



Mentat 3.1-MARC K7.1: New Features

User Guide





Copyright © 1997 MARC Analysis Research Corporation Printed in U. S. A. This notice shall be marked on any reproduction of this data, in whole or in part.

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 3.1-MARC K7.1: New Features

Part Number: UG-3012-01 Revision Date: September, 1997

PROPRIETARY NOTICE

MARC Analysis Research Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice.

ALTHOUGH DUE CARE HAS BEEN TAKEN TO PRESENT ACCURATE INFORMATION, MARC ANALYSIS RESEARCH CORPORATION DISCLAIMS ALL WARRANTIES WITH RESPECT TO THE CONTENTS OF THIS DOCUMENT (INCLUDING, WITHOUT LIMITATION, WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE) EITHER EXPRESSED OR IMPLIED. MARC ANALYSIS RESEARCH CORPORATION SHALL NOT BE LIABLE FOR DAMAGES RESULTING FROM ANY ERROR CONTAINED HEREIN, INCLUDING, BUT NOT LIMITED TO, FOR ANY SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF, OR IN CONNECTION WITH, THE USE OF THIS DOCUMENT.

This software product and its documentation set are copyrighted and all rights are reserved by MARC Analysis Research Corporation. Usage of this product is only allowed under the terms set forth in the MARC Analysis Research Corporation License Agreement. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MARC Analysis Research Corporation is prohibited.

RESTRICTED RIGHTS NOTICE

This computer software is commercial computer software submitted with "restricted rights." Use, duplication, or disclosure by the government is subject to restrictions as set forth in subparagraph (c)(i)(ii) or the Rights in technical Data and Computer Software clause at DFARS 252.227-7013, NASA FAR Supp. Clause 1852.227-86, or FAR 52.227-19. Unpublished rights reserved under the Copyright Laws of the United States.

TRADEMARKS

All products mentioned are the trademarks, service marks, or registered trademarks of their respective holders.

Contents

1. Introduction

About This User Guide

What's Covered in This User Guide 15 About Mentat 3.1 Enhancements 15 About MARC K7.1 Enhancements 15 Questions or Comments 19

Supporting Documentation

About Printed Documentation for Mentat 3.1 **20**Displaying Online Help in Mentat 3.1 **20**Printed and Online Documentation for MARC K7.1 **21**Periodic Updates for Mentat 3.1 and MARC K7.1 Documentation **21**

2. Getting Started

Running Mentat 3.1

Running Mentat 3.1 at the Command Line **25**New Command Line Parameters Featured in Mentat 3.1 **25**

Running MARC K7.1

About Shell Scripts **26**Submitting a Job in MARC **26**

Understanding the Mentat 3.1 Menu System

About Enhancements in the Mentat 3.1 Menus 28

Resizing a Mentat 3.1 Window 29
Using the Root Window for Parenting 31

Dynamic Viewing

About the Dynamic Viewing Feature 33
Moving an Object or a Model 33
About Dynamic Viewing Options 34
Translating a View 34
Rotating a View 34
Zooming a View In 34

3. Basic Procedures

File Browser

About the File Browser 39 Selecting a File 39

Import-Export Utility

About the Import Feature 42
Importing an ACIS File 43
Modifying the ACIS File 44
Importing an IGES File 47
Importing a NASTRAN or a PATRAN File 48
Exporting an IGES, a FIDAP, or a NASTRAN File 50

Adaptive Plotting

About Adaptive Plotting **52**Tolerance and Modes of Tolerances **52**About Absolute Mode of Tolerance **53**Safety Features **54**Default Settings **55**Changing the Default Settings **55**About Pre-Defined Settings **56**Additional Information **56**

View Snapshot

About View Snapshot 57
Creating a View Snapshot in IRIS RGB, TIFF, BMP, GIF 57
Setting the PostScript Plotting Attributes 58
Creating View Snapshots in PostScript 59
Changing the Default JPEG Attributes 60
Creating View Snapshots in JPEG 60

Plotting PostScript Image Files

About Plotting a Graphics Image 62
Printing PostScript Image Files 62
Sending a PostScript Output Directly to a Printer 63
About the Resolution of PostScript Images 64
Increasing the Font Size of Text Matter 66
Additional Information 67

Arrow Settings

About Arrow Settings **68**Specifying the Length of Preprocessing Arrows Manually **68**About Arrow Modes **69**Additional Information **70**

Element Extrapolation

About Element Extrapolation for Display Purposes **71** Selecting an Element Extrapolation Option **72** About Nodal Averaging **72**

4 General Technology

Using N to 1 and N to N Options in Links

About N to 1 and N to N Options 75
About Node Lists and Node Paths 77
Using the N to N Feature in Servo Links 81
About Resetting Parameters 84
Additional Information 85

Specifying Node Lists and Node Paths

Specifying a Node List **86**Specifying a Node Path **88**Additional Information **89**

Buckle Solutions Using Lanczos Method

About Lanczos Method for Buckle Solution **90**Specifying a Buckle Solution Method **90**Additional Information **91**

Adaptive Load Stepping

About Adaptive Load Stepping 92
Loadcase Types Featuring Adaptive Load Stepping Parameters 92

Displaying the Adaptive Load Stepping Parameters 93
About Load Stepping Criteria 94
About Specifying Multiple Ranges for a Criterion 96
Applications of Multiple Ranges for Criteria 96
Additional Information 97

Extended Precision Input

About the Extended Precision Input Feature **98**Activating the Extended Precision Input Feature **98**

Constant Dilatation

When to Use the Constant Dilatation Parameter 100 Analysis Classes Featuring the Constant Dilatation Parameter 100 Additional Information 101

Assumed Strain Formulation

About the Assumed Strain Formulation 102
Analysis Classes That Feature Assumed Strain 102
Additional Information 103

Numerical Preferences

About the Numerical Preferences Option 104 Additional Information 105

User-Defined Post Variables

About User Subroutine to Write to the Post File 106
Activating the User Subroutine UPOSTV 106
Additional Information 107

5. Mesh Generation

Repair Geometry

About Repair Geometry Operations 111
Trimming Surfaces 112
Removing Free Curves 112
Cleaning 2-D Curve Loops 113
Splitting Curves 115
Checking Curve or Surface Geometry 117
About Tolerance Value for Curve Intersection 118
Checking for Intersecting Loops Using a Tolerance Value 119

Curve Divisions

About Curve Divisions 120
Applying Fixed Divisions to a Curve 121
Applying Fixed Average Length Divisions 121
About Curvature-Dependent Curve Divisions 121
Applying Curvature-Dependent Divisions to a Curve 122
When to Break or Match Curves 124
Input Geometry Considerations for 2-D Planar and Surface Meshing 125
Guidelines for Inner and Outer Loops 126
Checklist for the Advancing Front and Delaunay Meshers 126

Using the Advancing Front Mesher

About the Advancing Front Meshers 127
Special Considerations for Quad Meshing 127
About Distortion Parameters in Quad/Tri Meshing 128
Considerations for Specifying Distortion Parameters 128

Using the Delaunay Mesher

About the Delaunay Mesher 131
Features of the Delaunay mesher 131
Additional Information 132

Expanding Elements Along a Curve

About Expanding Elements Along a Curve in Mentat 3.1 133

Creating a Node or Point at Midpoint

Creating a Node at Midpoint 136 Creating a New Point 136

6. Contact

Performing Contact Analyses

Components of a Contact Analysis 139
About Physical Bodies 140
About Contact Bodies 140
Choosing a Contact Body Type 141
About Flip Elements, Curves, and Surfaces 143
Finite Element Mesh for a Deformable Body 143
Analysis Parameters for Deformable Bodies 143
About Discrete and Analytical Descriptions 144
About the Analytical Desc. Discontinuity Option 145
About Rigid Bodies and Their Representation 146

Motion Control of Rigid Bodies 147
About the Centroid and Rotation Axis Options 147
About Load as Rigid Body Motion Control 149
Coupled Analysis Considerations 150
Special Assumptions for Symmetry Bodies 151
About Rigid w/Heat Transfer Bodies 151
About Contact Tables 152
Table Properties of a Contact Table 152
About Touching Bodies 153
About Contact Areas 154

Specifying Contact Information in Loadcases

About Contact Information and Loadcases in Analysis 156
Using Contact Information in Loadcase Types 156
About Contact Body Release 157
Using Contact Body Releases 157
Preventing Separated Nodes from Contacting Again 157

Specifying Contact Information in Jobs

About Contact Control in Jobs 159
About User Subroutines 160
About Deformable-Deformable Contact Control 163
About Separation Procedures 163
About Max # Separations/Inc. 164
About Separation Increment 164
About Friction Models 164
About Stick-slip 166
Additional Information 168

7• Design Sensitivity and Optimization

Design Sensitivity and Optimization

About Design Sensitivity and Optimization 171 Components of Design Optimization 171 When to Apply Design Optimization 172 Where You Can Apply Design Optimization 172

Design Variables

Types of Design Variables 175
Linking and Unlinking Design Variables 175
About Bounds and Design Sensitivity 176
Using Composites as Design Variables 177
Geometry Design Variables 178

Types of Geometry Design Variables 178 Using Geometry as Design Variables 179

Design Constraints

Types of Constraints 180 About Bounds 180 Setting Displacement Constraints 181 Adding or Removing Nodes 182 Specifying the Vector Direction/Plane Normal 182 Setting Stress or Strain Constraints 184 Types of Eigenvalue Constraints 185 Setting the Eigenvalue Constraints 185

Using Design Variables and Constraints

Associating Constraints with a Loadcase 186 Design Sensitivity and Optimization as Mechanical Analysis Options 186 Using Design Variables and Constraints in Design Sensitivity 187 About Maximum Active Set Size 189 About Maximum Cycles 190

Post-Processing of Sensitivity Results

About the Plotting of Sensitivity Results 192 About Constraint Reference Numbers 192 About Response Gradient Plots of Sensitivity Results 193 About Bar Charts Showing the Gradient of a Response Quantity 194 Plotting the Gradient of a Response Quantity with Respect to Design Variables 195 Plotting Bar Charts Showing Gradient of Objective Function 196

About Contour Plots 197 Creating Contour Plots of Element Contributions 197

Post-Processing of Optimization Results

About Plotting of Optimization Results 199 About History Plots of Optimization Results 199 Creating History Plots of Objective Function Values 200 About History Plots of Design Variable Values over Optimization Cycles

Additional Information 204

8. Element Technology

New Shell Elements

About the New Shell Elements 207

About Element 138 207
About Element 139 208
About Element 140 208
Associating Elements with a 3-D Membrane/Shell 209
Additional Information 209

New Rebar Elements

About Rebar Elements 210
Types of Rebar Elements 210
Generating and Identifying Rebar Elements 211
Specifying Rebar Material Properties 212
Associating Double Elements with Nodes 213
Additional Information 214

9. Fluid Mechanics

Fluid Analysis

About Fluid Analysis 217
About the Lumped Mass Option 218
About Mixed and Penalty Method Procedures 219
About Strongly-Coupled and Weakly-Coupled Parameters 220
About the Solver Option for Fluid Analysis 221
Valid Element Types 221

Boundary Conditions for Fluid Analysis

About Boundary Conditions for Fluid Analysis 222
About Fixed Velocity 222
About Point Loads 224
About Global Loads 226
About Gravity Loads 227
About Centrifugal Loads 227

Initial Conditions for Fluid Analysis

About Types of Initial Conditions 228
Coupled Analysis Considerations 229
About Limitations on Large Displacements for Solids 229

Material Properties for Fluid Analysis

About Material Properties in a Fluid Analysis 230 Mass Density and Volumetric Expansion 233

Loadcases for Fluid Analysis

About Loadcases for Fluids 234
Applying the Fluid Steady State Parameters 235
About Control Tolerances 237
Viewing the Convergence Testing Parameters 238
Additional Information 239

10 Material Modeling

Using the Narayanaswamy Model

About Thermo-Rheologically Simple Materials and the Narayanaswamy Model 243

Mechanical Material Types for the Narayanaswamy Model 243 Parameters for the Narayanaswamy Model 243 Additional Information 246

New Approaches in Plasticity Modeling

Multiplicative Decomposition and New Plasticity Procedures 247 Considerations for Elastic Data 248 Considerations for Work-Hardening Data 248 Element Considerations 249

Mooney-Rivlin and Ogden Formulations

About the Updated Lagrange Procedure 251 Considerations for Mooney-Rivlin and Ogden Formulations 251 Additional Information 252

User Subroutines in Hypoelastic Properties

About User Subroutines in Hypoelastic Properties 253
About HYPELA and HYPELA2 253
Choosing a User Subroutine for Hypoelastic Properties 254
Additional Information 255

Experimental Data Fitting

About Experimental Data Fitting 256
About Experimental Tests 257
Reading in Experimental Data 257
About Forms of Data 258
Deciding Which Modes to Use 262
Additional Information 263
About the Continuous and Discontinuous Damage Models 264
Setting Damage Control Parameters 264
Considerations for Data Sets 265
Checklist for Using the Continuous Damage Model 265

Additional Information 265
About the Viscoelasticity Model and Relaxation Spectra 266
Determining the Coefficients for an Energy Relaxation Test 268
Considerations for Data Sets 268
Additional Information 269

11. Analysis Integration

Using Radiation Viewfactors

About the Radiation View Factor Feature 273
About Viewfactors and Computation 273
Applying Radiation As Boundary Conditions 274
About Emitting and Absorbing Roles 275
About Emitting and Incident Objects 275
Indicating the Location of the Viewfactor File 279
Additional Information 280

Appendixes

A: User Enhancements

List of User Enhancements

MARC New Parameters 285
MARC New Model Definition 285
MARC New History Definition 286
New User Subroutines 286
New Element Types 286

B: Fluid Elements

Fluid Element Types

Planar Element Types 291
Axisymmetric Element Types 291
Three-Dimensional Element Types 292

C: MARC Data Reader Support

Supported Parameters

List of Supported Parameters 295

Supported Model-Definition Options

List of Supported Model-Definition Options 297

D: NASTRAN Writer Data Entries

Minimum File Requirements

Minimum File Requirements 303

Finite Element Connectivity

Bulk Data Cards Written by Mentat 3.1 304

Node Coordinates/Transformations

Node Coordinates / Transformations 305

Materials and Geometry

Material ID's Included in NASTRAN Geometry Cards 306

Loads and Boundary Conditions

Loads and Boundary Conditions 308

Sample File

Sample NASTRAN Bulk Data File 309

E: Command Line Parameters

Mentat 3.1 Command Line Parameters

Table of Additional Command Line Parameters 313

MARC K7.1 Command Line Parameters

Table of MARC K7.1 Command Line Parameters 315

F: Demonstration Problems

Demonstration Problems

Lists of Demonstration Problems in MARC K7.1 319

Index

1. Introduction

About This User Guide

What's Covered in This User Guide 15 About Mentat 3.1 Enhancements 15 About MARC K7.1 Enhancements 15 Conventions Used in This Guide 18

Supporting Documentation

About Printed Documentation for Mentat 3.1 20 Displaying Online Help in Mentat 3.1 20 Printed and Online Documentation for MARC K7.1 21 Periodic Updates for Mentat 3.1 and MARC K7.1 Documentation 21

About This User Guide

What's Covered in This User Guide

This user guide describes the major enhancements in Mentat 3.1-MARC K7.1 and how to use them in nonlinear finite element analysis.

About Mentat 3.1 Enhancements

The Mentat 3.1 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. 271).

About MARC K7.1 Enhancements

The MARC K7.1 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. 137).

- 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. 169).
- *Element Technology*—describes the new shell and rebar elements (*p. 169*).
- Fluid Mechanics—describes the solution of fluid mechanics problems using the Navier-Stokes equations for planar, axisymmetric and 3-D geometries (p. 215).
- *Material Modeling*—provides instructions on how to use the new material models, subroutine HYPELA2, and the Experimental Data Fitting feature (p. 241).

About the Appendixes

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

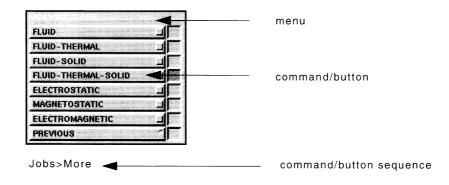
19-300-44	Refer to Appendix,
lists of new or modified:	A (p. 283)
 parameters model definitions history definitions user subroutines element types 	
types of elements that you can represent in the fluid region in analysis,	B (p. 289)
lists of parameters and model-definition options supported by the MARC Reader,	C (p. 293)
data issues relating to NASTRAN Writer; bulk data entries,	D (p. 301)
descriptions of the command line parameters and options in MENTAT and MARC,	E (p. 311)
lists of demonstration problems associated with selected options in MARC,	F (p. 317)

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 3.1, MARC K7.1); button names	<pre>small caps, bold (e.g., MENTAT, MARC; LOADCASE)</pre>
Main menus and windows	title case (e.g., Analysis Class)
Left-mouse click	Click; <ml></ml>
Middle mouse click; right mouse click	<mm>;<mr></mr></mm>
Cross-references	italicized text (e.g., See "Running Mentat 3.1" 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 3.1 and MARC K7.1 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.

