
MAX DEPTH	7	5

Changing the Default Settings

Although the adaptive plotting feature works automatically based on default settings, you can change the settings to reflect your needs more accurately. This enables you to minimize the number of segments or patches drawn while still retaining a given tolerance.



Plot>More>Divisions

To change the default settings:

1. Choose Plot>More>Divisions
2. Choose either **RELATIVE** or **ABSOLUTE**.

Basic Procedures

3. Choose one of the following options:

- To change the settings for curves, choose **CURVE FACETTING.**
- To change the settings for surfaces, choose **SURFACE FACETTING.**

4. Click **TOLERANCE** and type in a value.
5. Click **MIN DEPTH** and type in a value.
6. Click **MAX DEPTH** and type in a value.

About Pre-Defined Settings

Depending on the degree of accuracy of plotting that you are aiming for, you can choose from among the following pre-defined settings:

- high
- medium
- low

Use the “high” setting when the accuracy of the plotting of surfaces and curves is critical. Use the “low” setting when the accuracy of plotting is not so critical and you wish to speed up the display of the model.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
description of buttons in the Adaptive Plotting menus	*15:Visualization*	Mentat 3.1 Command Reference

View Snapshot

About View Snapshot

Use the View Snapshot feature to capture the graphics area and save it in a file in a specific file format.

You can save a file in the following formats:

- IRIS RGB
- PostScript
- TIFF
- MS-Windows BMP
- JPEG
- GIF

Creating a View Snapshot in IRIS RGB, TIFF, BMP, GIF

To create a view snapshot in IRIS RGB, TIFF, BMP, GIF:

1. Choose Utils>Snapshot.
2. Click the relevant format button (i.e., **IRIS RGB, TIFF, MS-WINDOWS BMP, GIF**) in the Create sub-menu.
3. Type the view no. (i.e.,1,2,3 or 4) in the command prompt.
4. Press Enter.
5. Type the image file name that you wish to save the view to.
6. Type "y" if you wish to save it or "n" to cancel.
7. Press Enter.

During this operation, the data is read directly from the screen. So you should have the desired view completely visible during this operation.

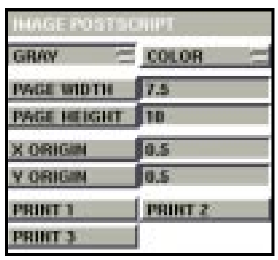
PostScript Files

Creating a view snapshot in PostScript involves the following stages:

- Setting the PostScript plotting attributes
- Creating the view snapshot file

Setting the PostScript Plotting Attributes

You can specify the PostScript plotting attributes in the **IMAGE POSTSCRIPT** menu or use the default settings in **MENTAT.**



Utils>Snapshot>PostScript

You can specify the following plotting attributes:

To...	Choose...
use color or grayscale in your file respectively	**GRAY** or **COLOR**
specify page dimensions (in inches)	**PAGE WIDTH** and/or **PAGE HEIGHT**

To...	Choose...
specify the origin of the image, in inches	**X ORIGIN** and/or **Y ORIGIN**
send a specific image file to a specific printer	one of the following: • **PRINT 1** • **PRINT 2** • **PRINT 3**
save to file	**GRAY FILE** or **COLOR FILE**

Creating View Snapshots in PostScript

To create view snapshots in PostScript:

1. Click **UTILITIES** in the static menu area.
2. Click **SNAPSHOT** in the **UTILITIES** menu.
3. Click **POSTSCRIPT** in the **CREATE** sub-menu.
4. Type the view no. (i.e.,1,2,3 or 4) in the command prompt.
5. Press Enter.
6. Type the image file name that you wish to save the view to.
7. Type “y” if you wish to save it or “n” to cancel.
8. Press Enter.

JPEG Files

Creating a view snapshot in JPEG involves the following stages:

- Setting the JPEG attributes
- Creating the view snapshot file

Basic Procedures

Setting the JPEG Attributes

Use the following table to set the JPEG attributes:

To set the JPEG attribute...	with a range of values...	Click...
Quality (the higher the value, the greater the file size and better the image quality)	1-100	JPEG QUALITY
Smoothing (reduces the distortion of edges when you shrink or expand an image)	5-30	JPEG SMOOTHING

The default attributes are:

- JPEG QUALITY—75
- JPEG SMOOTHING—15

Changing the Default JPEG Attributes

To change the default JPEG attributes:

1. Click JPEG QUALITY in the Attributes sub-menu.
2. Enter a new value.
3. Click JPEG SMOOTHING.
4. Enter a new value.

Creating View Snapshots in JPEG

To create view snapshots in JPEG:

1. Choose Utils>Snapshot.
2. Click JPEG in the CREATE sub-menu.
3. Type the view no. (i.e.,1,2,3 or 4) in the command prompt.

4. Press Enter.
5. Type the image file name that you wish to save the view to.
6. Type “y” if you wish to save it or “n” to cancel.
7. Press Enter.

Basic Procedures

Plotting PostScript Image Files

About Plotting a Graphics Image

You can plot grayscale or color graphics images in MENTAT using either of the following options:

- send a PostScript file (color or grayscale) representing the current graphics image to a specific printer. To use this option, you must configure the printer by editing the appropriate file in the bin directory.
- write a PostScript output (color or grayscale) to a file.

Printing PostScript Image Files

After you write a PostScript output to a file, you can print the image in one or more of the following ways:

- send the PostScript file to a printer that supports PostScript.
- import the PostScript file as an object into an application that reads PostScript files as input (e.g. FrameMaker). Depending on the application, you can alter the object attributes (e.g., size, rotation, aspect ratio, etc.) before you print the image.

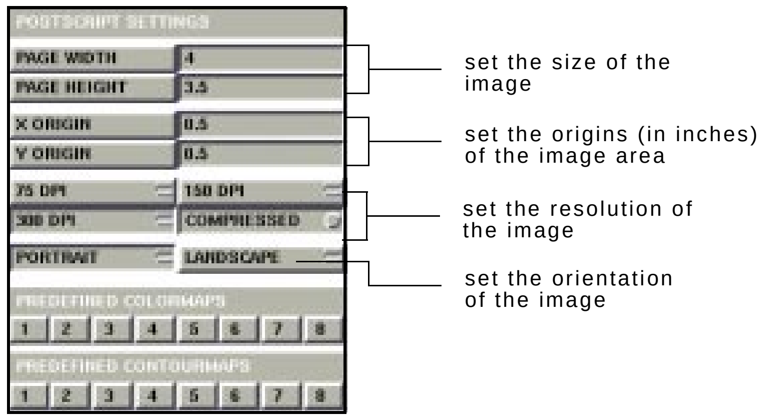
The image files shown here were written as PostScript output files in MENTAT and read into FrameMaker, a document publishing software.

Basic Procedures

Setting PostScript Image File Attributes

To set the image attributes of a PostScript file before you write it to a file or a printer:

1. Click UTILS in the static menu area.
2. Click SETTINGS and enter the image attributes.

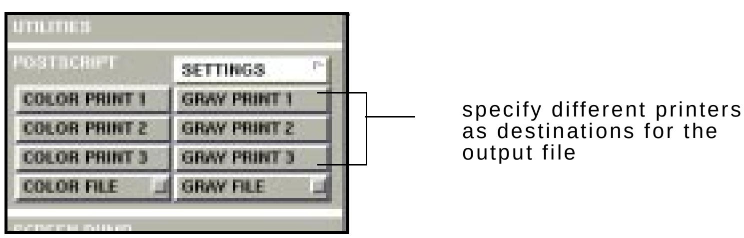


Utils>(PostScript)Settings

Sending a PostScript Output Directly to a Printer

To send a PostScript image file directly to the printer (you must configure your printer before you send the file):

1. Click UTILITIES in the static menu area.



Utilities

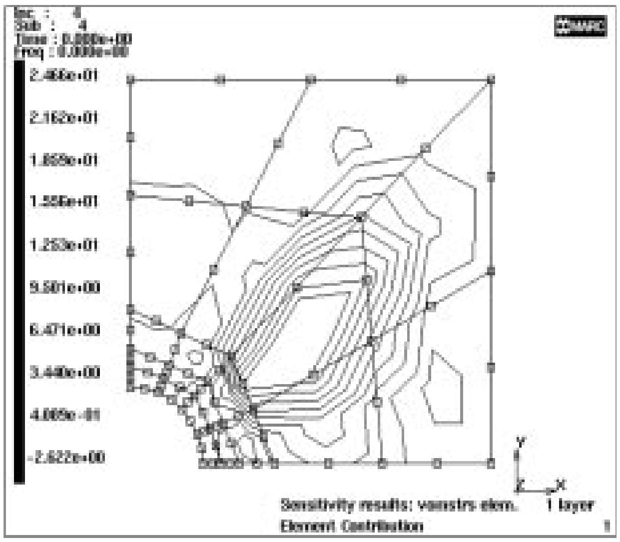
2. Choose a printer destination (e.g., Color Print 1) for the output file.

About the Resolution of PostScript Images

You can use the Settings menu to specify a resolution, in dots per inch (dpi), for the PostScript image. Higher resolutions result in greater size of the PostScript file.

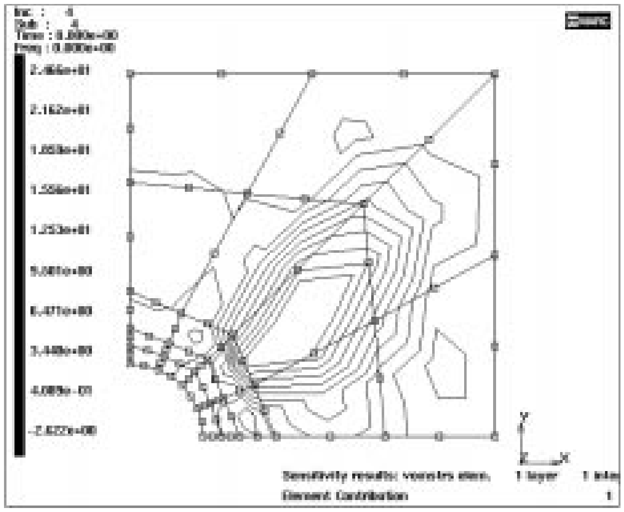
Here are examples of PostScript images with three different dpi settings in **MENTAT**:

- 75 dpi (lines are jagged; smallest size of file)



Grayscale File of Image Plotted at 75 dpi

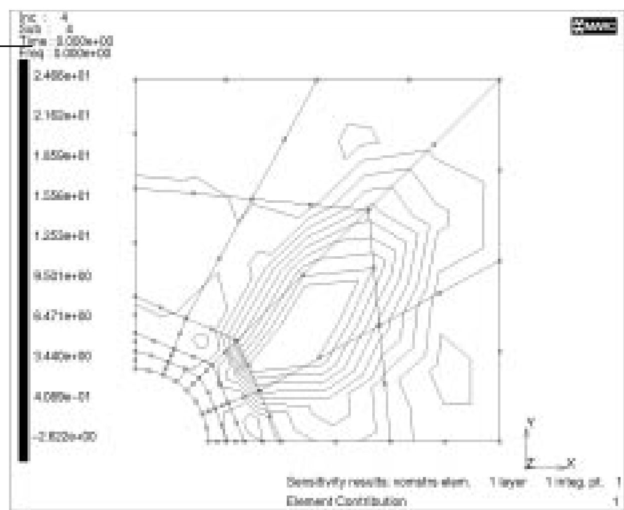
- 150 dpi



Gray File of Image Plotted at 150 dpi

- 300 dpi (lines are smooth; smallest size of text matter; largest size of file)

Increased font size (20 points) to improve readability of text matter



Gray File of Image Plotted at 300 dpi

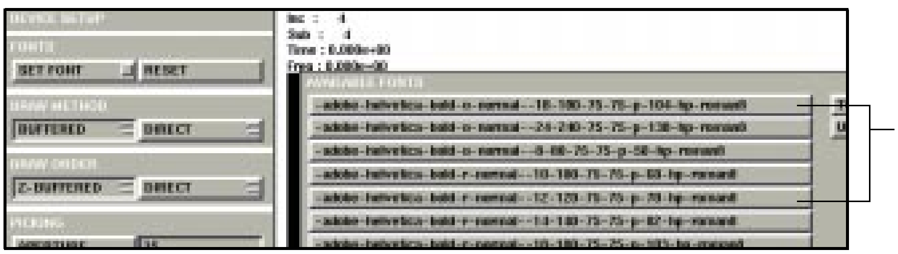
For quality images, choose a resolution that is higher than 75 dpi.

Increasing the Font Size of Text Matter

When you choose a higher resolution (e.g., 300 dpi) for the image, the text matter in the image area appears relatively smaller. You can offset this by increasing the font size for the text matter in the image area before you write the PostScript output file.

To increase the font size of the text matter in the image area:

1. Choose Device>Set Font.



list of available fonts in your local environment or platform

Device>Set Font

2. From the list of available fonts, choose a larger font size (e.g., 32).

The list of available fonts and font sizes vary depending on your local environment/platform.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
instructions on how to configure your destination printer(s) for your PostScript files	*Mentat Interfaces*	Mentat Installation Guide(s)

Arrow Settings

About Arrow Settings

Use the Arrow Settings feature to indicate boundary conditions on your finite element model. You can specify the following attributes for the arrows:

- length—for preprocessing arrows
- mode—for all arrows

To view the Arrow Settings options, choose Boundary Conditions>Arrow Settings.

Boundary Conditions>Arrow Settings

Specifying the Length of Preprocessing Arrows Manually

To specify the length of preprocessing arrows in the user coordinate system manually:

1. Choose Boundary Conditions>Arrow Settings.
2. Choose **MANUAL**.
3. Click **LENGTH** and specify a value.

Scaling the Length of Arrows Automatically

You can specify the length of arrows to be a percentage of the graphics screen. The arrows are then automatically scaled with respect to the graphics screen. Use the **FACTOR** button to set the percentage of the graphics screen.

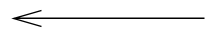
To scale the lengths of arrows automatically:

1. Choose Boundary Conditions>Arrow Settings.
2. Click **AUTOMATIC**.
3. Click **FACTOR** and enter a fraction (e.g., 0.2).

About Arrow Modes

You can specify how the arrows are drawn by choosing one of the following modes:

- wireframe



wireframe arrow

- solid

solid arrows

Use the **#FACETS** button to specify the number of facets for an arrow.

Basic Procedures

Additional Information

Use the following table to find more information:

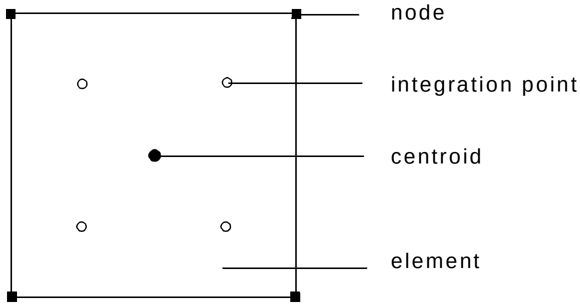
For...	refer to Chapter(s)...	in...
description of buttons in the Arrow Settings menu	*4:Boundary Conditions– Arrow Settings*	Mentat 3.1 Command Reference

Basic Procedures

Element Extrapolation

About Element Extrapolation for Display Purposes

Use the Element Extrapolation feature to control extrapolation of the integration point values to the nodes of an element for display.



Components of Element Extrapolation

There are three element extrapolation methods in MENTAT:

- Linear—the average of the integration point values is calculated and placed at the centroid. Then MARC performs a linear extrapolation of the values from the centroid through the integration point to the nodes.
- Translate—the values at the integration points are simply copied to the nearest nodes. If there are fewer integration points than nodes, MARC averages the values of neighboring integration points. When combined with the Isolate feature (Results>More>Isolate) to isolate just one element, this enables you to see the exact integration point values produced by MARC in history plots.
- Average—MARC computes the average of all the values at the integration points and assigns an equal value to all the nodes.

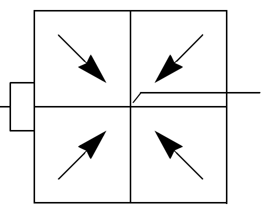
Selecting an Element Extrapolation Option

To select an element extrapolation method in MENTAT, choose Results>Scalar Plot Settings>Extrapolation.

Results>(Scalar Plot) Settings>Extrapolation

About Nodal Averaging

Use the Nodal Averaging feature to control the inter-element averaging of the nodal data after extrapolation. To ensure that the contour lines are continuous, choose **ON**.



Nodal Averaging Enabled

When you choose (nodal averaging) **OFF**, each element is independently contoured and the contour lines are discontinuous.

When you notice large differences between the averaged and the non-averaged results, the analysis is probably not very accurate.

4• General Technology

• General Technology

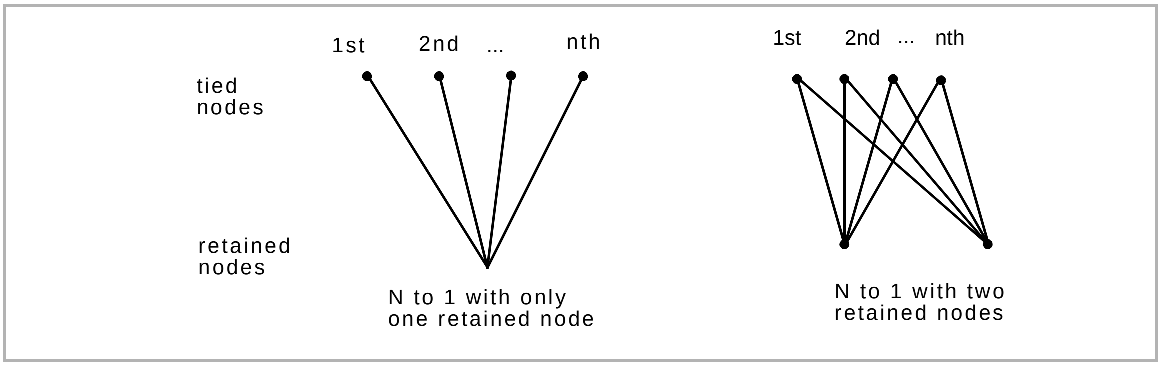
Using N to 1 and N to N Options in Links

About N to 1 and N to N Options

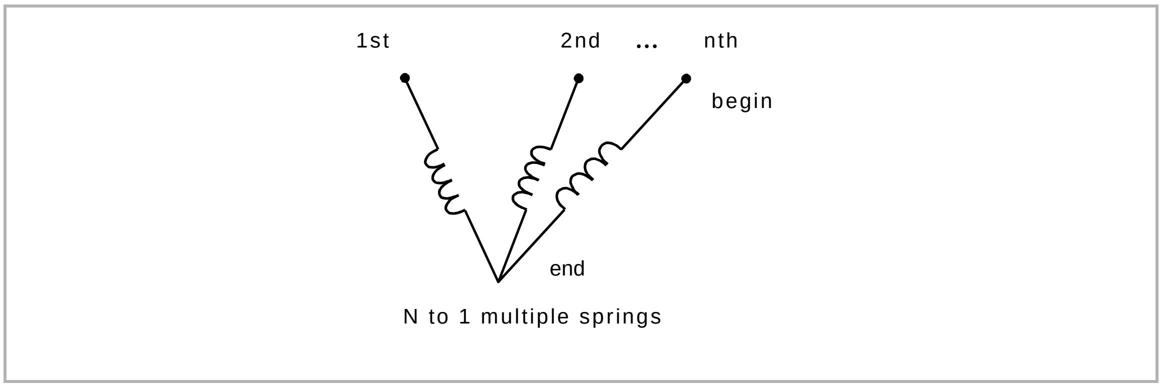
The N to 1 and N to N options are useful when you are applying constraint equations (ties or servo links) or springs that are similar in nature. A typical application is when you are modeling generalized plane strain conditions and you want multiple nodes to move as one node.

Here are the situations where you can use these options:

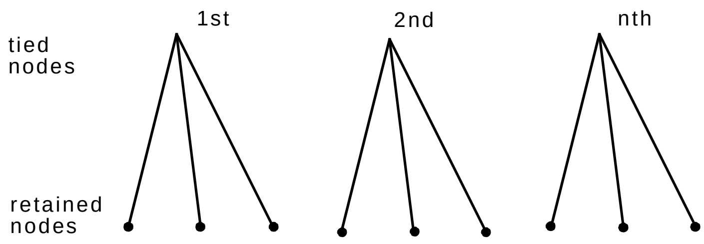
- N to 1—multiple ties between N tied nodes and common retained node(s).



- N to 1—multiple springs between N begin nodes and one common end node.

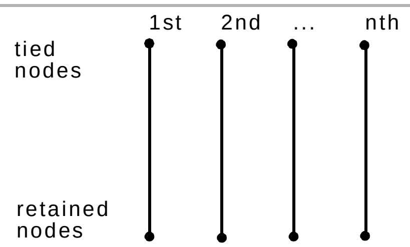


- N to N multiple ties (servo links)—N tied nodes with αN retained nodes, where α is the number of retained nodes per constraint



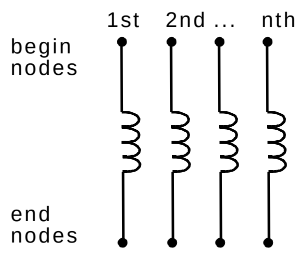
N-N servo links: three retained nodes per constraint

- N to N nodal ties—multiple links between N tied nodes and N retained nodes with no common nodes.



N to N nodal ties

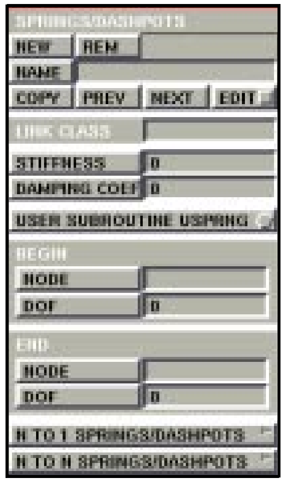
- N to N springs—multiple springs, each with the same stiffness/dashpot values, between two lists of nodes.



N to N springs/dashpots

Locating the N to 1 and N to N features

The N to 1 and N to N features are located in the Nodal Ties, Servo Links and Springs/Dashpots menus.



Location of the N to 1 and N to N buttons in the Links menus

Links>(Springs/Dashpots)

About Node Lists and Node Paths

You specify node lists and node paths to establish links between nodes (*See "Specifying Node Lists and Node Paths" on page 86*). Use the following table to determine which entity to specify with a feature:

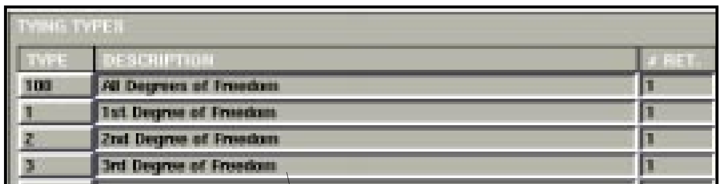
When using the feature,...	specify...
N to 1	node list
N to N	node path(s)

Using the N to 1 Feature for Nodal Ties

Use the N to 1 feature for Nodal Ties to make multiple similar ties for a list of N different tied nodes that have one common retained node.

To use the N to 1 feature for nodal ties:

1. Choose Links>Nodal Ties.
2. Click **N TO 1 NODAL TIES** in Nodal Ties.
3. Click **TYPE** and choose a tying type in Tying Types.



Scroll this list to choose a tying type

Links>Nodal Ties>Type

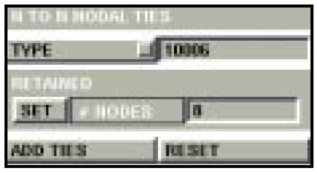
4. For user-defined ties, click **SET** and enter the number of retained nodes.
5. Specify the retained node(s).
6. Click **ADD TIES**.
7. Enter the node list for the tied nodes.

Using the N to N Feature in Nodal Ties

Before you create the multiple ties using the N to N feature, you must identify the retained nodes. The number of retained nodes varies with the your choice of a tying type.

To use the N to N feature for nodal ties:

1. Choose Links>Nodal Ties.
2. Click **N TO N NODAL TIES** in Nodal Ties.



Links>Nodal Ties

3. Click **TYPE** and choose a tying type in Tying Types.
4. For user-defined ties, click **SET** and enter the number of retained nodes.
5. Click **ADD TIES**.
6. Enter the node path(s) for the tied nodes.
7. Enter the node path(s) for the retained nodes.

The number of times that the system prompts you for the retained node path depends on the number of retained nodes in the tying type that you selected.

Using the N to 1 Feature in Servo Links

The servo link constraint is of the general form:

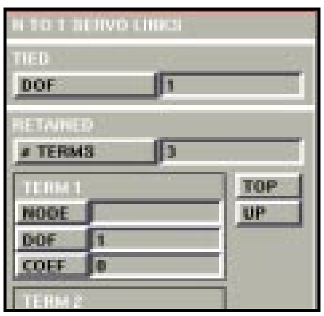
$$U^{T}_{DOF} = C_1 U^{R_1}_{DOF} + C_2 U^{R_2}_{DOF}$$

where U^{T}_{DOF} is the displacement of the tied node, $U^{R_1}_{DOF}$ is the displacement of the R_1 retained node, and C_1 is the corresponding coefficient.

Use the N to 1 feature in servo links to create a servo link having multiple tied nodes and one common set of retained nodes.

To use the N to 1 feature in servo links:

1. Choose Links>Servo Links.
2. Click **N TO 1 SERVO LINKS** in Servo Links.



Links>Servo Links>N to 1 Servo Links

3. Specify the degree of freedom for the tied node.
4. Enter the number of terms in the servo link equation.
5. Define the following for each term:

- retained node
- degree of freedom
- coefficient

6. Click **ADD TIES**.
7. Enter the node list for the tied nodes.

Using the N to N Feature in Servo Links

To use the N to N feature in servo links:

1. Choose Links>Servo Links.
2. Click N TO N SERVO LINKS in Servo Links.
3. Specify the degree of freedom for the tied node.
4. Enter the number of terms in the servo link equation.
5. Define the following for each term:

- degree of freedom
- coefficient

6. Click ADD SERVOS.
7. Enter the node path for the tied nodes.
8. Enter the node path(s) for the tied nodes followed by the retained nodes.

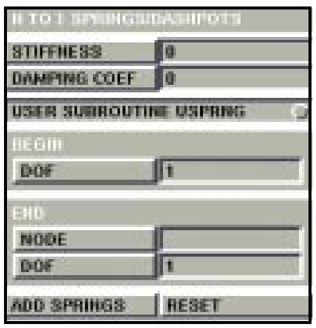
> **The number of times that the system prompts you for the retained node path depends on the number of terms that you specify.**

Using the N to 1 Feature for Springs/ Dashpots

Use the N to 1 feature for springs/dashpots in situations where you have multiple similar springs that have different begin nodes and a common end node.

To use the N to 1 feature for springs/dashpots:

1. Choose Links>Springs/Dashpots.
2. Click **N TO 1 SPRINGS/DASHPOTS** in Springs/ Dashpots.



Links>Springs/Dashpots>N to 1 Springs/Dashpots

3. Flag the subroutine, USPRNG, or enter the following parameters:

- stiffness
- damping coefficient

4. Enter the degree of freedom for the end node.
5. Specify the end node.
6. Enter a degree of freedom for the begin nodes.
7. Click **ADD SPRINGS**.
8. Enter the node list for the begin nodes.

Using the N to N Feature for Springs/ Dashpots

When you use the N to N feature for springs/dashpots, you specify node paths for begin and end nodes instead of a list of nodes. This ensures that there is a one-to-one correspondence between the begin nodes and end nodes.

To use the N to N Feature in Springs/Dashpots:

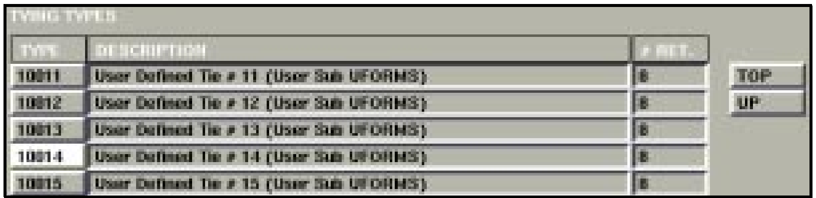
1. Choose Links>Springs/Dashpots.
2. Click **N TO N SPRINGS/DASHPOTS** in Springs/ Dashpots.

Links>Springs/Dashpots>N to N Springs/Dashpots

3. Flag the subroutine, USPRNG, or enter the following parameters:
- stiffness
- damping coefficient
4. Enter the degree of freedom for the begin node.
5. Enter the degree of freedom for the end node.
6. Click **ADD SPRINGS.**
7. Enter the node path for the begin nodes.
8. Enter the node path for the end nodes.

About User-Defined Ties

Tying Types, 10001-10020, are user-defined tying types where you can change the number of retained nodes. You cannot override the retained node settings for any typing type other than the user-defined tying types.



To scroll this list, use the TOP, UP, or DOWN buttons

Links>Nodal Ties>Type

About Resetting Parameters

The reset button resets the program parameters to the default value that existed before you clicked on the **ADD TIES** (or **ADD SERVOS**, or **ADD SPRINGS**) button.

The parameters that you reset for an N to N link type (e.g., servo links) are shared with the N to 1 menus for that link type only and not with any other link type. So when you press the reset button for a particular link type, you reset the N to 1 and N to N parameters for that link type only. The parameters in other link types are not affected.

Additional Information

Use the following table to find more information:

For...	refer to Chapter/s...	in...
description of buttons in the Links menus	*6: Links*	Mentat 3.1 Command Reference
description of tying types	*9:Boundary Conditions–Kinematic Constraints*	Vol. A
MARC input description	*3:Model Definition Options*	Vol. C
UFORMS	*2: User-Defined Loading, Boundary Conditions and State Variables Subroutines*	Vol. D
demonstration problems (*See "Lists of Demonstration Problems in MARC K7.3" on page 323*)		Vol. E

General Technology

Specifying Node Lists and Node Paths

Specifying a Node List

Use the following table to choose a method for specifying a node list:

Method	Procedure
A	1. Use <ML> to pick nodes (to undo your last pick operation, click <MM>). 2. Indicate end of list by choosing one of the following: • Click END LIST (#). • Click <MR> in the graphics area. • Type “#” in the dialogue area.
B	Type in the name of a node set in the dialogue area.

Method	Procedure
C	1. Type in the node ID’s separating each entry by a space or a comma. 2. Indicate end of list by choosing one of the following: • Click **END LIST (#)**. • Click <**MR**> in the graphics area. • Type “#” in the dialogue area.
D	1. Use the Box Pick or Polygon Pick method to pick the nodes. To undo the last Box Pick or Polygon Pick, click <**MM**>. 2. Indicate end of list by choosing one of the following: • Click **END LIST (#)**. • Click <**MR**> in the graphics area. • Type “#” in the dialogue area.
E	Use the List buttons (see Mentat User Guide: Chapter 2.5, *List Specification*).

General Technology

Specifying a Node Path

Use the following table to choose a method for specifying a node path:

Method	Procedure
A	1. Click <ML> to pick the nodes defining the node path. 2. Indicate end of list by choosing one of the following: • Click END LIST (#). • Click <MR>. • Type “#” in the dialogue area. The program automatically detects all the nodes in the node path.
B	1. In the dialogue area, type in the nodes defining the node path. 2. Indicate end of list by choosing one of the following: • Click END LIST (#). • Click <MR>. • Type “#” in the dialogue area.
C	Type in the name of a node path set in the dialogue area.

Storing a Node Path Set

You can also store a node path set and use it in commands where you need to enter node path information.

To store a node path set:

1. Click **SELECT** in the Static Menu area.
2. Click **STORE NODE PATH.**
3. Enter a name for the set.
4. Specify a node path (*See "Specifying a Node Path" on page 88*).

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
list specifications and mouse button functions	2, *Mechanics of Mentat*	Mentat User Guide
	1, *Introduction*	Mentat 3.1 Command Reference

Buckle Solutions Using Lanczos Method

About Lanczos Method for Buckle Solution

You can now use the Lanczos method for buckle eigenvalue extraction analysis in MARC. There are now two methods for specifying the buckle parameters:

- Lanczos
- Inverse Power Sweep

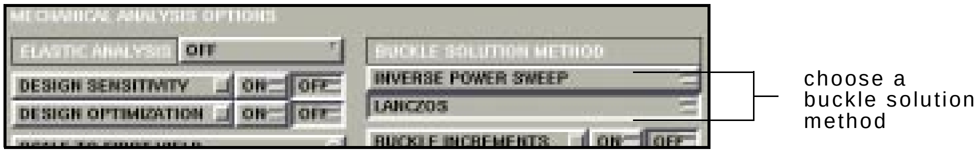
The Lanczos method is more computationally efficient than the Inverse Power Sweep method for relatively larger problems and small problems that require more than five buckle modes.

Specifying a Buckle Solution Method

To specify a buckle solution method for your job:

1. Choose Jobs>Mechanical>Analysis Options.
2. In the Buckle Solution Method menu area, choose either

   LANCZOS or INVERSE POWER SWEEP.



Jobs>Mechanical>Analysis Options

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
background and reference information	*Dynamic Eigenvalue Extraction*	Vol. A
input options	*Buckle Parameter Card/Buckle History Definition Card*	Vol. C

General Technology

Adaptive Load Stepping

About Adaptive Load Stepping

Use the Adaptive Load Stepping procedure to control the time step of nonlinear analyses. The criteria that you can control using the Adaptive Load Stepping procedure include the following:

- total strain
- plastic strain
- creep strain
- strain energy
- temperature
- displacement
- rotation

The procedure is best suited for problems with instabilities such as, snap through, necking, or cracking. You can use the procedure with all nonlinear structural mechanics procedures in MARC.