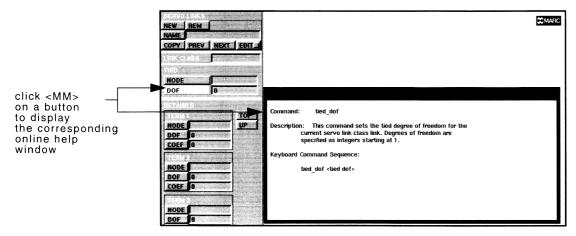
About Printed Documentation for Mentat 3.1

Use the following table to locate additional Mentat 3.1 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 3.1 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 3.1

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



Links>Servo

Printed and Online Documentation for MARC K7.1

Volumes A, B, C, and D contain the documentation for MARC K7.1. 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 K7.1 documents that address your specific needs:

For was a second	refel 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 3.1 and MARC K7.1 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

•	
	Montat 3 1-MARC K7 1 · Now Features

2• Getting Started

Running Mentat 3.1

Running Mentat 3.1 at the Command Line 25

Running MARC K7.1

About Shell Scripts 26 Submitting a Job in MARC 26

Understanding the Mentat 3.1 Menu System

Setting Stand-Alone Static Menus 30 Using the Root Window for Parenting 31

Dynamic Viewing

Moving an Object or a Model 33 About Dynamic Viewing Options 34

Running Mentat 3.1

Running Mentat 3.1 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 3.1

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. 311);

Operation: To	use the command line parameter	Option
use the OpenGL graphic system	-ogl	
create a log file with the name <i>lfile</i> (the default filename is <i>mentat.log</i>)	-If	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.

Running MARC K7.1

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 K7.1 Command Line Parameters (p. 315).

Additional Information

Use the following table to find more information:

For	refer to the Chapter	lo _C
installation instructions for Mentat 3.1	Running Mentat	Mentat 3.1 Installation Guide
installation instructions for MARC K7.1	Running MARC	MARC K7.1 Installation Guide
a description of how to execute MARC K7.1 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 K7.1 Installation Guide

Understanding the Mentat 3.1 Menu System

About Enhancements in the Mentat 3.1 Menus

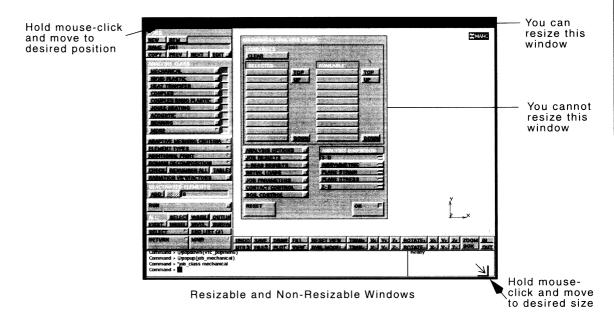
Here are some of the key enhancements to the Mentat 3.1 menu system:

- 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 3.1 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 3.1" 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 3.1" 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 3.1 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.

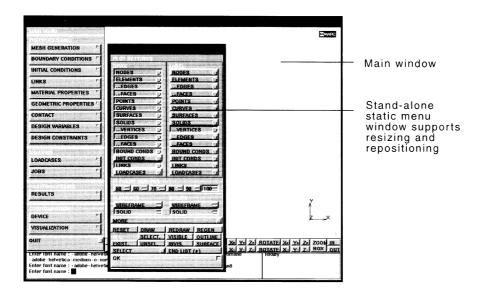
During a MENTAT session, you can resize a window parented by the root window as you would any X-window in your environment. The following figure shows a type of resizing supported by the root window of the window manager in a HP-UX 10.0 environment.



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

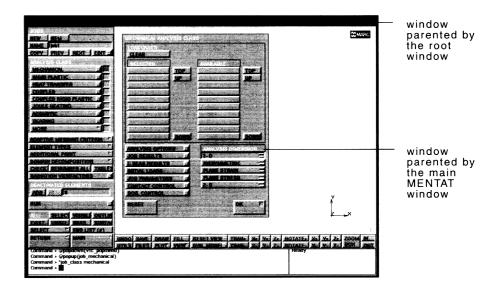
You can also enable the stand-alone feature by starting MENTAT with the following command line option:

-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 3.1

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
title "Mentat Graphics Window"
origin 32 0 size 96 112
background_color background
border_width 0 border_color border
buffering double
}

Comment this line
Add this line

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 3.1 supports more keyboard and mouse options than earlier versions of MENTAT.

Moving an Object or a Model

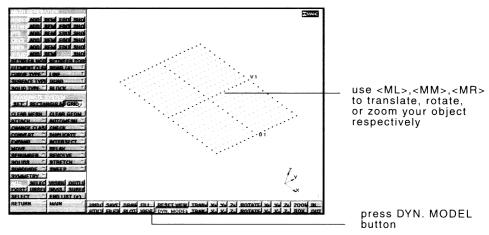
To move a grid object or a model:

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

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	The second secon
an introduction to the MENTAT environment	Mechanics of Mentat	Mentat 3.1 User's Guide

3• Basic Procedures

File Browser
About the File Browser 39
Import-Export Utilities
About the Import Feature 42
Adaptive Plotting
About Adaptive Plotting 52
View Snapshot
About View Snapshot 57
PostScript Image Files
About the Resolution of PostScript Images 64
Arrow Settings
About Arrow Settings 68
Element Extrapolation

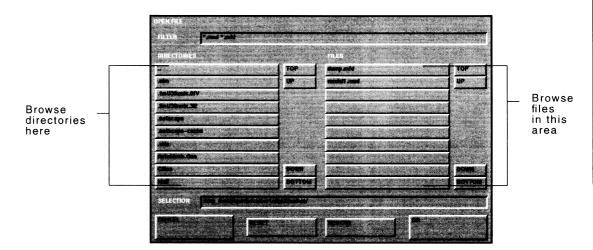
About Element Extrapolation for Display Purposes 71

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.

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 o K.

Buttons in the File Browser

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

10	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	воттом
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.

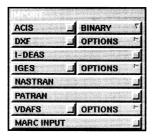
Import-Export Utility

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.



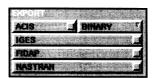
Files>File I/O>Import

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.

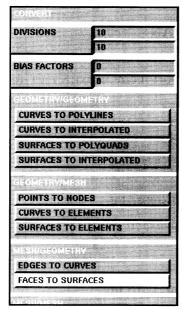
ACIS binary files have .sab extension. ACIS text files have a .sat extension.

- 3. Click ACIS.
- 4. Using the file browser, choose the ACIS file.
- 5. Click o K.

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:

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	REPORT
• DXF entity summary	
Mentat entity summary	

- 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 o K.

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:

[Our	Chocae
• turn on validation of IGES entities	VALIDATE
• correct invalid entities so that they	(ON)
can be processed	
check explicitly-defined semantics	
in the IGES specification	
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:	REPORT
• IGES version number	
• IGES entity summary	
Mentat entity summary	

- 3. Click IGES.
- 4. Using the file browser, choose the IGES file.
- 5. Click o K.

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 o K.

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:

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

- 3. Click VDAFS.
- 4. Choose the VDAFS file using the file browser.
- 5. Click o K.

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

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 o K.

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 o K.

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.

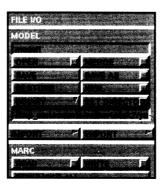
When you export using the NASTRAN option, the data file export writes bulk data entries (See "D: NASTRAN Writer Data Entries" on page 301) from the current Mentat database. The data apply to structural analyses only.

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. 293).

To import a MARC data file:

1. Choose FIles>Read.



Files

- 2. Choose the relevant file in the file browser.
- 3. Click o K.

Adaptive Plotting

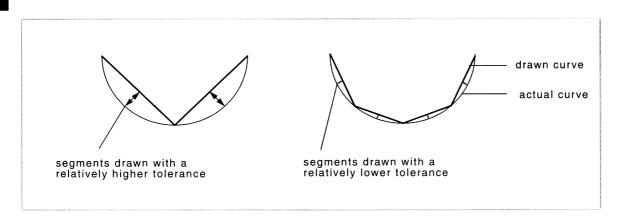
About Adaptive Plotting

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.



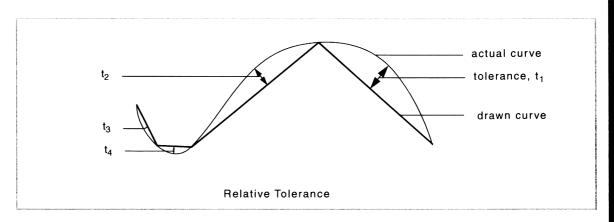
When you change the default tolerance settings in the adaptive plotting feature, you can choose either of the following types of tolerances:

- relative
- absolute

About Relative Tolerance

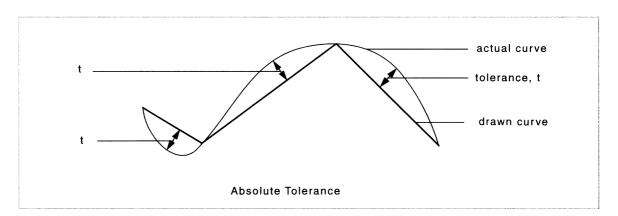
In relative mode, the tolerances are relative to the length of the curve and longer curves have larger margins of error.

For most cases, use the relative mode of tolerance.



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.



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	for the number	and the upper limit for the number of segments is squal 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		

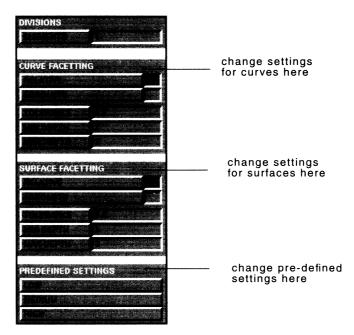
Default Settings

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.

- 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 predefined 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.

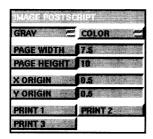
PostScript Files

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 OF COLOR
specify page dimensions (in inches)	PAGE WIDTH and/or PAGE HEIGHT

1000 - 10	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 OF 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

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.

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:

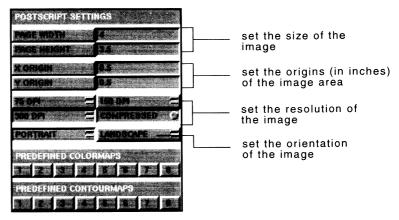
- send the PostScript file to a printer that supports PostScript.
- import the PostScript file as an object into an application that reads PostScript files as input (e.g. FrameMaker).
 Depending on the application, you can alter the object attributes (e.g., size, rotation, aspect ratio, etc.) before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

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.

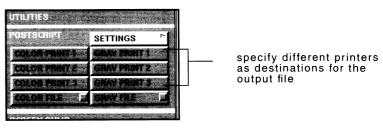


Utils>(PostScript)Settings

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

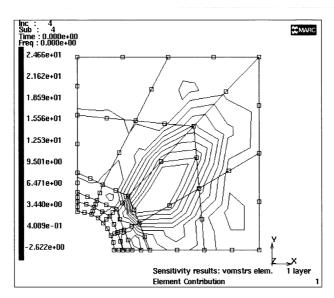
2. Choose a printer destination (e.g., Color Print 1) for the output file.

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.

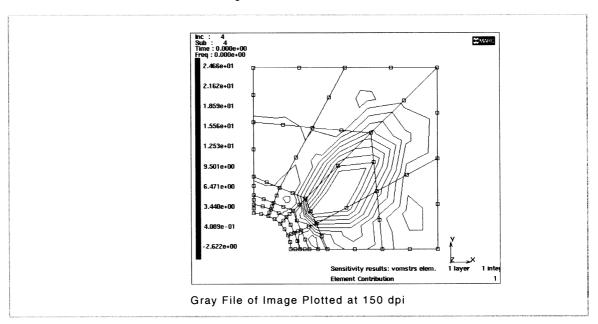
Here are examples of PostScript images with three different dpi settings in MENTAT:

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

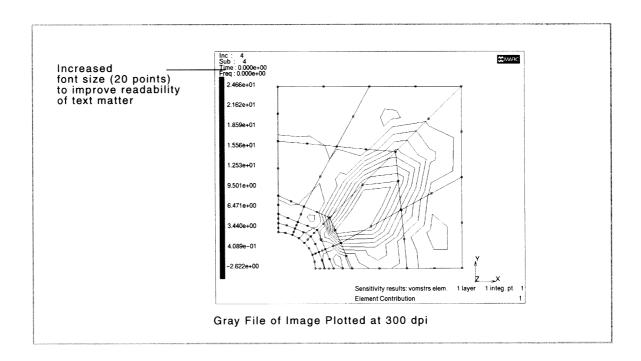


Grayscale File of Image Plotted at 75 dpi





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



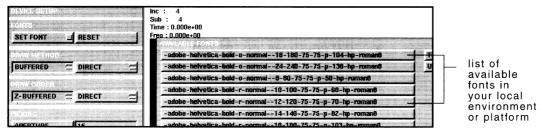
For quality images, choose a resolution that is higher than 75 dpi.

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.



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),,,	ln.,
instructions on how to configure your destination printer(s) for your PostScript files	Mentat Interfaces	Mentat Installation Guide(s)

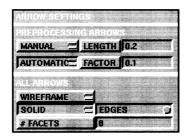
Arrow Settings

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).

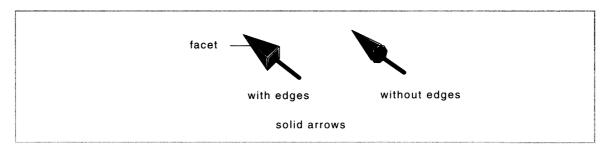
About Arrow Modes

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

wireframe



solid



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

Additional Information

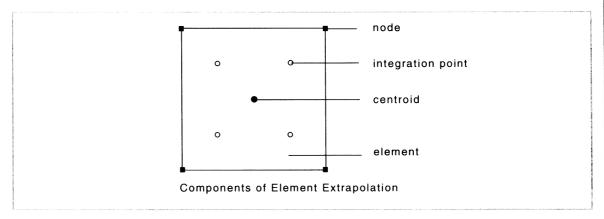
Use the following table to find more information:

description of buttons in the Arrow Settings menu	4:Boundary Conditions– Arrow Settings	Mentat 3.1 Command Reference

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.

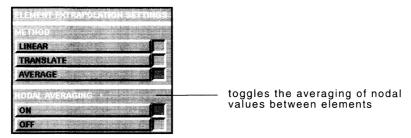


There are three element extrapolation methods in MENTAT:

- 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

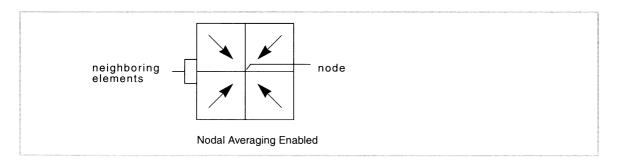
To select an element extrapolation method in MENTAT, choose Results>Scalar Plot Settings>Extrapolation.



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.



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

Using the N to 1 and N to N Options in Links
_
Specifying Node Lists and Node Paths
Buckle Solutions Using Lanczos Method
Adaptive Load Stepping
Extended Precision Input
Constant Dilatation
Assumed Strain
Numerical Preferences
User-Defined Post Variables

General Technology



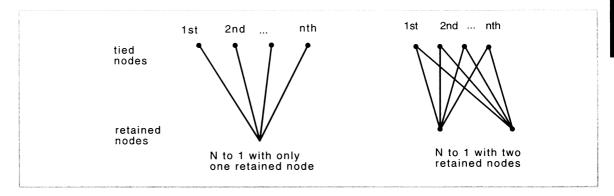
Using N to 1 and N to N Options in Links

About N to 1 and N to N Options

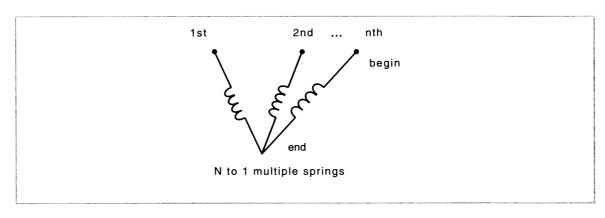
The N to 1 and N to N options are useful when you are applying constraint equations (ties or servo links) or springs that are similar in nature. A typical application is when you are modeling generalized plane strain conditions and you want multiple nodes to move as one node.

Here are the situations where you can use these options:

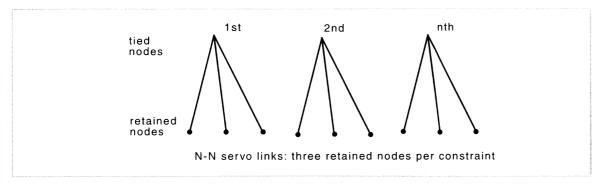
• N to 1—multiple ties between N tied nodes and common retained node(s).