Loadcase Types Featuring Adaptive Load Stepping Parameters

Use the following table to locate the loadcase types that feature the Adaptive Load Stepping parameters in MENTAT:

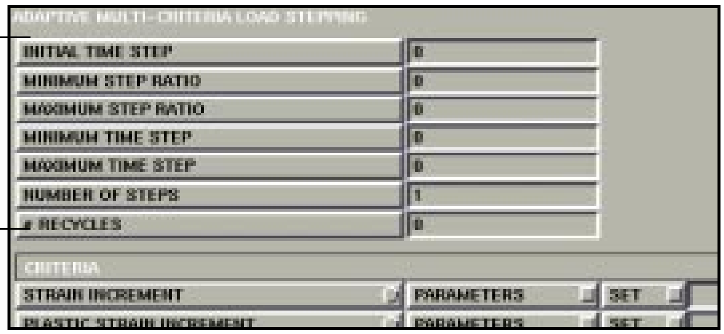
Analysis Class	Loadcase types that feature Adaptive Load Stepping parameters
Mechanical	• Static • Creep • Dynamic Transient • Rigid Plastic
Coupled	• Quasi-Static • Creep • Dynamic Transient • Rigid Plastic

Displaying the Adaptive Load Stepping Parameters

To display the Adaptive Load Stepping parameters:

1. Choose LOADCASES in Main Menu.
2. Choose either MECHANICAL or COUPLED.
3. Choose a loadcase type (e.g., Creep).
4. Click PARAMETERS in MECH. LOAD (MULTI-CRITERIA).

use these fields to set the general parameters for adaptive load stepping



Loadcases>(Mechanical)>(Creep)>
Mechanical Load (Multi-Criteria) Parameters

When you click MECH. LOAD (MULTI-CRITERIA), it generates the AUTO STEP **option in MARC. This option is similar to the mechanical load equilibrium procedure, which in MARC WAS called** AUTO TIME **as a method to control time steps, but is more powerful.**

General Technology

About Minimum and Maximum Step Ratios

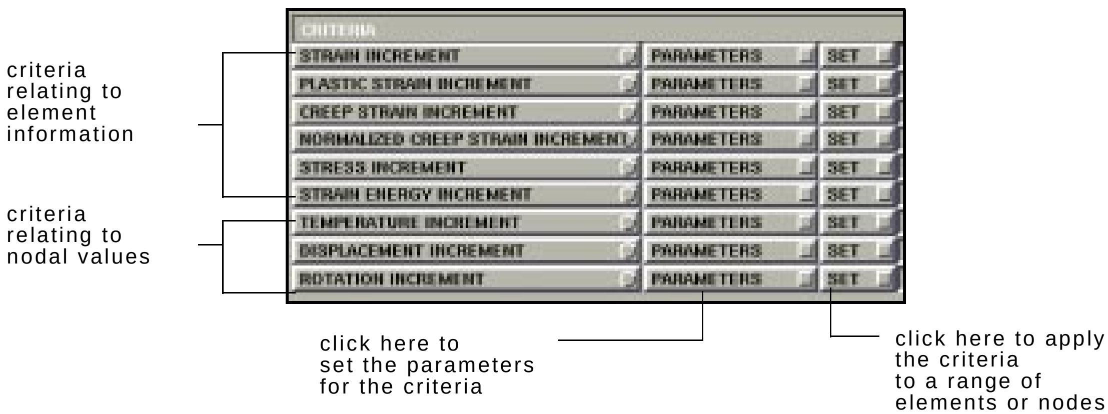
The minimum step ratio and the maximum step ratio are the ratios of the time step of the current increment to the time step of the next increment. Use these ratios to prevent the time steps from varying too sharply.

About Load Stepping Criteria

You can use a variety of criteria to control load stepping. You can either have one only one criterion or several criteria to govern the steps.

The Adaptive Load Stepping criteria are grouped into two sections:

- element information (e.g., strain increment)
- nodal values (e.g., temperature increment, displacement increment)

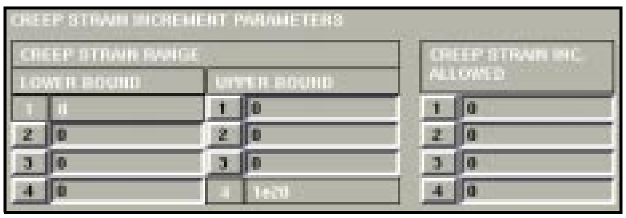


Loadcases>(Mechanical)>(Creep)>
Mechanical Load (Multi-Criteria) Parameters

Setting the Bounds Parameters for a Criterion

To set the bounds for a criterion:

1. Choose **LOADCASES** in Main Menu.
2. Choose either **MECHANICAL** or **COUPLED**.
3. Choose a loadcase type (e.g., Creep).
4. Click **PARAMETERS** in **MECH. LOAD (MULTI-CRITERIA)**.
5. Click parameters associated with an increment (e.g., Creep Strain Increment).



For every criterion that you select, you have to specify a bound on the allowable value of the criterion.

EXAMPLE

If you picked plastic strain increment, you might set a bound to specify that plastic strain increment within a step to be less than, or equal to, 1%. There are two possibilities:

1. If the program performs an increment of analysis and the plastic strain is greater than 1%, then that increment is recycled with a smaller step.

2. If the increment satisfies the criteria (i.e., strains are less than, or equal to, 1%), then in the subsequent increments, the time steps increase because the program can take larger time steps and still satisfy your criteria for accuracy.

About Specifying Multiple Ranges for a Criterion

To make adaptive load stepping even more effective, you can specify multiple ranges for a particular criterion.

EXAMPLE

For a plastic strain increment, you can specify the following:

- When the plastic strains are less than $^1/_{10}$ of 1%, the increments of plastic strain are less than $^1/_{100}$ of 1%.
- When the plastic strain is less than 2%, you might accept strain increments of 0.1%.
- If the plastic strains are very large (e.g., 10% or 100%), you might accept strain increments of 1% or 2%.

Applications of Multiple Ranges for Criteria

The criteria that you specify depends on where you are in the analysis or the type of material behavior in the analysis.

Multiple ranges are typically applicable to the following:

- manufacturing problems—where you want to force small strain increments at the initial yield point of the elastic-plastic material and you want larger time steps at the higher range of plastic strain.

- creep analysis—where the material has different behavior between primary and secondary creep stage or tertiary creep stage.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
definition of buttons in Adaptive Load Stepping	*11: Analysis: Loadcase—Adaptive Multi-criteria Load Stepping*	Mentat 3.1 Command Reference
description of the AUTO STEP input in MARC	*4: History Definition Options*	Vol. C
demonstration problems	e3x12b, e3x21f, e3x33, e4x7c, e5x8, e7x3b, e7x4b, e8x12d, e8x13b, e8x15d, e8x16b, e8x17b, e8x18d	Vol. E

Extended Precision Input

About the Extended Precision Input Feature

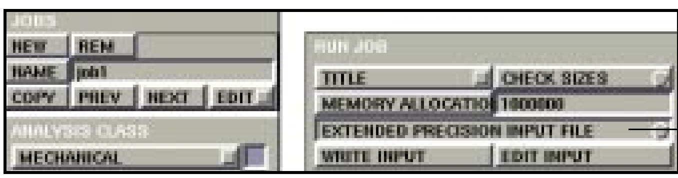
The Extended Precision Input feature ensures that the data which you enter into MENTAT, to be transferred into a MARC input file, has greater precision. This greater precision is useful in accurately modeling NURB surfaces for contact analysis (*p. 141*).

The Extended Precision Input feature also allows the solution of extremely large models. You must use the feature to create models with greater than 99999 elements or 99999 nodes.

Activating the Extended Precision Input Feature

To activate the Extended Precision Input Feature:

1. Choose Jobs>Run.



click to activate Extended Precision input

Jobs>Run

2. Click **EXTENDED PRECISION INPUT.**

Format Changes to Input Files

When you write a MARC input file using the Extended Precision Input feature, the following format changes occur:

- the EXTENDED parameter is included in the input file.
- all integers are written using the I10 format instead of the I5 format.
- all floating point numbers are written in the F20 or the F30 format instead of the F10 or the F15 format.
- all character strings are written using the A20 format instead of the A10 format.
- the input field widths (*see Vol. C: Chapter 2–Parameters*) are doubled when you include the EXTENDED parameter.

> **You can also mix the formats of an existing input file by using the NEW option (*see Vol. C: Chapter 4–History Definition Options*).**

General Technology

Constant Dilatation

When to Use the Constant Dilatation Parameter

Use the Constant Dilatation parameter with element types, 7, 10, 11, 19, or 20, to improve the behavior of these elements for modeling nearly incompressible materials. During elastic-plastic or creep analysis, use the Constant Dilatation parameter for these elements.

However, when you use the multiplicative decomposition, elastic-plastic formulation or the large-strain elasticity formulation in the Updated Lagrange framework, you do not require the Constant Dilatation parameter.

Analysis Classes Featuring the Constant Dilatation Parameter

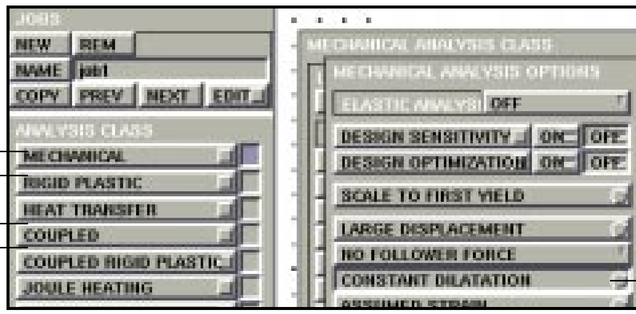
You can activate the Constant Dilatation parameter for the following analysis classes in MENTAT:

- Mechanical
- Rigid Plastic
- Coupled
- Coupled Rigid Plastic

Activating the Constant Dilatation Parameter

To activate the Constant Dilatation parameter for any of the above analysis classes:

1. Click Jobs.
2. Choose an analysis class (see list of relevant analysis classes above).
3. Click **ANALYSIS OPTIONS**.



Jobs>(Analysis Class)>Analysis Options

4. In the Analysis Options window, click **CONSTANT DILATATION** to enable it.

> **You can also activate the Constant Dilatation option by adding the** CONSTANT **parameter to the input file.**

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
description of the CONSTANT parameter,	*2: Parameters*	Vol.C

Assumed Strain Formulation

About the Assumed Strain Formulation

Use the Assumed Strain formulation in conjunction with elements 3, 7, and 11 to modify the conventional shape functions of the elements to improve the bending behavior. For linear elastic problems, the Assumed Strain formulation makes it possible to capture pure bending using a single element through the thickness.

Using the Assumed Strain formulation increases computational costs.

Analysis Classes That Feature Assumed Strain

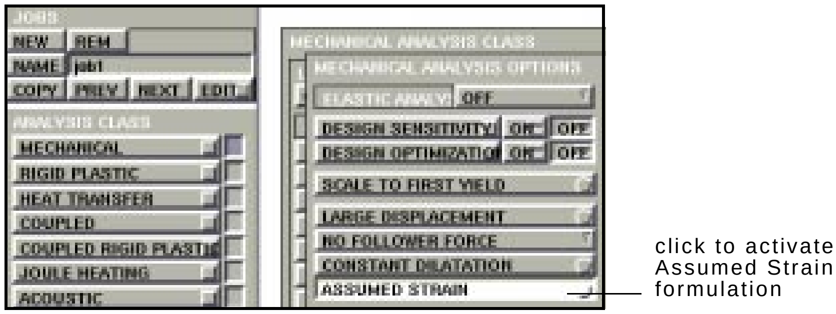
You can use the Assumed Strain formulation for the following analysis classes:

- Mechanical
- Rigid
- Plastic
- Coupled
- Coupled Rigid Plastic

Activating the Assumed Strain Formulation

To activate the Assumed Strain formulation:

1. Jobs>Analysis Class (see analysis classes above)>Analysis Options.



Jobs>(Analysis Class)>Analysis Options

2. Click **ASSUMED STRAIN** to activate formulation.

> **When you use the Assumed Strain option in Analysis Options, it activates the** ASSUMED **parameter in the input file. You can also add the** ASSUMED **parameter directly to the input file.**

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
description of the ASSUMED parameter	*2: Parameters*	Vol.C

General Technology

Numerical Preferences

About the Numerical Preferences Option

Use the Numerical Preferences option to change the default parameters that control the solution of a numerical analysis. You can use this option for the following analysis classes:

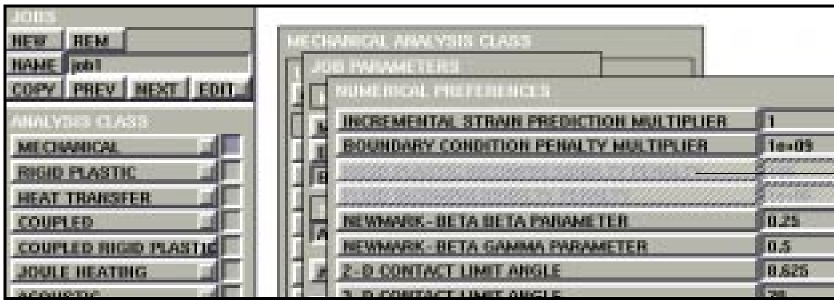
- Mechanical
- Rigid Plastic
- Coupled
- Coupled Rigid Plastic
- Fluid
- Fluid-Thermal
- Fluid-Solid
- Fluid-Thermal-Solid

The Numerical Preferences option activates the PARAMETERS **model definition or history definition option.**

Activating the Numerical Preferences Option

To activate the Numerical Preferences option:

1. Click **JOBS**.
2. Choose an analysis class (see valid analysis classes above).
3. Choose Job Parameters>Numerical Preferences.



options are available or grayed out based on the analysis class

Jobs>(Analysis Class)>Analysis Options>
Job Parameters>Numerical Preferences

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
description of the PARAMETER model definition option	*3: Model Definition Options*	Vol.C

General Technology

User-Defined Post Variables

About User Subroutine to Write to the Post File

There are three user subroutines to define the quantities to write to the post file during an analysis:

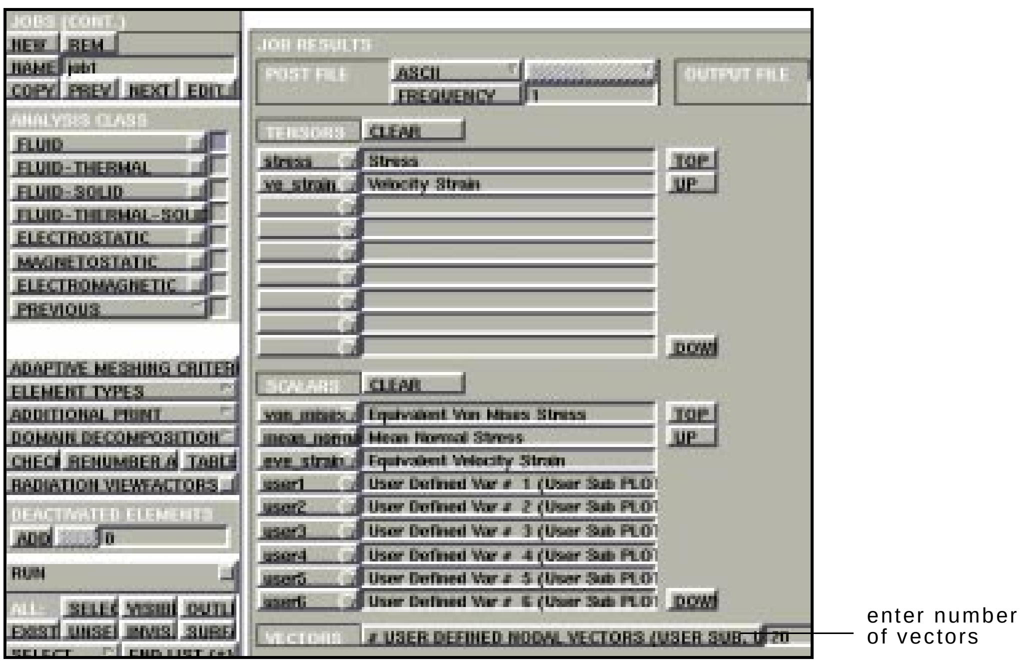
- ELEVAR—output of element quantities
- ELEVEC—output of element quantities in harmonic analysis
- UPOSTV—(*new user subroutine*) user-selected post-processing of nodal variables

The new user subroutine, UPOSTV, enables you to define nodal vectors.

Activating the User Subroutine UPOSTV

To activate the user subroutine UPOSTV:

1. Click Jobs>(Analysis Class)>Job Results.



Jobs>(Analysis Class)>Job Results

2. Click # **USER DEFINED NODAL VECTORS** and enter the number of vectors.

> **As an alternative, you can modify the POST model definition option (*See Vol. C: Chap.3–Program Control*). It is also necessary to write the user subroutine and include it with any other user subroutines.**

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
description of the user subroutine, UPOSTV	*7: Output Quantities*	Vol.D

5• Mesh Generation

Repair Geometry

Curve Divisions

Using the Advancing Front Mesher

Using the Delaunay Mesher

Expanding Elements Along a Curve

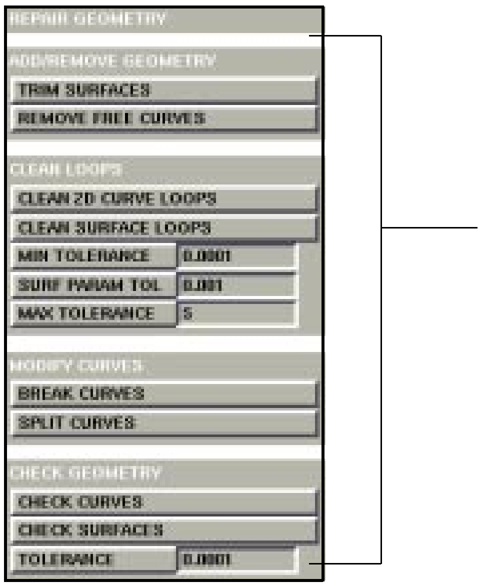
Creating a Node Point at Midpoint

• Mesh Generation

Repair Geometry

About Repair Geometry Operations

Use the Repair Geometry menus to clean up the geometric data for curves and surfaces when you import a CAD geometry.



use the Repair Geometry commands for either curves or surfaces

Mesh Generation>Automesh>Repair Geometry

The Repair Geometry operations change your model. Save the current model before you use the Repair Geometry commands.

Adding/Removing Geometry

Trimming Surfaces

Very often, when you import geometries into **MENTAT**, some or all of the trimming curves for some surfaces are missing. Use the Trim Surface command to place four trimming curves on the edges of surfaces.

To trim surfaces:

1. Choose Mesh Generation>Automesh>Repair Geometry.
2. Click **TRIM SURFACES**.
3. Enter a surface list.

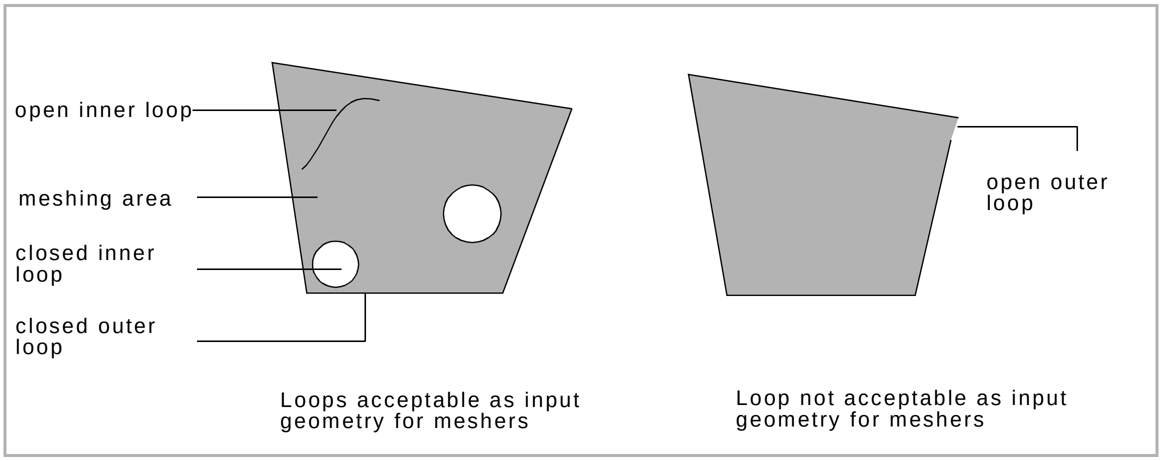
Removing Free Curves

To remove curves not attached to any surface, choose Mesh Generation>Automesh>Repair Geometry> Remove Free Curves.

Mesh Generation>Repair Geometry>Remove Free Curves

Cleaning Loops

Use the commands in Clean Loops to set tolerance values and clean loops in 2-D and 3-D geometries.



Cleaning 2-D Curve Loops

Use the Clean 2-D Curve Loops command to remove curves and close gaps in loops when the associated length is shorter than the minimum tolerance that you prescribe.

To clean 2-D curve loops:

1. Choose Mesh Generation>Automesh>Repair Geometry.
2. Click **MIN. TOLERANCE** and enter a value.
3. Click **CLEAN 2-D CURVE LOOPS**.
4. Enter a curve list.

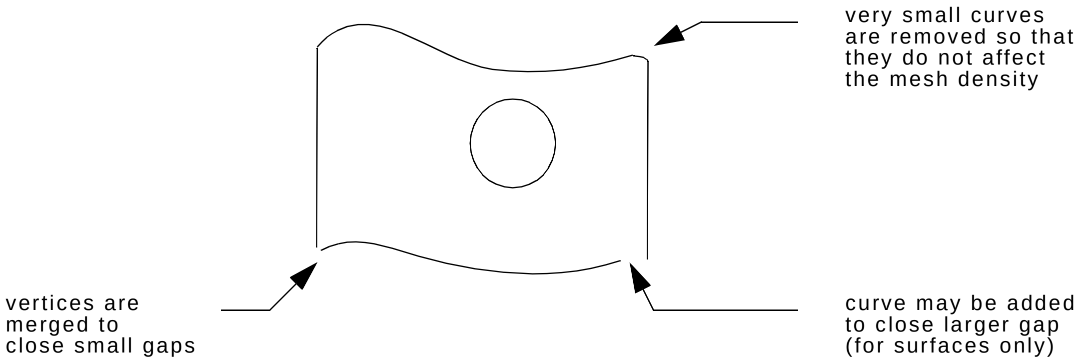
Cleaning Surface Loops

Use the Clean Surface Loops command to:

- remove trimming curves that are shorter than the minimum tolerance and the parametric tolerance (in surface parametric space) that you define.
- add trimming curves to connect points when the distance between them is shorter than the maximum tolerance that you define.

To clean surface loops:

1. Choose one or more of the following options:
   - Click **MIN TOLERANCE** and specify a value.
   - Click **MAX TOLERANCE** and specify a value.
   - Click **SURF PARAM TOL** and specify a value.
2. Click **CLEAN SURFACE LOOPS**.
3. Enter a surface list.

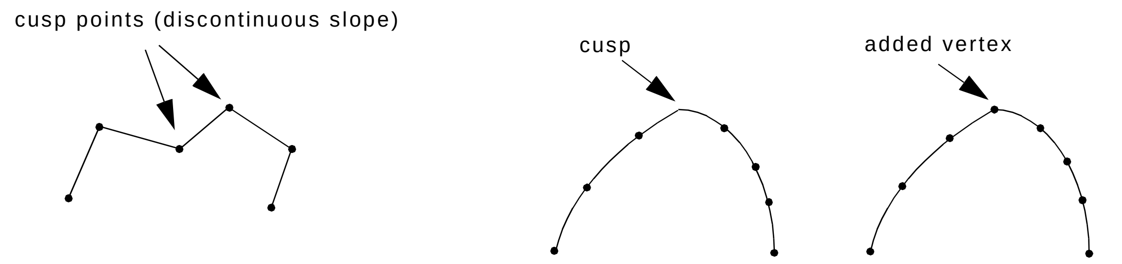


Possible operations supported by the Clean Loops features

Modifying Curves

Splitting Curves

Use the Split Curves command to split curves into multiple curves at cusp points and to ensure that vertices are located only at the end points.



Single curve with cusp points split into multiple curves. Example shown here using a polyline curve type

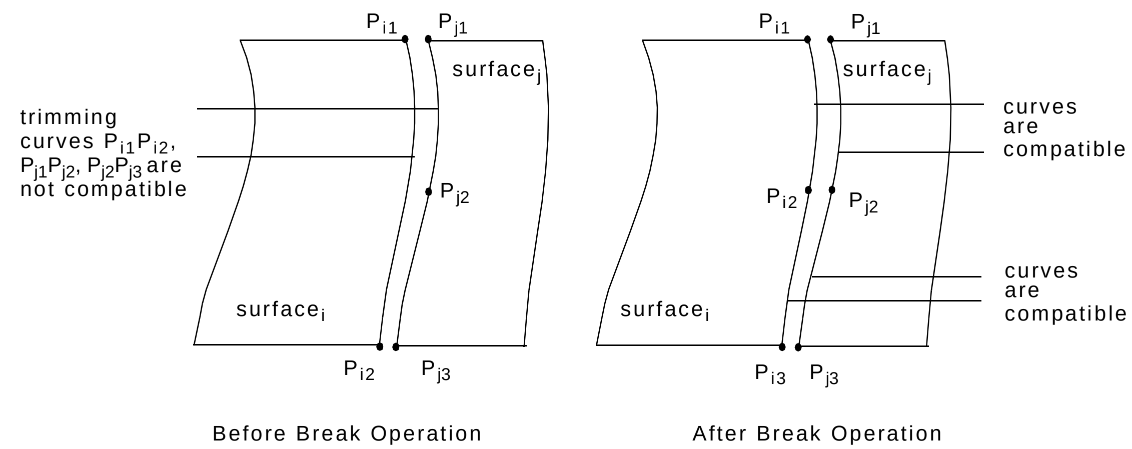
To split curves:

1. Choose Mesh Generation>Automesh>Repair Geometry.
2. Click **SPLIT CURVES**.
3. Enter a curve list.

Mesh Generation

Breaking Curves

There are cases where two trimming surfaces may share a common geometric curve but have different trimming curves. To make all trimming curves compatible, use the Break Curves command.



To break a curve:

1. Choose Mesh Generation>Automesh>Repair Geometry.
2. Click **BREAK CURVES**.
3. Enter a vertex tolerance.
4. Enter a curve list.

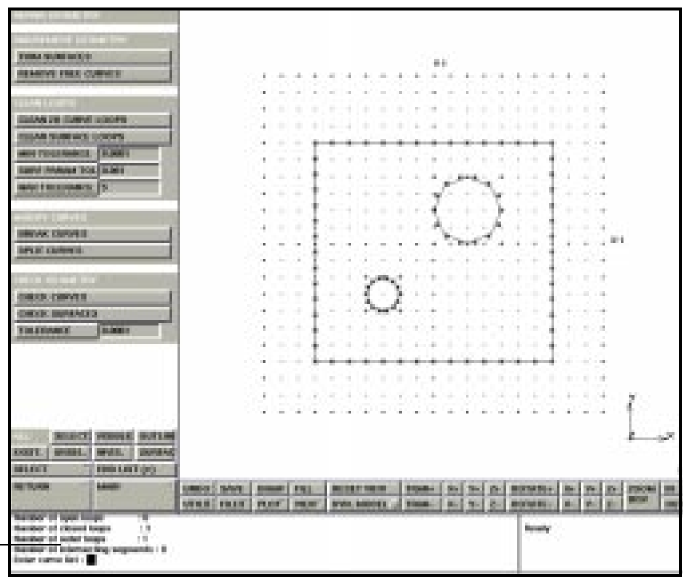
Checking Geometry

Checking Curve or Surface Geometry

Use the Check Geometry commands to check the topology information (*See Command Reference–Chap. 3: Preprocessing: Mesh Generation*) when you specify a list of curves or surfaces.

To check geometry of curves or surfaces:

1. Choose Mesh Generation>Automesh>Repair Geometry.
2. Choose **CHECK CURVES** or **CHECK SURFACES.**
3. Enter a curve or surface list.



summary information generated by the Check Curves option. Scroll down to view the full summary in the dialogue area.

When you choose the **CHECK CURVES** option, you get a summary in the dialogue area containing the following information:

- minimum and maximum curve lengths
- number of segments
- number of open and closed loops
- number of intersecting segments

Mesh Generation

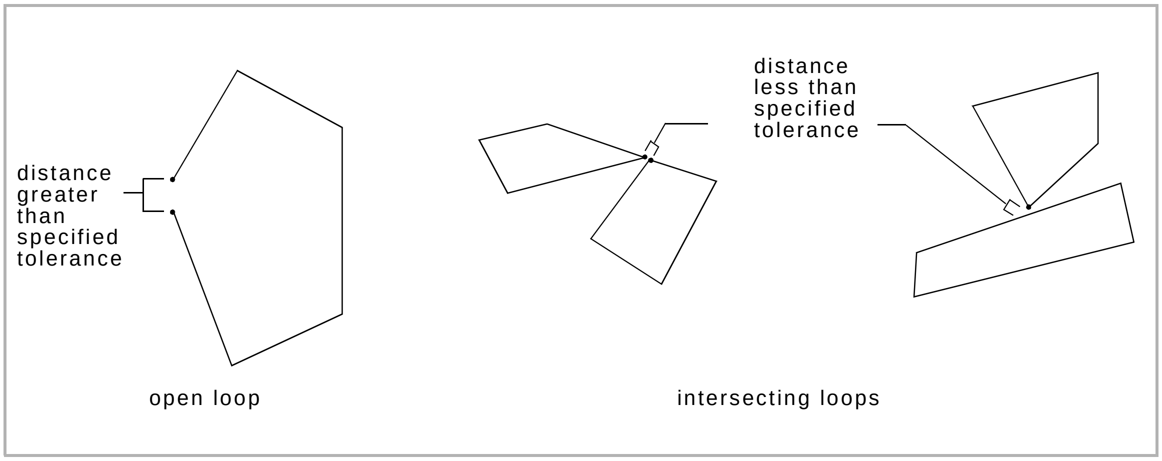
When you choose the **CHECK SURFACES** option, you get a summary in the dialogue area containing the following information:

- minimum and maximum trimming curve lengths
- number of segments
- number of open and closed loops
- number of outer and inner loops
- number of intersecting segments
- minimum length of shortest trimming curve
- maximum length of the longest trimming curve

About Tolerance Value for Curve Intersection

When you set a tolerance value for curve intersection, the program compares the specified tolerance and the distance between the end points in adjoining segments.

If the distance between the end points is greater than the specified tolerance, the program treats it as an open loop. If the distance from one end point to more than one geometric entity (i.e., end point, edge) is smaller than the specified tolerance, the program treats the geometry as having intersecting loops.



Checking for Intersecting Loops Using a Tolerance Value

To use a tolerance value to check for intersecting loops:

1. Click **TOLERANCE** in Check Geometry and enter a value.

   The default value is.0001.

default value is .0001

Mesh Generation>Automesh>Repair Geometry

2. Choose either **CHECK CURVES** or **CHECK SURFACES.**
3. Enter a curve or surface list.

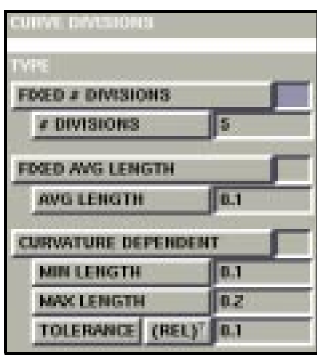
Mesh Generation

Curve Divisions

About Curve Divisions

You can specify three types of curve divisions in **MENTAT**:

- fixed number of divisions—the number of divisions along the curve is fixed by the number of divisions that you specify. All segments along the curve have equal length.
- fixed average length—each segment along the curve is set as close as possible to the value that you define as the average length.
- curvature dependent—the length of each segment varies with the curvature along the curve according to the parameters that you specify. All segment lengths are within the range prescribed in the **MIN LENGTH** and **MAX LENGTH** fields.



default setting is
FIXED # DIVISIONS

Mesh Generation>Automesh>Curve Divisions

You can choose only one type of curve division at a time.

Applying Fixed Divisions to a Curve

To apply a fixed division to curves:

1. Choose Mesh Generation>Automesh>Curve Divisions.
2. Click **FIXED # DIVISIONS**.
3. Click **# DIVISIONS** and enter a value.
4. Click **APPLY CURVE DIVISIONS**.
5. Enter a curve list.

Applying Fixed Average Length Divisions

To apply a fixed average length to a curve:

1. Choose Mesh Generation>Automesh>Curve Divisions.
2. Click **FIXED AVG. LENGTH**.
3. Click **AVG. LENGTH** and enter a value.
4. Click **APPLY CURVE DIVISIONS**.
5. Enter a curve list.

About Curvature-Dependent Curve Divisions

Use curvature-dependent curve divisions to produce curve divisions that are consistent with curve shape.

Here are some benefits of using curvature-dependent divisions:

- minimum number of divisions generated over straight portions of the curve
- more division edges generated over the high curvature portion of the curve

Mesh Generation

Parameters for Curvature-Dependent Curve Divisions

To apply curvature-dependent divisions to a curve, specify the following parameters:

- **MIN. LENGTH**: the minimum length of segments generated by the curve division operation.
- **MAX. LENGTH**: the maximum length of segments generated by the curve division operation.
- **TOLERANCE**: the maximum allowable deviation of straight division segments to the curve, given as an absolute distance value (**ABS**), or a value which is relative to the edge segment length (**REL**).

Applying Curvature-Dependent Divisions to a Curve

To apply curvature-dependent curve divisions to a curve:

1. Choose Mesh Generation>Automesh>Curve Divisions.
2. Click **CURVATURE DEPENDENT**.
3. Click **MIN LENGTH** and enter a value.
4. Click **MAX LENGTH** and enter a value.
5. Click **TOLERANCE** and enter a value.
6. Choose **REL** or **ABS** in the tolerance option (the default value is **REL**).
7. Click **APPLY CURVE DIVISIONS**.
8. Enter a curve list.

The tolerance options mentioned above work in the same manner as the tolerance options described in Adaptive Plotting (*See "Tolerance and Modes of Tolerances" on page 52*).