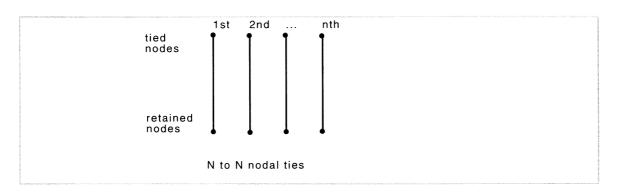
 N to 1—multiple springs between N begin nodes and one common end node.



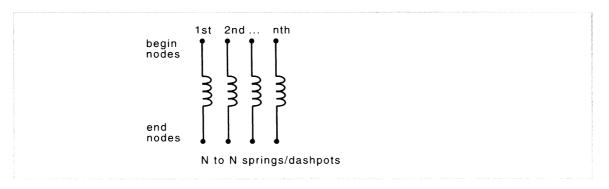
 N to N multiple ties (servo links)—N tied nodes with αN retained nodes, where α is the number of retained nodes per constraint



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

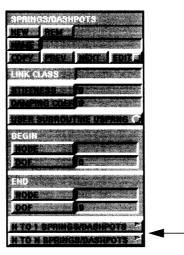


 N to N springs—multiple springs, each with the same stiffness/dashpot values, between two lists of nodes.



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.



Links>(Springs/Dashpots)

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

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:

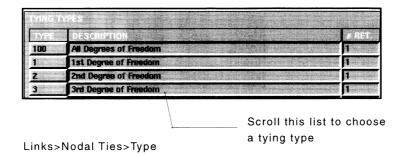
When using the feature,	specify:
N to 1	node list
N to N	node path(s)

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.



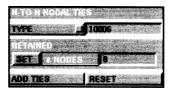
- 4. For user-defined ties, click SET and enter the number of retained nodes.
- 5. Specify the retained node(s).
- 6. Click ADD TIES.
- 7. Enter the node list for the tied nodes.

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:

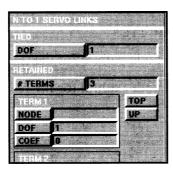
$$U_{DOF}^{T} = C_1 U_{DOF}^{R_1} + C_2 U_{DOF}^{R_2}$$

where U_{DOF}^{T} is the displacement of the tied node, $U_{DOF}^{R_1}$ 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.

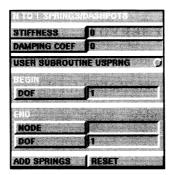
The number of times that the system prompts you for the retained node path depends on the number of terms that you specify.

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.
- Click N TO 1 SPRINGS/DASHPOTS in Springs/ Dashpots.



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

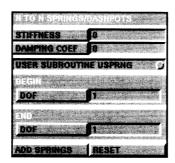
- 3. Flag the subroutine, USPRNG, or enter the following parameters:
- stiffness
- · damping coefficient
- 4. Enter the degree of freedom for the end node.
- 5. Specify the end node.
- 6. Enter a degree of freedom for the begin nodes.
- 7. Click ADD SPRINGS.
- 8. Enter the node list for the begin nodes.

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.

40044	Uses Defined To 4.11 (Uses Oak UPADVO)	в тор
10011	User Defined Tie # 11 (User Sub UFORMS)	
10012	User Defined Tie # 12 (User Sub UFORMS)	8 <u>UP</u>
10013	User Defined Tie # 13 (User Sub UFORMS)	8
10014	User Defined Tie # 14 (User Sub UFORMS)	8
10015	User Defined Tie # 15 (User Sub UFORMS)	8

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

For	refer to Chapter/s	ln
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 K7.1" on page 319)		Vol. E

Specifying Node Lists and Node Paths

Specifying a Node List

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

Method	Gr. Procedure
A	1. Use <ml> to pick nodes (to undo your last</ml>
	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.
В	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	 Use the Box Pick or Polygon Pick method to pick the nodes. To undo the last Box Pick or Polygon Pick, click < MM>. Indicate end of list by choosing one of the following: Click END LIST (#). Click < MR> in the graphics area. Type "#" in the dialogue area.
Е	Use the List buttons (see Mentat 3.1 User Guide: Chapter 2.5, <i>List Specification</i>).

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.
В	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.
С	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

For	refer to Chapter(s)	Section of the sectio
list specifications and mouse button functions	2, Mechanics of Mentat 3.1	Mentat 3.1 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:

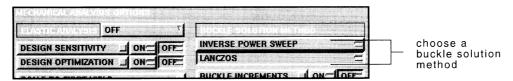
- Lanczos
- Inverse Power Sweep

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

Folia:	refer to Chepter(cher	Day
background and reference information	Dynamic Eigenvalue Extraction	Vol. A
input options	Buckle Parameter Card/ Buckle History Definition Card	Vol. C

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:

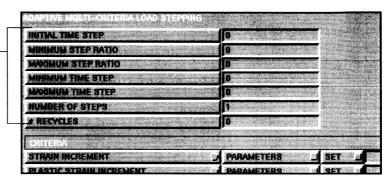
Analysis Class	Loadcase types that feature Adaptive Load Stepping parameters
Mechanical	StaticCreepDynamic TransientRigid Plastic
Coupled	 Quasi-Static Creep Dynamic Transient Rigid Plastic

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.

About Minimum and Maximum Step Ratios

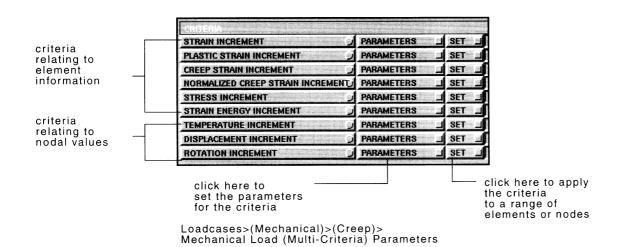
The minimum step ratio and the maximum step ratio are the ratios of the time step of the current increment to the time step of the next increment. Use these ratios to prevent the time steps from varying too sharply.

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:

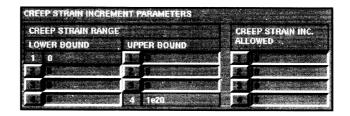
- element information (e.g., strain increment)
- nodal values (e.g., temperature increment, displacement increment)



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 elasticplastic 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

For	refer to Chapter(s)	Prince of the property of the second of the
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

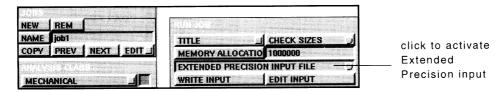
The Extended Precision Input feature ensures that the data which you enter into MENTAT, to be transferred into a MARC input file, has greater precision. This greater precision is useful in accurately modeling NURB surfaces for contact analysis (p. 139).

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.



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).

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

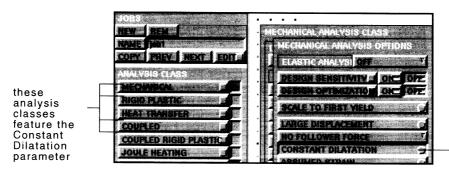
You can activate the Constant Dilatation parameter for the following analysis classes in MENTAT:

- 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.



click to activate Constant Dilatation

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

description of the	refer to Chapter(s)	In.
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

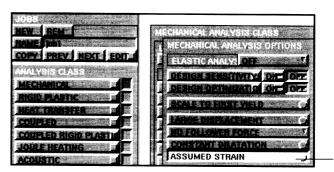
You can use the Assumed Strain formulation for the following analysis classes:

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

Activating the Assumed Strain Formulation

To activate the Assumed Strain formulation:

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



click to activate Assumed Strain formulation

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

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

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:

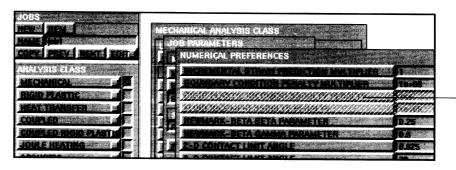
- Mechanical
- Rigid Plastic
- Coupled
- Coupled Rigid Plastic
- Fluid
- Fluid-Thermal
- Fluid-Solid
- Fluid-Thermal-Solid

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

FOR-	refer to Chapter(s)	ln
description of the	3: Model	Vol.C
PARAMETER model	Definition	
definition option	Options	

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:

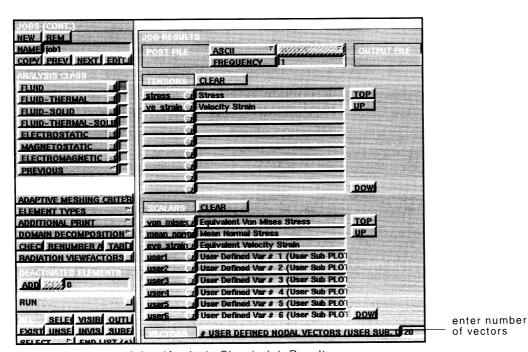
- ELEVAR—output of element quantities
- ELEVEC—output of element quantities in harmonic analysis
- UPOSTV—(new user subroutine) user-selected postprocessing of nodal variables

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

Trimming Surfaces 112
About Tolerance Value for Curve Intersection 118

Curve Divisions

Applying Fixed Divisions to a Curve 121 Parameters for Curvature-Dependent Curve Divisions 122

Using the Advancing Front Mesher

Special Considerations for Quad Meshing 127 Specifying Distortion Parameters 129

Using the Delaunay Mesher

Features of the Delaunay mesher 131 Applying the Delaunay Mesher 132

Expanding Elements Along a Curve

About Expanding Elements Along a Curve in Mentat 3.1 133

Creating a Node Point at Midpoint

Creating a New Point 136

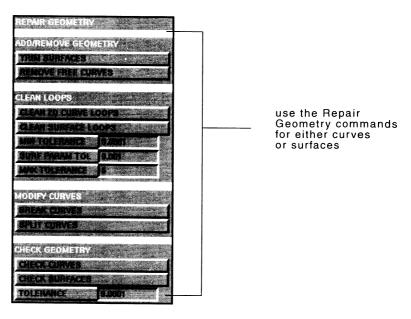
Mesh Generation



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.



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

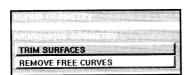
Very often, when you import geometries into MENTAT, some or all of the trimming curves for some surfaces are missing. Use the Trim Surface command to place four trimming curves on the edges of surfaces.

To trim surfaces:

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

Removing Free Curves

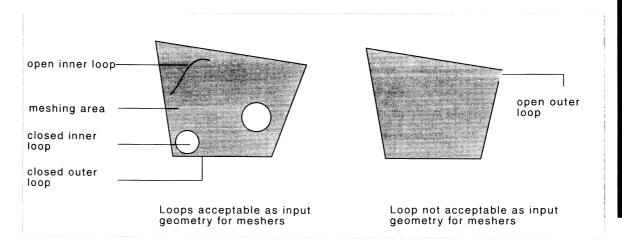
To remove curves not attached to any surface, choose Mesh Generation>Automesh>Repair Geometry> Remove Free Curves.



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.

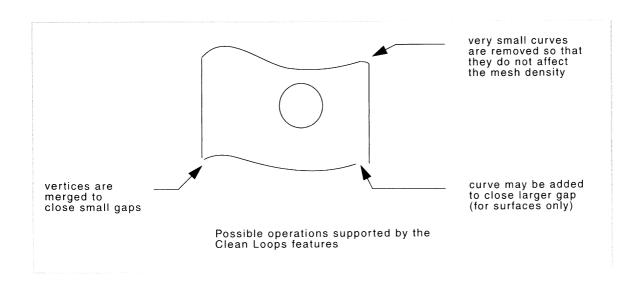
Cleaning Surface Loops

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:

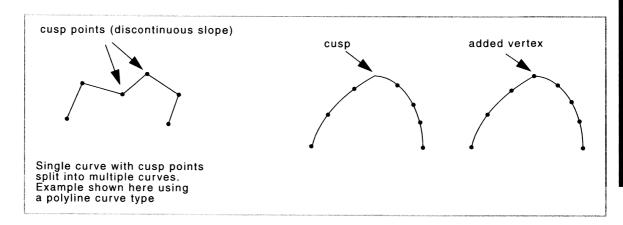
- 1. Choose one or more of the following options:
- Click MIN TOLERANCE and specify a value.
- Click MAX TOLERANCE and specify a value.
- Click SURF PARAM TOL and specify a value.
- 2. Click CLEAN SURFACE LOOPS.
- 3. Enter a surface list.



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.

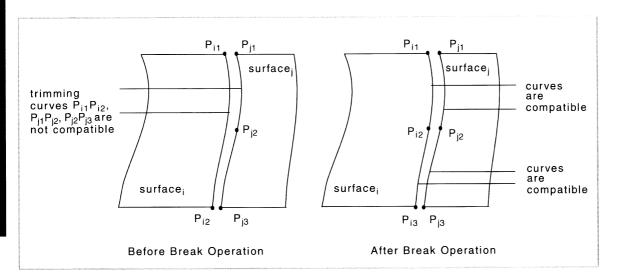


To split curves:

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

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.

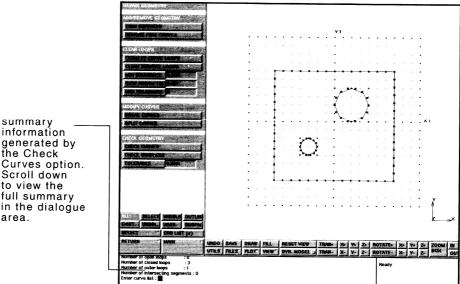
Checking Geometry

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.
- Choose CHECK CURVES OF 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

> 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

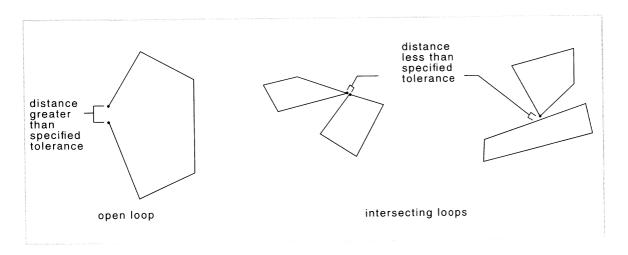
When you choose the CHECK SURFACES option, you get a summary in the dialogue area containing the following information:

- 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

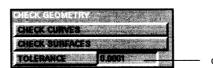
When you set a tolerance value for curve intersection, the program compares the specified tolerance and the distance between the end points in adjoining segments.

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

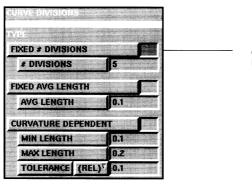
- 2. Choose either CHECK CURVES OF CHECK SURFACES.
- 3. Enter a curve or surface list.

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 divisions 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

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).

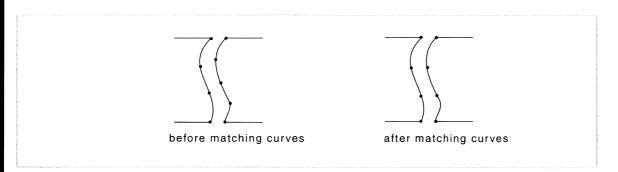
- 1. Choose Mesh Generation>Automesh>Curve Divisions.
- 2. In the Apply Restriction To area, choose individual curves or detected loops.
- 3. Choose FORCE EVEN DIV OF FORCE ODD DIV.
- 4. Click APPLY CURVE DIVISIONS.
- 5. Enter a curve list.

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 116) or match curves:

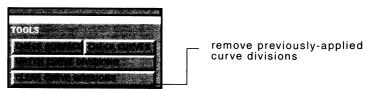
La company of the second of th	thon; use this :: operation:
neighboring trimming curves are not coincident over their entire lengths	Break
you applied curve divisions and there are coincident neighboring curves with non-conforming divisions	Match

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 112) as their input and generate meshes over the surface area bounded by trimming curves.

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:

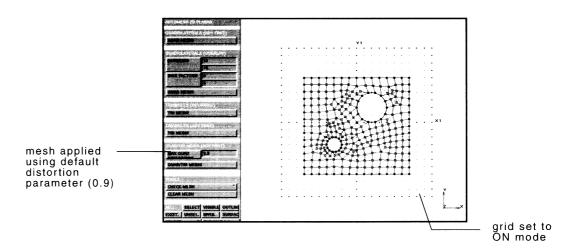
- there is at least one outer loop
- all open loops are contained within a closed loop
- all the curves have divisions applied to them
- no loops intersect

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.

So 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.

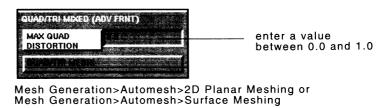
If you enter a very low value of the distortion parameter (i.e., a value closer to 0.0), it slows down the mesher considerably while generating a predominantly tri-mesh.

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 OF SURFACE MESHING.
- 2. Click MAX. QUAD DISTORTION and enter a value.



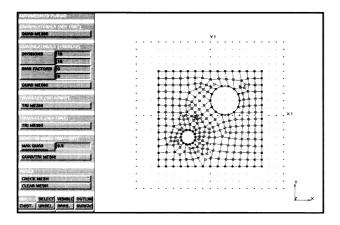
Applying the Advancing Front Meshers

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

- 1. Choose either REPAIR GEOMETRY OF CURVE

 DIVISIONS or both to prepare the model for meshing (See

 "Checklist for the Advancing Front and Delaunay Meshers" on
 page 126).
- 2. Choose either 2D PLANAR MESHING OF 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.