Applying Restrictions to Individual Curves and Detected Loops

Use the Restriction menus to force even or odd divisions on individual curves or detected loops (*See Mentat 3.1 Command Reference–Chapter 3, Preprocessing: Mesh Generation*).

1. Choose Mesh Generation>Automesh>Curve Divisions.
2. In the Apply Restriction To area, choose individual curves or detected loops.
3. Choose **FORCE EVEN DIV** or **FORCE ODD DIV**.
4. Click **APPLY CURVE DIVISIONS**.
5. Enter a curve list.

You can specify a fixed number of divisions, N, for curve divisions. If the number, N, conflicts with the restriction type, the resulting division has one more division than the value of N.

> **When you apply the even/odd restriction on detected loops within the given curves, the restriction is only enforced on entire closed loops and individual curves may not necessarily meet the criteria. This might cause problems when you try to match divisions for corresponding curves on surfaces.**

Matching Curves

Use the Match Curves command to ensure that the number of mesh divisions of coincident curves is equal. The curves are matched based on a vertex tolerance that you specify.

To match curves:

1. Choose Mesh Generation>Automesh>Curve Divisions.
2. Click **MATCH CURVE DIVISIONS.**
3. Enter a value for the vertex tolerance.
4. Enter a curve list.

The Match Curves command changes the divisions on certain curves where necessary and might disrupt the odd/even restrictions that you had applied when you created the curve divisions.

When to Break or Match Curves

Use the table to determine when to break (*See "Breaking Curves" on page 118*) or match curves:

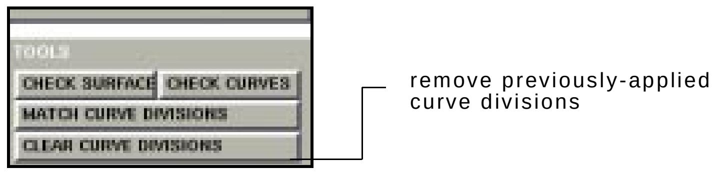
If...	then, use this operation:
neighboring trimming curves are not coincident over their entire lengths	Break
you applied curve divisions and there are coincident neighboring curves with non-conforming divisions	Match

Clearing Curve Divisions

Use the Clear Curve Divisions command to remove previously-applied curve divisions.

To clear curve divisions:

1. Choose **CLEAR CURVE DIVISIONS** in Curve Divisions.



Mesh Generation>Automesh>Curve Divisions

2. Enter a curve list.

If you do not specify any divisions for a list of curves, they are meshed with the following defaults:

- three divisions for closed curves
- one division for open curves

Input Geometry Considerations for 2-D Planar and Surface Meshing

You should apply curve divisions to all curves prior to meshing. Straight segments connecting the division points are used as boundary element edges.

Mentat 2-D meshers generate meshes in the domain bounded by loops of curves lying in a plane.

Surface meshers take trimmed surfaces (*See "Trimming Surfaces" on page 114*) as their input and generate meshes over the surface area bounded by trimming curves.

Mesh Generation

Guidelines for Using 2-D and Surface Meshers

For 2-D meshers, you should enter a list of coplanar curves. The mesh area lies within closed loops formed by the curves.

For surfaces meshers, you should enter a list of surfaces. The mesh area on each surface lies between closed loops formed by trimming curves attached to that surface.

Do not use 2-D planar meshers to generate meshes over planar surfaces in analyses that are not 2-D. Otherwise, the resulting mesh will not be attached to the surface and will not adhere to any boundary conditions for that surface.

Guidelines for Inner and Outer Loops

You can use the 2-D and surface meshers for models that contain:

- closed inner and outer loops
- open inner loops

When you use both inner and outer loops, the mesh area lies between them.

Checklist for the Advancing Front and Delaunay Meshers

Before using the 2-D or surface meshers on your model, make sure that:

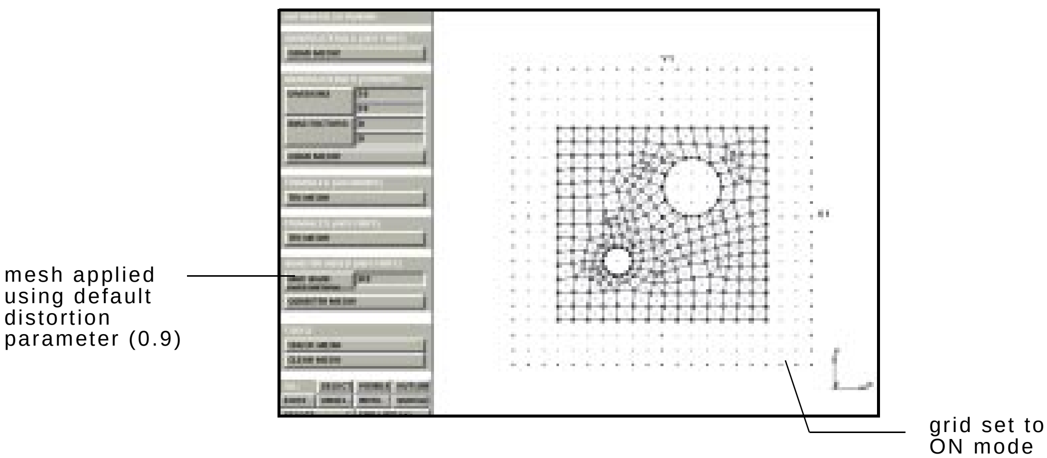
- there is at least one outer loop
- all open loops are contained within a closed loop
- all the curves have divisions applied to them
- no loops intersect

Using the Advancing Front Mesher

About the Advancing Front Meshers

Use the Advancing Front Meshers to automatically generate a mesh of quadrilateral or triangular elements in 2-D space or on surfaces. You can apply the following types of advancing front meshers on your model:

- Quad Mesh—meshes the specified list of curves or surfaces with quadrilateral elements.
- Tri Mesh—meshes the specified list of curves or surfaces with triangular elements.
- Quad/Tri Mesh—meshes the specified list of curves or surfaces with a combination of quadrilateral and triangular elements.



Advancing Front Quad/Tri Mesh applied to a 2-D planar model

Special Considerations for Quad Meshing

Before you apply the quad mesher, you should ensure that all closed loops contain an even number of divisions. If these conditions are not met, you may find triangular elements in the mesh or the mesher might fail.

About Distortion Parameters in Quad/Tri Meshing

When you use the Quad/Tri Mesher, you should specify the maximum allowable distortion of the quadrilateral elements in the mesh using the Max. Quad. Distortion parameter. A distortion parameter is a measure of how closely an element resembles its parent element which, for a quad element, is a rectangle. The value of a distortion parameter may range from 0.0 (perfect rectangle) to 1.0 (collapsed or obtuse).

When you mesh your model with the Quad/Tri mesher, the preferred elements are quadrilateral and no element is allowed to have a distortion parameter greater than the prescribed value of the Max. Quad. Distortion parameter.

If you set the value of the parameter to 1.0 (full quad distortion) and all the closed loops have an even number of boundary divisions, then the resulting mesh is an all-quad mesh. If you set the value to 0.0 (no quad distortion), the resulting mesh is an all-tri mesh.

Considerations for Specifying Distortion Parameters

Although the value of the Max. Quad. Distortion parameter ranges from 0.0 to 1.0, you may want to specify a value that is greater than 0.5. The higher the value of the distortion parameter, the more quads that make up the final mesh. The mesher always tries to place quad elements first.

If you enter a very low value of the distortion parameter (i.e., a value closer to 0.0), it slows down the mesher considerably while generating a predominantly tri-mesh.

You might discover that in most cases, the more practical values for the distortion parameter lie in the 0.9-1.0 range. In that range, the mesh is a predominantly-quad mesh.

Specifying Distortion Parameters

To specify a value for the distortion parameter:

1. Choose **2D PLANAR MESHING** or **SURFACE MESHING.**
2. Click **MAX. QUAD DISTORTION** and enter a value.

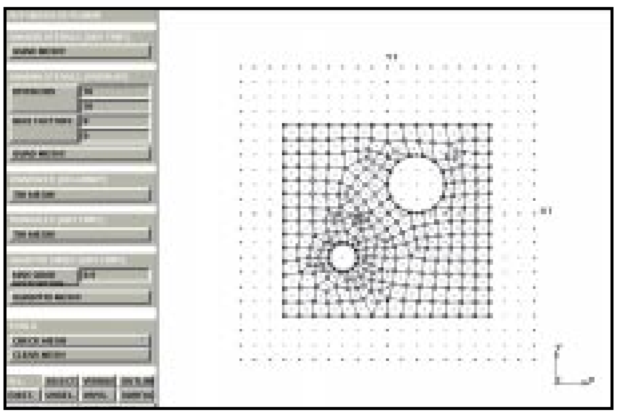
enter a value between 0.0 and 1.0

Mesh Generation>Automesh>2D Planar Meshing or
Mesh Generation>Automesh>Surface Meshing

Applying the Advancing Front Meshers

To apply the Advancing Front Quad or Quad/Tri Mesh for 2-D planar or surface meshing:

1. Choose either **REPAIR GEOMETRY** or **CURVE DIVISIONS** or both to prepare the model for meshing (*See "Checklist for the Advancing Front and Delaunay Meshers" on page 128*).
2. Choose either **2D PLANAR MESHING** or **SURFACE MESHING.**
3. Specify a distortion parameter or use the default value (*for quad/tri meshing*).
4. Choose a mesher (i.e., Quad Mesh).
5. Enter a list of curves or surfaces.

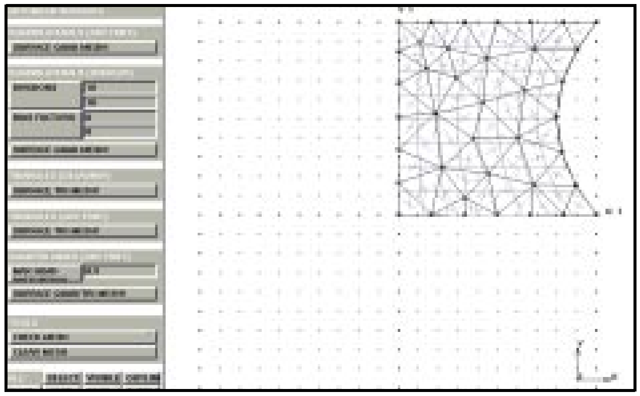


Advancing Front Quad mesh for 2-D planar model

Using the Delaunay Mesher

About the Delaunay Mesher

Use the Delaunay mesher to automatically generate triangular element meshes in 2-D space or on surfaces.



Delaunay surface tri mesh applied to a simple surface model

Features of the Delaunay mesher

A mesh generated by the Delaunay mesher contains the following attributes:

- shape—the triangles in the mesh tend to be equilateral.
- size—in critical areas, the density of mesh could be high; in non-critical areas, the density could be low which saves computing time.
- transition—the density of mesh changes gradually resulting in a smooth transition.

If the variation of stress is very high at a certain area, you can increase the density of mesh for that area. For areas that are flat and the variation of physical properties is not very high, you can generate a mesh that is less refined.

Mesh Generation

Applying the Delaunay Mesher

To apply the Delaunay mesher to your model:

1. Choose either **REPAIR GEOMETRY** or **CURVE DIVISIONS** or both to prepare the model for meshing (*See "Checklist for the Advancing Front and Delaunay Meshers" on page 128*).
2. Click on **2D PLANAR MESHING** or **SURFACE MESHING.**
3. Click on either **TRI MESH** (2-D) or **SURFACE TRI MESH** (3-D).
4. Enter a curve or surface list.

Additional Information

Use the following table to find more information:

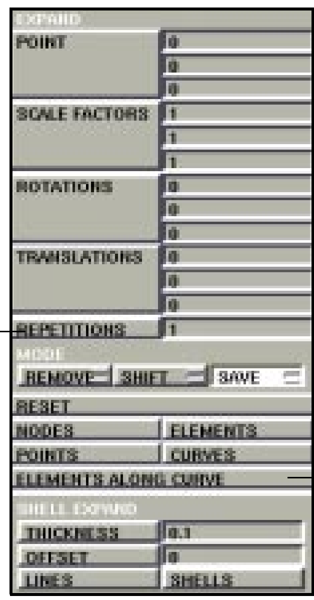
For...	refer to Chapter(s)...	in...
Definitions of individual buttons	*3–Preprocessing: Mesh Generation*	Mentat 3.1 Command Reference
a tutorial on how to apply the Advancing Front or Delaunay meshers	*16–Automatic Meshing*	Mentat User Guide

Expanding Elements Along a Curve

About Expanding Elements Along a Curve in Mentat 3.3

You can expand a one-dimensional mesh into a two-dimensional mesh by projecting the original mesh in the direction of the curve. You can also expand a two-dimensional mesh into a three-dimensional mesh by using the orientation of the curve.

To view the Expand menu, choose Mesh Generation>Expand.



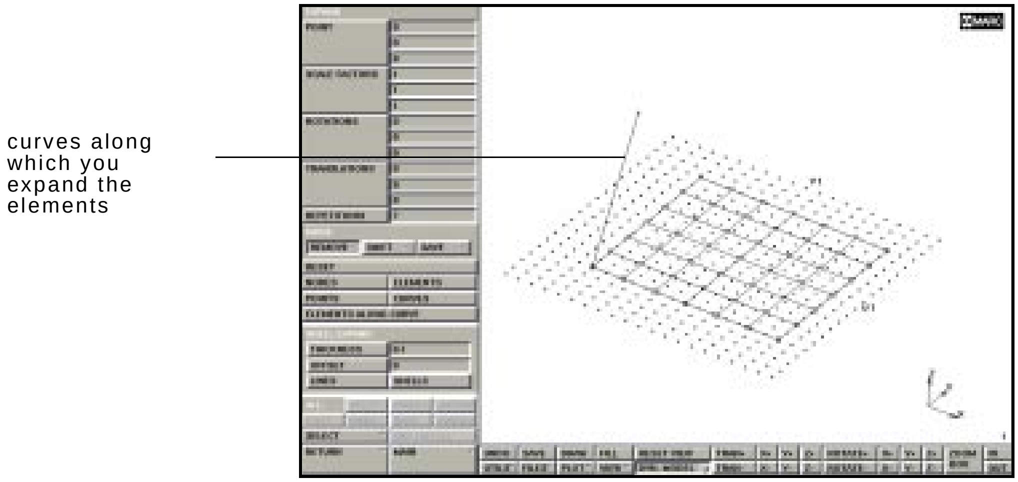
set the number of times that the elements expand

select a curve to expand elements along

Mesh Generation>Expand

Prerequisites for Expanding Elements Along a Curve

Before you use the Expand Elements Along a Curve feature, you should have already created a curve and a finite element mesh.



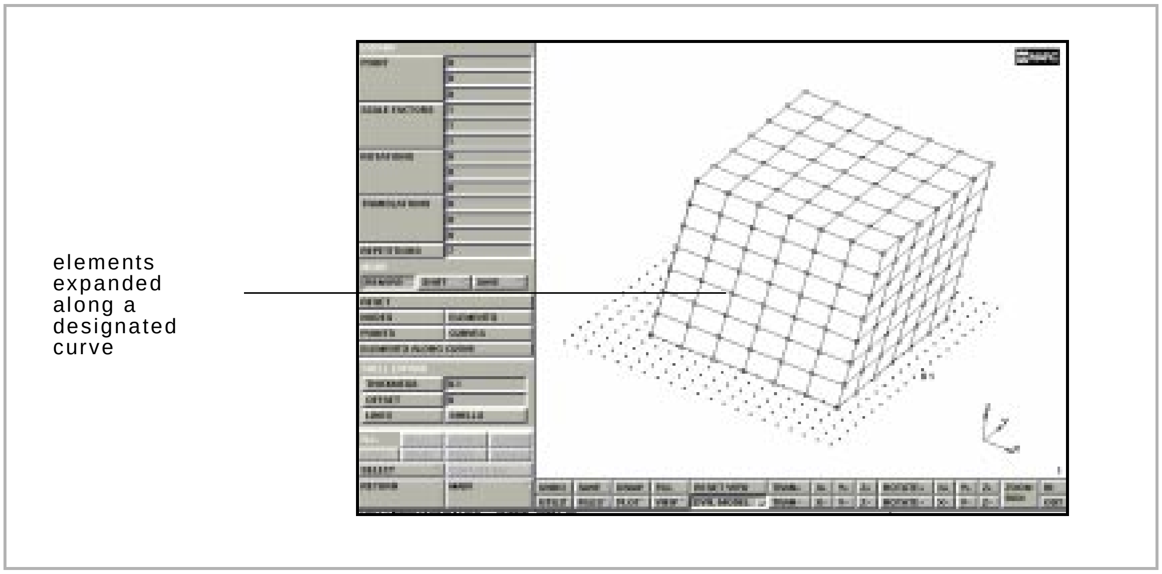
Use the REPETITIONS command to set the number of times that the elements are expanded. However, use this command with caution as it is easy to create a mesh which penetrates itself.

Mesh Generation

Creating a New Mesh Using Expand Elements Along a Curve

To create a new mesh using the Expand Elements Along a Curve feature:

1. Choose Mesh Generation>Expand.
2. Click **REPETITIONS** and enter the number of repetitions to create.
3. Click **ELEMENTS ALONG CURVE** and select the curve to use and the elements to expand.



If you picked the REMOVE mode, the original elements are deleted.

Creating a Node or Point at Midpoint

Creating a Node at Midpoint

To create a new node at midpoint between two locations:

1. Click **MESH GENERATION**.
2. Click **BETWEEN NODES**.

Mesh Generation

3. Enter the first and the second point coordinates to specify the two locations.

Creating a New Point

To create a new point between two locations:

1. Click **MESH GENERATION**.
2. Click **BETWEEN POINTS**.
3. Enter the first and the second point coordinates to specify the two locations.

6• Contact

Performing Contact Analyses

Specifying Contact Information in Loadcases

Specifying Contact Information in Jobs

- **Contact**

Contact

Performing Contact Analyses

Components of a Contact Analysis

The contact analysis process has the following components:

- define different contact bodies—bodies can be conventional deformable bodies made up of finite elements or rigid bodies made up of either curves for 2-D, or surfaces for 3-D analyses (*See "Choosing a Contact Body Type" on page 143*).

- determine the contact tolerances—establish how close the body needs to be with another body to conclude that there is contact between the two bodies (*See "About Contact Tolerances" on page 163*).

- define where the contact potentially occurs—by default, the system assumes that any point on the outside surface of a body can contact any other body. While this makes it easier to define contact, it might lead to higher computational costs (*See "About Contact Areas" on page 156*).

- prescribe different contact procedures in MARC—how the numerical algorithms are implemented in the analysis (*See "About Contact Control in Jobs" on page 161*).

- prescribe separation procedures—how a body separates from another body (*See "About Separation Procedures" on page 165*).

- specify the type of friction—which type of friction model to use; assign the coefficients of friction between the surfaces of the two different bodies (*See "About Friction Models" on page 166*).

- assign the parameters for coupled analysis—evaluate the thermal interactions between two adjacent bodies (*See "Coupled Analysis Considerations" on page 152*).

About Physical Bodies

When you are defining physical bodies, you can specify one or more of the following conditions:

- one of the bodies is deformable
- multiple bodies are deformable
- one deformable body is in self-contact
- multiple bodies are rigid

You should define at least one deformable body in the analysis.

About Contact Bodies

In MENTAT, there are four different types of contact bodies:

- Deformable
- Rigid—the body is either moving in space or is a special type of rigid body.
- Symmetry—special type of rigid body which is a symmetry line or a symmetry surface.
- Rigid with Heat Transfer—a body made up of finite elements and is not deformable; used only in coupled thermal stress analysis. This is a region comprising of

elements that do not distort but are used in heat transfer analysis so that you can model thermal behavior in a particular region

Choosing a Contact Body Type

To choose a contact body type:

1. Click **CONTACT** in Main Menu.
2. Click **CONTACT BODIES** in Contact.

Contact>Contact Bodies

3. Choose a contact body type.
4. Define which elements, curves, or surfaces are part of the contact body.

To add elements, curves, or surfaces refer to the following table:

If you choose, body type...	then use...
deformable or rigid with heat transfer	**ELEMENTS**
rigid or symmetry	• **CURVES** (2-D) • **SURFACES** (3-D)

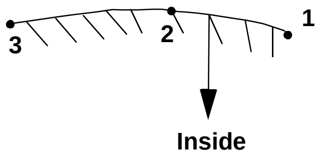
About I.D.Contact and I.D. Backfaces

Use the **ID CONTACT** and **ID BACKFACES** buttons in Contact Bodies to turn on the identification of contact bodies and backface polygons respectively in **MENTAT**.

Use the **I.D. CONTACT** feature to identify all the deformable and the rigid bodies.

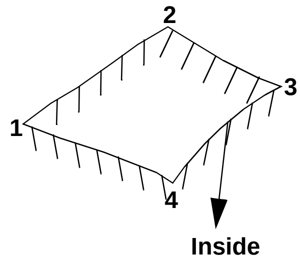
Use the **I.D. BACKFACES** feature to indicate the direction or the orientation of the rigid surface.

For rigid bodies, define the inside and outside surface. For 2-D analyses, the interior side (inside) is formed by the right-hand rule when moving along the body.



interior side
for 2-D analyses

For 3-D analyses, the interior side is formed by the right-hand rule along a patch. This surface, colored pink in **MENTAT**, is the front surface. Engineering hatch marks indicate the solid regions for rigid surfaces in 2-D while color codes are used in 3-D. Color codes are also used to identify the back surfaces of the body. A body is able to come into contact with the brown side of a 3-D rigid surface.



interior side
for 3-D analyses

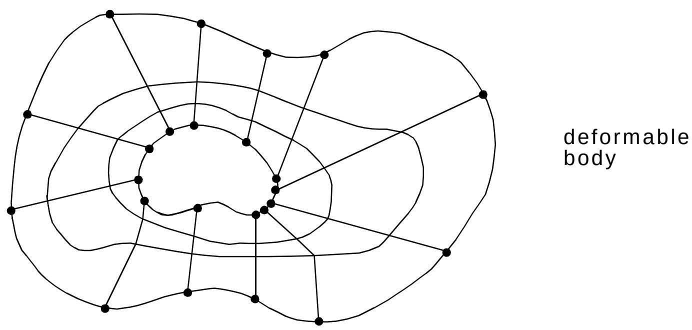
About Flip Elements, Curves, and Surfaces

If the surfaces on the contact bodies are not in the correct orientation, then, during the analysis procedure, a deformable body will penetrate a rigid surface.

Use the **FLIP CURVES** (2-D) or **FLIP SURFACES** (3-D) buttons in Contact Bodies to correct the orientation of the contact bodies.

Finite Element Mesh for a Deformable Body

You can generate the finite element mesh for a deformable body using the mesh generation options in the Main Menu in **MENTAT.**



Analysis Parameters for Deformable Bodies

The analysis parameters for deformable bodies are grouped under the following categories:

- Friction
- Description
- Heat Transfer Data for Coupled Analyses

Use the **FRICTION COEFFICIENT** option in Friction to associate a friction coefficient with a deformable body. You define the friction model in the **JOBS** option in Main Menu.

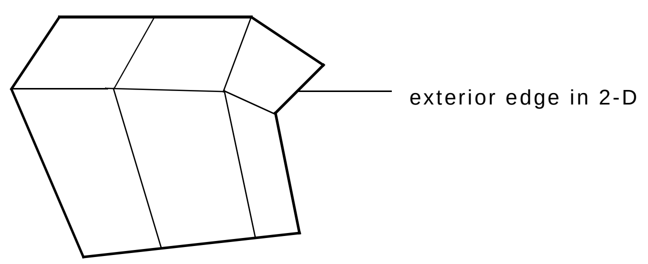
Use the Description field to prescribe how the surface of the deformable body is represented in your contact analysis.

For a heat transfer or coupled analysis, use the option in Heat Transfer Data for Coupled Analyses to prescribe the heat transfer parameters of the body.

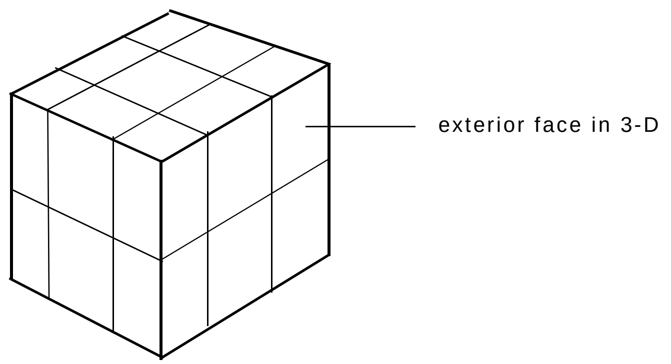
About Discrete and Analytical Descriptions

When you choose **DISCRETE** as the description option for a deformable body, this is how the discrete representations work for 2-D and 3-D situations:

- 2-D—the exterior edges of the deformable body represent the body. If another deformable body contacts this deformable body, the program bases the contact on those discrete representations.

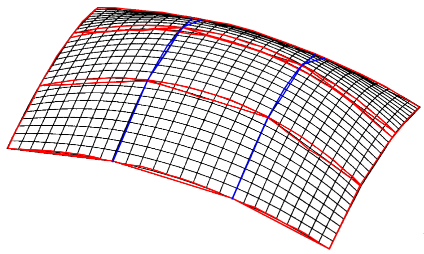


- 3-D—the exterior faces of the finite element mesh represent the body.



When you choose **ANALYTICAL** as your description option for a deformable body, here is how the analytical representation works on the exterior of the body:

- 2-D—puts a spline through the nodes on the exterior.
- 3-D—puts Coons patches through the nodes on the exterior.

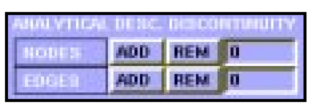


representation
based on
Coons surface

Use the Analytical description option for analysis of smooth bodies where you wish to improve the accuracy of the solution. The analytical solution does not add significant computational cost to your analysis.

About the Analytical Desc. Discontinuity Option

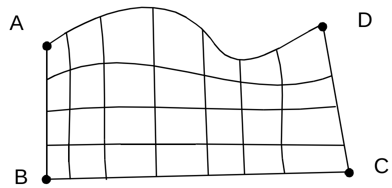
If you are using the Analytical description option and the deformable body has a real edge or node, you should activate the **ANALYTICAL DESC. DISCONTINUITY** option to define that edge.



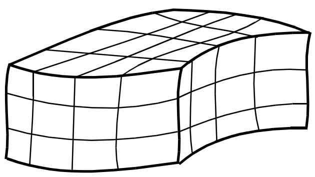
The **ANALYTICAL DESC. DISCONTINUITY** option is activated whenever you choose analytical description for a deformable body. Use this option to prevent the system from fitting a spline or a Coons patch through a particular region.

For 2-D meshes, specify the nodes at which discontinuities occur.

Contact



For 3-D meshes, specify the edges at which discontinuities occur.



About Rigid Bodies and Their Representation

Rigid bodies are a representation of reality because bodies are never absolutely rigid. But some bodies are substantially stiffer than other bodies and we can model them as rigid bodies.

A rigid body is represented by curves (2-D) and surfaces (3-D).

You can generate the finite element mesh for a rigid body using the mesh generation options (*See "Mesh Generation" on page 109*) in the Main Menu in MENTAT.

You can:

- define as many regions as necessary by constructing curves (2-D) and surfaces (3-D).
- read in geometry from a CAD system where you have already defined the curves and surfaces (*See "Import-Export Utility" on page 42*).

Motion Control of Rigid Bodies

Use the following options to control the motion of a rigid body:

- velocity
- position
- force on the rigid surface (i.e., load)

To choose a motion control option for a rigid body:

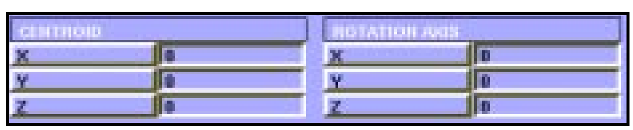
1. Choose Contact>Contact Bodies.
2. Click **RIGID** in Contact Body Type.

Contact>Contact Bodies>Rigid

3. Choose a body control option.

About the Centroid and Rotation Axis Options

To control the motion of a rigid body, you associate a point and an axis with the rigid body using the **CENTROID** and **ROTATION AXIS** options in Rigid Body.



Contact>Contact Bodies>Rigid

For 2-D analyses, you prescribe the x and y positions of the centroid. For 3-D analyses, you prescribe the x, y, z, and the rotational axis information.

About Velocity as Rigid Body Motion Control

Very often, the rigid surface is not in initial contact with the deformable surfaces, and you may not be exactly sure when or where the rigid surfaces first touch. To account for this, use the **INITIAL VELOCITY** option in Rigid Body to prescribe an initial velocity of the body such that it will have rigid body motion until it just comes into contact.

Contact>Contact Bodies>Rigid

When you choose velocity as your rigid body motion control option, you can prescribe the velocity to be a function of time.

Contact>Contact Bodies>Rigid

If the velocity varies with time, you could construct a table that is a function of time and then reference the table. All the different components of velocity (x, y, z, and rotational) can have their own unique functions of time.

About Position as Rigid Body Motion Control

When you choose the position as rigid body motion control, use the **POSITION** option in Rigid Body to describe the position (x, y, and z and rotational) of the reference point as a function of time.

rotation in fraction of a cycle about the rotation axis

reference your currently-defined tables here

Contact>Contact Bodies>Rigid

About Load as Rigid Body Motion Control

To specify load as control for a rigid body, you should associate an additional node with the rigid body. Using three different menus, you should specify the following three attributes for that additional node:

Attribute No....	Description...	Use the menu(s)...
1	position of the node	Mesh Generation
2	point load to prescribe that the force is a function of time and gets transferred to the rigid surface. If the force is not uniform, you can create a table that describes the force as a function of time	Boundary Conditions> Mechanical> Point Load
3	node number associated with the rigid surface	Contact Bodies> Control Node

Contact

Analytical and Discrete Description of Rigid Bodies

After you have defined the motion control parameters of the rigid body, you prescribe how the rigid surface is modeled numerically.

In MENTAT, all surfaces are represented as NURB curves or NURB surfaces. For your contact analysis, you can decide whether to use the analytical (NURB) description of the rigid body or a discrete description of body. The default setting is the analytical description.

The analytical description is more accurate but results in higher computation cost per iteration. But in many problems, you might require fewer iterations when you use the analytical procedure.

Coupled Analysis Considerations

When you use rigid surfaces in a coupled analysis, the rigid surface is treated as a heat source or a heat sink. The rigid body is considered to be a fixed body of infinite mass so that the temperature does not change in it.

To set the parameters for heat transfer in a coupled analysis, use the Heat Transfer Data for Coupled Analysis area in Rigid Body.

To enter specific parameters for heat transfer in a coupled analysis, refer to the following table:

To prescribe...	Use the button,...
the sink temperature of the rigid body	ENVIRONMENT SINK TEMPERATURE
how much of the temperature differential is going to result in a flux	CONTACT HEAT TRANSFER COEFF.

About Symmetry Bodies and Surfaces

Symmetry bodies are a subset of rigid bodies which have a unique behavior. Use symmetry surfaces when you have a body with a plane of symmetry associated with it.

In most cases, symmetry bodies are flat surfaces with a given orientation. You can model a given symmetry surface using either the analytical or discrete surface type. Symmetry surfaces are usually flat patches and you can use the analytical or discrete surface with little variance in results.

Special Assumptions for Symmetry Bodies

When you use a symmetry body, you make the following assumptions:

- the deformable body is in contact with the symmetry body from the beginning to the end of analysis. This assumption is necessary to maintain that there are no cracks or holes generated during the analysis.
- the symmetry bodies are frictionless.
- there is no flux across a symmetry surface during a heat transfer analysis (you don't need to provide any thermal information associated with a symmetry line or a symmetry plane.).

About Rigid w/Heat Transfer Bodies

The analysis considerations for rigid bodies with heat transfer are very similar to considerations for a rigid body (*See "About Rigid Bodies and Their Representation" on page 148*)

For the motion control of a rigid body with heat transfer, you can prescribe the velocity or position only. You cannot prescribe a load (force) to the region.

About Contact Tables

Use the Contact Tables options in MENTAT to determine the location of potential contact. By identifying a new contact table, you can indicate which body is potentially going to contact another body.

enter table properties here

Contact>Contact Tables

The Contact Table option is a very flexible way to enter complex contact conditions. You can use contact tables to specify that two bodies have particular attributes that are not shared with other pairs of contact bodies. These attributes include:

- contact distance
- separation force
- friction coefficient
- interface closure distance

Table Properties of a Contact Table

The TABLE PROPERTIES feature in Contact Tables is a flexible way to specify individual contact relationships between bodies.

Use the contact tables in your analysis to determine areas of potential contact which can help you reduce your computational costs. You can specify any of the following conditions:

- every body potentially touches every other body.
- every body touches every other body but does not touch itself.
- a body touches selected other bodies.
- a body is glued to another (or other) body (or bodies).

Contact

EXAMPLE

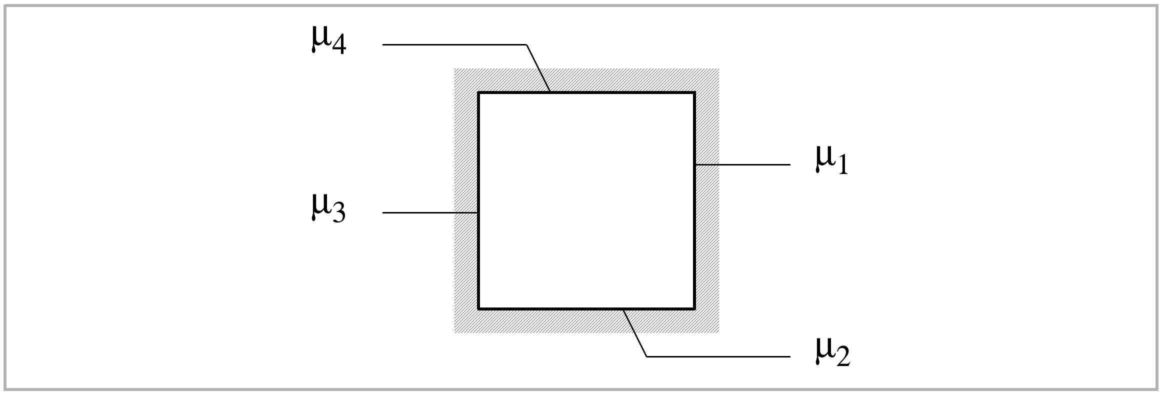
You might have a sheet forming problem where the default settings prescribe that every node in the sheet can contact some other region in the sheet. As long as the sheet does not fold or wrinkle unto itself, it is computationally effective to indicate that the sheet will not contact itself.