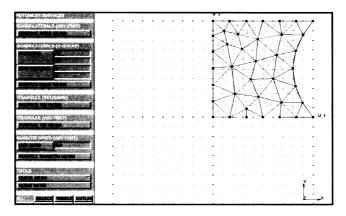


Advancing Front Quad mesh for 2-D planar model

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.

Applying the Delaunay Mesher

To apply the Delaunay mesher to your model:

- 1. Choose either REPAIR GEOMETRY OF CURVE

 DIVISIONS or both to prepare the model for meshing (See

 "Checklist for the Advancing Front and Delaunay Meshers" on
 page 126).
- 2. Click on 2D PLANAR MESHING OF 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

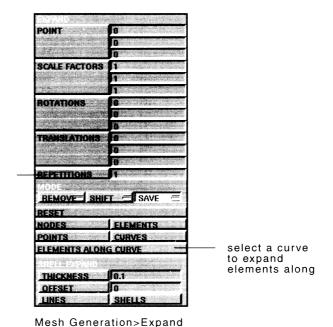
Use the following table to find more information:

For	refer to Chapter(s)	and Mesagerage
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 3.1 User Guide

Expanding Elements Along a Curve

About Expanding Elements Along a Curve in Mentat 3.1 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.

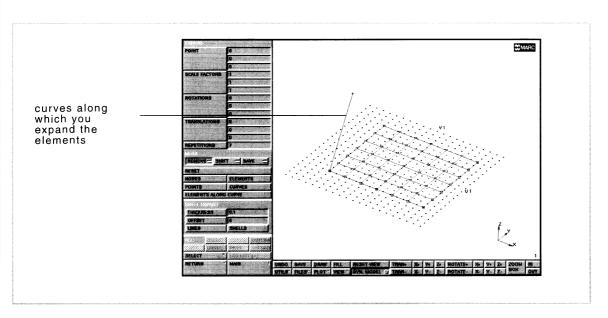
To view the Expand menu, choose Mesh Generation>Expand.



set the number of times that the elements expand

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.

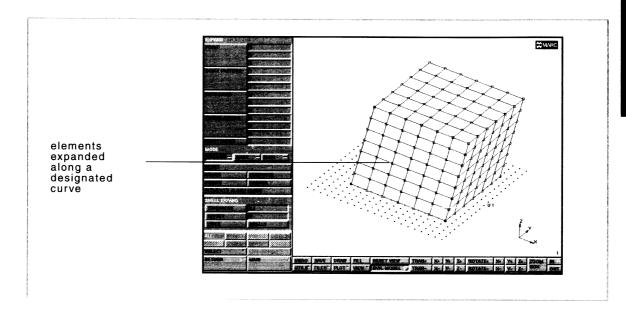


Use the REPETITIONS command to set the number of times that the elements are expanded. However, use this command with caution as it is easy to create a mesh which penetrates itself.

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

Components of a Contact Analysis 139
Choosing a Contact Body Type 141
Analysis Parameters for Deformable Bodies 143
About Rigid Bodies and Their Representation 146
About Velocity as Rigid Body Motion Control 148
Coupled Analysis Considerations 150
About Contact Tables 152
About Exclude Segments 155

Specifying Contact Information in Loadcases

About Contact Information and Loadcases in Analysis 156

Specifying Contact Information in Jobs

About Contact Control in Jobs 159 About User Subroutines 160 About Increment Splitting 162 About Separation Procedures 163 About Friction Models 164 About Stick-slip 166

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 141).
- 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 161).
- 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 154).
- prescribe different contact procedures in MARC—how the numerical algorithms are implemented in the analysis (See "About Contact Control in Jobs" on page 159).
- prescribe separation procedures—how a body separates from another body (See "About Separation Procedures" on page 163).

- 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 164).
- assign the parameters for coupled analysis—evaluate the thermal interactions between two adjacent bodies (See "Coupled Analysis Considerations" on page 150).

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:

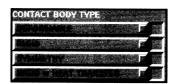
- Deformable
- Rigid—the body is either moving in space or is a special type of rigid body.
- Symmetry—special type of rigid body which is a symmetry line or a symmetry surface.
- Rigid with Heat Transfer—a body made up of finite elements and is not deformable; used only in coupled thermal stress analysis.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

	thus are a	
deformable or rigid with heat transfer	ELEMENTS	
rigid or symmetry	• CURVES (2-D)	
	• SURFACES (3-D)

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

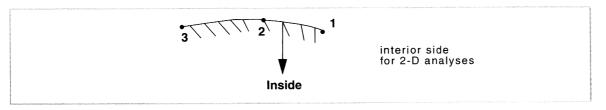
Use the ID CONTACT and ID BACKFACES buttons in Contact Bodies to turn on the identification of contact bodies and backface polygons respectively in MENTAT.

ID CONTACT J ID BACKFACES

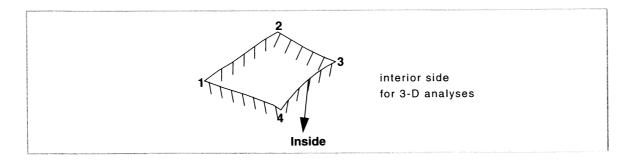
Use the I.D. CONTACT feature to identify all the deformable and the rigid bodies.

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.



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.



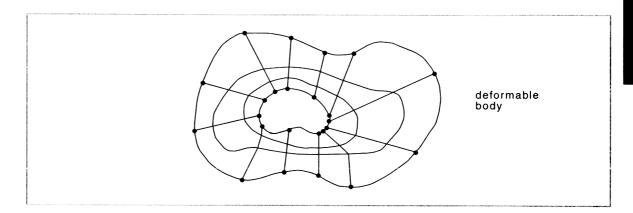
About Flip Elements, Curves, and Surfaces

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.

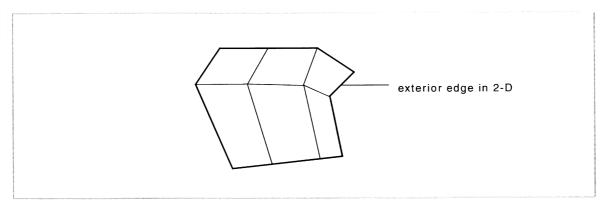
Use the Description field to prescribe how the surface of the deformable body is represented in your contact analysis.

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.

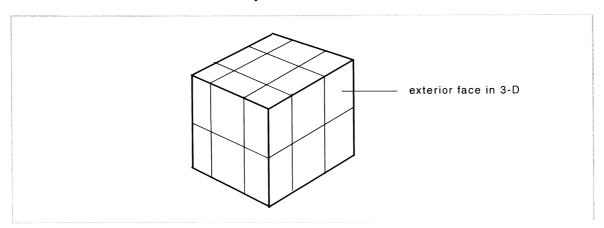
About Discrete and Analytical Descriptions

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.

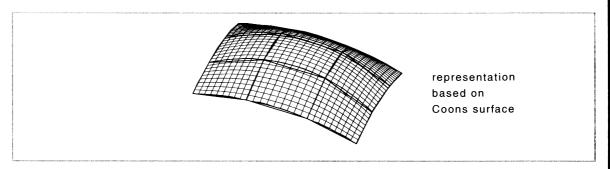


• 3-D—the exterior faces of the finite element mesh represent the body.



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.



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.

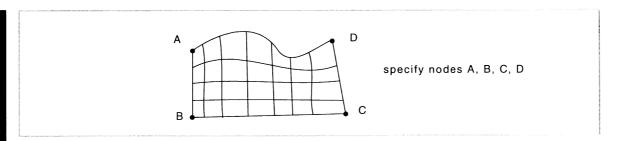
About the Analytical Desc. Discontinuity Option

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.

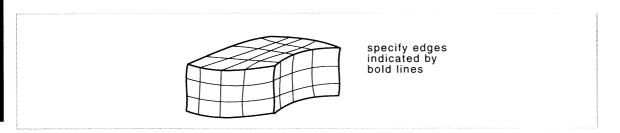


The ANALYTICAL DESC. DISCONTINUITY option is activated whenever you choose analytical description for a deformable body. Use this option to prevent the system from fitting a spline or a Coons patch through a particular region.

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



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*).

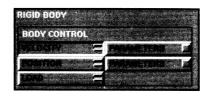
Motion Control of Rigid Bodies

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.

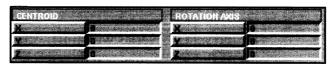


Contact>Contact Bodies>Rigid

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.

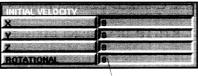


Contact>Contact Bodies>Rigid

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

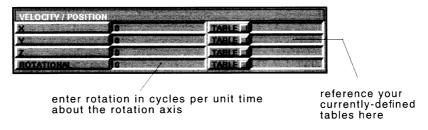
Very often, the rigid surface is not in initial contact with the deformable surfaces, and you may not be exactly sure when or where the rigid surfaces first touch. To account for this, use the INITIAL VELOCITY option in Rigid Body to prescribe an initial velocity of the body such that it will have rigid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



enter rotation in cycles per unit time about the rotation axis

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.

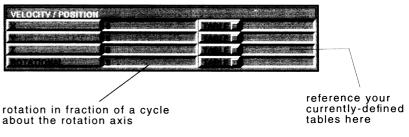


Contact>Contact Bodies>Rigid

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.



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	However the second seco
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

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:

is presented.	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.).

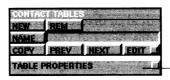
About Rigid w/Heat Transfer Bodies

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 146)

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).

EXAMPLE

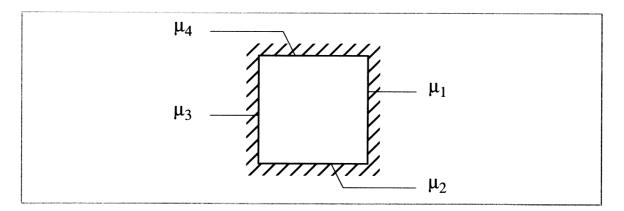
You might have a sheet forming problem where the default settings prescribe that every node in the sheet can contact some other region in the sheet. But as long as the sheet does not fold or wrinkle unto itself, it is computationally effective to indicate that the sheet will not contact itself.

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 (μ_1 , μ_2 , μ_3 , μ_4) of friction for each side of a square with the rigid surface.

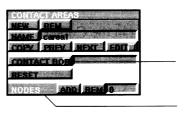


Using contact tables, you:

- 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

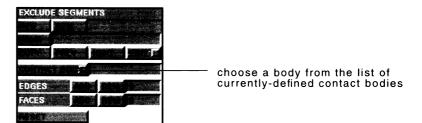
Using the options in Contact Areas, you can select:

- 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.

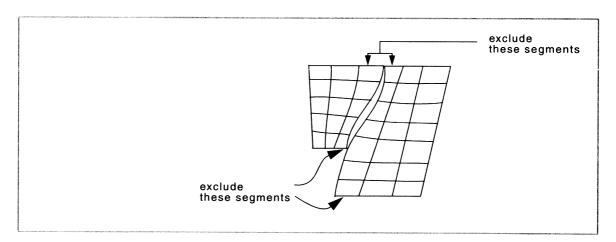
About Exclude Segments

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

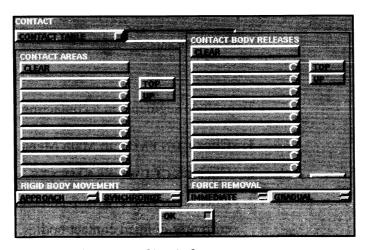
Use the Loadcase option to apply the contact information from your contact tables, contact areas, and exclude segments to your analysis. You can apply the contact information to loadcase types for the following loadcase classes:

- 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 Class.
- 2. Choose a loadcase class (e.g., Mechanical).
- 3. Choose a loadcase type (e.g., Static).
- 4. Choose CONTACT.



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.

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:

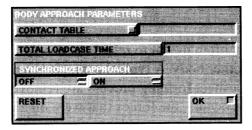
- 1. In the Contact menu, select the body from which the nodes are to be separated.
- 2. Choose a type of release, IMMEDIATE or GRADUAL.

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

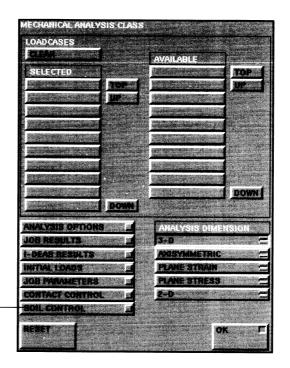
About Contact Control in Jobs

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.

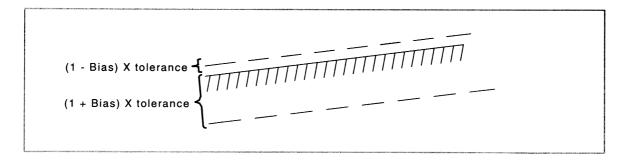


Jobs>(Analysis Class)>Contact Control

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:

REPORTED AND THE PROPERTY OF THE PARTY OF TH	Henry Cabour Street
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

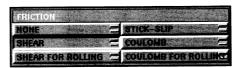
The default setting for the Separation Increment option is CURRENT. When separation occurs, the program separates the bodies within the increment.

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 141), 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:

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 164*) indicates the velocity at which they begin to slip.



Jobs>(Analysis Class)>Contact Control

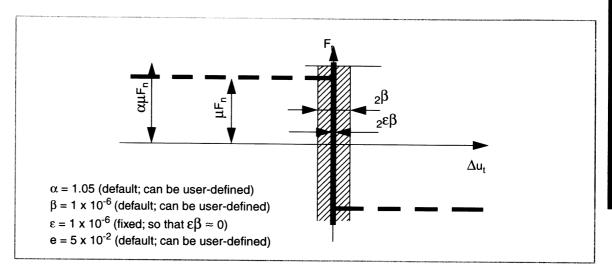
Using the Stick-Slip friction model, three parameters $(\alpha, \beta, and e)$ are available to control the numerical behavior. a represents a tolerance on the frictional force before sliding occurs. The node changes from stick to slip when

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

 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

Additional Information

Use the following table to find more information:

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

About Design Sensitivity and Optimization 171 Application of Design Variables and Design Constraints to Jobs 173

Design Variables

Types of Design Variables 175 About Bounds and Design Sensitivity 176

Design Constraints

Types of Constraints 180 Setting Displacement Constraints 181

Using Design Variables and Constraints

Associating Constraints with a Loadcase 186

Post-Processing Sensitivity and Optimization Results

About the Plotting of Sensitivity Results 192 About History Plots of Optimization Results 199 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

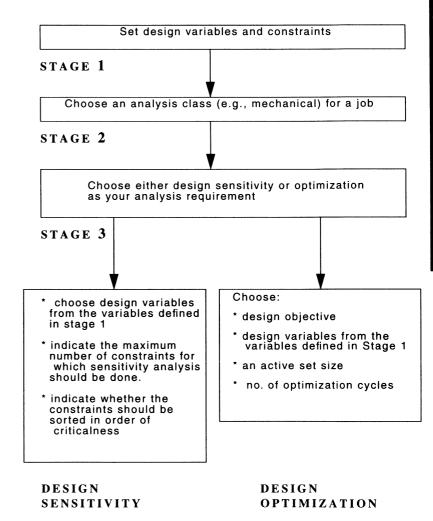
Where You Can Apply Design Optimization

Use the Design Optimization feature in structures that involve:

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

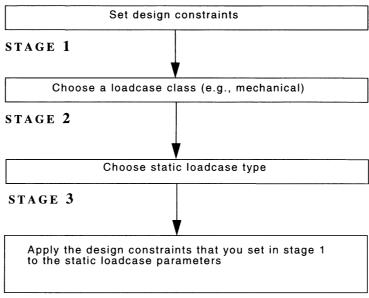
Application of Design Variables and Design Constraints to Jobs

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:



STAGE 4

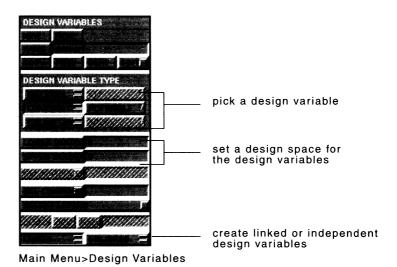
Design Variables

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.



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.

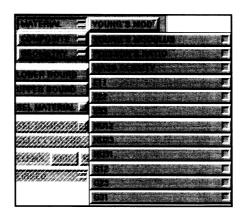
Choosing tight bounds constrains the optimization, but, in general, leads to a faster analysis and a more accurate design.

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.

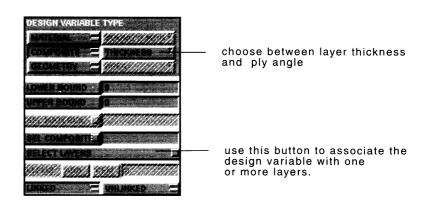


Design Variables>Material

- 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 oneto-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

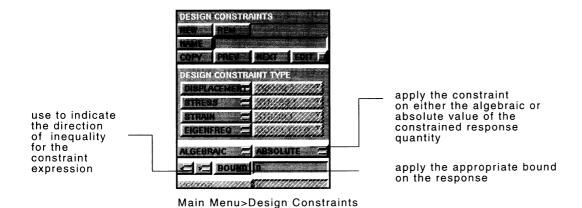
Types of Constraints

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.



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.

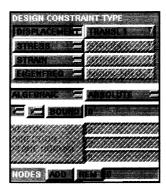
- 5. Choose algebraic or absolute value.
- 6. Add a node list to associate with the constraint.

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

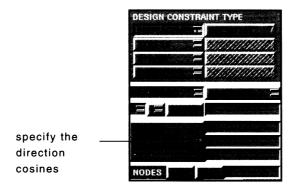
You must specify a vector direction when you use any of the following design constraints:

- 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.

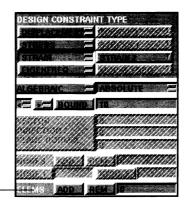


Main Menu>Design Constraints

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.



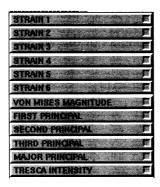
associate an element with, or dissociate an element from, a constraint

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 OF 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.

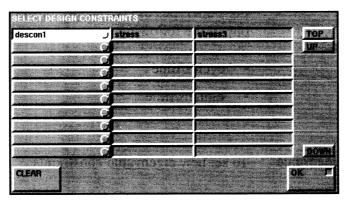
Using Design Variables and Constraints

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.

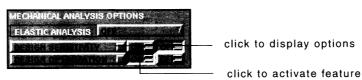


Loadcases>Mechanical>Static>Design Constraints

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

Design Sensitivity and Optimization as Mechanical Analysis Options

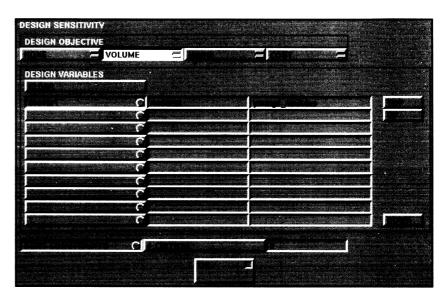
The design sensitivity and optimization features are paired in the Mechanical Analysis Options window.



Jobs>Mechanical>Analysis Options

Using Design Variables and Constraints in Design Sensitivity To use design variables and constraints in a design sensitivity analysis:

 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:

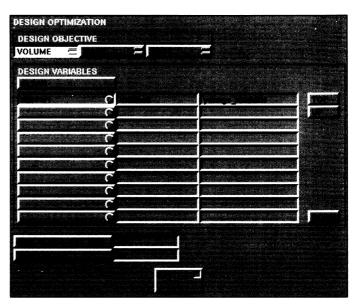
- per unit volume
- · per unit mass

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:

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 o K.

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:

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 DISPLACEMENT 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.

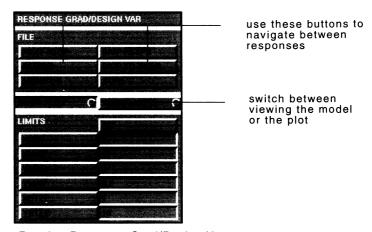
EXAMPLE

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:

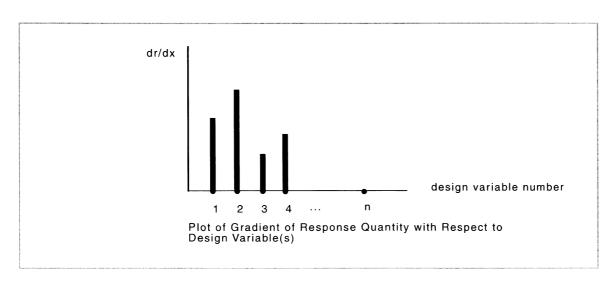
- bar charts—the gradient of a response quantity
- bar charts—the gradient of the objective function
- contour plots (similar to stress contours)—showing, on the finite element model, the element contributions to the specific response quantity.

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:

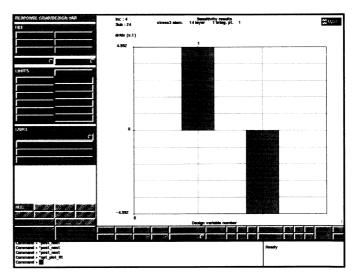
$$\nabla r_i = \begin{bmatrix} (dr_i)/(dx_1) \\ \dots \\ (dr_i)/(dx_n) \end{bmatrix} \quad \begin{array}{l} \text{where } x_j \text{ are the design variables} \\ \end{array}$$

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



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

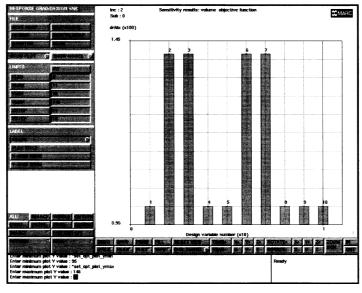
To obtain a meaningful plot, you might have to adjust the upper and lower limits.

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

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.

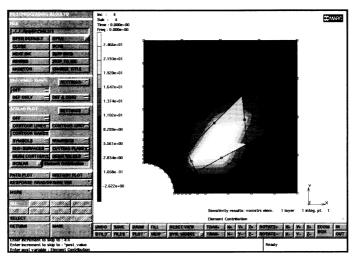
Creating Contour Plots of Element Contributions

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:

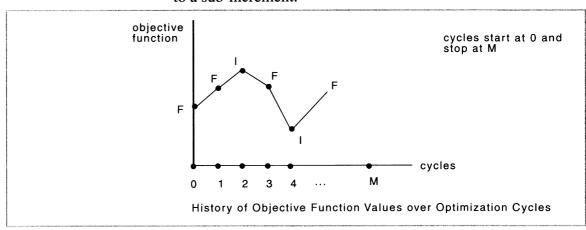
- history plots—the change in a response quantity or objective function over the optimization cycles.
- bar chart—the values of the design variables at a cycle.

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.

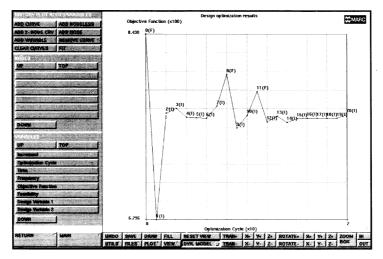
The program traces history plots over the optimization cycles (sub-increments), with each integer x-axis value corresponding to a sub-increment.



Creating History Plots of Objective Function Values

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

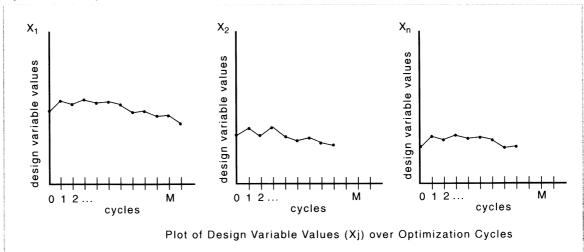
- 1. Choose Results>Open.
- 2. Using the file browser, open the appropriate optimization post file.
- 3. Click SCAN to view the increments.
- 4. Choose History Plot>Collect Data and enter the following information:
- first increment
- last increment
- step size
- 5. Choose Nodes/Variables>Add Nodeless
- 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

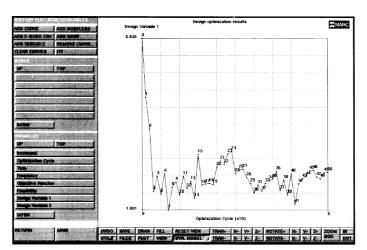
You can plot the change in the values of design variables over the optimization cycles using the same commands as in creating a plot for objective function.



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.

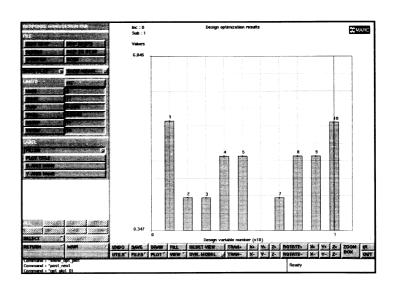


History Plot of Design Variable Values with Respect to Optimization Cycles

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., same
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

About the New Shell Elements 207
About Element 138 207
About Element 139 208
About Element 140 208
Associating Elements with a 3-D Membrane/Shell 209

New Rebar Elements

About Rebar Elements 210
Types of Rebar Elements 210
Generating and Identifying Rebar Elements 211
Specifying Rebar Material Properties 212
Associating Double Elements with Nodes 213

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:

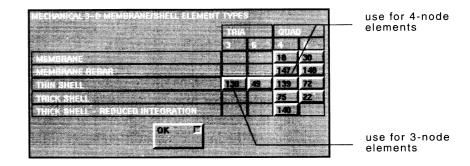
- six degrees of freedom
- uses hourglass stiffness matrix to insure that it is a stable element
- cheaper to use but not as accurate in the case of plasticity analysis
- about two-and-a-half times faster than Element 75 during the element assembly phase

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

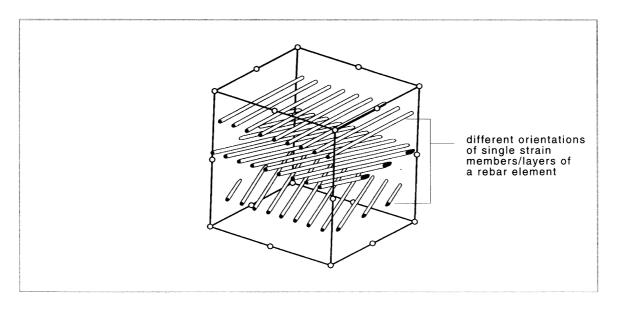
New Rebar Elements

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).

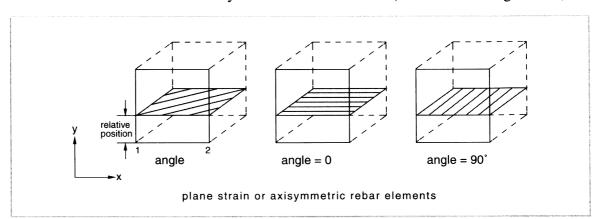
The orientation of single strain members in different layers of a rebar element may be different. Each rebar element may have up to five rebar layers oriented in a particular direction. Each one of the layers has a different material behavior.



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)



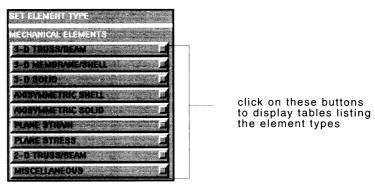
• axisymmetric rebar elements (four-node and eight-node)

- 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.



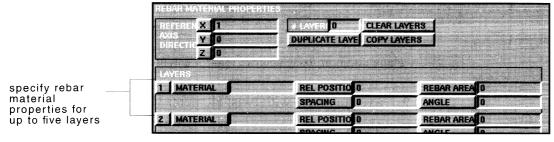
Jobs>Element Type

Specifying Rebar Material Properties

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

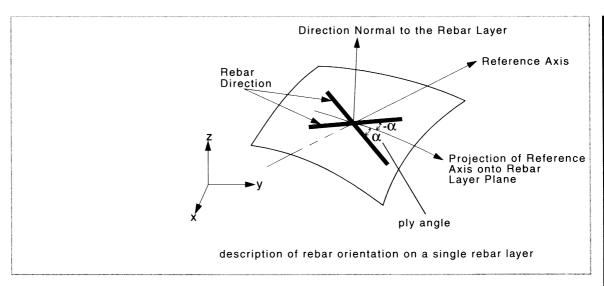
To specify rebar material properties:

1. Choose Material Properties>New Rebar-Define.



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)



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:

707	roter 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

About Fluid Analysis Options 218 About the Solver Option for Fluid Analysis 221

Boundary Conditions for Fluid Analysis

About Boundary Conditions for Fluid Analysis 222 About Centrifugal Loads 227

Initial Conditions for Fluid Analysis

About Velocity as Initial Condition Type 229 Coupled Analysis Considerations 229

Material Properties for Fluid Analysis

About Fluid Properties and Viscosity Models 231 Mass Density and Volumetric Expansion 233

Loadcases for Fluid Analysis

Applying the Fluid Steady State Parameters 235
About Iterative Procedures in Solution Control 237

Fluid Mechanics



Fluid Analysis

About Fluid Analysis

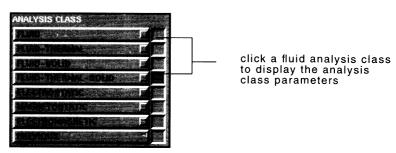
You can obtain solutions to problems in fluid mechanics by solving the Navier Stokes equations for the following fluid analyses in MARC:

- 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.



Jobs >More

About Fluid Analysis Options

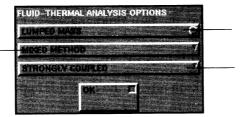
These are the fluid analysis options in MARC:

- 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.

click here to toggle between the Mixed and Penalty methods



click here to activate the lumped mass option

click here to toggle between Strongly Coupled and Weakly Coupled

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 fluidthermal 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.

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.

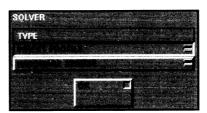
About the Solver Option for Fluid Analysis

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.

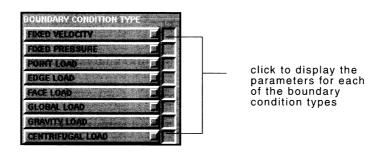
For a table of valid fluid element types, see Appendix B (p. 289).

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.

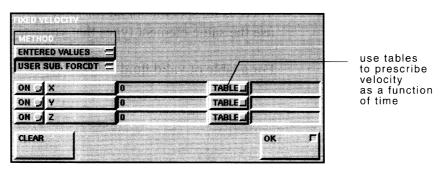


Boundary Conditions>Fluid

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.

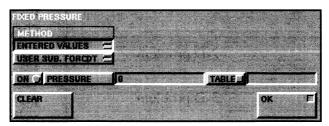
For transient analyses, you can use the user subroutine FORCDT (*See Vol. D: User Subroutines–FORCDT*) or tables to specify the time-dependent velocity.

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 219*). 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.

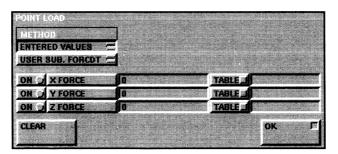


Boundary Conditions>Fluid>Fixed Pressure

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.

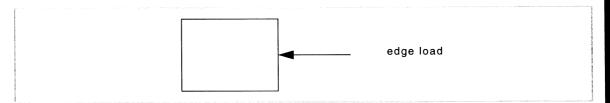
To prescribe the parameters for the Point Load boundary condition, choose Boundary Conditions>Fluid>Point Load.



Boundary Conditions>Fluid>Point Load

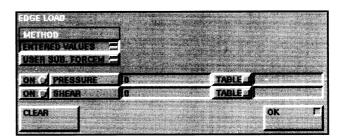
About Edge Loads

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.

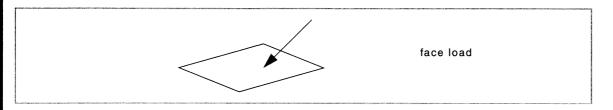
To prescribe the parameters for the Edge Load boundary condition, choose Boundary Conditions>Fluid>Edge Load.



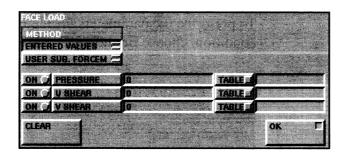
Boundary Conditions>Fluid>Edge Load

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.

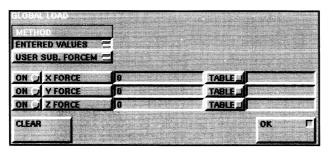


Boundary Conditions>Fluid>Face Load

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.

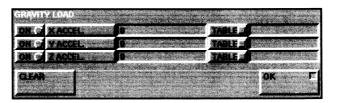


Boundary Conditions>Fluid>Global Load

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.

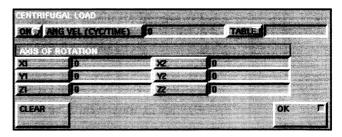


Boundary Conditions>Fluid>Gravity Load

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

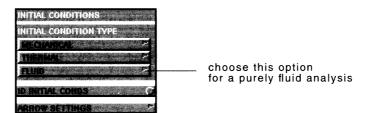
Initial Conditions for Fluid Analysis

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.

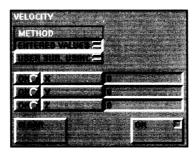
Picking good initial conditions (conditions that are close to the final solution) can result in improved convergence in the numerical analysis. However, they will not influence the final results.

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.

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 217), 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,	Action and the second s
Fluid	FLUIDS and assign all the fluid properties (See "About Fluid Properties and Viscosity Models" on page 231).
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

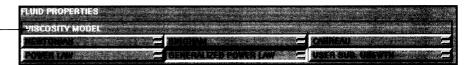
Use the Fluid Properties menus to assign the fluid properties for an analysis. Since the material is incompressible, you need to define the viscosity and the mass density of the fluid. To neglect the inertia effects, specify a small value for the mass density.