About Touching Bodies

The default setting in the Contact Tables menu is that the bodies do not touch one another. If you want all the bodies to contact and touch each other in your analysis, choose **TOUCH ALL** in the Table Properties window.

EXAMPLE

You might have a situation where deformable bodies are in contact with a square with one or more rigid surfaces. You can prescribe different coefficients $(\mu_1, \mu_2, \mu_3, \mu_4)$ of friction for each side of a square with the rigid surface.



Using contact tables, you:

- select the different touching bodies.
- specify the friction coefficient between the two touching bodies.

About Contact Areas

Use the CONTACT AREAS option in Contact to indicate contact regions in a contact analysis. The default setting enables all nodes on the exterior surfaces to touch other bodies. This might lead to a modest increase in computational costs. To reduce the computational cost, select only the nodes that might contact another body.

choose a body from the list of currently-defined bodies

choose nodes in the body that potentially touch other regions

Contact>Contact Areas

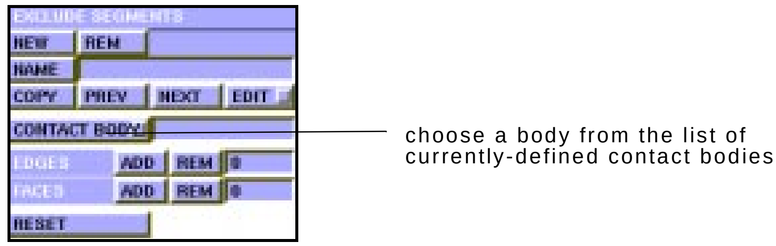
Using the options in Contact Areas, you can select:

- a deformable body.
- which nodes to associate with this deformable body that may touch other regions

You can prescribe that a node never come into contact in an analysis. However, that node could later penetrate another deformable or rigid body. Since there is no checking mechanism for such a situation, use this feature with caution.

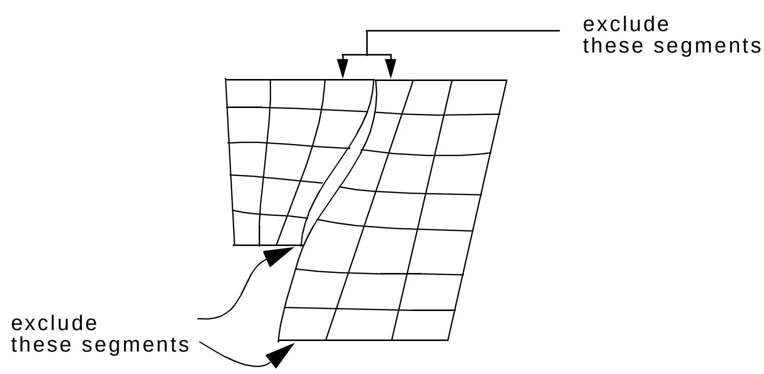
About Exclude Segments

Whereas the Contact Areas options enable you to exclude nodes in a contacting body, the Exclude Segments options lets you exclude certain segments of the contacted body.



Contact>Exclude Segments

Use the Exclude Segments options in situations where a node in a body makes contact at exactly the intersection of two segments or faces and you want the node to slide along one of the segments only.



To exclude segments in a contacting body:

1. Click **CONTACT** in Main Menu.
2. Click **EXCLUDE SEGMENTS**.
3. Click **CONTACT BODY** and choose a body from the list of currently-defined contact bodies.
4. Choose the edges (2-D) or faces (3-D) to exclude.

Specifying Contact Information in Loadcases

About Contact Information and Loadcases in Analysis

Use the Loadcase option to apply the contact information from your contact tables, contact areas, and exclude segments to your analysis. You can apply the contact information to loadcase types for the following loadcase classes:

- Mechanical (for Static, Creep, Dynamic Transient, and Rigid Plastic classes)
- Coupled
- Fluid-Solid
- Fluid-Thermal-Solid

Using Contact Information in Loadcase Types

To apply contact information to a loadcase:

1. Choose Loadcase>Loadcase Class.
2. Choose a loadcase class (e.g., Mechanical).
3. Choose a loadcase type (e.g., Static).
4. Choose **CONTACT**.

Loadcase>(Loadcase Class)>Contact

5. Choose from the list of contact tables and areas that you defined.

In a loadcase, you can have only one contact table active; you can also activate multiple contact areas.

Contact

About Contact Body Release

It is often useful to release nodes which are in contact with another body. A typical example would be when the rigid dies in a sheet forming process are removed from the sheet. When this occurs, the forces that were on a deformable body must be eliminated because the surface is now free. This results in a distribution of stress (in sheet forming, it results in spring back). It may also lead to a poor convergence if the forces are all removed at once. You can prevent this by removing the forces gradually using the **GRADUAL** option in the Contact menu.

Using Contact Body Releases

To release nodes that are in contact with another body:

1. In the Contact menu, select the body from which the nodes are to be separated.
2. Choose a type of release, **IMMEDIATE** or **GRADUAL**.

Preventing Separated Nodes from Contacting Again

After a node separates from a body, it may re-contact the same body. You can prevent this by using the Contact Table option to specify that the nodes cannot contact the body that they have been separated from. As an alternative approach, you can prescribe a time-dependent velocity or position to the surface such that it moves away from the nodes that have been separated from it.

About Synchronized Approach as a Body Approach Parameter

It is often useful to bring new surfaces into contact at the beginning of an increment. Rigid surfaces which are not currently in contact are moved with a prescribed velocity such that contact occurs. You can control this movement of rigid surfaces in the Body Approach Parameters window in **MENTAT.**

Loadcase>Loadcase Type>Body Approach

When you set synchronized approach to **OFF**, as soon as one rigid body makes contact, the other rigid bodies are stopped and the increment begins.

When you set the synchronized approach to **ON**, the rigid bodies with non-zero velocities that are not in contact are moved such that they all come into contact with a deformable body.

Specifying Contact Information in Jobs

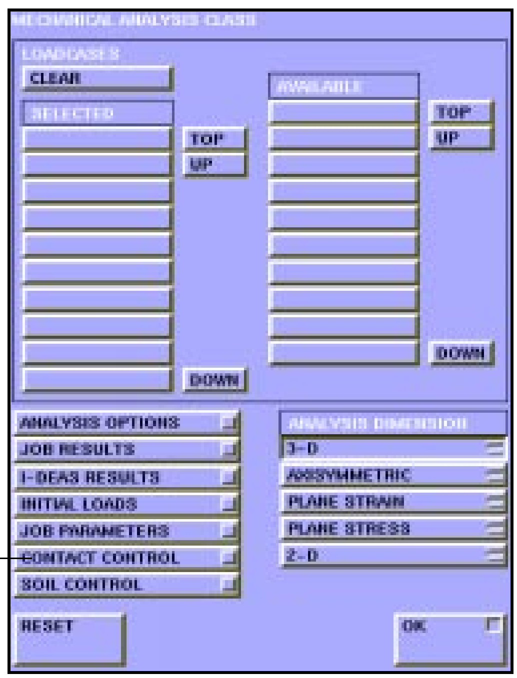
About Contact Control in Jobs

You can apply contact control parameters to your job for the following analysis classes:

- Mechanical
- Rigid Plastic
- Coupled
- Coupled Rigid Plastic
- Fluid-Thermal-Solid

To display your contact control options:

1. Choose Jobs>(Analysis Class).



click here for contact control options

Jobs>(Analysis Class)

2. Click **CONTACT CONTROL**.

About User Subroutines

Use the User Subroutine options in Contact Control to determine which user subroutines to activate in your contact analysis.

Jobs>(Analysis Class)>Contact Control

You can use one or more of the following user subroutines in your contact analysis:

- **MOTION**—prescribe either the velocity of the rigid surfaces or the position of the rigid surfaces.
- **SEPFOR—**specify the force required for separation.
- **SEPSTR**—prescribe the stress required for separation.
- **UCONTACT**—prescribe your own contact procedure.
- **UFRIC** (an alternative to specifying the coefficient of friction)—use in problems where friction coefficient is a function of the pressures between the bodies or if the friction coefficient was a function of the surface roughness and varied over the deformable region.
- **UHTCOE**—prescribe the heat transfer coefficient between the deformable body and the environment and the temperature of the environment.
- **UHTCON**—specify, in coupled analyses, the thermal coefficient between two bodies that are in contact.

About Contact Tolerances

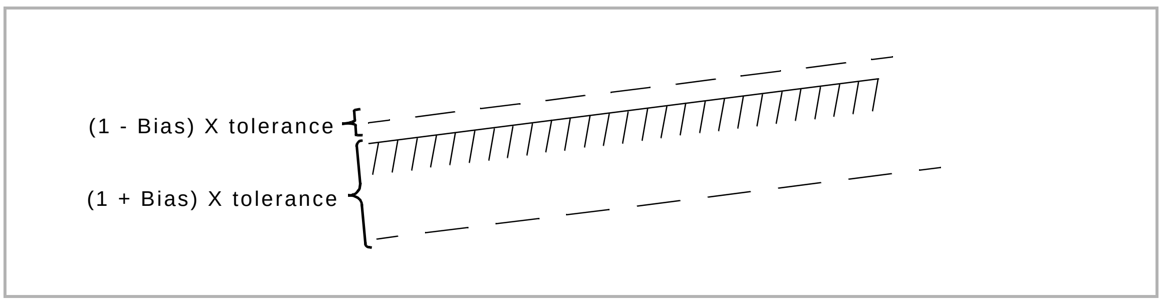
During the contact process, it is unlikely that a node exactly contacts the surface. So a contact tolerance is associated with each surface (*See also Vol. A: Chapter 8–Contact*). Use the contact tolerance options in Contact Control to specify the tolerance for the two bodies to contact each other.

Jobs>(Analysis Class)>Contact Control

When you leave the default value of the **DISTANCE TOLERANCE** field as zero, the program calculates the distance tolerance based on, either:

- the smallest element in the analysis, or
- the thickness of shell elements or beam elements

You can also specify a distance tolerance bias value to indicate that the tolerance is different on the backsurface of the body.



About Increment Splitting

Use the increment splitting options in Contact Control to specify the procedure when a node of one body penetrates another body.

Jobs>(Analysis Class)>Contact Control

You can use one of the following increment splitting procedures:

- Allowed—the increment is split into, at least, two parts. In the first part of the increment, no contact occurs. In the second part, the node is in contact with the rigid body or another deformable body. Use this procedure with the Fixed Time-Step procedure.
- Suppressed—when the node penetrates the surface, the penetration is ignored during that increment. In the following increment, the node is brought back on to the rigid or deformable surface.
- Iterative—satisfies the contact condition simultaneously with the Newton-Raphson iteration procedure. Use the Iterative procedure with the Fixed Time-Step procedure.

Of all the increment splitting procedures, the Suppressed option is the least expensive but the least accurate. The Allowed option is moderately expensive and relatively more accurate.

When you use the iterative procedures, less iterations are required for the analysis. However, if you use complex NURBS, each iteration may be more computationally costly.

About Deformable-Deformable Contact Control

The default setting in the Deformable-Deformable option is such that when you use single-sided contact, a node on the lowered-numbered body is checked for contact with the segments on the higher-numbered body.

The body with the higher I.D. and the body with the lower I.D. are termed master body and slave body respectively. If you choose **SINGLE-SIDED**, then there is contact between the master body and the slave body. The Single-Sided option can often reduce your computational cost.

About Separation Procedures

Separation of a node with a body in **MARC** is based upon either of the following procedures:

- nodal forces
- nodal stress

To use the **SEPARATION FORCE** button based on forces or on stresses, refer to the following table:

If you choose a separation procedure based on...	then, use the SEPARATION FORCE button to enter the...
forces	force at which two bodies separate.
stresses	stress at which separation occurs.

In theory, separation occurs when there is a positive tensile force. However, from a numerical perspective, you may wish to prescribe a large separation force. If you give too small a separation force, you might be faced with a higher computational cost because the nodes separate, return to contact and separate again.

About Max # Separations/Inc.

The default value for the maximum number of separations per increment is 9,999. This means that every time the system finds a node which separates within an increment, it will recycle that increment. This parameter limits the number of times that this will occur. You can reduce computing costs by using this feature.

About Separation Increment

The default setting for the Separation Increment option is CURRENT. When separation occurs, the program separates the bodies within the increment.

You can use the NEXT button to ignore the separation and only perform the separation at the beginning of the next increment. This is an analogous to suppressing increment splitting.

You might have decreased computational costs with the NEXT option, but it might lower the accuracy of your solution.

About Friction Models

In the Contact Bodies menus (*See “Choosing a Contact Body Type” on page 143*), you prescribed the coefficient of friction between contacting bodies. In the Friction option in Contact Control, you prescribe which friction model to use.

Jobs>(Analysis Class)>Contact Control

You can choose the following friction models:

- Shear—the friction stress is based upon the coefficient of friction and the equivalent von Mises stress in the material.
- Shear for Rolling—similar to Shear but modified for rolling.

- Coulomb—the friction stress is based upon the coefficient of friction and the normal stress at the surface.
- Coulomb for Rolling
- Stick-Slip

If you specify any of the friction models other than the Stick-Slip model, you have to specify the relative sliding velocity.

You also choose which of the following methods to use:

- Nodal Stress
- Nodal Force

EXCEPTIONS:

Refer to the following table for exceptions:

For...	the program uses this method...
Shear or Shear for Rolling models	Nodal Stress
Shell elements	Nodal Force
Stick-Slip model	Nodal Force

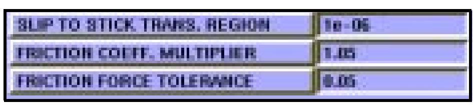
Coulomb for Rolling in Nodal Stresses and Nodal Forces

When you use the Coulomb or Coulomb for Rolling option, then your choice of the Nodal Stress or Nodal Force method actually makes a difference as shown in the following table:

If you choose the method...	then the program calculates...
Nodal Stress	a stress normal to the surface at the node and the frictional stress is proportional to that normal stress and the coefficient of friction.
Nodal Force	the reaction forces of the node which is in contact and uses the coefficient of friction and that reaction force to obtain a frictional force.

About Stick-slip

Contacting bodies never purely stick; there is always some slip associated with them. The relative sliding velocity (*See "About Friction Models" on page 166*) indicates the velocity at which they begin to slip.



Jobs>(Analysis Class)>Contact Control

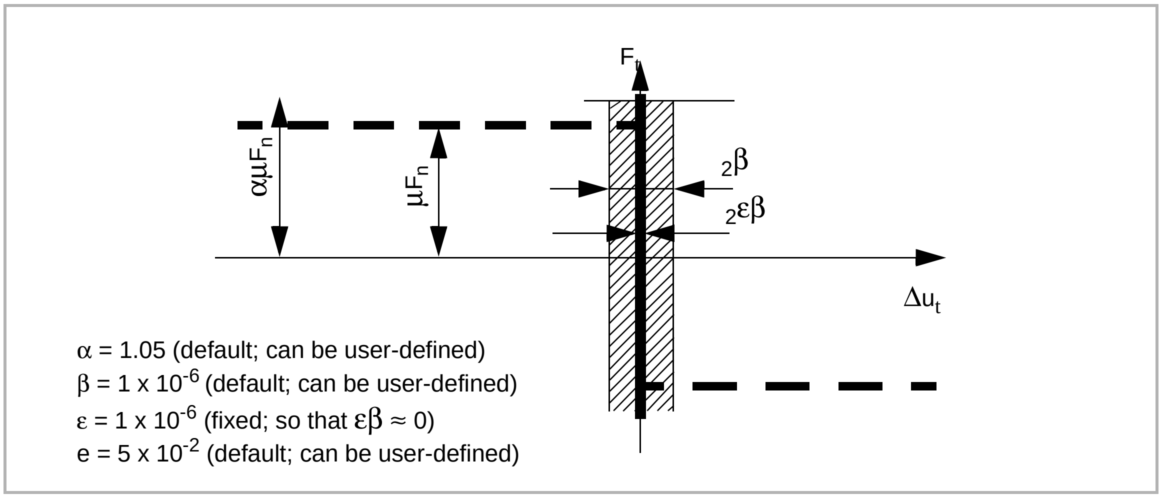
Using the Stick-Slip friction model, three parameters (α, β, and e) are available to control the numerical behavior. a represents a tolerance on the frictional force before sliding occurs. The node changes from stick to slip when

$$|F_t| > \alpha\mu F_n$$

β represents the amount of relative displacement needed to create slipping conditions. e is a tolerance on the convergence of the solution. The constitutive relationship is given by:

$$1 - e \le \frac{F_t}{F_t^{\,p}} \le 1 + e$$

where $F_t^{\,p}$ is the tangential force at the previous iteration.



The Stick-slip option enables you to completely model a sticking option. Use the option to prescribe the following parameters for contact bodies:

- Slip-to-Stick to Transition Region
- Friction Coefficient Multiplier
- Friction Coefficient Tolerance

Contact

Contact

Additional Information

Use the following table to find more information:

For...	refer to Chapter/s...	in...
background information and theoretical discussion	*8: Contact*	Vol. A
description of input	*3:Model Definition Option*	Vol. C
description of user subroutines, such as: MOTION (2D) MOTION (3D)	*2: User-Defined Loading, Boundary Conditions, and State Variables Subroutines*	Vol. D
demonstration problems	*8:Contact*	Vol. E

7• Design Sensitivity and Optimization

Design Sensitivity and Optimization

Design Variables

Design Constraints

Using Design Variables and Constraints

Post-Processing Sensitivity and Optimization Results

• Design Sensitivity and Optimization

Design Sensitivity and Optimization

About Design Sensitivity and Optimization

The relevant concepts relating to design optimization are:

- Analysis—evaluating the values of response quantities and the objective function based on a given set of design variable values.
- Sensitivity—evaluating the tendency of response quantities to change with respect to the design variables.

Components of Design Optimization

The three components of design optimization that you can apply to design sensitivity and optimization jobs in MENTAT are:

- Design Objective—either a direct or an indirect function of design variables; currently, the design objective can be to minimize the material mass, the volume, or the cost of a structural design.

- Design Variables—quantifiable, variable parameters of design. The design variable is associated with a physical quantity of design that you can specify (e.g., shell thickness, Young’s modulus, material density, or composite ply angles).

- Design Constraints—direct or indirect limitations on design variables or response quantities. The constraints are used to place bounds on design variables or impose limits on the response. For example, you would constrain the response such that a resultant stress (or deflection) is less than a certain value.

In MENTAT, the specification of design constraints and design variables is independent of the types of jobs. So, for example, you could specify several design variables or constraints and use only some of them in one job and others, including some of the previous ones, in another job.

When to Apply Design Optimization

Use the Design Optimization feature to:

- achieve a good approximation of the final design
- gauge the direction of the final design

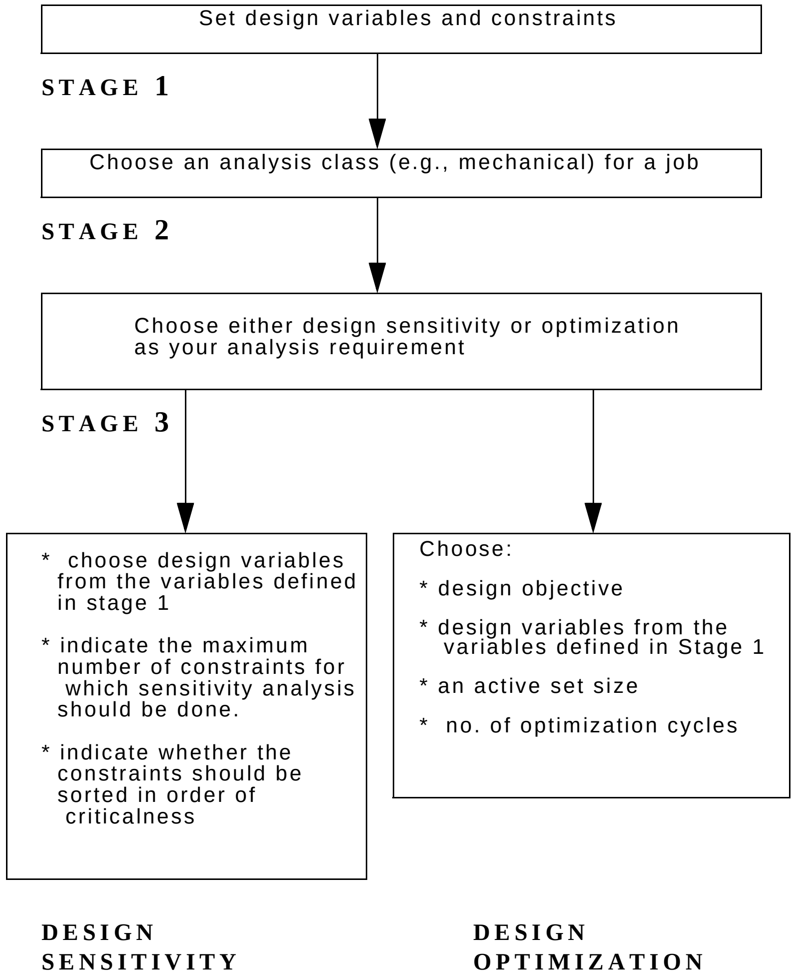
Where You Can Apply Design Optimization

Use the Design Optimization feature in structures that involve:

- static loads
- linear behavior
- multiple loadcases
- multiple constraints on stress, strain, and eigenfrequency response

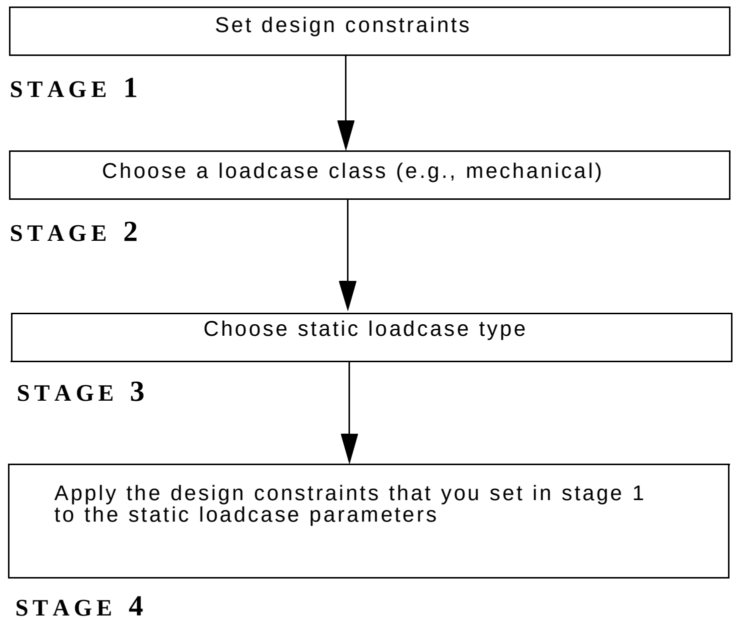
Application of Design Variables and Design Constraints to Jobs

The process of applying design variables and design constraints to jobs is shown in the following flow diagram:



Application of Design Constraints to Related Loadcases

The process of applying design constraints to related loadcases is shown in the following flow diagram:



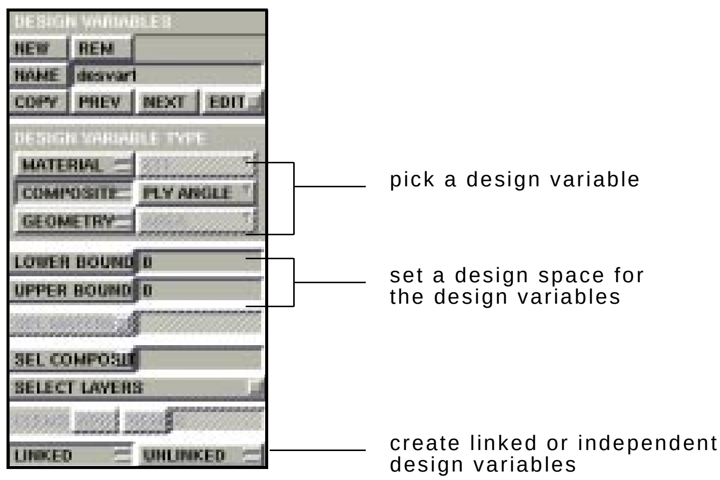
Design Variables

Types of Design Variables

You can use the following types of design variables in MENTAT:

- material
- composite
- geometry

To pick a design variable type, choose **DESIGN VARIABLES** in the Main Menu.



Main Menu>Design Variables

Linking and Unlinking Design Variables

Use the **LINKED** feature to associate one geometry design variable with more than one entity, such as an element or layer. To create geometry design variables that are independent of each other, use **UNLINKED**.

EXAMPLE:

You can define plate thickness as a design variable. If you specify certain elements (say, 1-10) with the design variable and you choose **LINKED**, then all of the elements, 1-10, have the same thickness. When the program changes the value of plate thickness, the elements, 1-10, reflect that change.

You can also use the linking options to specify that the ply angles or layer thicknesses of multiple layers should be the same.

When you use material design variables, the design variable for all elements associated with the chosen material are automatically linked.

About Bounds and Design Sensitivity

Use the **LOWER BOUND** and **UPPER BOUND** buttons to establish a design space for your design variables. The upper and lower bounds create a range for the design variables used in the optimization analysis.

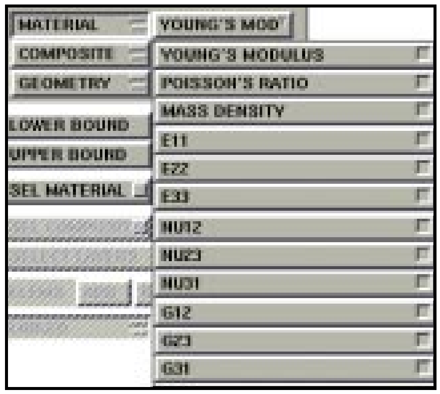
Choosing tight bounds constrains the optimization, but, in general, leads to a faster analysis and a more accurate design.

Within the design space, you can pick certain points and perform sensitivity analyses at those points. The sensitivity analysis is performed using the prescribed values of the design variables. In an optimization analysis, these prescribed values are not relevant.

Using Materials as Design Variables

To specify a material property as a design variable:

1. Choose Design Variables>Material.

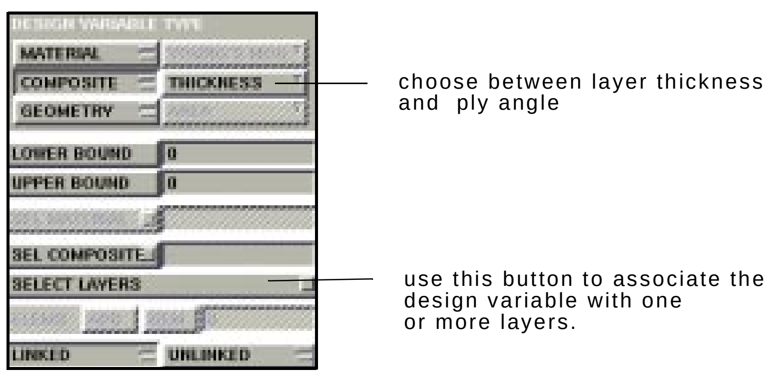


Design Variables>Material

2. Enter the upper and lower bound.
3. Use SEL MATERIAL to pick materials from the Currently Defined Base Materials window.

Using Composites as Design Variables

Use the Composites option to define the design variables for a composite group.



Design Variables>Composites

The design variables for composites are:

- ply angle
- layer thickness

Use the **SEL COMPOSITE** button to choose a composite material to apply the design variables to.

Geometry Design Variables

The geometry design variables for design optimization relate to the element sizing parameters. These variables must have a one-to-one correspondence with the geometry constants that you define for each particular element.

EXAMPLE:

IXX and IYY are the moments of inertia along the X and Y axes respectively for beam elements. If an element does not have the moment of inertia defined in the geometry option, you cannot have it as a design variable for that element.

Types of Geometry Design Variables

You can also define the following variables as geometry variables:

- Area (cross-sectional area of a truss element or a beam element)
- Beam height (also occurs as *thickness* in the element library)
- Beam width
- Radius of hollow cross-sections
- Wall thickness of hollow cross-sections
- Constant thickness (for shells, plane stress elements, membrane elements, generalized plane strain elements)

Except in the case of axisymmetric shells, avoid specifying thickness as a separate variable at every node of an element. For axisymmetric shells, you can specify thickness as a separate variable at each node of an element.

Using Geometry as Design Variables

To use geometry design variables:

1. Choose Design Variables>Geometry in Main Menu.

Design Variables>Geometry

2. Choose **LINKED** or **UNLINKED** option for the variable.
3. Choose a geometry design variable.
4. Enter the upper and lower bound.
5. Choose:

- **ADD** to associate elements with the design variable
- **REM** to dissociate elements from a design variable.

Design Constraints

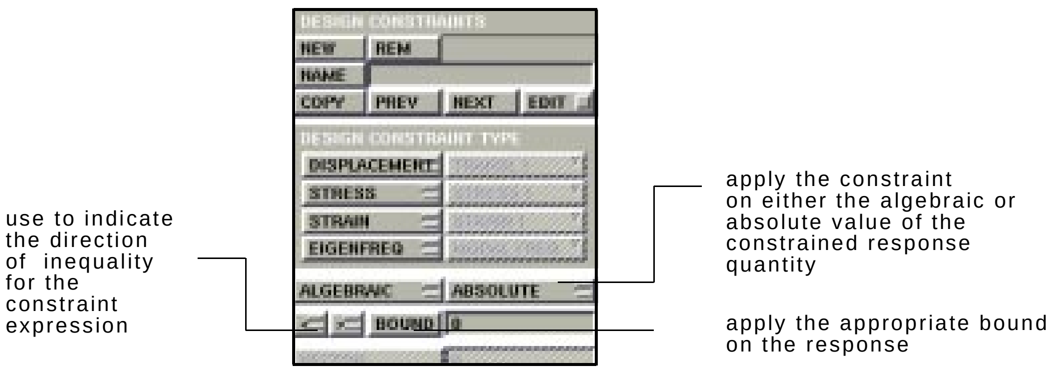
Types of Constraints

You can use the following types of constraints on your finite element model:

- displacement
- stress
- strain
- eigenfrequency

The constraints apply to the solution of loadcase or eigenfrequency analysis.

To set up constraints in **MENTAT**, choose **DESIGN CONSTRAINTS** in the Main Menu.



Main Menu>Design Constraints

About Bounds

Use the **BOUNDS** feature to establish a range of response for the design constraints. To set up the range of response for the value of a bound, use the > (greater than or equal to) or the < (less than or equal to) buttons.

Algebraic and Absolute Values

To put a constraint on the algebraic value, choose the **ALGEBRAIC** feature. This feature gives you the flexibility to set bounds on both the positive and the negative sides.

A typical algebraic constant would be:
$U_x < U_{max}$

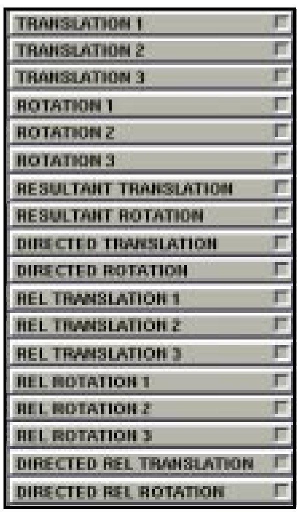
When you choose the **ABSOLUTE** button, the program picks the absolute, positive value of the response quantity.

A typical absolute value of constraint would be:
$|\sigma| < \sigma_{max}$

Setting Displacement Constraints

To set new displacement constraints:

1. Click **DESIGN CONSTRAINTS** in the main menu.
2. Click **DISPLACEMENT**.



Design Constraints>Displacement

3. Choose a constraint from the drop-down menu.
- For directed displacement constraints (e.g., directed rotation), enter the vector direction/plane normal.
4. Enter the bounds information and assign a range.

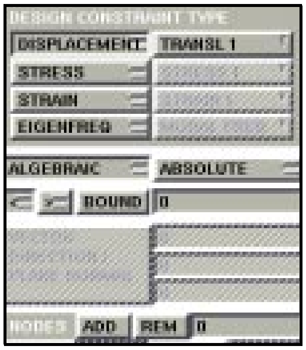
5. Choose algebraic or absolute value.
6. Add a node list to associate with the constraint.

Adding or Removing Nodes

Use the **ADD** and **REM** buttons to apply or remove a displacement constraint from a node.

To add or remove nodes:

1. Choose Design Constraints>Displacement.



Design Constraint>Displacement

2. Click **ADD** or **REM**.
3. Enter a node list.

Specifying the Vector Direction/Plane Normal

You must specify a vector direction when you use any of the following design constraints:

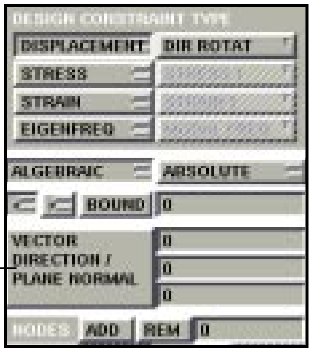
- directed translation (translation parallel to a vector direction)
- directed rotation (rotation about a vector direction)
- directed relative rotation (relative rotation about a vector direction)

- directed relative translation (relative translation parallel to a vector direction)
- normal stress on a plane
- maximum stress on a plane

To specify the vector direction for any of the above constraints:

1. Choose **VECTOR DIRECTION/PLANE NORMAL.**

specify the direction cosines



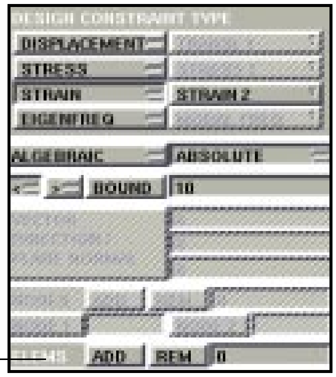
Main Menu>Design Constraints

2. In the dialogue area, enter the direction cosines for the X, Y, and Z components of the vector.

Adding or Removing Elements for Stress and Strain Constraints

To associate one or more elements with a stress or strain constraint, use the **ADD** button. To dissociate one or more elements from any of the constraints, use the **REM** button.

associate an element with, or dissociate an element from, a constraint

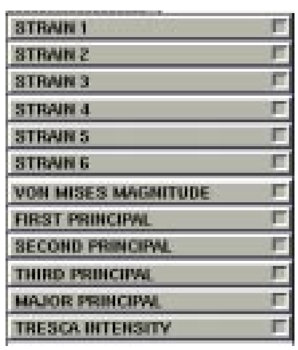


Main Menu>Design Constraints

Setting Stress or Strain Constraints

To set new stress or strain constraints:

1. Click **DESIGN CONSTRAINTS** in the main menu.
2. Click **STRESS** or **STRAIN**.



Design Constraints>Strain

3. Choose a constraint from the stress or strain constraints drop-down menu.
4. Enter the bounds information and assign a range.
5. Choose algebraic or absolute value.
6. Add an element list to associate with the constraint.

Types of Eigenvalue Constraints

You can specify the following types of eigenvalue constraints in MENTAT:

- Modal Frequency
- Frequency Difference

Use the MODAL FREQUENCY feature to prescribe frequencies of individual modes to be greater than certain values.

Use the FREQUENCY DIFFERENCE feature to prescribe a separation between different modes to prevent two modes from being too close to each other.

For both types of eigenvalue constraints, you can choose either one of the following units of frequency:

- cycles/time
- radians/time

Setting the Eigenvalue Constraints

To set new eigenvalue constraints:

1. Click Design Constraints>Eigenvalue.

Design Constraints>Eigenvalue

2. Choose one of the two eigenvalue constraint types.
- If you choose modal frequency, enter the mode number.
- If you choose frequency difference, enter the two mode numbers that you wish to associate with the constraint.
3. Enter the bounds information and specify a range.
4. Choose algebraic or absolute value.
5. Choose the unit of frequency.

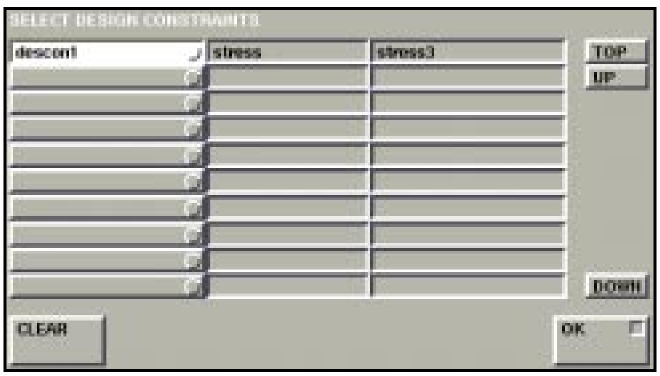
Using Design Variables and Constraints

Associating Constraints with a Loadcase

You can associate design constraints on static response with a particular loadcase in your analysis. Although an eigenvalue analysis is also considered to be a loadcase, currently all eigenfrequency constraints are automatically associated with a prescribed eigenvalue analysis ("modal shape") and are thus handled internally.

To associate constraints with a particular loadcase:

1. Choose Loadcases>Mechanical>Static>Design Constraints.

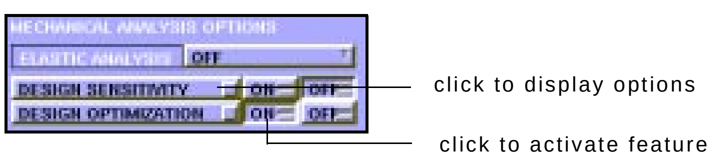


Loadcases>Mechanical>Static>Design Constraints

2. Choose the relevant constraints from the list of constraints.
3. Click OK.

Design Sensitivity and Optimization as Mechanical Analysis Options

The design sensitivity and optimization features are paired in the Mechanical Analysis Options window.

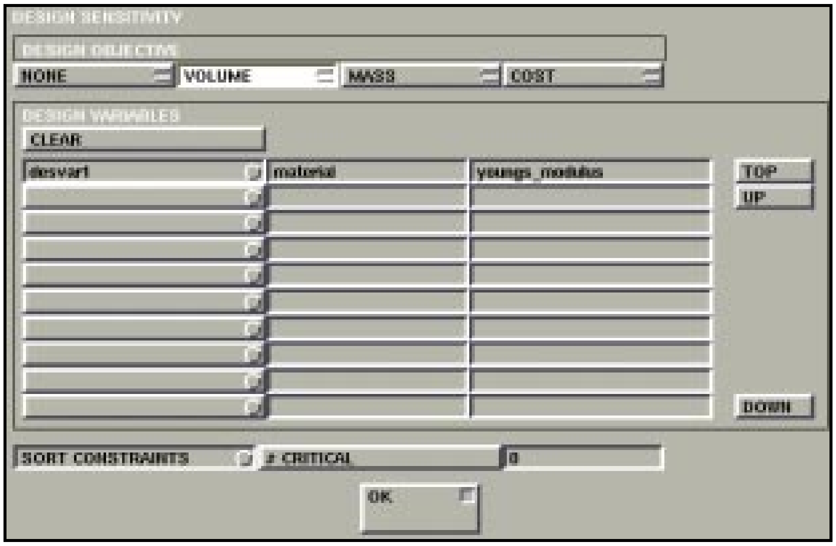


Jobs>Mechanical>Analysis Options

Using Design Variables and Constraints in Design Sensitivity

To use design variables and constraints in a design sensitivity analysis:

1. Choose Jobs> Mechanical> Analysis Options> Design Sensitivity



Jobs>Mechanical>Analysis Options>Design Sensitivity

2. Choose the relevant variables from the list of design variables.
3. Click **SORT CONSTRAINTS** and enter a value for the number of critical constraints to sort.
4. Click **OK**.
5. In the Design Sensitivity area, click **ON**.

Design Objective Considerations

If your design objective is mass, then you should provide the material unit mass for each of the elements.

You can specify the cost of a material as either of the following:

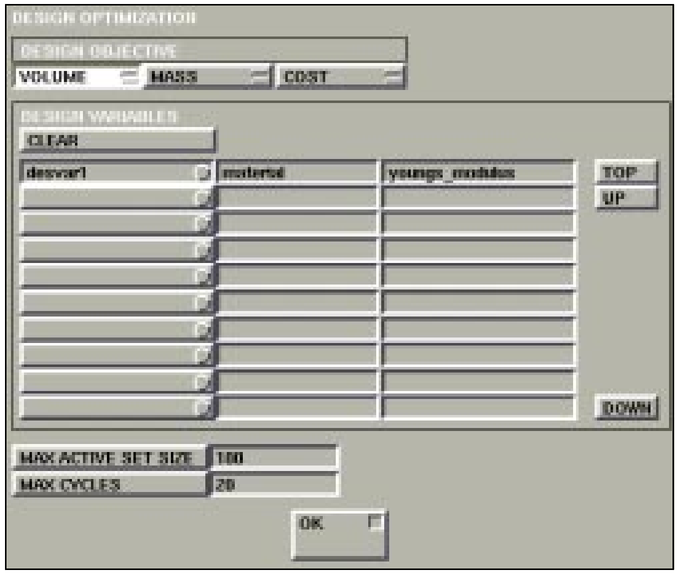
- per unit volume
- per unit mass

When you prepare the material data, remember to provide the material cost per unit mass or volume. If your material data is per unit mass, then you should also provide the mass density of the material.

Using Design Variables in Design Optimization

To use design variables in design optimization analysis:

1. Choose Jobs>Mechanical>Analysis Options>Design Optimization.



Jobs>Mechanical>Analysis Options>Design Optimization

2. Choose a design objective.
3. Choose the relevant variables from the list of design variables.
4. Enter a constraint set size or use the default set size.
5. Enter the number of cycles or use the default number of cycles.
6. Click **OK**.

About Maximum Active Set Size

Use the **MAX. ACTIVE SET SIZE** feature to work with a certain size of a constraint set. The constraint set determines the maximum number of constraints for sensitivity analysis during optimization.

The default set size is 100. You can alter the size of the active set to obtain the best results for your design optimization job. The larger the size of the active constraint set, the greater the computational costs.

About Maximum Cycles

A cycle comprises of an exact analysis and a completed cycle of design iteration. The default number of cycles is 20. You can alter the number of cycles depending on your:

- optimization results
- computational resources

Additional Information

Use the following table to find more information:

For...	refer to Chapter/s...	in...
definitions of individual buttons	*10: Design Variables and Constraints*	Mentat 3.1 Command Reference
background information and theoretical discussion	*5: Structural Procedure Library*	Vol. A
description of design sensitivity and design optimization parameters	*2:Parameters*	Vol. C
description of the following options: • DESIGN OBJECTIVE • DESIGN VARIABLES • DESIGN DISPLACE-MENT • CONSTRAINTS • DESIGN STRESS CONSTRAINTS • DESIGN STRAIN CONSTRAINTS • DESIGN FRE-QUENCY CON-STRAINTS	*3: Model Definition Options–Program Control*	Vol. C
list of demonstration problems	*Appendix F: Demonstration Problems*	this guide

Post-Processing of Sensitivity Results

About the Plotting of Sensitivity Results

In plotting of the results of a sensitivity analysis, the increment number associated with the plot is one increment higher than the highest increment number relating to loadcases.

EXAMPLE

Under the MARC increment numbering system, if there are three static analyses (the program does not count the eigenvalue analysis as an increment), the system adds one to that number. So all the sensitivity-related plots are shown as sub-increments of a fourth increment number.

When you read in the MARC post file of the sensitivity results in MENTAT, the program displays the last available increment in the post file. Each of the responses is a sub-increment, starting from first to the *n*th sub-increment. The only exception is the objective function, which, if it exists, appears in the zeroth sub-increment.

The objective function gradient is given as the zeroth sub-increment. There is no zeroth sub-increment if you did not specify an objective function.

About Constraint Reference Numbers

The sub-increment numbers for sensitivity analysis have a one-to-one correspondence with the constraint reference numbers in the output file for sensitivity results. A constraint reference number is a unique number associated with a response for which a sensitivity analysis is done. Therefore, for this particular increment, the sub-increment numbers are neither necessarily contiguous nor in increasing order.

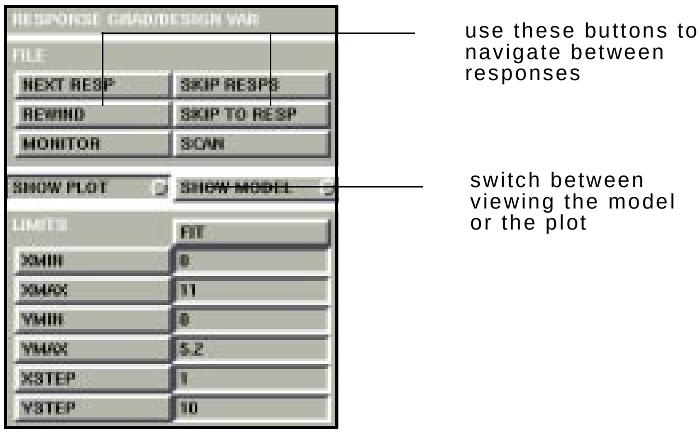
EXAMPLE

Consider that Stress 5 at Element 1, Layer 1, Integration Point 20, for Loadcase 2 happens to be a static loadcase (i.e., first increment if preceded by an eigenvalue analysis). If this particular constraint has the reference number 3, then to display the plots for this constraint reference number in MENTAT, look for sub-increment 3.

About Response Gradient Plots of Sensitivity Results

The response gradient plots provide the gradients of response quantities with respect to the design variables. From this information, you can determine which of the design variables the response quantity is more (or less) sensitive to. The data for the plots is available as sub-increments of the highest-numbered increment during post-processing.

Use the Response Gradient/Design Variable menus to plot the derivatives of each response with respect to the design variables at the prescribed design.



Results>Response Grad/Design Var

The results are only valid as first derivatives (i.e., as first order information). They are usually meaningful in a small region of the design space around the prescribed design.

Types of Plots Associated with Response Quantities

There are three types of plots associated with response quantities (essentially, a sub-increment in the t19, or t16 post file) and the objective function:

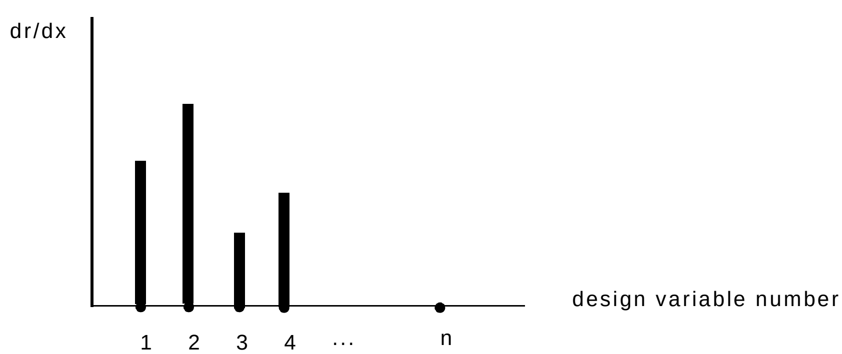
- bar charts—the gradient of a response quantity
- bar charts—the gradient of the objective function
- contour plots (similar to stress contours)—showing, on the finite element model, the element contributions to the specific response quantity.

About Bar Charts Showing the Gradient of a Response Quantity

When you plot a bar chart showing the gradient of a response quantity r_i, you can view all the derivatives, $(dr)/(dx)$, of that response quantity with respect to each of the design variables. So, for a number of response quantities:

$$\nabla \underset{\sim}{r_i} = \begin{bmatrix} (dr_i)/(dx_1) \\ \ldots \\ (dr_i)/(dx_n) \end{bmatrix} \quad \text{where } x_j \text{ are the design variables}$$

The plot of the gradient of the response quantity with respect to the design variables is represented in a bar chart, as shown here:



Plot of Gradient of Response Quantity with Respect to Design Variable(s)

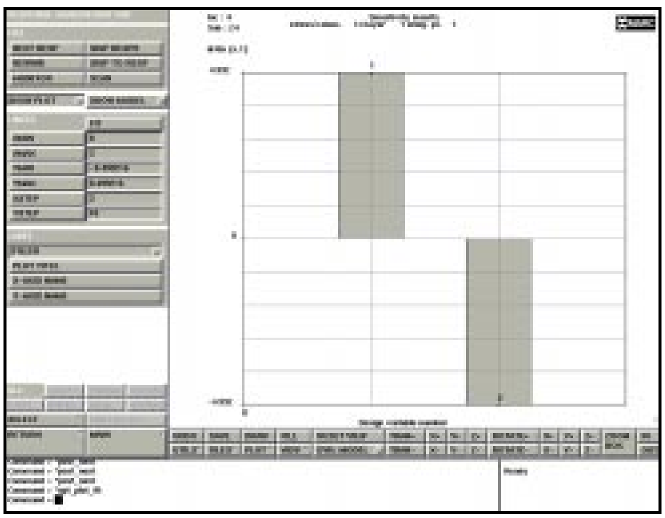
Plotting the Gradient of a Response Quantity with Respect to Design Variables

To plot the gradient of a response quantity from a **MARC** post file:

1. Choose Results>Open.
2. Using the file browser, open the appropriate **MARC** post file.
3. Choose Response Grad/Design Var>Show Plot.
4. To skip to a specific response quantity, use either of the following buttons:

- **SKIP TO RESP**
- **SKIP RESPS**
- **NEXT RESP**

5. Click **FIT** to fit all the values into the screen.



Gradient of Response Quantity with Respect to Design Variables

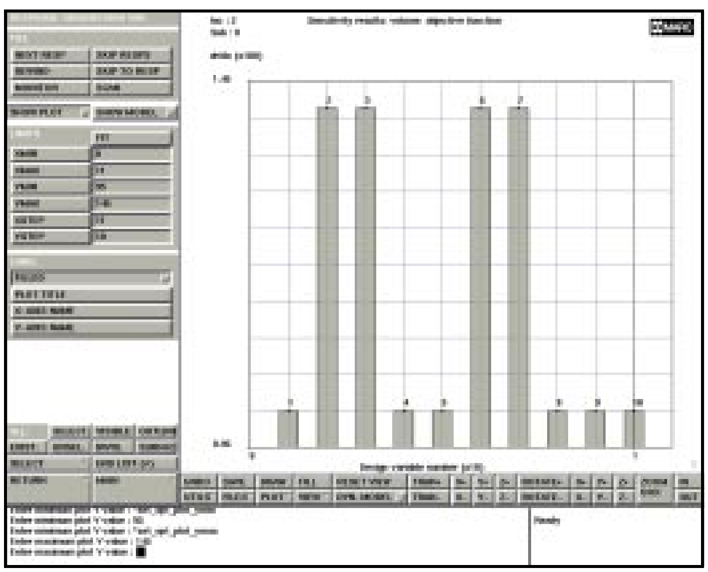
To obtain a meaningful plot, you might have to adjust the upper and lower limits.

Plotting Bar Charts Showing Gradient of Objective Function

You can plot bar charts showing the gradient of the objective function (mass, volume, cost) with respect of each of the design variables. For the program to display a gradient of the objective function, you must specify an objective function for the analysis. During post-processing, related results appear as the zeroth sub-increment of the last increment.

To plot a bar chart showing the gradient of the objective function with respect to each of the design variables:

1. Choose Results>Open.
2. Using the file browser, open the MARC sensitivity results post file.
3. Click SCAN to view the sub-increments.
4. Click SKIP TO INC and enter the last increment and the zeroth subincrement (e.g. 4:0).
5. Choose Response Grad/Design Var>Show Plot.
6. Click FIT to fit all the values into the screen.



Plot of Gradient of Objective Function with Respect to Design Variables

About Contour Plots

Contour plots display, on the finite element model, the element contributions to the specific response quantity. The element contributions are those numbers, which when added, usually sum up to the response itself.

Here the response (r_i) is represented by:

$$r_i = \sum_{j=1}^{n} c_{ij} + C$$

constant
is a
work term
that may
be due to,
say, support
settlement

where c_{ij} are the element contributions.

In some cases, such as when there is support settlement (i.e., some work is done at the support), there is another constant term (such as C, above) that is taken into consideration. So, in special cases, such as support settlement, the element contributions do not add up to the response itself due to the presence of this additional work term.

Creating Contour Plots of Element Contributions

To create contour plots of elements contributions from a sensitivity post file:

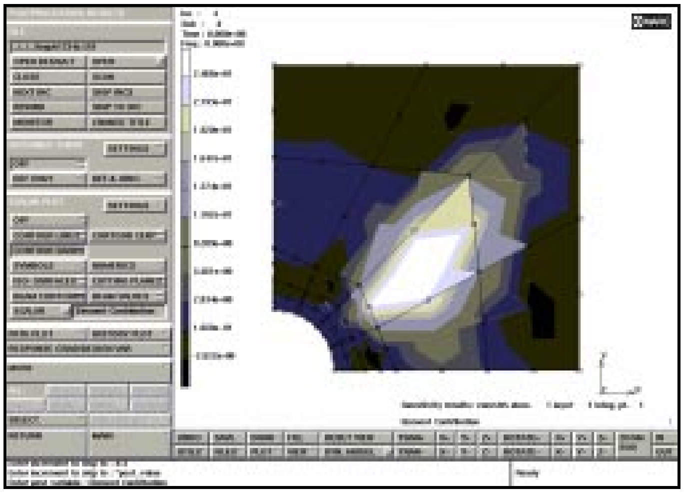
1. Choose Results>Open.
2. Using the file browser, open the appropriate **MARC** sensitivity post file.
3. Choose one of the contour plot options (e.g., contour bands).
4. Skip to a sub-increment containing the gradient information using one of the following buttons:

- **SKIP TO INC**
- **SKIP INCS**
- **NEXT INC**

5. Click **SCALAR** and choose element contribution from the Select Post Scalar menu.

6. Click **RESPONSE GRAD/DESIGN VAR.**
7. Click **SHOW MODEL.**

.



Contour Plot of Element Contributions from a Sensitivity Analysis

Post-Processing of Optimization Results

About Plotting of Optimization Results

You can create the following plots for optimization results in MENTAT:

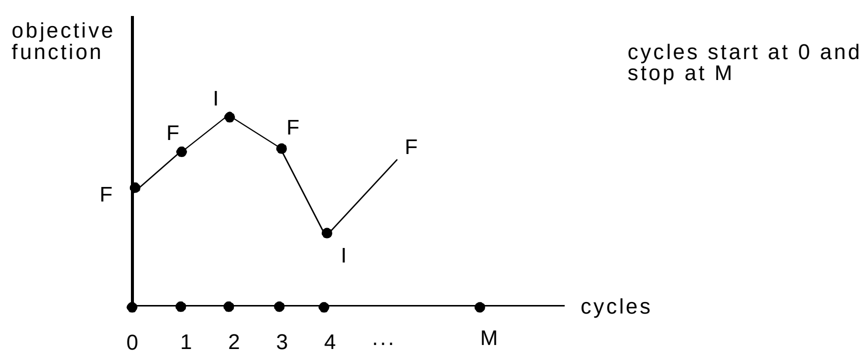
- history plots—the change in a response quantity or objective function over the optimization cycles.
- bar chart—the values of the design variables at a cycle.

About History Plots of Optimization Results

History plots of optimization results show the change in the objective function, or a design variable, with the optimization cycles. In the post file, the optimization cycles occur as sub-increments of the zeroth increment. The history plots are also paired with information on whether each point on the plot represents a feasible (F), or an infeasible (I) design.

A design is considered feasible when it satisfies all of the given constraints. An infeasible design is one which violates one or more of the constraints. If the violation is reasonably small, you can decide whether the design is acceptable or not. Generally, if the violation is less than the absolute, normalized value, 1×10^{-4}, the design is considered feasible.

The program traces history plots over the optimization cycles (sub-increments), with each integer x-axis value corresponding to a sub-increment.



History of Objective Function Values over Optimization Cycles

Design Sensitivity and Optimization

Creating History Plots of Objective Function Values

To create a history plot of objective function values over optimization cycles:

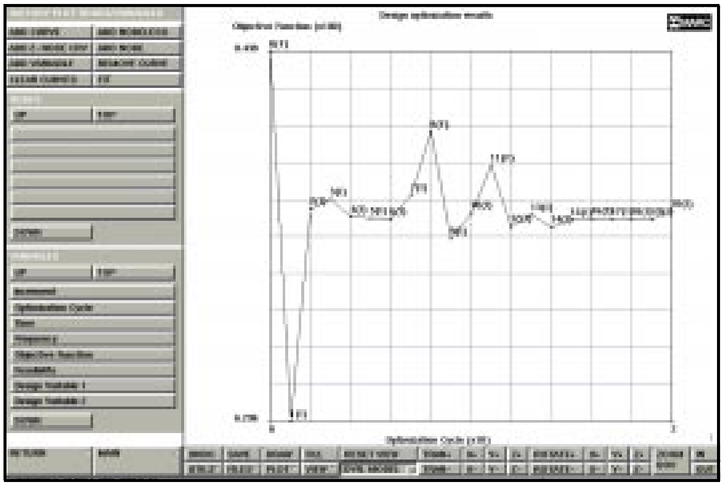
1. Choose Results>Open.
2. Using the file browser, open the appropriate optimization post file.
3. Click **SCAN** to view the increments.
4. Choose History Plot>Collect Data and enter the following information:

- first increment
- last increment
- step size

5. Choose Nodes/Variables>Add Nodeless
6. From the list of design variables, choose:

- the plot variable for the X-axis (optimization cycles)
- the plot variable for the Y-axis (objective function)

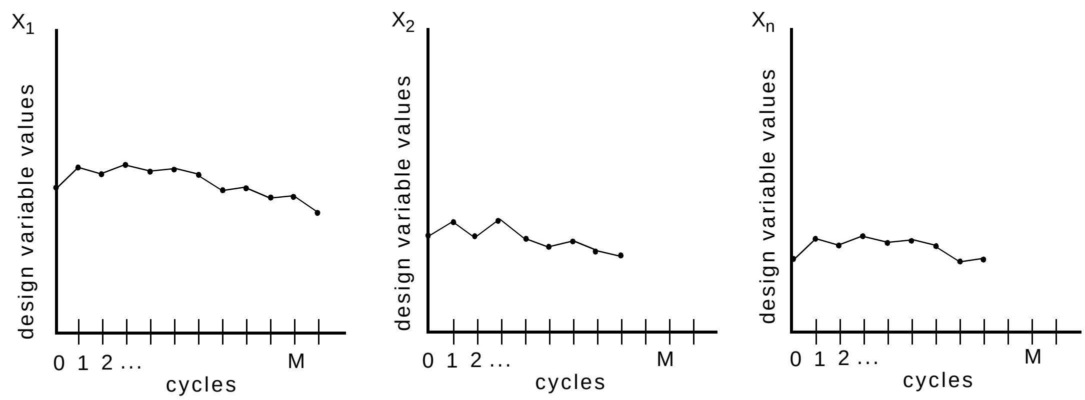
7. Click **FIT** to fit all the values into the screen.



History Plot of Objective Function with Respect to Optimization Cycles

About History Plots of Design Variable Values over Optimization Cycles

You can plot the change in the values of design variables over the optimization cycles using the same commands as in creating a plot for objective function.



Plot of Design Variable Values (Xj) over Optimization Cycles

Creating History Plots of Design Variable Values

To create history plots of design variable values over optimization cycles:

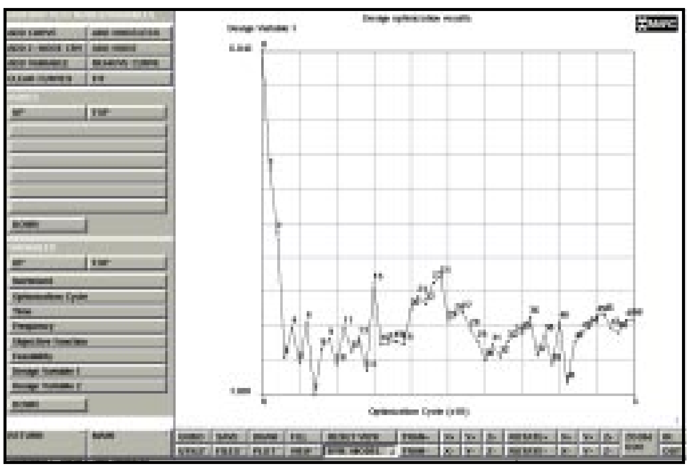
1. Choose Results>Open.
2. Using the file browser, open the optimization results file.
3. Click **COLLECT DATA** and enter the following information:

- first increment
- last increment
- step size

4. Choose Nodes/Variables>Add Nodeless
5. From the list of design variables, choose:

- the plot variable for the X-axis (optimization cycles)
- a design variable for the Y-axis

6. Click **FIT** to fit all the values into the screen.



History Plot of Design Variable Values with Respect to Optimization Cycles

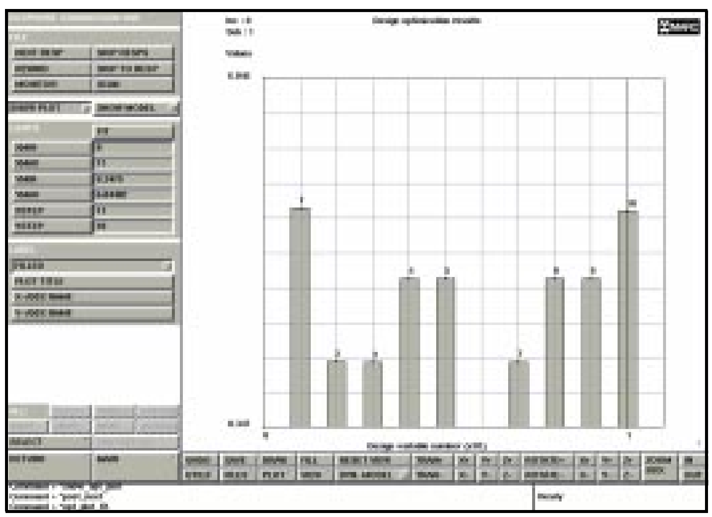
Creating Bar Charts of Design Variables Values

To create bar charts of design variable values at the end of a cycle:

1. Choose Results>Open.
2. Using the file browser, open the appropriate optimization file.
3. Choose Response Grad/Design Var>Show Plot.
4. To skip to a specific sub-increment, use either of the following buttons:

- **SKIP TO INC**
- **SKIP INCS**
- **NEXT INC**

5. Click **FIT** to fit all the values into the screen.



Design Variable Values at the End of a Cycle

Additional Information

Use the following table to find more information:

For...	refer to Chapter/s...	in...
definitions of individual buttons in the Results menus	*13: Results*	Mentat 3.1 Command Reference
background information and theoretical discussion	*5: Structural Procedure Library*	Vol. A

8• Element Technology

New Shell Elements

New Rebar Elements

• Element Technology

New Shell Elements

About the New Shell Elements

The following element types are now available in MARC:

- 138
- 139
- 140

Like all other MARC shell-elements, these three elements use a numerical integration through the thickness. The default number is 11.

You can modify the number of layers per shell by using either of the following options:

- composites
- job parameters

About Element 138

Element 138 is a four-node, thin-shell element based on the Discrete Kirchoff theory.

These are some of the salient features of Element 138:

- triangular element; works very well with the triangular mesh generators that are available in Mentat or in other CAD programs
- ease of use
- six degrees of freedom per node
- requires no tying
- applicable to all the material models in MARC

About Element 139

Element 139 is a four-node, thin-shell element based on the Discrete Kirchoff shell theory.

These are some of the salient features of Element 139:

- similar to Element 75, but uses thin-shell theory as opposed to thick-shell theory
- suitable for thin-shell structures
- applicable to all the material models that are in **MARC**
- six degrees of freedom per node
- four integration points per layer
- about five times faster than Element 75 during the element assembly phase

About Element 140

Element 140 is a four-node, thick-shell element similar to Element 75 but uses a single integration point per element.

These are some of the salient features of Element 140:

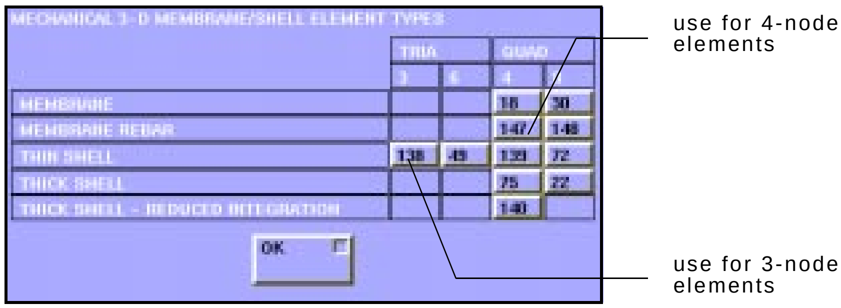
- six degrees of freedom
- uses hourglass stiffness matrix to insure that it is a stable element
- cheaper to use but not as accurate in the case of plasticity analysis
- about two-and-a-half times faster than Element 75 during the element assembly phase

Associating Elements with a 3-D Membrane/ Shell

Before you associate elements types with your 3-D Membrane/ Shell element, you should create your mesh using three-node or four-node elements.

To list elements with a 3-D Membrane/Shell element:

1. Choose Jobs>Element Types.
2. Click **3-D MEMBRANE/SHELL.**
3. Choose the elements to associate.



4. Click **OK.**
5. Select elements.

Additional Information

Use the following table to find more information:

For...	refer to Chapter/s...	in...
notes on shell orientation, layers	*1:MARC System*	Vol. A
description of the elements	*3: Element Library*	‘Vol. B

Element Technology

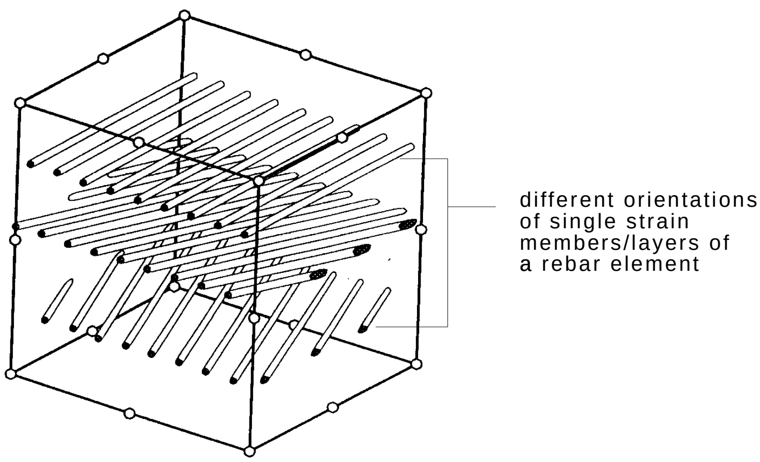
New Rebar Elements

About Rebar Elements

Rebar elements are, essentially, hollow elements with uniaxial stiffness. You can place single strain members in rebar elements.

Use rebar elements to represent stiffness elements that are present in a single direction through the matrix material. Although the rebar layer concept was originally applied to steel rebars in concrete civil engineering applications, you can use rebar elements in composite materials—composites containing small filaments (e.g., tires in which there are belts running through the rubber material).

The orientation of single strain members in different layers of a rebar element may be different. Each rebar element may have up to five rebar layers oriented in a particular direction. Each one of the layers has a different material behavior.