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:

Folia	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 behaviorrate (strain rate) behavior		
	as given by the relation:		
	$\tau_{ij} = \mu_0 d_{ij} + g D^{-1} d_{ij} \text{ if } \tau \ge g$		
	$d_{ij} = 0$ if $\tau < g$		
Carreau model fluids	The values of K, D, and n as given by the relation: $\mu = \mu_{\infty} + (\mu_0 - \mu_{\infty})(1 + K^2D^2)^{(n-1)/2}$		
Power Law	K, n, and D as given by the relation:		
fluids	$\tau_{ij} = \mu_0 K D^{n-1} d_{ij}$		
Generalized Power Law	K, η, D, A1, A2, A3, A4 as given by the relation:		
fluids	$\int \mu_0 K \exp(A_1 T + A_2 T^2) D_0^P D < D_0$		
	$\mu = \begin{cases} \mu_0 K \exp(A_1 T + A_2 T^2) D_0^P & D < D_0 \\ \mu_0 K \exp(A_1 T + A_2 T^2) D^P & D \ge D_0 \end{cases}$		
	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 fluidthermal 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

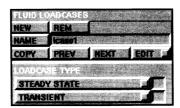
For each of the above loadcase classes, you can specify the following loadcase types:

- 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).

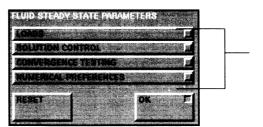


Loadcase >(Loadcase Class)

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.



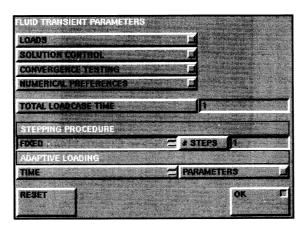
click on these buttons to define additional data

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 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

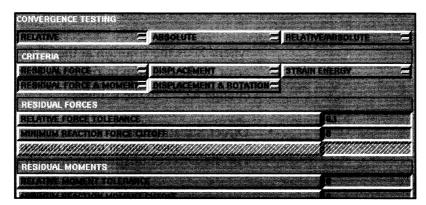
When you perform a coupled analysis with either the Strongly-Coupled or Weakly-Coupled method you should specify two separate control tolerances for the:

- 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 OF 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-	(Blatic Charleto)	
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

Mechanical Material Types for the Narayanaswamy Model 243

New Approaches in Plasticity Modeling

Multiplicative Decomposition and New Plasticity Procedures 247

Mooney-Rivlin and Ogden Formulations

About the Updated Lagrange Procedure 251

User Subroutines in Hypoelastic Properties

About HYPELA and HYPELA2 253

Experimental Data Fitting

About Experimental Data Fitting 256
Setting Damage Control Parameters 264
About the Viscoelasticity Model and Relaxation Spectra 266

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 thermorheologically 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.

- 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 o K.

- 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 o K.
- 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)	Marine Sale
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:

Form	Chapter(s)	line.
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.

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:

Folio:	refer to Chapter(s)	The same of the sa
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:

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

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:

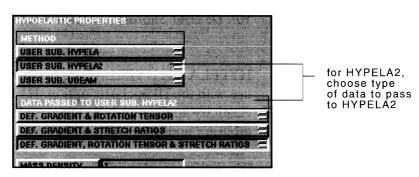
$$\underline{F} = \underline{R} \underline{U}$$

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	Mare Visited Commission of the Commission of the Commission of the
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

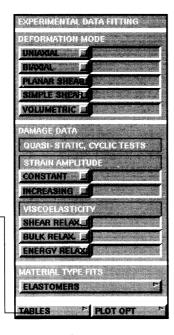
About Experimental Data Fitting

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

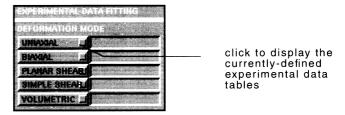
While it is possible to use just uniaxial data for curve fitting and extract the constants for elastomeric material models, you should also have experimental data for multiple types of experiments. This enables you to achieve material constants that can be used in practical engineering analysis. You 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 (£) and the engineering stress (σ) measures. The Foam model requires dilatational information (*See "About Forms of Data" on page 258*). The experimental data is read in as £, 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	$ \varepsilon_{\text{max}}, \sigma _{\varepsilon = \text{fixed}} $
Increasing Strain Amplitude	cycle count, $\sigma _{\epsilon = \text{fixed}}$ equispaced
Relaxation	time, σ equispaced

Although your data set may have three components, you'll see only the X and Y components in Mentat.

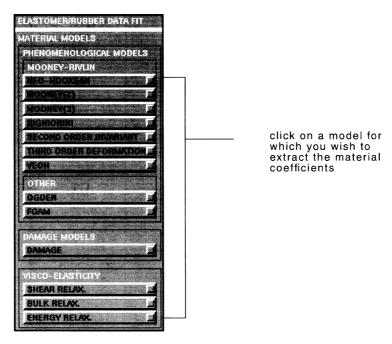
For Foam models, the Z-component is the dilatational information, (J). The relation between the deformation mode and form of J is given below:

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.

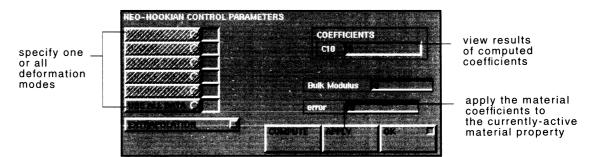
To display the Material Models menu, choose Material Properties>Experimental Data Fitting>Elastomers.



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.

 ${\bf MARC}$ 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.

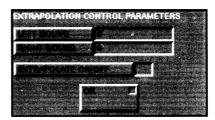
Use the following table to determine which deformation modes (or their combinations) to use:

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)	The second secon
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.

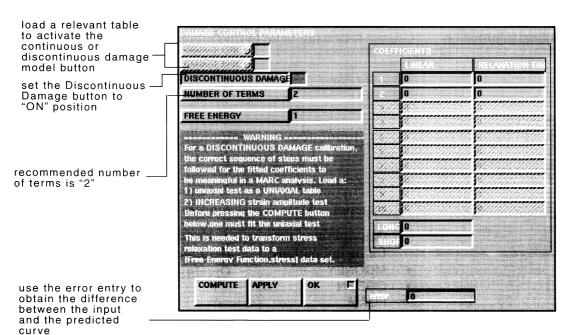
Use the Continuous damage model in MARC to calibrate the continuous damage terms. The model uses the "constant strain amplitude" data set.

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.



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, $(\epsilon_i^{max}, \sigma_i)$ 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, $(\boldsymbol{\epsilon}_{fix},\,\boldsymbol{\sigma}_0).$
- set the DISCONTINUOUS DAMAGE button to "Off" position.

Additional Information

Use the following table to find more information:

definitions of buttons	7, Material Properties	Mentat 3.1 Command
background information	7, Material Properties	Reference Vol. A
description of the option, DAMAGE	3, Model Definition Options	Vol. C
demonstration problem	E 7x-30	Vol. E

Viscoelasticity Model

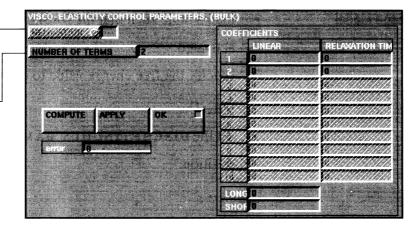
About the Viscoelasticity Model and Relaxation Spectra

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"

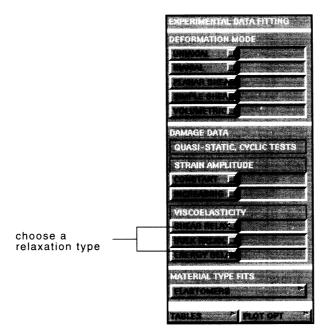


Material Properties>Experimental Data Fitting>Bulk Relax.

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:

- 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:

- 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, $(time_i, \sigma_i)$ to a table, $(time_i, W_i)$ where i = 1, 2, 3,...N.

Additional Information

Use the following table to find more information:

Por	chapter(s)	A Company of the Comp
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

About the Radiation View Factor Feature 273
Applying Radiation As Boundary Conditions 274
About Emitting and Incident Objects 275
About the Compute Radiation Viewfactors Window 276
Computing Viewfactors in Analysis 279

Analysis Integration

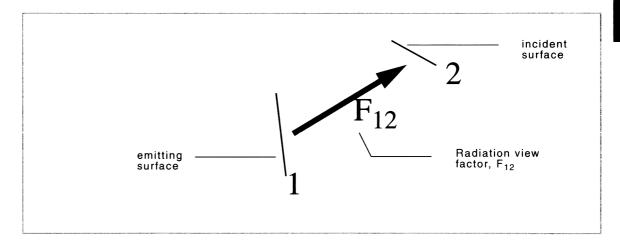
Using Radiation Viewfactors

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

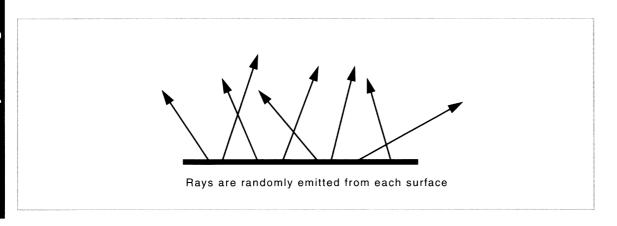
The radiation viewfactor is the fraction of radiation leaving the emitting surface that reaches an incident surface and depends solely on the geometric configuration of the surfaces.



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.

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

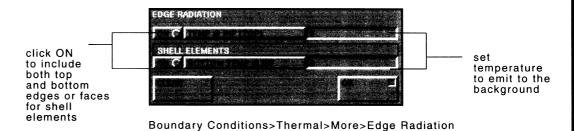


Applying Radiation As Boundary Conditions

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.



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.

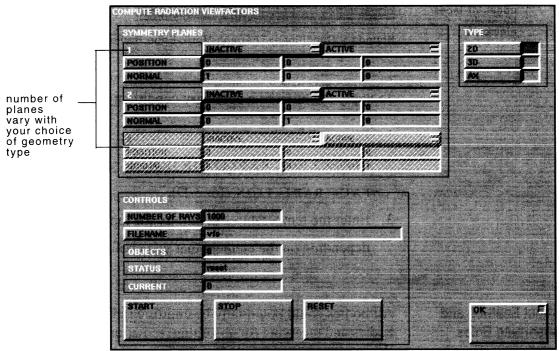
About Emitting and Incident Objects

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.



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	ТҮРЕ	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	

To 17	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.
- 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.

Additional Information

Use the following table to find more information:

For.	refer to Chapter(s)/ Section(s)	in
the description	12, Jobs/	Mentat 3.1
and functions of	Radiation	Command
the buttons	View Factors	Reference
description of the RADIATION parameter	2: Parameters	Vol. C
description of the VIEWFACTOR option	3: Model Definition Options	Vol. C

Appendixes

- A: User Enhancements 283
- B: Fluid Elements 289
- C: MARC Data Reader Support 293
- D: NASTRAN Writer Data Entries 301
- E: Command Line Parameters 311
- F: Demonstration Problems 317

Appendixes, A-F

A: User Enhancements

List of User Enhancements

MARC New Parameters 285
MARC New Model Definition 285
MARC New History Definition 286
New User Subroutines 286
New Element Types 286

 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)

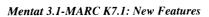
ı	
•	



B: Fluid Elements

Fluid Element Types

Planar Element Types 291 Axisymmetric Element Types 291 Three-Dimensional Element Types 292



Fluid Element Types

You can represent the fluid region in analysis using the following element types:

Planar Element Types

Nodes	Element	Туре
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	Туре	
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	Туре
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

List of Supported Parameters 295

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

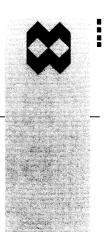
These are the model-definition options that the MARC Reader currently supports (for information on options, see *Volume C: Program Input, Chapter3–Model Definition Options*):

- 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)

- 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

- 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

- 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.

Minimum File Requirements 303
Bulk Data Cards Written by Mentat 3.1 304
Node Coordinates / Transformations 305
Material ID's Included in NASTRAN Geometry Cards 306
Loads and Boundary Conditions 308
Sample NASTRAN Bulk Data File 309

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:

Poulton	Valid Entry
At the start of the bulk data	BEGIN BULK
At the end of the bulk data	END DATA

Finite Element Connectivity

Bulk Data Cards Written by Mentat 3.1 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:

Mentat	NASTRAL TI	Noos
node coordinates	GRID	Transformations become coordinate systems
transformations	CORD2R	and their ID's are included in each GRID card.

Materials and Geometry

Material ID's Included in NASTRAN Geometry Cards

These are the material ID's included in NASTRAN geometry cards.

Mentat	NASTRAN STRAN	Notes	
Isotropic	MAT1	no table support; no plasticity	
Orthotropic	MAT3	no table support; no plasticity	
Anisotropic	MAT9	no table support; no plasticity	
3-D GEOMETR	IES	The second secon	
Truss	PBAR		
Elastic Beam	PBAR		
General Beam	PTUBE	only circula X-section supported	
Membrane	PSHELL		
Shear Panel	PSHELL		
Shell	PSHELL		
Solid	PSOLID		

	ann -
	A88
	2007
	A800
	200
	A800
Contract Con	286
	A80
	<i>1</i> 000
	. SEC
	200
	200
	2000
	<i>m</i>
	200
	600E

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

\$ NASTRA \$ BEGIN BU		Data File	e create	ed by Men	tat II==		
	NT CONNE	ECTIVITY==		=======	======		
CTETRA \$	12	8	126	127	125	117	
\$ TRANS	FORMATIO	ONS=====	======				=======================================
\$ CORD2R* * * \$	1	-4.40000 -2.20000	-	-2.20000	-	-2.200000 4.919350 -9.319350	0.000000 -4.400000
\$ NODAL	COORDI	NATES====	=====		======	:=======	
\$ GRID* *	10	0.000000	nn			-1.400000	-1.400000
GRID*	20					-0.200000	-2.000000
* GRID* *	117	-3.200000				-2.200000	-1.000000
GRID*	125					-2.200000	0.000000
* GRID* *	126	-4.400000				-2.200000	-1.000000
GRID* *	127	-4.400000				-1.300000	-1.000000
	IALS===			=======	======		
\$ MAT1* * \$	1	1.00000	o	29.00000			0.300000
\$ GEOME	TRY====		======	=======	======		
\$ PSOLID* *	8		1				
•	ARY CON	DITIONS==:	=====	=======			
\$ FORCE*	1		125		0		1.000000
* MOMENT*	1	10.000000	0 125	10.00000	0	0.000000	1.000000
* PLOAD*	4	20.00000)	0.00000		0.000000	25
* GRAV*	127 5		0			1.000000	0.320000
* \$	3	0.100000	-	0.10000	0	2.00000	3.323000
			=====	======	======		
MPC ENDDATA	4	10	1	1.00	020	0	0.001



E: Command Line Parameters

Mentat 3.1 Command Line Parameters

Table of Additional Command Line Parameters 313

MARC K7.1 Command Line Parameters

Table of MARC K7.1 Command Line Parameters 315

Command Line Parameters



Mentat 3.1 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 3.1" on page 25):

Operation: To specify	use the command line parameter	Option
aspect ratio of the screen that Mentat uses (0 <frac≤1)< td=""><td>-ar</td><td>frac</td></frac≤1)<>	-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_STA NDALONE
read all the menu files at startup,	-ra	

Operation: To specify	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 K7.1 Command Line Parameters

Table of MARC K7.1 Command Line Parameters For a description of the command line parameters in the MARC K7.1 shell script, refer to the following table:

Parameter	Rossible Names or Yalues	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.