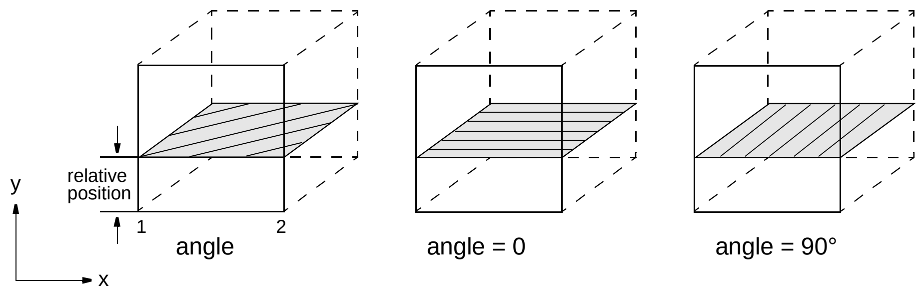


Types of Rebar Elements

Here are the types of rebar elements available in MARC/MENTAT:

- plane-strain rebar elements (four-node and eight-node)
- generalized plane-strain elements (eight-node)

- axisymmetric rebar elements (four-node and eight-node)



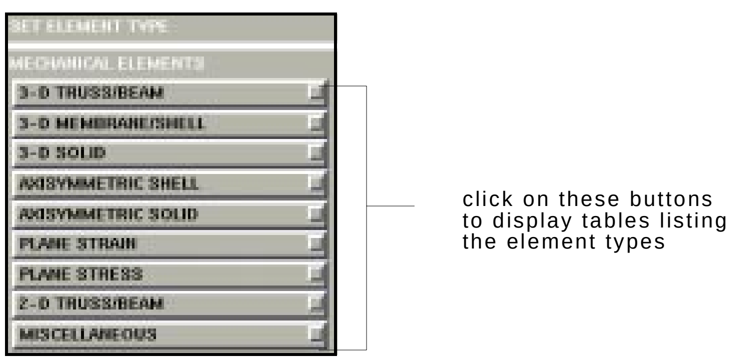
plane strain or axisymmetric rebar elements

- axisymmetric with twist elements (four-node and eight-node)
- brick rebar elements (eight-node and twenty-node)
- membrane rebar elements (four-node and eight-node)

Generating and Identifying Rebar Elements

To generate and identify the rebar elements:

1. Use the mesh generation tools in MENTAT to generate the elements.
2. Choose Jobs>Element Type to identify the rebar elements.



Jobs>Element Type

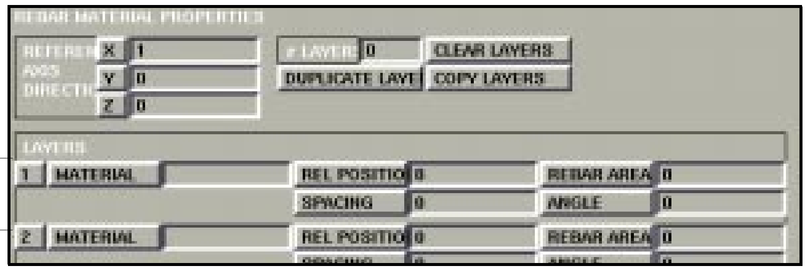
Specifying Rebar Material Properties

Use the Materials Properties menus to specify the properties of rebar elements including the direction of uniaxial behavior.

To specify rebar material properties:

1. Choose Material Properties>New Rebar-Define.

specify rebar material properties for up to five layers

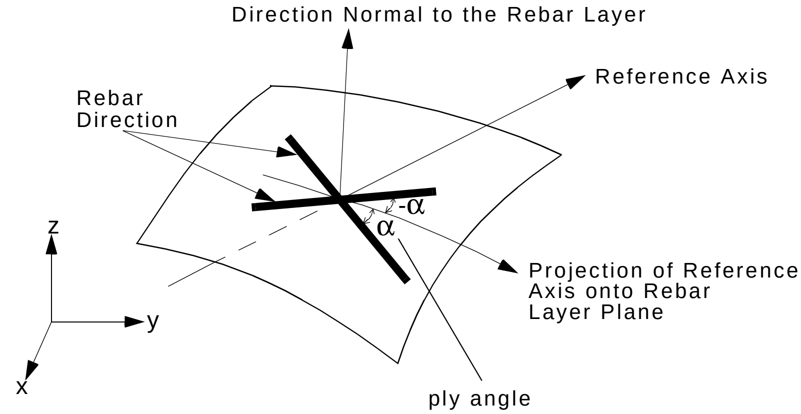


Material Properties>New Rebar>Define

2. Specify the orientation of uniaxial behavior.
3. Choose the number of layers for the rebar element.
4. Specify the following properties for one or more layers:

- relative position of this rebar layer in the element (this number is between 0.0 and 1.0).
- cross-sectional area of the rebar.

- ply angle (the angle with respect to the projection of reference axis onto rebar layer plane)



description of rebar orientation on a single rebar layer

Associating Double Elements with Nodes

By associating double elements with nodes, you specify dissimilar materials (e.g., rubber and steel) to concurrent elements.

To duplicate elements where you want rebar behavior:

1. Generate a finite element mesh.
2. Using the Mesh Generation menus, duplicate those elements where you want rebar behavior.
3. Specify the following settings to create elements in the identical location(s):

- Scale factor: 1
- Locations: 0,0,0
- Translations: 0,0,0

4. Sweep nodes.

Use the SWEEP command to remove the duplicate nodes only. Do not sweep the elements.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
definitions of individual buttons in the new rebar menus	*7: Material Properties*	Mentat 3.1 Command Reference
background information and theoretical discussion	*10: Special Elements*	Vol. A
description of individual rebar elements	*3: Element Library*	Vol. B
descriptions on how to define rebar positions, areas, and orientations in MARC	*3: Material Properties–Rebar*	Vol. C
description of alternative user subroutine	*6: Geometry Modifications and Subroutines–Rebar*	Vol. D

9• Fluid Mechanics

Fluid Analysis

Boundary Conditions for Fluid Analysis

Initial Conditions for Fluid Analysis

Material Properties for Fluid Analysis

Loadcases for Fluid Analysis

• Fluid Mechanics

Fluid Analysis

About Fluid Analysis

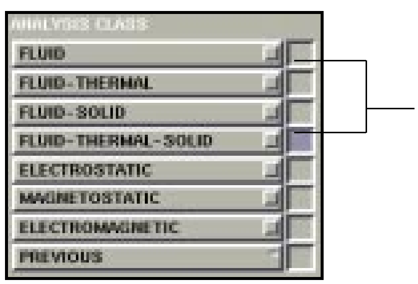
You can obtain solutions to problems in fluid mechanics by solving the Navier Stokes equations for the following fluid analyses in MARC:

- purely fluid
- fluid coupled with heat transfer (thermal)
- fluid-solid
- fluid-thermal-solid

> **The procedure is limited to problems where the fluid may be treated as a single-phase, incompressible material. The solution is also limited to problems where turbulence does not develop in the flow.**

To display the four fluid analyses class parameters in MENTAT:

1. Choose Jobs>More.
2. Click on any of the four fluid analyses class buttons.



click a fluid analysis class to display the analysis class parameters

Jobs >More

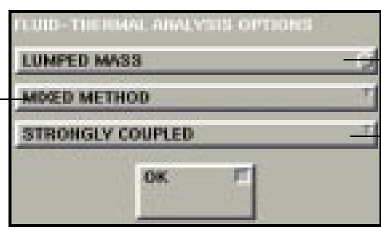
About Fluid Analysis Options

These are the fluid analysis options in **MARC**:

- Lumped Mass
- Mixed Method
- Penalty Method
- Strongly Coupled
- Weakly Coupled

To use the fluid analysis options in **MENTAT**:

1. Choose Jobs>More.
2. Choose one of the analysis classes (e.g., Fluid, Fluid-Thermal, etc.).
3. Click **ANALYSIS OPTIONS**.



Jobs>More>(Analysis Class)>Analysis Options

About the Lumped Mass Option

Use the Lumped Mass parameter to control whether the solver uses the consistent mass matrix or the lumped mass matrix. The application of the Lumped Mass parameter for fluids is similar to its application in heat transfer analysis. Instead of forming a consistent mass matrix, the program now formulates a lumped mass matrix which affects the inertia term associated with the fluid behavior.

When you choose the lumped mass parameter for a fluid-thermal or a fluid-thermal-solid analysis, you also lump the specific heat matrix.

About Mixed and Penalty Method Procedures

You can toggle the buttons between the Mixed Method and Penalty Method parameters in MENTAT to select a procedure that ensures that the fluid is computationally treated as incompressible.

When you use the Mixed Method, there is an additional degree of freedom–a pressure degree of freedom–at all the nodes. So the degrees of freedom involve the velocities and pressure.

When you specify the Penalty Method parameter, the degree of freedom involves the velocities only and an additional penalty function. The penalty function is a scalar function and you specify the value of the function in the Numerical Preferences option.

The Mixed Method procedure is more accurate but requires more computational resources because there is an additional degree of freedom for every node. This makes the stiffness matrix larger and the solution of the equation more computationally expensive.

In the Penalty Method, if the value of the Penalty function is too large, it might affect the accuracy of the solution process. This is reflected in a singularity ratio that might be extremely small. If the value of the penalty function is too small, the fluid will not satisfy the incompressibility requirement.

Fluid Mechanics

Specifying a Value for Penalty Function

To specify the value of the Penalty function for fluid analysis:

1. Click **JOBS**.
2. Choose a fluid analysis class (e.g., Fluid, Fluid-Thermal, etc.)
3. Choose Job Parameters>Numerical Preferences.

Jobs>(Analysis Class)>Job Parameters>Numerical Preferences

4. Click **FLUID INCOMPRESSIBILITY PENALTY** and specify a value.

About Strongly-Coupled and Weakly-Coupled Parameters

The Strongly-Coupled parameters apply to fluid-thermal and fluid-thermal-solid analyses only. Using these parameters, you can simultaneously solve the velocity of the fluid and the temperature of the fluid. This simultaneous solution also results in an additional degree of freedom per node.

Use the Weakly-Coupled parameters to solve the following in sequence:

- velocity
- temperature of the fluid

The Weakly-Coupled procedure is also known as the “Staggered” procedure.

The Strongly-Coupled procedure is better suited for thermally-driven fluid problems. The Weakly-Coupled procedure might run into stability problems when the temperature changes very quickly resulting in changes in viscosity. A change in viscosity affects the velocity of the fluid which, in turn, affects the temperature of the fluid.

About the Solver Option for Fluid Analysis

Since fluid mechanics involves non-symmetric analysis, the solver options in MARC are fewer compared to a structural-mechanics analysis. MARC features the Direct Profile solver for non-symmetric analyses.

The Hardware Sparse solver is another solver available for fluid analyses on selected platforms. If the Hardware Sparse solver is not available on your platform, MARC defaults to the Direct Profile Solver.

To display the fluid analysis solver options for your platform:

1. Click JOBS.
2. Choose a fluid analysis class.
3. Choose Job Parameters>Solver.

Jobs>(Analysis Class)>Job Parameters>Solver

Valid Element Types

There is no specific menu for fluid element types since you can use the same element types that you use in solid analyses.

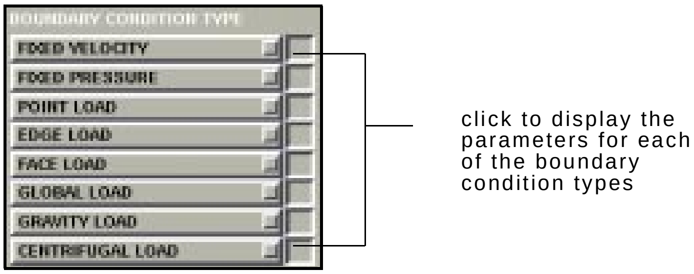
For a table of valid fluid element types, see Appendix B (*p. 293*).

Boundary Conditions for Fluid Analysis

About Boundary Conditions for Fluid Analysis

Use the Boundary Condition menus to define the boundary conditions for the type of your fluid analysis. For example, if you wanted to perform a fluid-thermal analysis, you would not only prescribe boundary conditions for fluids, but also for the thermal component of your analysis.

To display the boundary condition types for your fluid analyses, choose Boundary Conditions>Fluid.

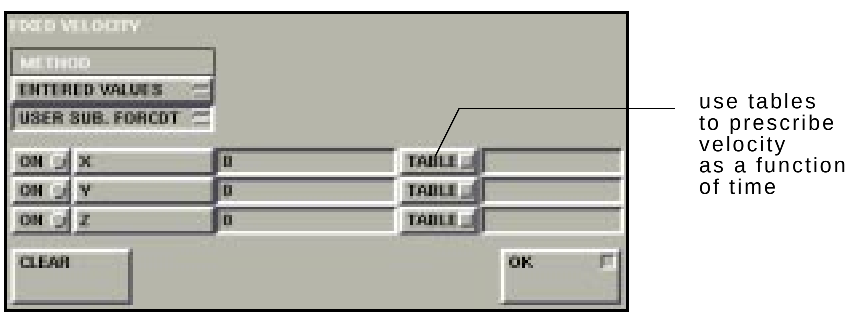


Boundary Conditions>Fluid

About Fixed Velocity

The Fixed Velocity option is the most common boundary condition for fluids. Use this option to prescribe the velocity at the nodal points.

To prescribe the parameters for the Fixed Velocity boundary condition, choose Boundary Conditions>Fluid>Fixed Velocity.



Boundary Conditions>Fluid>Fixed Velocity

To set the parameters, refer to the following table:

If the analysis is for a ...	then...
steady state fluid problem	prescribe the final velocities at the nodal points.
transient state fluids	you can enter a table prescribing how the velocities change with time.
2-D problem	prescribe the X and Y velocities.
3-D problem	prescribe the X, Y, and Z velocities.

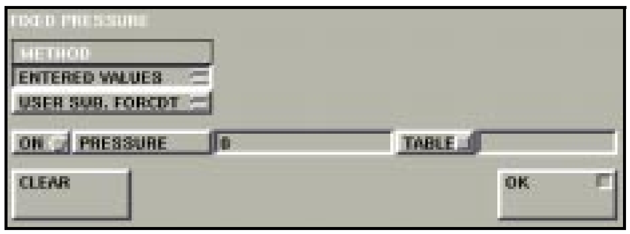
For transient analyses, you can use the user subroutine FORCDT (*See Vol. D: User Subroutines–FORCDT*) or tables to specify the time-dependent velocity.

To prevent the incidence of a single set of equations in a fluid analysis, you should apply some boundary condition to the fluid.

About Fixed Pressure

Use the Fixed Pressure option when you are applying the Mixed Method procedure for fluid analysis (*See "About Mixed and Penalty Method Procedures" on page 223*). The Fixed Pressure option applies a fixed pressure on the nodal points.

To prescribe the parameters for the Fixed Pressure boundary condition, choose Boundary Conditions>Fluid>Fixed Pressure.

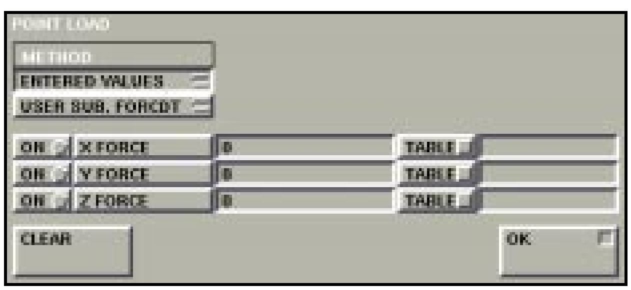


Boundary Conditions>Fluid>Fixed Pressure

About Point Loads

Point loads prescribe a force on a nodal point. In a steady-state analysis, the point load is fixed. For a transient analysis, you can prescribe a table to specify that the load is varying with time.

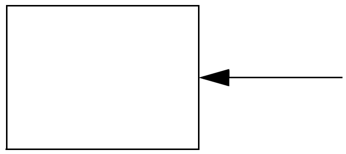
To prescribe the parameters for the Point Load boundary condition, choose Boundary Conditions>Fluid>Point Load.



Boundary Conditions>Fluid>Point Load

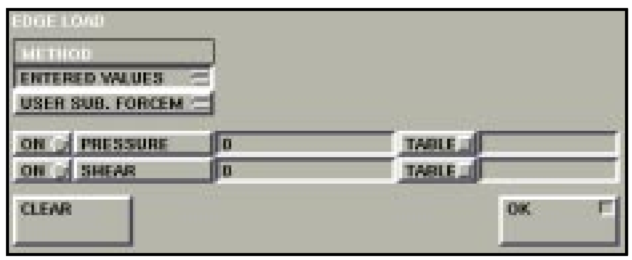
About Edge Loads

Edge loads are distributed loads that act upon the edge of a 2-D planar or axi-symmetric model where you prescribe a load per unit length.



In a steady-state analysis, the edge load is fixed. For a transient analysis, you can prescribe a table to specify that the load is varying with time.

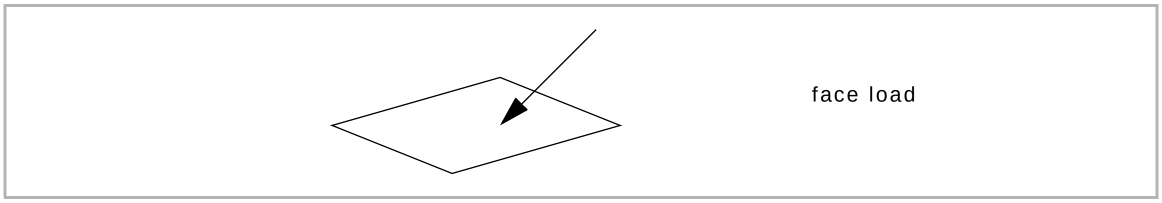
To prescribe the parameters for the Edge Load boundary condition, choose Boundary Conditions>Fluid>Edge Load.



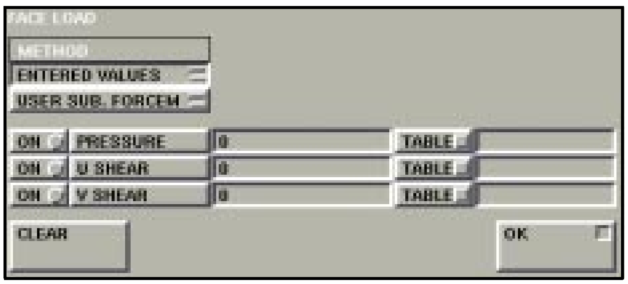
Boundary Conditions>Fluid>Edge Load

About Face Loads

Face loads are loads acting upon a face for a 3-D problem. A positive load implies that the load is directed into the element.



To prescribe the parameters for the Face Load boundary condition, choose Boundary Conditions>Fluid>Face Load.

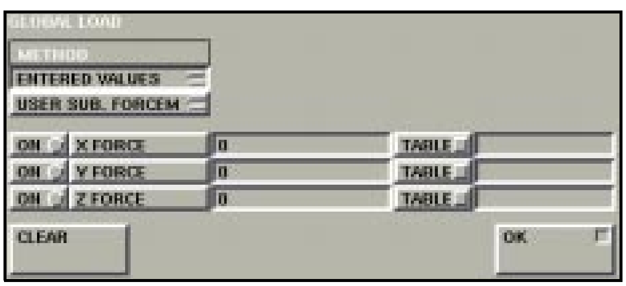


Boundary Conditions>Fluid>Face Load

About Global Loads

Global loads are loads per unit volume.

To prescribe the parameters for the Global Load boundary condition, choose Boundary Conditions>Fluid>Global Load.

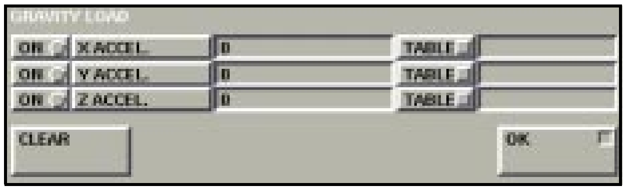


Boundary Conditions>Fluid>Global Load

About Gravity Loads

Gravity loads are loads per unit mass. Generally, this value of the load would be the acceleration due to gravity.

To prescribe the parameters for the Gravity Load boundary condition, choose Boundary Conditions>Fluid>Gravity Load.

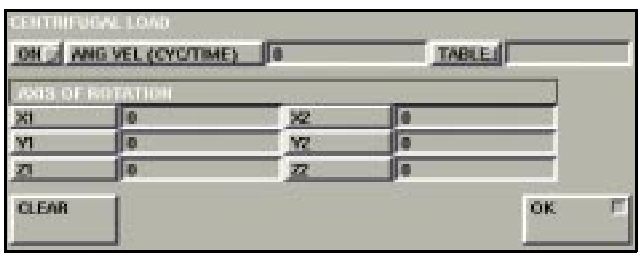


Boundary Conditions>Fluid>Gravity Load

About Centrifugal Loads

You can apply centrifugal loads by specifying the angular velocity and the axis of rotation of your analysis model. You should also prescribe the two points that define the rotational axis.

To prescribe the parameters for the Centrifugal Load boundary condition, choose Boundary Conditions>Fluid>Centrifugal Load.



Boundary Conditions>Fluid>Centrifugal Load

Initial Conditions for Fluid Analysis

About Types of Initial Conditions

There are three types of initial conditions that you can specify in a fluid analysis:

- Mechanical
- Thermal
- Fluid

To display the types of initial conditions in **MENTAT**, choose Initial Conditions in Main Menu.

Initial Conditions

Depending on the type of analysis, you may require initial conditions for one or all of the types. For example, if it is a purely fluid analysis, you may need initial conditions for fluids only.

If you perform a steady state analysis, the initial conditions are not as relevant as they would be in a transient state analysis.

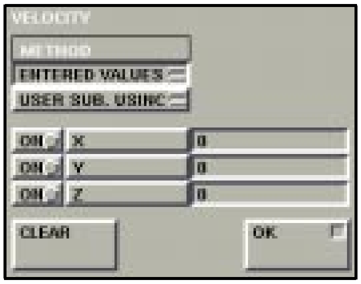
> **Picking good initial conditions (conditions that are close to the final solution) can result in improved convergence in the numerical analysis. However, they will not influence the final results.**

About Velocity as Initial Condition Type

For a purely fluid analysis, the initial condition associated with the fluid is its velocity. You prescribe the velocity at the nodal points in the model.

To prescribe velocity as an initial condition:

1. Choose Initial Conditions>Fluid.
2. Click **VELOCITY**.



Initial Conditions>Fluid>Velocity

Coupled Analysis Considerations

For fluid-solid analysis in steady state, you should prescribe the solid behavior as linear elastic. If any material or geometric nonlinearity occurs in the solid, then you have to take an incremental procedure for the solid and perform a complete transient analysis.

In a fluid-solid analysis, you also have to specify a convergence testing associated with fluid and solid region. For the solid region, you specify the residual or displacement parameters.

About Limitations on Large Displacements for Solids

There are limits to the amount of displacement that can occur in solids. The motion of the free surface of a fluid or the motion of the solid is limited to small displacements. Large displacements in solids might imply that the fluid is having large motion which requires a new mesh to be placed in the fluid region—a capability not included in this version of **MARC**.

Fluid Mechanics

Material Properties for Fluid Analysis

About Material Properties in a Fluid Analysis

The fluid region consists of one type of material and you can perform single phase fluid analysis with no mixing in MARC.

Depending on the type of your fluid analysis (*See "About Fluid Analysis" on page 221*), you may have to prescribe more than one type of material property. Use the following table to assign the material properties for different types of fluid analyses.

For the analysis type,	choose...
Fluid	FLUIDS and assign all the fluid properties (*See "About Fluid Properties and Viscosity Models" on page 235*).
Fluid-Thermal	FLUIDS and HEAT TRANSFER for those regions that are fluids. There should be at least two material types.
Fluid-Solid	FLUIDS for the fluid region and MECHANICAL for the solid region.
Fluid-Thermal-Solid	FLUIDS and HEAT TRANSFER for the fluid region; MECHANICAL and THERMAL for the solid region.

About Fluid Properties and Viscosity Models

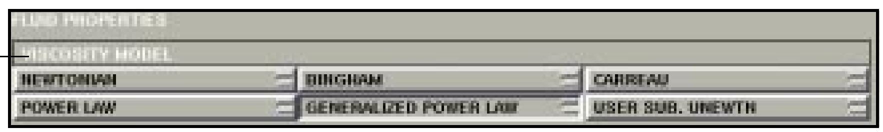
Use the Fluid Properties menus to assign the fluid properties for an analysis. Since the material is incompressible, you need to define the viscosity and the mass density of the fluid. To neglect the inertia effects, specify a small value for the mass density.

The fluid models featured in **MARC** include:

- Newtonian
- Bingham
- Carreau
- Power Law
- Generalized Power Law
- User-Defined

To display the Fluid Properties menus, choose Material Properties>Fluid.

choose a viscosity model or the user subroutine, UNEWTN



Material Properties>Fluid

Use the following table to define the parameters for each of the viscosity models:

For...	define...
Newtonian fluids	the viscosity or a table that represents the temperature effects on viscosity.
Bingham fluids	the viscosity and the following tables: • temperature-dependent behavior • rate (strain rate) behavior as given by the relation: $\tau_{ij} = \mu_0 d_{ij} + g D^{-1} d_{ij}$ if $\tau \geq g$ $d_{ij} = 0$ if $\tau < g$
Carreau model fluids	The values of K, D, and n as given by the relation: $\mu = \mu_\infty + (\mu_0 - \mu_\infty)(1 + K^2 D^2)^{(n-1)/2}$
Power Law fluids	K, n, and D as given by the relation: $\tau_{ij} = \mu_0 K D^{n-1} d_{ij}$
Generalized Power Law fluids	K, η, D, A1, A2, A3, A4 as given by the relation: $\mu = \begin{cases} \mu_0 K \exp(A_1 T + A_2 T^2) D_0^P & D < D_0 \\ \mu_0 K \exp(A_1 T + A_2 T^2) D^P & D \geq D_0 \end{cases}$ where $P = n - 1 + A_3 \ln D + A_4 T$

You can use the subroutine, UNEWTN, to define the viscosity at a particular spatial location. For more information, see Vol. D: UNEWTN: Input of Viscosity in Flow Analysis

Mass Density and Volumetric Expansion

You must enter the mass density for fluids and, if it is a fluid-thermal analysis, the volumetric expansion coefficient. When you enter a value for the volumetric expansion, the fluid is allowed to change its volume owing to a change in temperature based upon the temperature coefficient.

The MARC models of fluids assume that the mass density is constant and thus exclude gases.

Loadcases for Fluid Analysis

About Loadcases for Fluids

You can choose the following loadcase classes depending on the type of fluid analysis in **MENTAT**:

- Fluid
- Fluid-Thermal
- Fluid-Solid
- Fluid-Thermal-Solid

For each of the above loadcase classes, you can specify the following loadcase types:

- Steady State—you define the tolerance value to control convergence and accuracy
- Transient—you define the tolerance value to control convergence and accuracy, the time steps, and the total loadcase time.

The parameters and menus for the Fluid loadcase classes are similar to Heat Transfer parameters and menus (*See Mentat Command Reference Manual, Chapter 11: Analysis-Loadcases*).

To display the Loadcase Type menu:

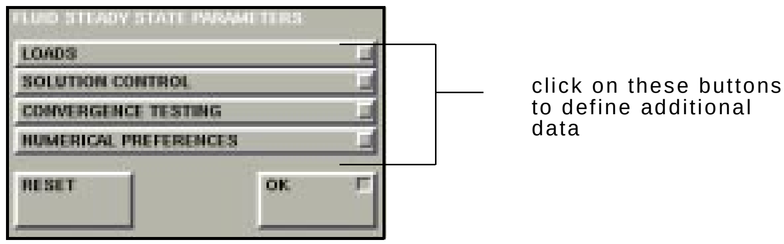
1. Click **LOADCASE** in Main Menu.
2. Choose a loadcase class (e.g., Fluid, Fluid-Solid).

Loadcase>(Loadcase Class)

Applying the Fluid Steady State Parameters

To apply the fluid steady state parameters:

1. Choose **LOADCASE** in Main Menu.
2. Choose a loadcase class (e.g., Fluid, Fluid-Solid).
3. Choose **STEADY STATE**.

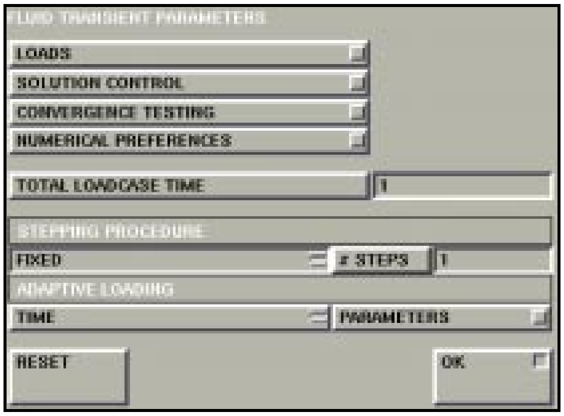


Loadcase>(Loadcase Class)>Steady State

Applying the Fluid Transient Parameters

To apply the transient parameters for a fluid analysis:

1. Choose **LOADCASE** in Main Menu.
2. Choose a loadcase class (e.g., Fluid, Fluid-Solid).
3. Choose **TRANSIENT**.



Loadcase>(Loadcase Class)>Transient

4. Click **TOTAL LOADCASE TIME** and specify the total loadcase time period.
5. Choose fixed or adaptive time steps.

Since fluid analyses is a highly nonlinear phenomenon, you might need to take small steps to achieve convergence for a transient analysis.

About Iterative Procedures in Solution Control

You can apply either of the iterative procedures for solution control in **MARC**:

- Full Newton-Raphson

  At each iteration of the Newton-Raphson procedure, you solve:

$$O(X)\delta X^{i+1} = R^{i}$$

$$X^{i+1} = X^{i} + \delta X^{i+1}$$

  where X^i is the solution at the i^{th} iteration, and R^i is the error in the solution or residual. Each iteration results in the change δX^{i+1} in the solution.

- Direct Substitution

  For the Direct Substitution procedure, you solve:

$$O(X^{i})X^{i+1} = F$$

  where the force remains for all iterations, and the operator matrix is obtained from the last solution of X.

When the Full Newton-Raphson procedure converges, it converges faster. However, the Direct Substitution procedure is often numerically more stable.

About Control Tolerances

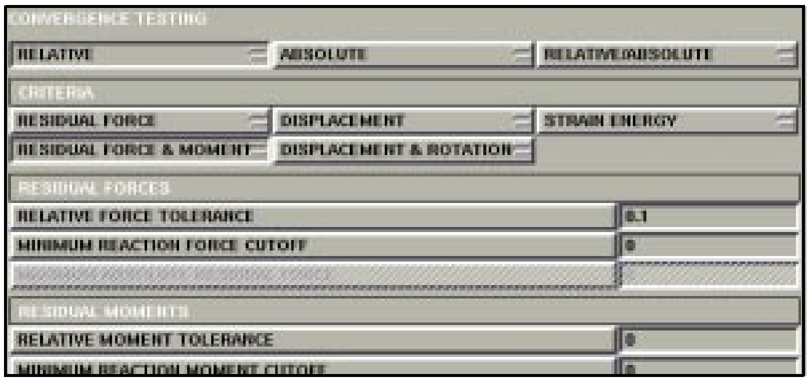
When you perform a coupled analysis with either the Strongly-Coupled or Weakly-Coupled method you should specify two separate control tolerances for the:

- velocity
- temperature change in the fluid

Viewing the Convergence Testing Parameters

To view the convergence testing parameters for the solid region:

1. Choose **LOADCASES** in Main Menu.
2. Choose either **FLUID-SOLID** or **FLUID-THERMAL-SOLID**.
3. Choose **TRANSIENT**.
4. Choose **CONVERGENCE TESTING SOLID REGION**.



Loadcases>Fluid-Solid or Fluid-Thermal-Solid>Transient>
Convergence Testing Solid Region

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
theoretical description of the Fluid capability and material models for fluids	*6: Non-Structural Procedure Library*	Vol. A
description of the parameter, FLUID	*2:Parameters*	Vol. C
description of the model definitions: • REGION • FIXED VELOCITY • ISOTROPIC • STRAIN RATE • TEMPERATURE EFFECTS • INITIAL VELOCITY • INITIAL TEMPERATURE	*3:Model Definition Options*	Vol. C
description of the user subroutine, UNEWTN	*3:User-Defined Anisotropy and Constitutive Relation Subroutines*	Vol. D
demonstration problems	*9*	Vol. E

10 • Material Modeling

Using the Narayanaswamy Model

New Approaches in Plasticity Modeling

Mooney-Rivlin and Ogden Formulations

User Subroutines in Hypoelastic Properties

Experimental Data Fitting

• Material Modeling

Using the Narayanaswamy Model

About Thermo-Rheologically Simple Materials and the Narayanaswamy Model

There is a large class of polymers adequately represented by linear viscoelastic laws at uniform temperature. With a change in temperature, these polymers exhibit an approximate translational shift of all the characteristic response functions along a logarithmic time axis. The shift occurs without a change of shape and these temperature-sensitive viscoelastic materials are termed, Thermo-Rheologically Simple (*See Vol. A, Chap. 7: Thermo-Rheologically Simple Behavior; Narayanaswamy Model*)

You can apply the Narayanaswamy model to define the thermo-rheologically simple properties of glass-type materials.

Mechanical Material Types for the Narayanaswamy Model

Use the Narayanaswamy Model in conjunction with the following mechanical material types:

- Isotropic
- Orthotropic
- Mooney
- Ogden

Parameters for the Narayanaswamy Model

Here are the parameters associated with the Narayanaswamy model:

- Activation energy/Gas const (H/R)—(*See Vol. A, Chap. 7: Narayanaswamy Model*)
- Glass Transition Temperature (T_g)—the temperature below which the material behaves as a solid and above which the material is a liquid.

Material Modeling

- Fraction Parameter—enables you to apply a mixture of fictive and real temperatures and control the participation of the fictive temperature.
- Absolute Temperature Shift—the shift between the temperatures that you are using in your model and absolute temperature; if you are using temperature in Rankine or Kelvin, then the shift would be equal to zero.

Before you apply Narayanaswamy model parameters for your finite element model, you should define the model geometry.