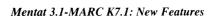


F: Demonstration Problems

Demonstration Problems

Lists of Demonstration Problems in MARC K7.1 319

MARC K7:1– Demonstration Problems



Demonstration Problems

Lists of Demonstration Problems in MARC K7.1

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 22quadratic 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	quad mesh 127
	quad/tri mesh 127
absolute distance value. See curvature-dependent di-	tri mesh 127
visions	algebraic value, putting a design constraint on 181
absolute temperature shift 244	allowable distortion, of quadrilateral elements. See
absolute tolerance	distortion parameters
application of 53	Allowed procedure, for increment splitting 162
when to use 53	all-quad mesh 128
see also adaptive plotting	Analytical Desc. Discontinuity option 145
absolute value	analytical description, of a deformable body 144
for design constraints 181	annotate, string to X-Windows file 314
see also algebraic value	appendixes, about 17
absorbing role, in heat transfer analysis 275	applying
ACIS	fixed divisions to curves 121
binary files 43	restrictions to individual curves 123
text files 43	ar command line parameter 313
see also Import-Export utility activating	arrow settings 68–70
constant dilatation 100	specifying arrow attributes 68
extended precision input 98	aspect ratio of the screen, command line parameter
activation energy/gas constant (H/R) 243	313
	associating elements with stress or strain constraints
Adaptive Load Stepping procedure 92–97	184
about 92	ASSUMED parameter 103
associated loadcase types 92	Assumed Strain formulation 102–103
general parameters for 93	activating 103
adaptive plotting 52–56	analysis classes 102
changing default settings 55	associated elements 102
default settings 55	AUTO STEP option. See Adaptive Load Stepping
default tolerance settings 52	procedure
pre-defined settings 56	average length, for curves 120
safety features 54	averaging, of nodal values 72
added vertex 115	axisymmetric element types, for fluids 291
adding file extensions. See filter button 41	axisymmetric rebar elements 211
additive decomposition 247	axisymmetric shells, specifying thickness for 178
Advancing Front mesher 127	
checklist for 126	axisymmetric with twist elements 211

Check Surfaces option 118 checklist, for meshers 126 back command line parameter (MARC) 315 back surfaces, identifying 142 backface polygons, identifying 142 backface polygons, identifying 142 backfaces. See backface polygons biaxial deformation mode 259 Bingham fluid model 231 bitplane threshold, specifying as a command line parameter 314 Body Approach parameter 158 Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser Check Surfaces option 118 checklist, for meshers 126 choosing a resolution 66 cleaning curves and surfaces. See repair geometr cleaning, loops 2-D curve loops 114 clearing curve divisions 125 coefficients, of energy relaxation tests 268 color codes, in contact analysis 142 Color Print button 63 command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatatior parameter	
back surfaces, identifying 142 backface polygons, identifying 142 backfaces. See backface polygons biaxial deformation mode 259 Bingham fluid model 231 bitplane threshold, specifying as a command line parameter 314 Body Approach parameter 158 Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves backface polygons cleaning curves and surfaces. See repair geometrocleaning, loops 2-D curve loops 113 surface loops 114 clearing curve divisions 125 coefficients, of energy relaxation tests 268 color codes, in contact analysis 142 Color Print button 63 command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser	
backface polygons, identifying 142 backfaces. See backface polygons biaxial deformation mode 259 Bingham fluid model 231 bitplane threshold, specifying as a command line parameter 314 Body Approach parameter 158 Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves browsing. See file browser cleaning, loops 2-D curve loops 113 surface loops 114 clearing curve divisions 125 coefficients, of energy relaxation tests 268 color codes, in contact analysis 142 Color Print button 63 command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command line parameters 25, 313 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
backface polygons, identifying 142 backfaces. See backface polygons biaxial deformation mode 259 Bingham fluid model 231 bitplane threshold, specifying as a command line parameter 314 Body Approach parameter 158 Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves browsing. See file browser cleaning, loops 2-D curve loops 113 surface loops 114 clearing curve divisions 125 coefficients, of energy relaxation tests 268 color codes, in contact analysis 142 Color Print button 63 command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100–101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	y
backfaces. See backface polygons biaxial deformation mode 259 Bingham fluid model 231 bitplane threshold, specifying as a command line parameter 314 Body Approach parameter 158 Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves brown side, of a 3-D rigid surface 142 browsing. See file browser 2-D curve loops 113 surface loops 114 clearing curve divisions 125 coefficients, of energy relaxation tests 268 color codes, in contact analysis 142 Color Print button 63 command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
Bingham fluid model 231 bitplane threshold, specifying as a command line parameter 314 Body Approach parameter 158 Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes breaking curves 116 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser clearing curve divisions 125 coefficients, of energy relaxation tests 268 color codes, in contact analysis 142 Color Print button 63 command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
bitplane threshold, specifying as a command line parameter 314 Body Approach parameter 158 Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes breaking curves 116 when to use 124 see also matching curves browsing. See file browser coefficients, of energy relaxation tests 268 color codes, in contact analysis 142 Color Print button 63 command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command line parameters 25, 313 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
bitplane threshold, specifying as a command line parameter 314 Body Approach parameter 158 Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes breaking curves 116 when to use 124 see also matching curves browsing. See file browser coefficients, of energy relaxation tests 268 color codes, in contact analysis 142 Color Print button 63 command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command line parameters 25, 313 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
Body Approach parameter 158 Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes breaking curves 116 when to use 124 see also matching curves brown side, of a 3-D rigid surface 142 browsing. See file browser Color Print button 63 command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command line parameters 25, 313 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
Bottom Temperature at Infinity parameter, for controlling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes breaking curves 116 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser command line description of parameters 26 minimum requirements 26 submitting an actual job 26 command line parameters 25, 313 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
trolling emission of radiation 274 boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser description of parameters 26 minimum requirements 26 submitting an actual job 26 command line parameters 25, 313 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
boundary conditions, for fluid analysis 222 bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser minimum requirements 26 submitting an actual job 26 command line parameters 25, 313 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100–101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
bounds for design constraints 180 for design variables 176 Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves brown side, of a 3-D rigid surface 142 browsing. See file browser submitting an actual job 26 command line parameters 25, 313 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100–101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
for design constraints 180 command line parameters 25, 313 for design variables 176 command/button sequence 18 Box Pick method. See picking nodes bp command line parameter 313 common retained node 75 composite design variable about 175 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser command line parameters 25, 313 command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100—101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
for design variables 176 Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser command/button sequence 18 common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100–101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
Box Pick method. See picking nodes bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser common retained node 75 composite design variable about 175 using 177 Constant Dilatation parameter 100–101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
bp command line parameter 313 breaking curves 116 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser composite design variable about 175 using 177 Constant Dilatation parameter 100–101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
breaking curves 116 when to use 124 see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser about 175 using 177 Constant Dilatation parameter 100–101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
when to use 124 see also matching curves Constant Dilatation parameter 100–101 brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser using 177 Constant Dilatation parameter 100–101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
see also matching curves brick rebar elements 211 brown side, of a 3-D rigid surface 142 browsing. See file browser Constant Dilatation parameter 100–101 when to use 100 CONSTANT parameter. See Constant Dilatation parameter	
brick rebar elements 211 when to use 100 brown side, of a 3-D rigid surface 142 CONSTANT parameter. See Constant Dilatation parameter	
brown side, of a 3-D rigid surface 142 CONSTANT parameter. See Constant Dilatation parameter	
browsing. See file browser parameter	
-	
buckle solutions, using Lanczos method 90–91 constant strain amplitude, form of experimental d	ata
about 90 258	
parameters 90 contact 137–168	
bulk data entries, for NASTRAN writer. See NAS- contact analysis	
TRAN writer components of 139	
bulk relaxation tests, computing 268 physical bodies 140	
separation procedures 139	
Contact Areas option, to indicate contact region	154
contact bodies, types of 140	
contact body release, types of 157	
calibrating quasi-static, stress-strain response curves 257 contact body, choosing a type of 141	
Carreau fluid model 231 contact control	
in jobs 159	
centrifugal loads 227 user subroutines for 160	
centroid 71 contact distance, in contact table 152	
Centroid option, for rigid bodies 147 Contact Heat Transfer Coeff. button 150	
changing the default parameters. See Numerical contact information	
Preferences option 104 applying to jobs 159	
Check Curves option 117 applying to loadcases 156	

Contact Tables option 152	cusp points 115
contact tolerances 161	
continuous contour lines 72 Continuous Damage model about 264	D
checklist for 265	damage models
control tolerances, for coupled analysis 237	considerations for data sets 265
controlling load steps. See Adaptive Load Stepping	setting control parameters 264
procedure	dashpot
controlling the emission of radiation, boundary con-	values 76
ditions for 274	see also springs/dashpots
conventions, in this guide 18	data types. See Import-Export utility
convergence testing parameters, for solid region in	def command line parameter (MARC) 315
coupled analysis 238	define contact bodies 139
Coons patches 145	defining nodal vectors. See user-defined post vari-
correcting orientation of contact bodies 143	ables deformable bodies
cost, minimizing. See design objective	
Coulomb friction model 164	analysis parameters 143
Coulomb-for-Rolling friction model 165	discrete and analytical descriptions 144
coupled analysis 100	meshing 143
coupled analysis considerations, for fluid analysis 229	deformation mode, form of experimental data 258 Delaunay mesher
coupled analysis, using rigid surfaces 150	about 131
creating a view snapshot 57	advantages of 131
curvature-dependent divisions	using 131
applying 122	deleting file extensions. See filter button.
benefits of 121	density
curve divisions 120–126	of mesh 131
curvature-dependent 121	design constraints 171, 180–191
defaults 125	applying to related loadcases 174
fixed average length 121	types of 180
menu 120	design objective 171
types of 120	design optimization applications of 172
curve facetting. See adaptive plotting, default settings	components of 171
curve geometry, checking 117	design sensitivity and optimization 171-204
curve intersection, setting tolerance values for 118	design variables 171, 175–179
curve, expanding elements along 134	linking and unlinking 175
curves	picking 175
breaking 116	types of 175
modifying 115	detected loops 123
splitting 115	determining contact tolerances 139
curves, drawn and actual. See Adaptive Plotting fea-	determining number of segments 54
ture	df command line parameter 313

Direct Profile solver. See solver options Direct Profile solver. See solver options	zooming a view in 34
Direct Substitution procedure, for solution control	
237 directed	E
relative rotation 182	E
relative rotation 183	adga lands 225
rotation 182	edge loads 225
translation 182	edge radiation, menu 274 edge segment length. <i>See</i> curvature-dependent divi-
direction cosines. See vector direction	sions
directories. See file browser.	eigenfrequency analysis. See design constraints
discontinuous contour lines 72	eigenfrequency design constraints 180
Discontinuous Damage model 264	eigenvalue constraints
	setting 185
discrete description, of a deformable body 144 Discrete Kirchoff theory. <i>See</i> Element 138	types of 185
displacement design constraints	units of frequency 185
about 180	elastic data, considerations for 248
removing from a node 182	elastomer analysis. See experimental data fitting
displaying	element extrapolation 71–72
changes to files 41	components of 71
directories or files 41	methods 71
distance tolerance bias value 161	
	options 72
distortion parameters about 128	element types
	new in MARC 286
all-tri mesh 128	see also fluid element types 286
practical values for 128	elements adding or removing, for stress or strain
range of values 128	constraints 184
specifying 128	see also new shell elements
documentation	elements, associating with a 3-D membrane or shell
additional MARC K7.1 documents 21	209
online 21	emitting roles, in heat transfer analysis 275
online help 20	energy relaxation tests, coefficients for 268
printed 20	enhancements
dots per inch (dpi) 64	MARC K7.1 15
double elements with nodes, associating 213	Mentat 3.1 15
DXF	
entity summary 45	entity 175
options 45 , 47	entity summary, for VDAFS 49
summary reports 45	Environment Sink Temperature button 150
version number 45	equispaced, forms of data 258
see also Import-Export utility	Exclude Segments option 155
dynamic viewing 33	expanding elements along a curve 133–135
rotating a view 34	experimental data fitting 256–269
translating a view 34	about 256
using the Alt key (for UNIX systems only) 33	applying material coefficients 262

damage models 264	types of 217
deciding on a mode 262	fluid element types 221, 291-292
determining the material coefficients 261	fluid models. See viscosity models
experimental tests 257	fluid properties, assigning 231
forms of data 258	fn command line parameter 313
reading in experimental data 257 viscoelasticity model 266	font name, specifying as a command line paramete 313
EXTENDED parameter. See extended precision input	font size, increasing 66 FORCDT user subroutine 223
extended precision input 98–99	forcing, even or odd divisions 123
about 98	format changes. See extended precision input
activating 98	fraction parameter 244
format changes 99	free curves, removing 112
extrapolating, integration point values 71	Frequency Difference feature 185
Extrapolation feature, for experimental data fitting	Friction Coefficient option 143
263	friction coefficient, in contact bodies 152
F	friction model, specifying for contact 140 friction models
•	about 164
face loads 226	types of 164 Full Newton-Raphson procedure, for solution con-
facets, of arrows 69	trol 237
FIDAP. See Import-Export utility	401207
file browser 39–41	
about 39	G
buttons 41	G.
operations 39	gaps in loops. See cleaning loops
files	generalized plane-strain rebar elements 210
name and location of 40	Generalized Power Law fluid model 231
pathname of 40	generating, meshes 109
selecting 40	geometry
filter button 41	adding or removing 112
fixed 121	checking 117
fixed divisions for curves 120	summary information 117
Fixed Velocity option 222	geometry design variable 175
flipping elements, curves and surfaces 143	types of 178
fluid analysis 215–239	using 179
about 217	GIF 57
boundary conditions 222	glass transition temperature (Tg) 243
initial conditions 228	glass-type materials. See Narayanaswamy model
loadcases for 234	Gradual option, for removing forces 157
material properties for 230	gravity loads 227
options 218	Gray button. <i>See</i> PostScript plotting attributes grayscale, run Mentat in 25

guidelines	Image Postscript menu. See PostScript plotting at- tributes
for 2-D and surface meshers 126	Import-Export utility 42–50
guidelines, for loops 126	export feature 43
	import feature 42
Н	importing a MARC data file 51
11	incident surfaces 273
Hardware Sparse solver. See solver options	increasing font size 66
help directory, specifying path to 313	increasing strain amplitude, form of experimental
high setting. See adaptive plotting, pre-defined set-	data 258
tings	increment splitting options 162
higher ID, of a contacting body 163	individual curves, applying restrictions to 123
history definitions, new in MARC 286	individual modes, prescribing frequencies for 185
hollow cross-section 178	initial conditions, for fluid analysis 228
hollow elements, with uniaxial stiffness. See rebar	Initial Velocity option 148
elements	input geometry considerations 125
hourglass stiffness matrix. See Element 140 208	input/output (I/O). See file browser operations
hp command line parameter 313	integration point values 71
HYPELA and HYPELA2, about 253	inter-element averaging. See nodal averaging
hypoelastic properties, user subroutines for 253	interface closure distance, in contact bodies 152
	interior side
	for 2-D contact analysis 142
I	for 3-D contact analysis 142
is a summand line management 214	intersecting loops 118
ic command line parameter 314	IRIS RGB 57
ID Contact button 142 I-DEAS. <i>See</i> Import-Export utility	isolate, an element 71
identify (MARC command line parameter)	Iterative procedure, for increment splitting 162
database file for substructures 315	iterative procedures, for solution control 237
job that created temperature file 315	
job/input file 315	_
previous job for RESTART 315	J
viewfactor file 315	
IGES	jid command line parameter (MARC) 315
summary report 47	jobs, contact control in 159
see also Import-Export utility	JPEG 57
IGES input file	changing default attributes 60
getting a color 47	creating view snapshot file 59
space curves 47	default attributes 60
specify a tolerance 47	Quality button 60
version number 47	setting file attributes 59
image file	Smoothing button 60
setting attributes 63	
image files	

printing 62

K	when to use 124
V 1 F	see also breaking curves
Kachanov Factor coefficients, computing 264	material design variable 175
	material mass, minimizing. See design objective material properties, for fluid analysis 230
L	maximum depth. See adaptive plotting safety fea- tures
Lanczos method. See buckle solutions, using Lanczos method	Mean-Normal procedure, for plasticity modeling 247
large-strain formulation 248	membrane rebar elements 211
layer thickness 177	Mentat entity summary. See DXF options
length, for preprocessing arrows 68	menu files, read at startup. See command line pa-
limitations, on large displacements of solids 229	rameters
linking design variables 175	menu system
links 75–85	enhancements 28
multiple springs 76	resizing windows 29
resetting program parameters 84	stand-alone static menus 30
types 75	window parenting 31
UFORMS subroutine 85	mesh generation 109–132
listing elements with a 3-D membrane/shell elements 209	mesh, projecting in the direction of a curve 133 meshers
load stepping criteria	Advancing Front 127
about 94	Delaunay 131
for element information 94	minimum depth. See adaptive plotting safety features
for nodal values 94	missing surfaces. See trimming surfaces
settings bounds for 95	Mixed method 218
specifying multiple ranges for 96	mixing formats. See extended precision input
types of 94	ml command line parameter 313
load, as rigid body motion control 149	Modal Frequency feature 185
Loadcase option, for contact analysis 156	mode, for all arrows 68
loadcases, for fluids 234	model definitions, new in MARC 285
log file, creating 25	modes, prescribing separations between 185
lower ID, of a contacting body 163	modifying, curves 115
Lumped Mass method 218	Monte Carlo method, of viewfactor computation 273
	Mooney-Rivlin formulations
M	about 251
	considerations for 251
MARC Data Reader support 293	motion control, of rigid bodies 147
model-definition options 297	MOTION subroutine. See user subroutines, for con-
parameters 295	tact control
MARC Input. See Import-Export utility	mp command line parameter 313
matching curves	MS-Windows BMP 57
about 124	multiple springs. See links 75