Applying Narayanaswamy Model Parameters

To apply the Narayanaswamy model parameters:

1. Click **MATERIAL PROPERTIES** in the main menu.
2. Choose a mechanical material type.
3. Choose **THERMAL EXP.** in the properties window.
4. Choose **VISCOELASTIC** and enter the liquid and solid coefficients of thermal expansion.

For your options, refer to the following table:

If the thermal expansion property is...	then...
Isotropic	1. Type in a value for the thermal expansion coefficient. 2. Click on **TABLE** to associate a table with this property. You would typically use tables here to provide temperature-dependent data. 3. Click **OK**.
Viscoelastic	1. Type in a value for the coefficient of thermal expansion in the liquid and solid states. 2. Click **OK**.

5. Choose **RATE EFFECTS>VISCOELASTIC** in the properties menu.
6. Choose **THERMO-RHEOLOGICALLY SIMPLE** in the viscoelastic properties window.
7. Click the **NARAYANASWAMY** shift function.

8. Type in the values for the parameters.
9. Click **OK**.
10. Choose **OK** in the **VISCOELASTIC PROPERTIES** window.
11. Choose **OK** in the **MECHANICAL MATERIAL TYPE** window.
12. Add the material property to the desired elements.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
definitions of buttons	*7, Material Properties*	Mentat 3.1 Command Reference
theoretical description and background information	*7, Material Library*	Vol. A
description of the following options: • SHIFT FUNCTION • VISCEL EXP	*3, Model Definition Options*	Vol. C
user demonstration problem	E 7.x-32	Vol. E

New Approaches in Plasticity Modeling

Multiplicative Decomposition and New Plasticity Procedures

There are five different plasticity procedures in MARC:

1. Small Strain–Mean Normal
2. Small Strain–Radial Return
3. Large Strain–Mean Normal–Additive Decomposition
4. Large Strain–Radial Return–Additive Decomposition
5. Large Strain–Radial Return–Multiplicative Decomposition

The difference between the first two procedures is the numerical implementation and determination of the normal to the yield surface. The Mean-Normal procedure is a more stable procedure, but it usually requires more iterations in the analysis process. The Radial Return procedure is less stable but in theory, it converges faster.

A new fifth procedure uses the multiplicative decomposition of the deformation gradient. This model enables you to take large increments of strain with greater accuracy and better convergence.

Selecting a Plasticity Procedure

To select a plasticity procedure:

1. Click **JOBS** in Main Menu.
2. Choose a relevant analysis class (e.g., Mechanical).
3. Click **ANALYSIS OPTIONS** in Analysis Class.
4. Choose a plasticity procedure or click again to choose the next procedure in the list of five procedures.

click again to display the next procedure

Jobs>(Analysis Class)>Analysis Options

Considerations for Elastic Data

When you define the material relationships for these plasticity models, the Young's modulus and the Poisson's ratio is always the same regardless of which plasticity procedure that you choose.

Considerations for Work-Hardening Data

In the case of the work-hardening data, when you define a table of the work-hardening/strain hardening information, here are some important considerations:

When you use the...	enter the data in terms of...
small-strain formulation	engineering stress and strain
large-strain formulation	Cauchy stress and logarithmic or true strain

The data that you enter is independent of the decomposition approach, multiplicative or additive, that you use.

Element Considerations

When a material is in the fully plastic range, the material is almost completely incompressible. This has a consequence on the type of elements that you can use to achieve an accurate solution.

When you use the Large Strain-Mean Normal-Additive Decomposition or the Large Strain-Radial Return-Additive Decomposition, MARC automatically invokes the Constant Dilatation procedure for lower order elements (types 3, 7, 10, 11, 19, 20). This improves the accuracy of the analysis.

When you use the Large Strain-Radial Return-Multiplicative Decomposition procedure, MARC uses an alternative, three-field variational approach that is applicable to most displacement-based elements.

For problems involving large strain, avoid using three-node triangular elements or four-node tetrahedral elements.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
theoretical concepts,	*5, Structural Procedure Library* and *7, Material Library*	Vol. A
element considerations,	*1, Introduction*	Vol. B
description of the option, PLASTICITY	*3, Model Definition*	Vol. C
demonstration problems,	E 3x-19, E 3x-21, E 3x-33, E 3x-34, E 3x-35, E 3x-36, E 3x-37, E 3x-38, E 8x-12, E 8x-15, E 8x-16, E 8x-17 E 8x-18, E 8x-60	Vol. E

Mooney-Rivlin and Ogden Formulations

About the Updated Lagrange Procedure

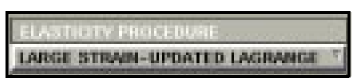
In addition to the total Lagrange procedure, you can now use the Updated Lagrange procedure for analyses that include Mooney-Rivlin or Ogden material models. When you employ the Updated Lagrange procedure for rubber formulation, you no longer need to use Herrmann elements to model the incompressibility of the rubber materials.

The Updated Lagrange procedure enables you to use the rubber materials with explicit dynamics and since the number of degrees of freedom is less in the model, your analysis is computationally more efficient. Also, since no Herrmann elements are required, the numerical procedure associated in solving a linear equation is more stable when you apply the iterative solvers for rubber analysis.

So for Mooney-Rivlin and Ogden formulations, you now have a choice of the Updated Lagrange or Total Lagrange procedure.

To select the Updated Lagrange procedure:

1. Choose Jobs>Mechanical>Analysis Options.



Jobs>(Analysis Class)>Analysis Options

2. Click the Elasticity Procedure cycle button and choose

   LARGE-STRAIN UPDATED LAGRANGE.

Considerations for Mooney-Rivlin and Ogden Formulations

If you had a region in your model that was a Mooney-Rivlin type material and another material that was not Mooney Rivlin, then the large-strain elasticity formulation works on the region with the Mooney-Rivlin type material only.

The Updated Elasticity Lagrange procedure works for the Mooney-Rivlin materials and Ogden materials only.

Material Modeling

You can define the Mooney-Rivlin material or the Ogden material regardless of which formulation (Total or Updated Lagrange) that you chose. The material constants that you enter are the same.

You can apply the rubber damage models to your Mooney-Rivlin or Ogden formulation when using either the Total Lagrange or Updated Lagrange procedure.

You can use the large-strain viscoelastic model with the total Lagrange formulation only.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
theoretical concepts	5, *Structural Procedure Library* and 7, *Material Library*	Vol. A
element considerations	1, *Introduction*	Vol. B
description of the parameter, ELASTICITY	2, *Parameters*	Vol. C
description of the options: • MOONEY • OGDEN	3, *Model Definition Options*	Vol. C
demonstration problems	E 7x-12, E 7x-14, E 7x-20, E 7x-28, E 7x-29, E 7x-30, E 7x-31, E 7x-43	Vol. E

User Subroutines in Hypoelastic Properties

About User Subroutines in Hypoelastic Properties

When you now specify the parameters for hypoelastic material properties, you have the choice of three user subroutines:

- HYPELA
- HYPELA2
- UBEAM

Use HYPELA and HYPELA2 for the following elements:

- continuum elements
- shell elements
- all beam elements except beam elements, 52 and 98

For beam elements 52 and 98, use subroutine, UBEAM (*See Vol. D: User Subroutines–UBEAM*).

About HYPELA and HYPELA2

HYPELA and HYPELA2 are two user subroutines that you can use with the Hypoelastic option. HYPELA is typically used for elasticity applications. You can use HYPELA2 for modeling arbitrary nonlinear material behavior including elastoplasticity.

For both user subroutines, you should define L and g (where L is a function of the mechanical strain and g is a function of the temperature) such that:

$$\dot{\sigma}_{ij} = L_{ijkl}\dot{\varepsilon}_{kl} + g_{ij}$$

HYPELA2 conveys more information about the kinematics of deformation (i.e., the deformation gradient; stretch ratios: rotation tensor). This enables you to account for rigid rotations while modeling material behavior. Also, you can calculate any kinematic terms that you need to add to the tangent.

In HYPELA2, you obtain the information about the current deformation gradient (F) which is decomposed, using polar decomposition, into rotation tensor (R) and stretch tensor(U), as given by the relation:

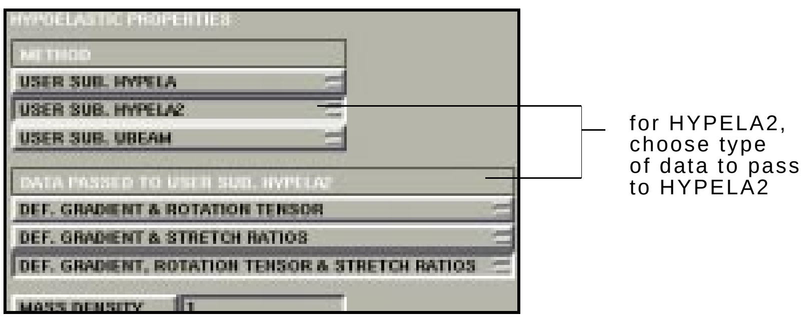
$$\underline{F} = \underline{R}\,\underline{U}$$

For reasons of maintaining backward compatibility, subroutine HYPELA is still supported in MARC. You do not need to change HYPELA.

Choosing a User Subroutine for Hypoelastic Properties

To select a user subroutine for hypoelastic properties:

1. Choose Material Properties>Hypoelastic.



Material Properties>Hypoelastic

2. Choose a user subroutine.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
constitutive theory and equations	*7, Material Library: Nonlinear Hypoelastic Material*	Vol. A
description of the option, HYPOELASTIC	*3, Model Definition Options*	Vol. C
description of the user routines and parameters	*3, User-defined Anisotropy and Constitutive Relations Subroutines*	Vol. D

Experimental Data Fitting

About Experimental Data Fitting

The Experimental Data Fitting feature enables you to extract the material constants for a variety of different constitutive laws from data that was experimentally obtained.

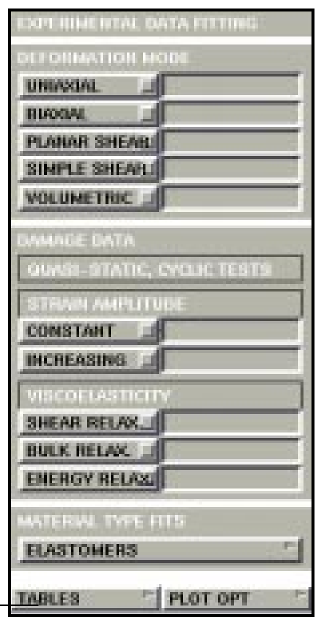
For an elastomer analysis, you can use the Experimental Data Fitting feature to calculate the following:

- for rate-independent behavior, the material data for

  elastomers for the rate-dependent, viscoelastic data.
- material data for damage data.

To view the experimental data fitting options, choose Material Properties>Experimental Data Fitting.

use tables to specify the variation of the material data with temperature, plastic strain, strain rate, and relative density



Material Properties>Experimental Data Fitting

You may want to perform several experimental tests before you extract the coefficients for a rubber material.

About Experimental Tests

The Experimental Data Fitting menus feature the following deformation modes for calibrating quasi-static stress-strain response curves (*See also Mentat Command Reference, Chap. 7: Experimental Data Fitting*):

- uniaxial
- biaxial
- planar shear (or pure shear)
- simple shear
- volumetric

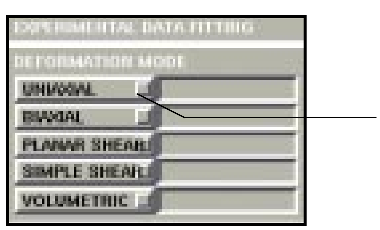
While it is possible to use just uniaxial data for curve fitting and extract the constants for elastomeric material models, you should also have experimental data for multiple types of experiments. This enables you to achieve material constants that can be used in practical engineering analysis. You cannot use shear experimental data for obtaining constants for the Foam model.

Reading in Experimental Data

Using the Experimental Data Fitting menus, you can read in your experimental data in a tabular form containing the engineering strain (ε) and the engineering stress (σ) measures. The Foam model requires dilatational information (*See "About Forms of Data" on page 262*). The experimental data is read in as ε, σ, J (Jacobian of deformation).

To read in your experimental data:

1. Choose Material Properties>Experimental Data Fitting.



Material Properties>Experimental Data Fitting

2. Choose a deformation mode (e.g., biaxial).
3. Click **TABLE** and choose a currently-defined table.

The steps described above ensure that the table is associated with the biaxial test.

Check to ensure that the table format is "RAW."

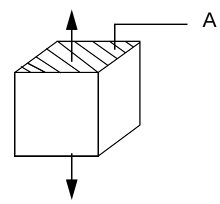
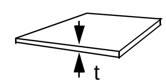
About Forms of Data

The forms of data in the tables is described here:

Category	Form of data
Deformation Mode	ε, σ, (J)
Constant Strain Amplitude	ε_{max}, $\sigma\|_{\varepsilon = fixed}$
Increasing Strain Amplitude	cycle count, $\sigma\|_{\varepsilon = fixed}$ equispaced
Relaxation	time, σ equispaced

Although your data set may have three components, you'll see only the X and Y components in Mentat.

For Foam models, the Z-component is the dilatational information, (J). The relation between the deformation mode and form of J is given below:

Deformation	J	where:
uniaxial	A/A_0	**A** is the cross-sectional area: 
biaxial	t/t_0	**t** is the thickness of the sheet: 

Types of Data Fitting Models

After reading in the experimental data, use the Material Models menu to select the type of material model.

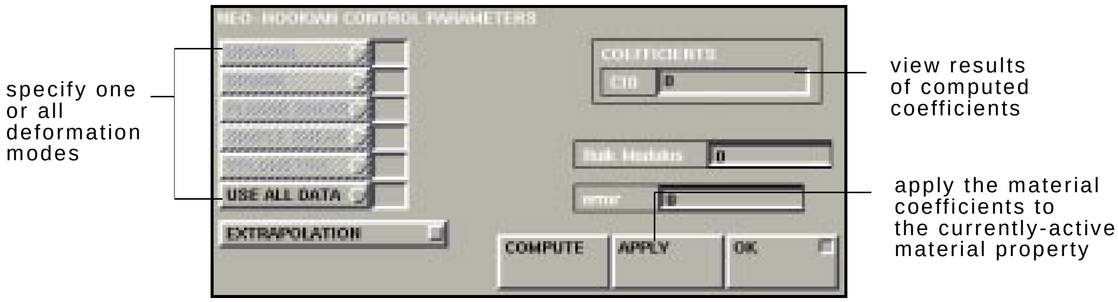
To display the Material Models menu, choose Material Properties>Experimental Data Fitting>Elastomers.

click on a model for which you wish to extract the material coefficients

Material Properties>Experimental Data Fitting>Elastomers

Determining the Material Coefficients

After specifying the deformation mode(s) and the type of model, use the control parameter windows to compute the material coefficients. You can use the material coefficients with either the total, or the updated, Lagrange procedure.



Material Properties>Experimental Data Fitting>Mooney(2)

To compute the material coefficients in the control parameters window:

1. Choose one or more deformation modes.
2. Click **COMPUTE**.

If you gave the data for more than one deformation mode (e.g., uniaxial and biaxial), you can choose to derive the coefficients based on any subset of the experimental data by picking the relevant deformation modes in the Experimental Data Fitting Menu.

Before performing multiple fits in one session, be sure to deselect deformation modes between models.

Applying Material Coefficients to the Currently-Active Property

You must press the **APPLY** button to apply the material coefficients to your model. The material coefficients are then applied to the currently-active material property (e.g. Mooney). You would still need to use the **ADD ELEMENTS** button in the Material Properties menu to associate part of the model to that particular material.

MARC does not require a bulk modulus for the Mooney-Rivlin material models. You do not need to supply a volumetric test for these models.

Deciding Which Modes to Use

After you load the tables for the individual deformation modes, you can decide, using the control parameter windows, which data to use with respect to the deformation modes.

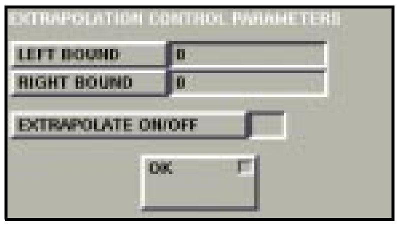
Use the following table to determine which deformation modes (or their combinations) to use:

To load data for...	In the control parameters windows, click...
one deformation mode (e.g., uniaxial)	the button corresponding to the deformation mode (e.g., **UNIAXIAL**)
all four deformation modes	**USE ALL DATA**

About the Extrapolation Feature

Use the Extrapolation feature to extrapolate the computed response curves to the regions beyond the experimental data that you defined. You can apply this feature to observe the material behavior for regions outside the user-defined experimental data.

Use the Extrapolation Control Parameters windows to specify the lower and the upper bounds of the new strain levels.



Material Properties>Experimental Data Fitting>
Control Parameters>Extrapolation

If you set the bounds such that they are within the range of the data set, the bounds are ignored.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
definitions of buttons	7, *Material Properties*	Mentat 3.1 Command Reference
description of material models	7, *Material Library*	Vol. A
description of the following options: • MOONEY • OGDEN • FOAM • VISCEL PROP	3, *Model Definition Options*	Vol. C

Damage Models

About the Continuous and Discontinuous Damage Models

In addition to the existing Discontinuous damage model in MARC, you can now use a Continuous damage model for elastomers.

Use the Continuous damage model in MARC to calibrate the continuous damage terms. The model uses the “constant strain amplitude” data set.

The existing Discontinuous Damage model uses the “increasing strain amplitude” data set.

Setting Damage Control Parameters

After loading the appropriate damage data set, you can use the Damage Control Parameters window to compute the Kachanov Factor coefficients.

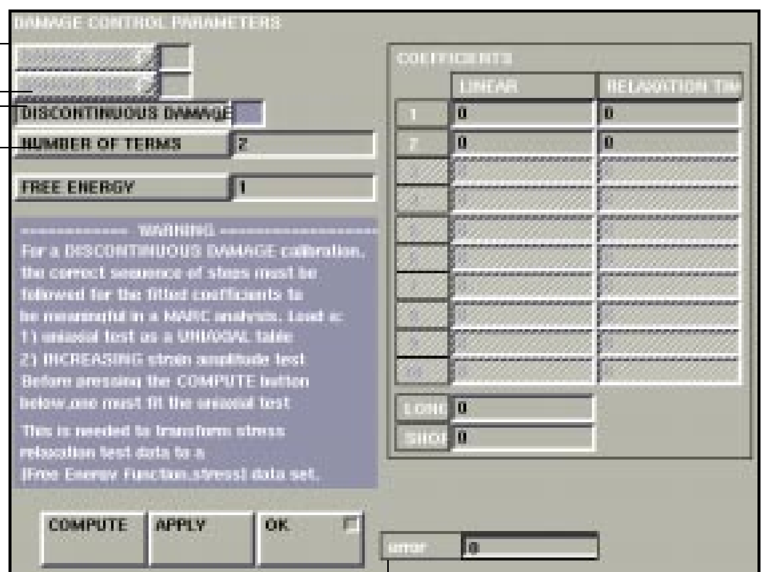
To display the Damage Control parameters, choose Material Properties>Experimental Data Fitting>Elastomers>Damage.

load a relevant table to activate the continuous or discontinuous damage model button

set the Discontinuous Damage button to “ON” position

recommended number of terms is “2”

use the error entry to obtain the difference between the input and the predicted curve



Material Properties>Elastomer>Damage

Considerations for Data Sets

You must calibrate a constitutive model from a uniaxial, biaxial or planar-shear data set in the step immediately before setting the control parameters. The program needs this sequence of actions to convert the table, $(\varepsilon_i^{max}, \sigma_i)$ to a table, (W_i, σ_i).

The transformation to the table, (W_i, σ_i) results in a non-equispaced data set. The program automatically smoothens this set into an equispaced set.

Checklist for Using the Continuous Damage Model

When using the Continuous Damage model, you should:

- load a table such that the **DAMAGE CON** button is activated.
- specify the free energy value to correspond to the point, $(\varepsilon_{fix}, \sigma_0)$.
- set the **DISCONTINUOUS DAMAGE** button to "Off" position.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
definitions of buttons	*7, Material Properties*	Mentat 3.1 Command Reference
background information	*7, Material Properties*	Vol. A
description of the option, DAMAGE	*3, Model Definition Options*	Vol. C
demonstration problem	E 7x-30	Vol. E

Viscoelasticity Model

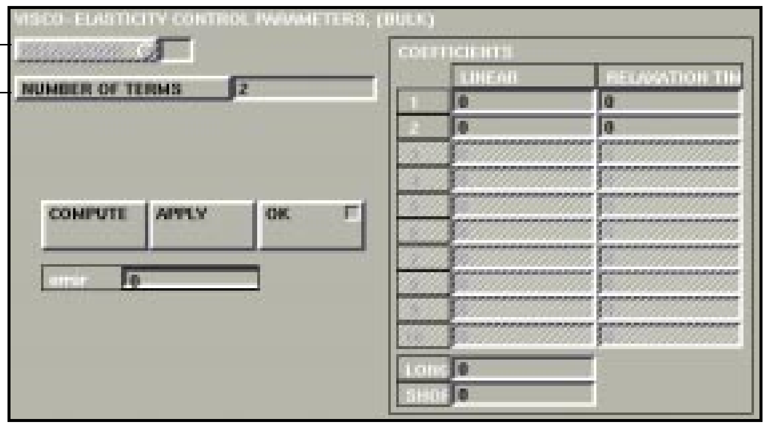
About the Viscoelasticity Model and Relaxation Spectra

After you load the appropriate stress relaxation data set, you can use the viscoelasticity control parameters to compute the shear, bulk, or energy relaxation spectra.

To display the viscoelasticity control parameters for a bulk test, choose Material Properties> Experimental Data Fitting> Elastomers> Bulk Relax.

load a relevant stress relaxation table to activate the RELAXATION button

recommended number of terms is “2”



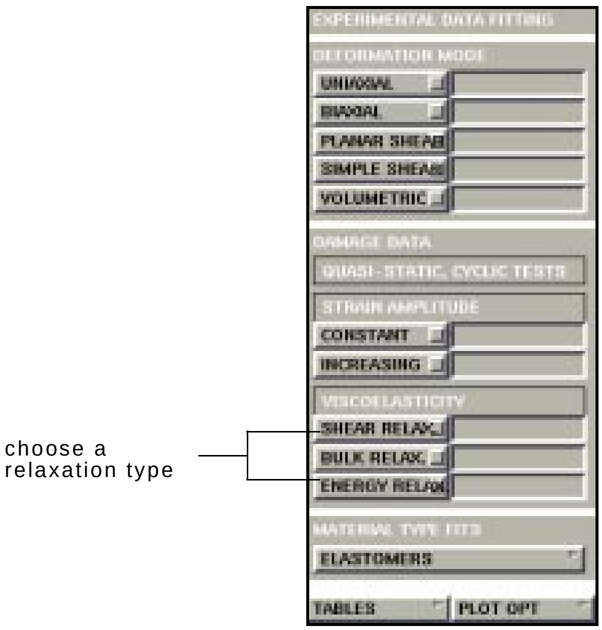
Material Properties>Experimental Data Fitting>Bulk Relax.

Using a Relaxation Table for Viscoelasticity Control Parameters

Use a relaxation table to enter the appropriate stress relaxation data set.

To select a relaxation table:

1. Choose Material Properties>Experimental Data Fitting.



Material Properties>Experimental Data Fitting

2. Choose a button for the relaxation type (e.g., **SHEAR RELAX.**) and select a table from the currently-defined tables which contain datasets from the respective test (e.g., shear-relaxation test).

Computing the Coefficients for Shear, or Bulk Relaxation Tests

Use the Viscoelasticity Control Parameters window to compute the coefficients for shear, or bulk relaxation tests.

To compute the coefficients:

1. Choose Material Properties>Experimental Data Fitting>Elastomers.
2. Choose either the **SHEAR RELAX.** or the **BULK RELAX.** button.
3. Click **COMPUTE**.

Determining the Coefficients for an Energy Relaxation Test

Use the Viscoelasticity Control Parameters (Free Energy) window to compute the coefficients for an energy relaxation test.

To compute the coefficients:

1. Choose Material Properties>Experimental Data Fitting>Elastomers.
2. Click **ENERGY RELAX.**
3. Click **COMPUTE**.

Considerations for Data Sets

You must calibrate a constitutive model from a uniaxial test in the step immediately before setting the control parameters. The program requires this sequence of actions to convert the table, $(\mathrm{time}_i, \sigma_i)$ to a table, (time_i, W_i) where $i = 1, 2, 3, \ldots N$.

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)...	in...
definitions of buttons	*7, Material Properties*	Mentat 3.1 Command Reference
background information	*7, Material Library*	Vol. A
description of the options: • VISCELPROP • VISCELORTH • VISCELMOON • VISCELOGDEN	*3, Model Definition Options*	Vol. C
demonstration problems	E 7x-12, E 7x-14, E 7x-18, E 7x-22	Vol. E

11 • Analysis Integration

Using Radiation Viewfactors

- # Analysis Integration

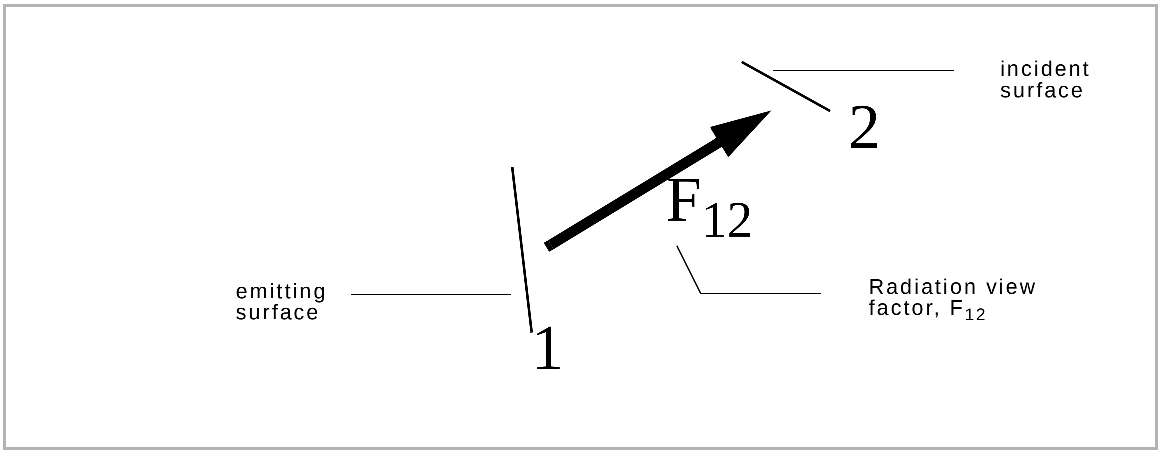
Using Radiation Viewfactors

About the Radiation View Factor Feature

To achieve accurate solutions of many heat transfer problems, you need to include the effects of radiative heat exchange. The Radiation Viewfactor feature in MENTAT computes the viewfactors that you can apply to the analysis of heat transfer problems. This feature also includes, automatically, the effects of shadowing for 2-D, 3-D, and axisymmetric geometries.

About Viewfactors and Computation

The radiation viewfactor is the fraction of radiation leaving the emitting surface that reaches an incident surface and depends solely on the geometric configuration of the surfaces.

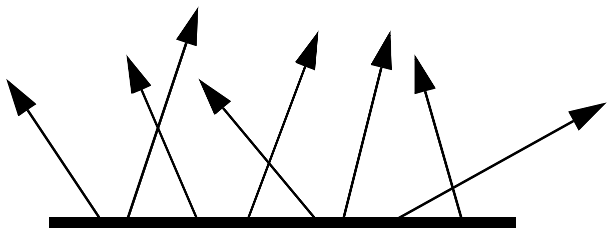


The computation of viewfactors in MARC uses the Monte Carlo method. This method handles the shadowing effects very efficiently and is superior to integration techniques in terms of accuracy and cost of computation.

Analysis Integration

About Distribution of Rays

Rays are randomly emitted from each surface in the analysis. Typically, these are faces of elements (3-D), or edges of elements (2-D or axisymmetric). While the origins of each of the rays must be uniformly distributed over the areas of the entire surface, the direction of rays are random distributed



Rays are randomly emitted from each surface

Applying Radiation As Boundary Conditions

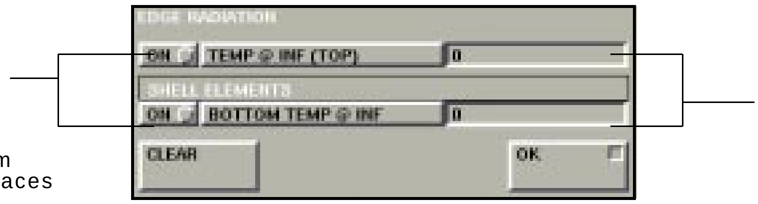
For an analysis using radiation, apply the boundary conditions on all element edges (2-D, axisymmetric) or element faces (3-D) involved in the radiation heat transfer computation.

Use the boundary condition parameters to control the emission of radiation. The two boundary condition parameters for edges and faces are:

- Temperature at Infinity (Top)—if the radiation is emitted from a surface and does not hit any other surface or body, it has to exchange heat with a background temperature located at infinity.

- Bottom Temperature at Infinity—for shells, the top and the bottom edges or surfaces are treated separately. You can include the top and bottom of a shell element in your analysis.

click ON to include both top and bottom edges or faces for shell elements



set temperature to emit to the background

Boundary Conditions>Thermal>More>Edge Radiation

To apply radiation as a boundary condition:

1. Choose Boundary Conditions>Thermal>More.
2. Choose **EDGE RADIATION** (for 2-D, axisymmetric) or **FACE RADIATION** (for 3-D).
3. Enter the temperatures at infinity for top or bottom depending on the element that you choose.

About Emitting and Absorbing Roles

All bodies could potentially be emitting radiation to other bodies and receiving (or absorbing) radiation from other bodies. When you set the boundary conditions, all the edges or faces that you chose become objects and they play both emitting and absorbing roles in a heat transfer analysis.

About Emitting and Incident Objects

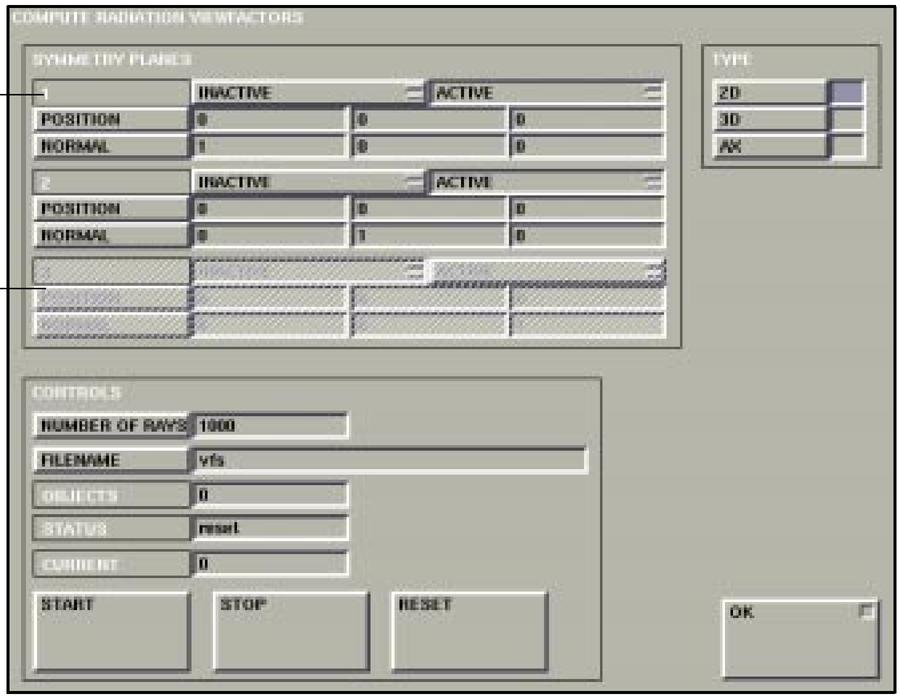
In the computation of radiation view factors, rays are cast at random from each emitting object. The default number of rays emitted for an object is 1000. For each ray emitted, **MARC** determines the first object (i.e., incident object) that the ray hits. The viewfactor is computed by summing up the number of rays hitting each object and dividing by the total number of rays cast.

About the Compute Radiation Viewfactors Window

Use the Compute Radiation Viewfactors window to specify the parameters for the computation of viewfactors.

To specify the parameters, choose Jobs>Radiation Viewfactors.

number of planes vary with your choice of geometry type



Jobs>Radiation Viewfactors

About the Parameters and Related Functions

Use the following table to locate the functions of selected parameters in the Compute Radiation Viewfactors window:

To...	use the parameter/ button,...	Additional Information
specify the geometric type of the computation	TYPE	2-D and axisymmetric geometries require element edges; 3-D geometries require element faces.
activate or deactivate symmetry plane(s) individually	SYMMETRY PLANES; ACTIVE; INACTIVE	Symmetry planes are defined by a position on the plane and a direction perpendicular to the plane. You can activate up to three planes.
specify the number of rays emitted	NUMBER OF RAYS	Larger numbers of rays increase the accuracy of viewfactor computation.
start or restart the viewfactor computation	START	

Analysis Integration

To...	use the parameter/ button,...	Additional Information
stop the computation and reset the controls to the beginning	**RESET**	
stop the computation	**STOP**	To restart the computation without losing the previously-computed results, press **START**.
specify the file to send the viewfactors to	**FILENAME**	This file is used by **MARC** during heat transfer analysis.

Before you submit an analysis job in MARC that involves the effects of radiation heat transfer, you should compute the radiation viewfactors and save it in a file.

Computing Viewfactors in Analysis

To compute the radiation viewfactors for existing radiation boundary conditions:

1. Choose Jobs>Radiation Viewfactors.
2. Choose the type (geometry) based on whether you set edges (2-D; axisymmetric) or faces (3-D) for your radiation boundary conditions.