multiple ties 76	node ID's 87
multiplicative decomposition 247	specifying 86
•	node number, associated with the rigid surface 149
	node path 86–89
N	picking 88
	specifying 88
N to 1 option 75	storing 89
N to N option 75	nodes, excluding 155
Narayanaswamy model 243–246	non-equispaced data set 265
applying model parameters 244	normal stress on a plane 183
mechanical material types 243	nosolids command line parameter 314
parameters for 243	Numerical Preferences option 104–105
NASTRAN writer, bulk data entries 301	about 104
bulk data cards 304	associated analysis classes 104
finite element connectivity 304	NURB description. See rigid bodies, description
geometry cards 306	
loads and boundary conditions 308	_
material ID's 306	0
minimum file requirements 303	
node coordinates and transformations 305	objective. See design objective
sample bulk data file 309	Ogden formulations
NASTRAN. See Import-Export utility	about 251
negative side, setting bounds on 181	considerations for 251
new enhancements, in MARC	open loop 118
element types 286	OpenGL graphic system, using. See command line
history definitions 286	parameters operations
model definitions 285	for cleaning loops 113
parameters 285	for geometry repair 111
user subroutines 286	optimization. See design sensitivity and optimiza-
new shell elements 207–209	tion
Element 138 207	
Element 139 208	_
Element 140 208	Р
new stress or strain constraints, setting 184	
Newtonian fluid model 231	Page Height button. See PostScript plotting at-
Next button, using to ignore separation 164	tributes Page Width button. See PostScript plotting at-
nodal averaging 72	tributes
Nodal Force method, usage 165	PARAMETER model definition option 105
Nodal Stress method, usage 165	parameters
nodal ties 76	command line 313
node at midpoint, creating 136	parameters, new in MARC 285
•	•
node lists 80, 86–89 indicating end of list 86	parenting of windows 31 path, specifying as a command line parameter

to directory containing executables 313	parameter 313
to directory containing material database 313 to help directory 313	prog command line parameter (MARC) 315
to menu directory 313	_
PATRAN. See Import-Export utility	R
penalty function 219	
Penalty method 218	ra command line parameter 313
physical bodies, in contact analysis 140 picking nodes	Radial Return procedure, for plasticity modeling 247
box pick 87	radiation viewfactors 273–280
polygon pick 87	radiation, applying as a boundary condition 274
pid command line parameter (MARC) 315	radius, of hollow cross-sections 178
planar element types, for fluids 291	ray distribution, for viewfactor computation 274
plane normal, specifying 182	rebar elements 210–214
plane-strain rebar elements 210	about 210
plasticity modeling 247–250	identifying 211
element considerations 249	orientation of single strain members 210
types of procedures 247	types of 210
plotting PostScript image files 62–67	rebar orientation 212
ply angle 177, 212	relative
point at midpoint, creating 136	rotation about a vector direction 182
point loads 224	translation parallel to a vector direction 183
polar decomposition 254	relative tolerance. See adaptive plotting
Polygon Pick method. See picking nodes	relaxation spectra 266
pop-up menus, treating as independent menus. See	relaxation table, using 267
command line parameters	relaxation, form of experimental data 258
position, as rigid body motion control 149	releasing nodes in contact 157
positive side, setting bounds on 181	removing parent attributes. See parenting of win-
POST model definition option 107	dows
post-processing of nodal variables. See user-defined	repair geometry 111–119
post variables PostScript 57	operations 111
	Repetitions button 134
creating the view snapshot file 58 plotting attributes 58	Report Files button. <i>See</i> VDAFS options rescan. <i>See</i> file browser
	resolution, of PostScript images 64
saving a file 59	response quantities. See design sensitivity and opti-
sending an image file to a printer 59 , 63	mization
specifying printer destinations 63	retained node per constraint 76
Power Law fluid model 231	retained nodes 75 , 76
procure docree of fine days G. Mindows I.	rid command line parameter (MARC) 315
pressure degree of freedom. See Mixed method	rigid bodies
previously-applied curve divisions, removing 125 procedure file 25	about 146
procedure file to run, specifying as a command line	description 150
procedure the to run, specifying as a command line	meshing of 146

motion control of 147	Shear-for-Rolling friction model 164
rigid body 140	shell scripts 26
rigid body, with heat transfer 140	sid command line parameter (MARC) 315
root window 29	single-sided contact, using 163
rotating a view. See dynamic viewing	size command line parameter 313
Rotation Axis option, for rigid bodies 147	size of window, specifying as a command line pa-
rotation tensor (R) 254	rameter 313
run (MARC command line parameter)	slipping conditions 166
program in background 315	small displacements of solid, in fluid analysis 229
program in foreground 315	small-strain formulation 248
program with or without user subroutines 315	Snapshot button. See creating a view snapshot
run Mentat, without ACIS solids modeling code.	solid arrows 69
See command line parameters	solution control, of fluid analysis 237
running MARC 26	solver options 221
running Mentat 3.1 25	specific heat matrix, lumping. See Lumped Mass method
	spline, through the nodes 145
S	Split Curves command 115
	splitting curves 115
save command line parameter (MARC) 315	springs
save the generated or existing module, MARC com-	N to N springs 76
mand line parameter. See save command line	springs/dashpots
parameter	menus 77
scaling, length of arrows. See arrow settings	using the N to 1 feature with 82
SDRC model 46	using the N to N feature with 83
segments, excluding. See Exclude Segments option	USPRNG subroutine 82
Sel Composite button 178	Staggered procedure. See weakly coupled
selecting a file. See file browser sensitivity. See design sensitivity and optimization	stand-alone static menus 30
separated nodes, preventing re-contact of 157	steady state fluid problem, prescribing final veloci-
	ties 223
separation force, in contact table 152	steady state, fluid loadcase type 234
separation procedures 163	step ratio, for adaptive load stepping 94
separations per increment 164 SEPFOR subroutine. <i>See</i> user subroutines, for con-	Stick-Slip friction model 165
tact control	Stick-Slip option
SEPSTR subroutine. See user subroutines, for con-	about 166
tact control	available parameters 166
servo links 76	strain design constraints 180
constraint equation 80	stress design constraints 180
using the N to 1 feature with 80	stress relaxation table 266
using the N to N feature with 81	stretch tensor(U) 254
setting a design space 175	strongly coupled 218
setting PostScript image file attributes 63	Suppressed procedure, for increment splitting 162
Shear friction model 164	surface facetting. See adaptive plotting, default set-
	tings
shear relaxation tests, computing 268	·o-

symmetry bodies 151 symmetry planes, in viewfactor computation 277 symmetry rigid body 140 synchronized approach, in contact analysis 158 Table Properties feature, in contact tables 152 tangential force, at previous iteration 167 Temperature at Infinity parameter, for controlling emission of radiation 274 thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tiri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tying types 78 UCONTACT subroutine. See user subroutines, for contact control UHTCOS subroutine. See user subroutines	surface geometry, checking 117 surface parametric space 114	U
symmetry planes, in viewfactor computation 277 symmetry rigid body 140 synchronized approach, in contact analysis 158 Table Properties feature, in contact tables 152 tangential force, at previous iteration 167 Temperature at Infinity parameter, for controlling emission of radiation 274 thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 point pages 78	•	UCONTACT subroutine. See user subroutines for
Symmetry rigid body 140 Synchronized approach, in contact analysis 158 Table Properties feature, in contact tables 152 tangential force, at previous iteration 167 Temperature at Infinity parameter, for controlling emission of radiation 274 thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 translating a view. See dynamic viewing translating dat types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 trimming, surfaces	-	
Table Properties feature, in contact tables 152 tangential force, at previous iteration 167 Temperature at Infinity parameter, for controlling emission of radiation 274 thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient parameters for fluid analyses, applying 236 transient parameters for fluid analyses, applying 236 transiating a view. See dynamic viewing translating a total types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimmting, surfaces 112 print threa 78	symmetry rigid body 140	
Table Properties feature, in contact tables 152 tangential force, at previous iteration 167 Temperature at Infinity parameter, for controlling emission of radiation 274 thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient parameters for fluid analyses, applying 236 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating a view. See of mport-Export utility version number 49 see also Import-Export utility version number 49 see also meshers trimming, surfaces 112 trime tynes 78	synchronized approach, in contact analysis 158	UHTCOE subroutine. See user subroutines, for con-
Table Properties feature, in contact tables 152 tangential force, at previous iteration 167 Temperature at Infinity parameter, for controlling emission of radiation 274 thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tviing types 78	Т	UHTCON subroutine. See user subroutines, for con-
tangential force, at previous iteration 167 Temperature at Infinity parameter, for controlling emission of radiation 274 thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 trine types 78		undo, box or polygon pick 87
Temperature at Infinity parameter, for controlling emission of radiation 274 thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 typing types 78	Table Properties feature, in contact tables 152	UNEWTN user subroutine, for fluid analysis 239
Temperature at Infinity parameter, for controlling emission of radiation 274 thermal expansion property, in Narayanaswamy model 245 thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 typing types 78		•
thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78		_
thermal expansion property, in Narayanaswamy model 245 thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 types 78e Updated Lagrange procedure 251 UPOSTV subroutine activating 106 additional information 107 Upper Bound button 176 user command line parameter (MARC) 315 user enhancements, list of 285–287 user subroutines for contact control 160 new in MARC 286 User-Defined fluid model 231 user-defined post variables 106–107 V validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		•
thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient parameters for fluid analyses, applying 236 transient parameters for fluid analyses, applying 236 transiating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78		
thermo-rheologically simple materials 243 three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 Tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tvine types 78 activating 106 additional information 107 Upper Bound button 176 user command line parameter (MARC) 315 user enhancements, list of 285–287 user subroutines for contact control 160 new in MARC 286 User-Defined fluid model 231 user-defined post variables 106–107 V validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		
three-dimensional element types, for fluids 292 ti command line parameter 314 tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 trime types 78 additional information 107 Upper Bound button 176 user command line parameter (MARC) 315 user enhancements, list of 285–287 user subroutines for contact control 160 new in MARC 286 User-Defined fluid model 231 user-defined post variables 106–107 V validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		activating 106
tied nodes 76 TIFF 57 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 Upper Bound button 176 user command line parameter (MARC) 315 user enhancements, list of 285–287 user subroutines for contact control 160 new in MARC 286 User-Defined fluid model 231 user-defined post variables 106–107 V validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		
translating data types. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tolerance options, for curvature-dependent curve divisions 122 tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating data types. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 translating at views 78 to the first of the first of 285–287 user subroutines for contact control 160 new in MARC 286 User-Defined fluid model 231 user-defined post variables 106–107 V validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		
tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78 user enhancements, list of 285–287 user subroutines for contact control 160 new in MARC 286 User-Defined fluid model 231 user-defined post variables 106—107 V validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		
tolerance options, for curvature-dependent curve divisions 122 tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 size 131 size also meshers trimming, surfaces 112 tving types 78 user subroutines for contact control 160 new in MARC 286 User-Defined fluid model 231 user-defined post variables 106—107 V validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		- · · · · · · · · · · · · · · · · · · ·
tolerance value for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78 Iuser-Defined fluid model 231 user-defined post variables 106—107 V validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		
for curve intersections 118 for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78		for contact control 160
for intersecting loops 119 tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78 User-Defined fluid model 231 user-defined post variables 106—107 V validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		new in MARC 286
tolerance, in adaptive plotting 52 topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78 user-defined post variables 106–107 Validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		User-Defined fluid model 231
topology information. See geometry touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78		
touching bodies about 153 selecting 153 transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 size 131 see also meshers trimming, surfaces 112 tving types 78 validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		F
about 153 selecting 153 transient parameters for fluid analyses, applying 236 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78 validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		
selecting 153 transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 size 131 see also meshers trimming, surfaces 112 tving types 78 validating IGES entities. See IGES options variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		V
transient parameters for fluid analyses, applying 236 transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78 variables. See design variables VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		•
transient, fluid loadcase type 234 translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 vector direction, specifying 182 velocity transition 131 see also meshers trimming, surfaces 112 tving types 78 VDAFS Mentat entity summary 49 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		validating IGES entities. See IGES options
translating a view. See dynamic viewing translating data types. See Import-Export utility tri-mesh shape 131 size 131 vector direction, specifying 182 velocity transition 131 as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		•
translating data types. See Import-Export utility tri-mesh shape 131 size 131 transition 131 see also meshers trimming, surfaces 112 tving types 78 version number 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		
tri-mesh shape 131 size 131 vector direction, specifying 182 velocity transition 131 see also meshers trimming, surfaces 112 tving types 78 velsion humber 49 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		•
shape 131 size 131 vector direction, specifying 182 velocity transition 131 see also meshers trimming, surfaces 112 tving types 78 see also Import-Export utility vector direction, specifying 182 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		
size 131 velocity transition 131 see also meshers trimming, surfaces 112 tving types 78 velocity as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		•
transition 131 see also meshers trimming, surfaces 112 tving types 78 as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229	-	
see also meshers trimming, surfaces 112 tving types 78 as rigid body motion control 148 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		•
trimming, surfaces 112 tving types 78 prescribing at nodal points 222 velocity as initial condition, in fluid analysis 229		- · · · · · · · · · · · · · · · · · · ·
tying types 78		•
ver command line parameter (MARC) 315	-	· · · · · · · · · · · · · · · · · · ·
verifying the input syntax in MARC command line		- · · · · · · · · · · · · · · · · · · ·

parameter for 315

vertex tolerance 124

vf command line parameter (MARC) 315

View Snapshot feature 57–60

viewfactors
about 273
computing 276
emitting surfaces 273

viscoelastic materials
thermo-rheologically simple 243

viscoelastic thermal expansion property 245

viscosity models 231

volume, minimizing. See design objective



wall thickness, of hollow cross-sections 178 weakly coupled 218 window manager attributes 31 wireframe arrows 69 work-hardening data, considerations for 248



X-Origin button. See PostScript plotting attributes



Y-Origin button. See PostScript plotting attributes

Z

Z-component, for Foam models **259** zooming a view in. *See* dynamic viewing



Research Corporation Document Title: Mentat 3.1-MARC K7.1: New Features

Part Number: UG-3012-01 Revision Date: September, 1997

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. 4350 LaJolla Village Drive, Suite 300 San Diego, CA 92122

Tel: 1 619 546 4414 Fax: 1 619 587 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. 25 Frank Lloyd Wright Drive Ann Arbor, MI 48106-0523

Tel: 1 313 998 0540 Fax: 1 313 998 0542 CZECH REPUBLIC MARC Overseas, Inc.

Podolska 50 147 00 Praha 4 Czech Republic

Tel: 011 420 (2) 61 21 41 23 Mobil: 011 420 (602) 21 23 26 Fax: 011 420 (2) 61 21 41 23

FRANCE MARC France S.A. 30, Rue de Rocroy

94100 Saint-Maur Des Fossés (near Paris)

France

Tel: 011 33 (1) 41 81 1 0 94 Fax: 011 33 (1) 41 81 1 0 95

GERMANY - Munich

MARC Software Deutschland GmbH Ismaninger Str. 9

85609 Aschheim (bie Munich) Germany

Tel: 011 49 (89) 904 50 33 011 49 (89) 903 06 76 Fax: Email: system@marc.co.de

GERMANY - Hannover

MARC Software Deutschland Gmbh

Alte Dohrener Str. 66

D 30173 Hannover, Germany Tel: 011 49 (511) 980 5182 Fax: 011 49 (511) 980 5187

ITALY

Espri-MARC s.r.l.

16129 Viale Brigata Bisagno 2/10

I-16129 Genova

Italy

Tel: 011 39 (10) 585 949 Fax: 011 39 (10) 585 949

THE NETHERLANDS

MARC Analysis Research Corporation

Dublinstraat 32 2713 HS Zoetermeer The Netherlands

Tel: 011 31 (79) 3510 411 Fax: 011 31 (79) 3517 560 Email: support@marc.nl

UNITED KINGDOM

MARC UK Ltd. 35, Shenley Pavilions Chalkdell Drive Shenley Wood

Milton Keynes, MK56LB Tel: 011 44 (1908) 506 505 Fax: 011 44 (1908) 506 522 **CHINA**

MARC Overseas, Inc.

Bright Chang An Building, Suite 321

7# Jian Guo Men Nei Street Beijing, China 10005

011 86 (10) 6510 2056 Tel: 011 86 (10) 6510 2057 011 86 (10) 6510 2058 Fax: 011 86 (10) 6510 2053

IAPAN - Nagoya Nippon MARC Co., Ltd.

Nagoya Nishiki Daiichi Seimei Bldg.

2F, 6-5 Nishiki 1-chome Naka-ku, Nagoya, Aichi 460 Tel: 011 81 (52) 202 5661 Fax: 011 81 (52) 202-5665

JAPAN - Osaka Nippon MARC Co., Ltd. 4F 2nd Kimi Building 2-11 Toyotucho Suita-shi

Osaka 564 Tel:

011 81 (6) 385 1101 Fax: 011 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

Tel: 011 81 (3) 3345 0181 Fax: 011 81 (3) 3345 1529 Email: system@marc.co.jp

KOREA

Dong Kyung Bldg. 7FL 824-19 Yuksam-Dong Kngnam-Ku Seoul, Korea

Tel: 011 82 (2) 561 7543 Fax: 011 82 (2) 561 7767

NOTES