> **If you defined edges only and you click 3-D as your choice of TYPE, the number of objects in the OBJECTS field will be zero.**

3. If you wish to use symmetry planes, click on **ACTIVE** in Symmetry Plane to activate the plane(s); use the **NORMAL** and **POSITION** buttons to define the plane(s).
4. Click **FILENAME** and specify a name of the file to store your results in.
5. Specify the number of emitted rays to be used for each element edge or face.
6. Click **START**.

Indicating the Location of the Viewfactor File

When you submit the heat transfer analysis through **MENTAT**, the location of the viewfactor file is provided to **MARC**.

When you use the **MARC** shell script to run the analysis, use the -vf filename option to indicate the location of the viewfactor file.

Analysis Integration

Additional Information

Use the following table to find more information:

For...	refer to Chapter(s)/ Section(s)...	in...
the description and functions of the buttons	*12, Jobs/ Radiation View Factors*	Mentat 3.1 Command Reference
description of the RADIATION parameter	*2: Parameters*	Vol. C
description of the VIEWFACTOR option	*3: Model Definition Options*	Vol. C

Appendixes

- **Appendixes, A-F**

A: User Enhancements

List of User Enhancements

• Appendix A: User Enhancements

List of User Enhancements

MARC New Parameters

These are the new parameters (*See Volume C: Program Input–Chap. 2*) in MARC:

- Constant Dilatation (define that elements are to use constant dilatation formulation)
- Assumed Strain (improved bending behavior)
- Fluid (for fluid, fluid-thermal, fluid-solid, and fluid-thermal-solid analysis)
- Plasticity (plasticity procedure)
- Elasticity (elasticity procedure)
- Design Sensitivity (perform sensitivity analysis only)
- Design Optimization (perform design optimization)
- Extended (extended precision of reading in data)

MARC New Model Definition

These are the new model definitions (*See Volume C: Program Input–Chap. 3*) in MARC:

- Region (electromagnetic–define elements in a region)
- Fixed Velocity (define fixed velocity)
- Viscoelastic Exp (viscoelastic thermal expansion)
- Spline (model definition–analytical surface used to represent a deformable body)
- Exclude (model definition–ignore contact with certain regions)
- Design Objective (define objective function to optimize)
- Design Variable (define variable design parameters)
- Design Stress Constraint (define limits on stress response)
- Design Strain Constraint (define limits on strain response)

- Design Frequency Constraint (define limits on eigenfrequency response)
- Design Displacement Constraint (define limits on displacement response)

MARC New History Definition

These are the new history definitions (*See Volume C, Chapter 4–History Definition: Static, Dynamic, Creep Analysis.*) in MARC:

- Auto Step (adaptive load step control)
- Approach (move rigid surfaces into position)
- Synchronize (move rigid surfaces into position)

New User Subroutines

Here are the new user subroutines (*See Volume D: User Subroutines*) in MARC:

- UPSTRECH (definition of generalized principal stretch-based elasticity models: *See Chap. 3*)
- HYPELA2 (user-defined material behavior: *See Chap. 3*)
- UPOSTV (user-selected post-processing of nodal variables: *See Chap. 7*)
- UFINITE (finite deformation isotropic material models: *See Chap. 3*)

New Element Types

Here are the new element types (*See Volume B: Element Library–Chap. 3*) available in MARC:

- 138 (bilinear thin-triangular shell element)
- 139 (bilinear thin-shell Element)
- 140 (bilinear thick-shell element with reduced integration)
- 142 (eight-node axisymmetric rebar element with twist)

- 143 (four-node plane strain rebar element)
- 144 (four-node axisymmetric rebar element)
- 145 (four-node axisymmetric rebar element with twist)
- 146 (three-dimensional eight-node rebar element)
- 147 (four-node rebar membrane)
- 148 (eight-node rebar membrane)

B: Fluid Elements

Fluid Element Types

• Fluid Elements

Fluid Element Types

You can represent the fluid region in analysis using the following element types:

Planar Element Types

Nodes	Element	Type
3-node	linear	3
4-node	isoparametric bilinear	11
4-node	isoparametric bilinear reduced integration	115
6-node	isoparametric triangle	125
8-node	isoparametric biquadratic	27
8-node	isoparametric biquadratic reduced integration	54

Axisymmetric Element Types

Nodes	Element	Type
3-node	linear	2
4-node	isoparametric bilinear	10
4-node	isoparametric bilinear reduced integration	116
6-node	isoparametric triangle	126
8-node	isoparametric biquadratic	28
8-node	isoparametric biquadratic reduced integration	55

Three-Dimensional Element Types

Nodes	Element	Type
4-node	tetrahedron	134
8-node	trilinear brick	7
8-node	trilinear brick with reduced integration	117
20-node	brick	21
20-node	brick with reduced integration	57

C: MARC Data Reader Support

Supported Parameters

Supported Model-Definition Options

- **MARC Data Reader Support**

Supported Parameters

List of Supported Parameters

These are the parameters that the MARC Reader currently supports (for information on parameters, see *Volume C: Program Input, Chapter 2–Parameters*):

- $NO LIST
- ACOUSTIC
- ALIAS
- ALL POINTS
- ASSUMED STRAIN
- BEARING
- BUCKLE
- CENTROID
- CONSTANT DILATATION
- COUPLE
- CREEP
- DIST LOADS
- DYNAMIC
- ELASTIC
- ELASTICITY
- ELECTRO
- ELEMENTS
- EL-MA
- END
- EXTENDED
- FINITE
- FLUID

List of Supported Parameters (Contd.)

- FLUXES
- FOLLOW FOR
- HARMONIC
- HEAT
- INPUT TAPE
- JOULE
- LARGE DISP
- LOAD COR
- LUMP
- MAGNETO
- NEW
- NO LOADCOR
- OLD
- PLASTICITY
- PORE
- PRINT
- PROCESSOR
- RADIATION
- RESTRICTOR
- R-P FLOW
- SETNAME
- SHELL SECT
- SIZING
- STATE VARS
- TIE
- TITLE
- TSHEAR
- UPDATE

Supported Model-Definition Options

List of Supported Model-Definition Options

These are the model-definition options that the MARC Reader currently supports (for information on options, see *Volume C: Program Input, Chapter3–Model Definition Options*):

- ANISOTROPIC (Mechanical)
- ANISOTROPIC (Thermal)
- COMPOSITE
- CONN FILL
- CONN GENE
- CONNECTIVITY
- CONTACT (2D)
- CONTACT (3D)
- CONTACT NODE
- CONTACT TABLE
- COORDINATES
- CRACK DATA
- CREEP
- CYLINDRICAL
- DEFINE (Sets)
- DIST CHARGE (Electromagnetic)
- DIST CHARGES (Electrostatic)
- DIST CURRENT (Electromagnetic)
- DIST CURRENT (Joule Heating)
- DIST CURRENT
- DIST FLUXES
- DIST LOADS
- DIST SOURCES (Model Definition)

List of Supported Model-Definition Options

- END OPTION
- FILMS (Model Definition)
- FIXED DISP (Fluid)
- FIXED DISP
- FIXED POTENTIAL (Electromagnetic)
- FIXED POTENTIAL (Electrostatic)
- FIXED POTENTIAL (Magnetostatic)
- FIXED PRESSURE
- FIXED TEMPERATURE
- FIXED VELOCITY
- FOAM
- FOUNDATION
- FXORD
- GEOMETRY
- HYPOELASTIC
- INITIAL DISP
- INITIAL VEL
- ISOTROPIC (Acoustic)
- ISOTROPIC (Electromagnetic)
- ISOTROPIC (Electrostatic)
- ISOTROPIC (Fluid)
- ISOTROPIC (Heat Transfer)
- ISOTROPIC (Hydrodynamic)
- ISOTROPIC (Magnetostatic)
- ISOTROPIC (Rigid-Plastic)
- ISOTROPIC (Stress)
- JOULE
- MASSES

List of Supported Model-Definition Options

- MERGE
- MOONEY
- NODE CIRCLE
- NODE FILL
- NODE GENER
- NODE MERGE
- OGDEN
- OPTIMIZE
- ORIENTATION
- ORTHO TEMP (Structural)
- ORTHO TEMP (Thermal)
- ORTHOTROPIC (Electrical)
- ORTHOTROPIC (Electromagnetic)
- ORTHOTROPIC (Magnetostatic)
- ORTHOTROPIC (Mechanical)
- ORTHOTROPIC (Thermal)
- PARAMETERS
- POINT CHARGE
- POINT CURRENT (Joule)
- POINT CURRENT (Magnetostatic)
- POINT CURRENT-CHARGE
- POINT FLUX
- POINT LOAD
- POINT SOURCE
- POINT TEMP
- POWDER
- RESTART LAST
- RESTART

List of Supported Model-Definition Options

- ROTATION A
- SOIL
- SOLVER
- SPLINE
- SPRINGS
- STRAIN RATE
- STRAIN RATE
- TEMPERATURE EFFECTS (Coupled Fluid-Thermal)
- TEMPERATURE EFFECTS (Coupled Thermal-Stress)
- TEMPERATURE EFFECTS (Heat Transfer)
- TEMPERATURE EFFECTS (Hydrodynamic)
- TEMPERATURE EFFECTS (Stress)
- TRANSFORMATION
- TYING
- UDUMP
- UFCONN
- UFRICTION
- UFXORD
- UHTCOEF
- UHTCON
- UMOTION
- VIEW FACTOR
- VOLTAGE
- WORK HARD

D: NASTRAN Writer Data Entries

NASTRAN Writer Bulk Data Entries

The NASTRAN bulk data file export writes the following bulk data entries from the current Mentat database. The data apply to structural analyses only.

- **NASTRAN Writer Bulk Data Entries**

Minimum File Requirements

Minimum File Requirements

Use the following table to determine the positions and corresponding valid entries for data files:

Position	Valid Entry
At the start of the bulk data	`BEGIN BULK`
At the end of the bulk data	`END DATA`

D: NASTRAN Writer Data Entries

Finite Element Connectivity

Bulk Data Cards Written by Mentat 3.3

These are the bulk data cards written by MENTAT:

Mentat Element/ Class	NASTRAN Bulk Data Entry	Notes
LINE (2)	CBAR	lacks orientation information
LINE (3)	CBEND	
TRIA (3)	CTRLA3	
TRIA (6)	CTRLA6	
QUAD (4)	CQUAD4	
QUAD (6)	CQUAD6	
QUAD (8)	CQUAD8	
QUAD (9)	CQUAD	
TETRA (4)	CTETRA	
TETRA (10)	CTETRA	
PENTA (6)	CPENTA	
PENTA (15)	CPENTA	
HEX (8)	CHEXA	
HEX (12)	CHEXA	
HEX (20)	CHEXA	
HEX (27)	–	no mapping

Node Coordinates/Transformations

Node Coordinates / Transformations

These are the corresponding data entries for node coordinates and transformations:

<table>
<tr><th>Mentat</th><th>NASTRAN</th><th>Notes</th></tr>
<tr><td>node coordinates</td><td><code>GRID</code></td><td rowspan="2">Transformations become coordinate systems and their ID’s are included in each <code>GRID</code> card.</td></tr>
<tr><td>transformations</td><td><code>CORD2R</code></td></tr>
</table>

D: NASTRAN Writer Data Entries

Materials and Geometry

Material ID's Included in NASTRAN Geometry Cards

These are the material ID's included in NASTRAN geometry cards.

Mentat	NASTRAN	Notes
Isotropic	MAT1	no table support; no plasticity
Orthotropic	MAT3	no table support; no plasticity
Anisotropic	MAT9	no table support; no plasticity
3-D GEOMETRIES		
Truss	PBAR	
Elastic Beam	PBAR	
General Beam	PTUBE	only circular X-section supported
Membrane	PSHELL	
Shear Panel	PSHELL	
Shell	PSHELL	
Solid	PSOLID	

Mentat	NASTRAN	Notes
2-D GEOMETRIES		
Plane Strain	PSHELL	
Plane Stress	PSHELL	

Loads and Boundary Conditions

Loads and Boundary Conditions

These are the corresponding NASTRAN data entries for loads and boundary conditions:

Mentat	NASTRAN	Notes
FIXED DISPLACEMENT	`SPC`	one card written per degree of freedom
POINT LOAD	`FORCE`	
POINT LOAD	`MOMENT`	
EDGE LOAD	`PLOAD1`	one card written per component (for beams only)
FACE LOAD	`PLOAD`	only normal pressures written
GRAVITY LOAD	`GRAV`	

Sample File

Sample NASTRAN Bulk Data File

```
$
$ NASTRAN Bulk Data File created by Mentat II=================================
$
BEGIN BULK
$
$ ELEMENT CONNECTIVITY=========================================================
$
CTETRA  12      8       126     127     125     117
$
$ TRANSFORMATIONS==============================================================
$
CORD2R* 1               0                               -2.200000       0.000000
*              -4.400000       -2.200000        4.919350       -4.400000
*              -2.200000        0.000000       -9.319350
$
$ NODAL COORDINATES============================================================
$
GRID*   10                                              -1.400000       -1.400000
*               0.0000000
GRID*   20                                              -0.200000       -2.000000
*               0.0000000
GRID*   117                                             -2.200000       -1.000000
*              -3.2000000
GRID*   125                                             -2.200000        0.000000
*              -4.4000001
GRID*   126                                             -2.200000       -1.000000
*              -4.4000000
GRID*   127                                             -1.300000       -1.000000
*              -4.4000000
$
$ MATERIALS====================================================================
$
MAT1*   1                      29.000000                                0.300000
*               1.000000        0.000000
$
$ GEOMETRY=====================================================================
$
PSOLID* 8               1
*
$
$ BOUNDARY CONDITIONS==========================================================
$
FORCE*  1               125             0                               1.000000
*              10.000000       10.000000        0.000000
MOMENT* 1               125             0                               1.000000
*              20.000000        0.000000        0.000000
PLOAD*  4                      10.000000117             125
*       127
GRAV*   5               0                        1.000000       0.320000
*               0.100000        0.100000
$
$ SERVO========================================================================
$
MPC     4       10      1          1.00020      0          0.001
ENDDATA
```

E: Command Line Parameters

Mentat 3.3 Command Line Parameters

MARC K7.3 Command Line Parameters

- # Command Line Parameters

Mentat 3.3 Command Line Parameters

Table of Additional Command Line Parameters

Use this table to browse the command line parameters that are in addition to the parameters described in Chapter 2, Getting Started (*See "New Command Line Parameters Featured in Mentat 3.3" on page 25*):

Operation: To specify...	use the command line parameter...	Option
aspect ratio of the screen that Mentat uses ($0<frac\leq1$)	-ar	*frac*
path to menu directory (defaults to standard location)	-mp	*"menu"*
path to help directory (defaults to standard location)	-hp	*"help"*
path to directory containing executables (defaults to standard location)	-bp	*"bin"*
font name	-fn	*name*
size of window	-size	*xs, ys*
procedure file to run when execution begins	-pr	*procname*
path to the directory containing material database	-ml	*mat*
treat pop-up menus as independent menus,(*See "Setting Stand-Alone Static Menus" on page 30*)	-df	STATIC_STANDALONE
read all the menu files at startup,	-ra	

Operation: To specify...	use the command line parameter...	Option
bitplane threshold	-ic	*number (6)*
annotate string to X-Windows file,	-ti	*string*
run Mentat without ACIS solids modeling code,	-nosolids	

MARC K7.3 Command Line Parameters

Table of MARC K7.3 Command Line Parameters

For a description of the command line parameters in the MARC K7.3 shell script, refer to the following table:

Parameter	Possible Names or Values	Description
-prog	marck7	run MARC with or without user subroutines; Run saved module "progname.marc"
-jid	jidname	identify job/input file
-rid	ridname	identify previous job for RESTART.
-pid	pidname	identify job that created temperature file.
-sid	sidname	identify database file for substructures.
-user	username	user subroutine username.f used to generate a new load module.
-back	yes (default)	run program in background.
	no	Run program in foreground.
-ver	yes (default)	verify the input syntax.
	no	don't verify the input syntax.
-save	yes	save the generated or existing module.
	no (default)	don't save the generated or existing module.
-vf	viewname	identify viewfactor file.
-def	def	default input file name.

F: Demonstration Problems

Demonstration Problems

- **MARC K7:1–Demonstration Problems**

Demonstration Problems

Lists of Demonstration Problems in MARC K7.3

This section lists selected demonstration problems for the following options in MARC:

Constant Dilatation

- *e3x21f.dat*–Necking of a cylindrical bar in tension

Design Sensitivity

- *e10x1a.dat*–Design sensitivity of the beam geometric properties of an alternator mount using element 52
- *e10x2a.dat*–Design sensitivity of the thickness for plane stress hole in plate problem
- *e10x3a.dat*–Design sensitivity of elastic material properties for the bending of a plate using element type 21
- *e10x4a.dat*–Design optimization of the thickness for Scordelis-Lo roof using element type 75
- *e10x5a.dat*–Design sensitivity of ply angle lay-up for simple composite plate
- *e10x6a.dat*–Design sensitivity of 10 bar truss problem
- *e10x7a.dat*–Design sensitivity of the beam geometric properties of an alternator mount using element 14

Design Optimization

- *e10x1b.dat*–Design optimization of the beam geometric properties of an alternator mount using element 52
- *e10x2b.dat*–Design optimization of the thickness for plane stress hole in plate problem

- *e10x3b.dat*–Design optimization of elastic material properties for the bending of a plate using element type 21
- *e10x4b.dat*–Design optimization of the thickness for a Scordelis-Lo roof
- *e10x5b.dat*–Design optimization of ply angle lay-up for simple composite plate
- *e10x6b.dat*–Design optimization of 10 bar truss problem
- *e10x7b.dat*–Design optimization of the beam geometric properties of an alternator mount using element 14

Elasticity

- *e7x20.dat*–Compression of O-ring using 3-term OGDEN model
- *e7x20b.dat*–Compression of O-ring using 3-term OGDEN model (includes follower force stiffness)
- *e7x20c.dat*–Compression of O-ring using 3-term OGDEN model (Adaptive Meshing)
- *e7x20d.dat*–O-ring, using Updated Lagrange procedure - element 10
- *e7x20e.dat*–O-ring, using Updated Lagrange procedure - element 116
- *e7x27.dat*–Twist of a tapered rod, using OGDEN in Updated Lagrange procedure
- *e7x28a.dat*–Expansion of a cylindrical tube, using OGDEN in Updated Lagrange element type 10
- *e7x28b.dat*–Expansion of a cylindrical tube, using OGDEN in Updated Lagrange element type 116

- *e7x28c.dat*–Expansion of a cylindrical tube, using OGDEN in Updated Lagrange element type 28
- *e7x28d.dat*–Expansion of a cylindrical tube, using OGDEN in Updated Lagrange element type 55
- *e7x29b.dat*–Hole in a plate, OGDEN in Updated Lagrange procedure - element type 117
- *e7x29_def.dat*–Defaults file for problems e7x29a and e7x29b
- *e7x30a.dat*–Damage model - using discontinuous model
- *e7x30b.dat*–Damage model - using continuous model
- *e7x31a.dat*–Rubber seal problem using Updated Lagrange procedure
- *e7x31b.dat*–Rezoning of Rubber seal problem using Updated Lagrange procedure
- *e8x43b.dat*–Rubber Seal - ELASTICITY,2, - element 10
- *e8x43c.dat*–Rubber Seal - ELASTICITY,2, - element 116

Plasticity

- *e3x19.dat*–Upsetting problem using FeFp procedure
- *e3x21a.dat*–Necking of a cylindrical bar in tension
- *e3x21d.dat*–Necking of a cylindrical bar in tension (Adaptive Meshing)
- *e3x21e.dat*–Necking of a bar using FeFp procedure
- *e3x33.dat*–Hole in plate - FeFp model - plane stress
- *e3x33b.dat*–Hole in plate - FeFp model - plane strain
- *e3x34.dat*–Cylinder expansion using membrane elements - FeFp model
- *e3x35.dat*–Cantilever beam - plane strain - FeFp procedure

- *e3x36.dat*–Hole in plate - brick elements - FeFp procedure
- *e3x37a.dat*–Elastic Deformation in a closed loop - Jaumann rates
- *e3x37b.dat*–Elastic Deformation in a closed loop - FeFp
- *e3x38a.dat*–Verify rotational invariance - tension w/rigid rotation - FeFp plane stress
- *e3x38b.dat*–Verify rotational invarience - tension w/rigid rotation - FeFp plane strain
- *e8x12d.dat*–Bolt forging - FeFp - AUTO STEP, no rezoning
- *e8x12dr.dat*–Bolt forging - FeFp - AUTO STEP, restart of e8x12d.dat - Rezoning
- *e8x12e.dat*–Bolt forging - FeFp - AUTO STEP - Adaptive
- *e8x15d.dat*–Double-sided contact - FeFp - AUTO STEP
- *e8x16b.dat*–Hook with release - FeFp - AUTO STEP - Release (Gradual method)
- *e8x17b.dat*–Extrusion - FeFp - AUTO STEP
- *e8x18c.dat*–3-D Forming of a circular blank using membrane elements - FeFp model
- *e8x19b.dat*–Indentation rolling - FeFp
- *e8x60.dat*–Plane strain Brake Forming

Extended

- *e8x45.dat*–Test of SPLINE option and EXTENDED - concentric cylinders
- *e8x46.dat*–Test of EXCLUDE - simple problem

Buckle (Lanczos)

- *e3x16b.dat*–Plastic buckling using Lanczos procedure
- *e4x1d.dat*–Shell buckling using Lanczos procedure

- *e4x4b.dat*–Buckling of a cylindrical tube using Lanczos procedure
- *e4x9b.dat*–Non-symmetric buckling of a ring using Lanczos procedure
- *e4x10b.dat*–Nonsymmetric buckling of a cylinder using Lanczos procedure
- *e4x15.dat*–Buckling of a cylinder tube using Lanczos procedure

Design Displacement Constraints

- *e10x1a.dat*–Design sensitivity of the beam geometric properties of an alternator mount using element 52
- *e10x1b.dat*–Design optimization of the beam geometric properties of an alternator mount using element 52
- *e10x3b.dat*–Design optimization of elastic material properties for the bending of a plate using element type 21
- *e10x5a.dat*–Design sensitivity of ply angle lay-up for simple composite plate
- *e10x5b.dat*–Design optimization of ply angle lay-up for simple composite plate
- *e10x7a.dat*–Design sensitivity of the beam geometric properties of an alternator mount using element 14
- *e10x7b.dat*–Design optimization of the beam geometric properties of an alternator mount using element 14

Design Frequency Constraints

- *e10x1a.dat*–Design sensitivity of the beam geometric properties of an alternator mount using element 52

- *e10x1b.dat*–Design optimization of the beam geometric properties of an alternator mount using element 52
- *e10x3a.dat*–Design sensitivity of elastic material properties for the bending of a plate using element type 21
- *e10x3b.dat*–Design optimization of elastic material properties for the bending of a plate using element type 21
- *e10x4a.dat*–Design optimization of the thickness for Scordelis-Lo roof using element type 75
- *e10x4b.dat*–design optimization of the thickness for a Scordelis-Lo roof
- *e10x7a.dat*–Design sensitivity of the beam geometric properties of an alternator mount using element 14
- *e10x7b.dat*–Design optimization of the beam geometric properties of an alternator mount using element 14

Design Strain Constraints

- *e10x2a.dat*–Design sensitivity of the thickness for plane stress hole in plate problem
- *e10x2b.dat*–Design optimization of the thickness for plane stress hole in plate problem
- *e10x5a.dat*–Design sensitivity of ply angle lay-up for simple composite plate
- *e10x5b.dat*–Design optimization of ply angle lay-up for simple composite plate

Design Stress Constraints

- *e10x7a.dat*–Design sensitivity of the beam geometric properties of an alternator mount using element 14

- *e10x7b.dat*–Design optimization of the beam geometric properties of an alternator mount using element 14

Exclude

- *e8x46.dat*–Test of EXCLUDE - simple problem

Parameters

- *e2x18.dat*–Elastic analysis of shell roof with element 22-quadratic thick shell element
- *e3x14a.dat*–Pre-bending of a prismatic beam
- *e3x23.dat*–Shell roof with geometric and material nonlinearity-element 75 AUTO INCREMENT
- *e4x5.dat*–Large displacement analysis of pinched cylinder
- *e7x3.dat*–Shell roof with material nonlinearity (cracking)-element 75
- *e8x18.dat*–Sheet forming of a circular blank with Coulomb friction
- *e8x18d*–3-D Forming of a circular blank using AUTO STEP
- *e9x7a.dat*–Natural convection in a square cavity (Ra:1000)
- *e9x7b.dat*–Natural convection in a square cavity (Ra:10,000)
- *e9x7c.dat*–Natural convection in a square cavity (Ra:100,000)

Rebar

- *e2x37b.dat*–Reinforced concrete beam - elements 11 & 143
- *e4x13a.dat*–Test of rebar elements type 142
- *e4x13b.dat*–Test of rebar elements type 144
- *e4x13c.dat*–Test of rebar elements type 145

- *e4x14a.dat*–Analysis of a thin cylinder with ply- element 18 & 147
- *e4x14b.dat*–Analysis of a thin cylinder with plys - element 30 & 148

Region

- *e9x10a.dat*–Fluid-solid - mixed approach
- *e9x10b.dat*–Fluid-solid - penalty approach
- *e9x10c.dat*–Fluid-thermal-solid - mixed approach
- *e9x10d.dat*–Fluid-thermal-solid - penalty approach
- *e9x11a.dat*–Fluid flow over a deformable cylinder
- *e9x11b.dat*–Transient analysis of fluid flow over a deformable cylinder
- *e9x12c.dat*–Fluid flow over a deformable cartridge and thermal effects
- *e9x13a.dat*–Fluid flow in a deformable tube
- *e9x13b.dat*–Fluid flow in a deformable tube

Spline

- *e8x45.dat*–Test of SPLINE option and EXTENDED - concentric cylinders

Viewfactors

- *e5x15b.dat*–Radiation analysis using Mentat calculated viewfactor file

Viscoelastic Exp.

- *e7x32.dat*–Structural relaxation of a glass cube

Adaptive (Merge)

- *e8x44c.dat*–Rolling example with ADAPTIVE, allow element merging

Damage

- *e7x30a.dat*–Damage model - using discontinuous model
- *e7x30b.dat*–Damage model - using continuous model

Hypoelastic (HYPELA2)

- *e7x29a.dat*–Hole in a plate, OGDEN in Updated Lagrange procedure - element type 7

Auto Step

- *e3x12b.dat*–Analysis of creep ring-element 10
- *e3x21f.dat*–necking of a cylindrical bar in tension
- *e3x33.dat*–Hole in plate - FeFp model - plane stress
- *e4x7c.dat*–deep arch analysis
- *e5x8e.dat*–Nonlinear heat conduction with AUTO STEP
- *e7x3b.dat*–Nonlinear heat conduction without AUTO STEP
- *e7x4b.dat*–Side pressing of a Rubber Hollow Cylinder using
- *e8x12d.dat*–Bolt forging - FeFp - no rezoning
- *e8x12e.dat*–Bolt forging - FeFp - Adaptive
- *e8x13b.dat*–Coupled analysis of ring compression
- *e8x15d.dat*–Double-sided contact - FeFp
- *e8x16b.dat*–Hook with release - FeFp - Release (Gradual method)
- *e8x17b.dat*–Extrusion - FeFp
- *e8x18d.dat*–Sheet forming with spherical punch

Shell to Shell Contact

- *e8x51.dat*–Test of shell-shell contact for a spring problem
- *e8x51b.dat*–Test of shell-shell contact, and no increment splitting for a spring problem
- *e8x52.dat*–Test of shell-shell contact - stiff hemisphere - contacts plate
- *e8x53a.dat*–Test of shell contact
- *e8x53b.dat*–Test of shell contact
- *e8x54.dat*–Test of shell contact

Stick to Shell Contact

- *e8x47.dat*–Test of stick-slip model - def - rigid
- *e8x48.dat*–Test of stick-slip model - def-def
- *e8x49.dat*–Test of stick-slip friction model - rubber problem
- *e8x50.dat*–Test of stick-slip friction model - upsetting problem

No Splitting

- *e8x15c.dat*–Double-sided contact - using no splitting procedure

Load-Controlled Dies

- *e8x55b.dat*–Deep drawing of a copper sheet - loads prescribed
- *e8x56b.dat*–Pressing of a piece - loads prescribed

New Elements

- *e2x72.dat*–Scordelis-Lo roof - element 138
- *e2x74.dat*–Scordelis-Lo roof - element 138

- *e2x75.dat*–Pinched cylinder - element 138
- *e4x2c.dat*–Square plate under load - element type 138
- *e8x57b.dat*–Adaptive meshing of simply supported square plate with point load element type 138
- *e2x73.dat*–Scordelis-Lo roof - element 139
- *e2x76.dat*–Pinched cylinder - element 139
- *e4x2d.dat*–Square plate under load - element type 139
- *e8x57c.dat*–Adaptive meshing of simply supported square plate with point load element type 139
- *e2x77.dat*–Pinched cylinder - element 140
- *e4x2e.dat*–Square plate under load - element type 140
- *e8x57d.dat*–Adaptive meshing of simply supported square plate with point load-element type 140
- *e4x13a.dat*–Test of rebar elements type 142
- *e2x37b.dat*–Reinforced concrete beam - elements 11 & 143
- *e4x13b.dat*–Test of rebar elements type 144
- *e4x13c.dat*–Test of rebar elements type 145
- *e2x14b.dat*–Reinforced concrete beam - elements 7 & 146
- *e4x14a.dat*–Analysis of a thin cylinder with plys - element 18 & 147
- *e4x14b.dat*–Analysis of a thin cylinder with plys - element 30 & 148

Index

A

B

C

D

Index

E

F

G

Index

K

L

M

N

O

P

R

S

Index

W

X

Y

Z

Analysis Research Corporation

Document Title: **Mentat 3.3-MARC K7.3:New Features**
Part Number: UG-3012-01
Revision Date: August, 1998

UNITED STATES

CALIFORNIA - Palo Alto
MARC Corporate Headquarters
260 Sheridan Avenue, Suite 309
Palo Alto, CA 94306, USA
Tel: 1 650 329 6800
Fax: 1 650 323 5892
Email: support@marc.com

CALIFORNIA - San Diego
MARC Analysis Research Corp.
4330 LaJolla Village Drive, Suite 320
San Diego, CA 92122
Tel: 1 619 658 9588
Fax: 1 619 758 8710

CONNECTICUT - E. Berlin
MARC Analysis Research Corp.
1224 Mill Street
E. Berlin, CT 06023
Tel: 1 860 828 2055
Fax: 1 860 828 2031

MICHIGAN - Ann Arbor
MARC Analysis Research Corp.
24 Frank Lloyd Wright Drive
Ann Arbor, MI 48106-0523
Tel: 1 734 998 0540
Fax: 1 734 998 0542

EUROPE

CZECH REPUBLIC
MARC Overseas, Inc.
Podolska 50
147 00 Praha 4
Czech Republic
Tel: 420 (2) 6121 4123
420 (2) 6121 4111 x252
Fax: 420 (2) 6121 4123
Email: support@marc.cz

FRANCE
MARC France S.A.
Bercy Expo SR 6101A
40 Avenue des Terroirs de France
75611 Paris Cedex 12, France
Tel: 33 (1) 4474 1550
Fax: 33 (1) 4474 1554
Email: support@marc.fr

GERMANY - Munich
MARC Software Deutschland GmbH
Ismaninger Str. 9
85609 Aschheim (bie Munich)
Germany
Tel: 49 (89) 904 50 33
Fax: 49 (89) 903 06 76
Email: support@marc.de

GERMANY - Hannover
MARC Software Deutschland GmbH
Alte Dohrener Str. 66
D 30173 Hannover, Germany
Tel: 49 (511) 980 5182
Fax: 49 (511) 980 5187

ITALY
Espri-MARC s.r.l.
Viale Brigata Bisagno 2/10
16129 Genova, Italy
Tel: 39 (10) 585 949
Fax: 39 (10) 585 949
Email: espri.marc@interbusiness.it

THE NETHERLANDS
MARC Analysis Research Corporation
Dublinstraat 32
2713 HS Zoetermeer
The Netherlands
Tel: 31 (79) 3510 411
Fax: 31 (79) 3517 560
Email: support@marc.nl

UNITED KINGDOM
MARC UK Ltd.
11 Linford Forum, Rockingham Drive
Linford Wood
Milton Keynes, MK14 6LY
Tel: 44 (1908) 606 070
Fax: 44 (1908) 606 633
Email: support@marc.co.uk

PACIFIC RIM

CHINA
MARC Overseas, Inc.
Bright Chang An Building, Suite 321
7# Jian Guo Men Nei Street
Beijing, China 100 005
Tel: 86 (10) 6510 2056
86 (10) 6510 2057
86 (10) 6510 2058
Fax: 86 (10) 6510 2053
Email: marcbj@public.bta.net.cn

JAPAN - Nagoya
Nippon MARC Co., Ltd.
Nagoya Nishiki Daiichi Seimei Bldg.
2F, 6-5 Nishiki 1-chome
Naka-ku, Nagoya, Aichi 460-0003
Tel: 81 (52) 202 5661
Fax: 81 (52) 202 5665

JAPAN - Osaka
Nippon MARC Co., Ltd.
4F 2nd Kimi Building
2-11 Toyotucho Suita-shi
Osaka 564-0051
Tel: 81 (6) 385 1101
Fax: 81 (6) 385 4343

JAPAN - Tokyo
(Asian Operations Headquarters)
Nippon MARC Co., Ltd.
Shinjuku Daiichi Seimei Building
P.O. Box 5056
2-7-1 Nishi-Shinjuku, Shinjuku-ku
Tokyo 163-0704
Tel: 81 (3) 3345 0181
Fax: 81 (3) 3345 1529
Email: support@marc.co.jp

KOREA
Dong Kyung Bldg. 7FL
824-19 Yuksam-Dong
Kangnam-Ku, Seoul, Korea
Tel: 82 (2) 561 7543
Fax: 82 (2) 561 7767
Email: marck@unitel.co.kr