Cimatron it

Cimatron
CAD/CAM Solutions for Manufacturing

Finite Element Modeling

Version 12

Data DI DFM NC Steel

Cimatron may make improvements and changes in the software described in this document at any time and without prior notice. These changes will be documented in future editions of this publication.

Disclaimer of Warranty and Liability.

> No representations or warranties, expressed or implied, of any kind are made by or with respect to anything in this document.
>
> In no event shall Cimatron Ltd., its employees or previous employees, be liable for any incidental, direct or indirect, special or consequential damages whatsoever (including but not limited to loss of profits) arising out of or related to this manual, and/or the product, or any use thereof.

Revised 2001.

Preface

Cimatron develops, markets and supports tools to automate the mechanical engineering process. Our systems support all phases of product development, with solutions for computer aided design (CAD) and manufacturing (CAM). **Cimatron**'s *integrated technology* approach combines design tools with optimized command output to computer-controlled manufacturing equipment. Drafting-table-to-shopfloor integration lets Cimatron clients realize dramatic efficiencies in product development and manufacturing.

Cimatron^{it} - Cimatron's flagship product - covers the entire spectrum of design, engineering and manufacturing processes, including:

- A complete range of **wireframe**, **surface** and **parametric solid modeling tools** with rendering capabilities;
- Advanced **assembly**, **sub-assembly** and **part management**, and **associative drafting functionality**;
- Comprehensive, accurate **data exchange interface utilities** covering **DXF**, **DWG**, **IGES**, **JAMA-IS**, **VDA**, **PTC**, **STEP**, **SAT**, **CATIA** and **UNIGRAPHICS**;
- Powerful and intelligent **NC** applications for precise multi-axis machining.

The modular yet integrated structure of Cimatron^{it} grows to accommodate cutting edge tools and techniques. These now include the new **Quick Tooling** applications:

- **QuickSplit**
  QuickSplit automates the search and separation of core, cavity and sliders to assist in determining the number of actions required to create a mold. After separating core, cavity and slides, QuickSplit identifies the parting lines and generates the parting surface.
  Automatic and interactive tools allow the construction of parting surfaces for any complex geometry. Embedded Draft Analysis enables designers to identify potential problems with undercuts and confirm minimum draft per side.
  QuickSplit is tolerant of surface models with gaps, mismatched boundaries or missing faces, therefore bypassing data corrections and saving precious time.
  Component motion animation, dynamic cross-sectioning and clipping planes, reduce human error and verify parting design. QuickSplit enables several trial and error iterations in a very short time - resulting in optimal draw directions.

- **QuickElectrode**
  QuickElectrode is an EDM electrode design solution used for shortening the electrode process. QuickElectrode is used for burn area selection, electrode design, management, documentation and manufacturing.
  The QuickElectrode Navigator enables full control over the display and activation of electrodes, while allowing several users to collaborate on the same part.

QuickElectrode's report generation features includes set-up sheets, burn location reports and a full electrode schedule, therby alleviating the tedious task of documenting the process .

- **QuickConcept**
  QuickConcept is a preliminary design and review package which allows tool designers and their suppliers to hold *virtual* review meetings over the Internet in real-time. Multiple users can connect to each other to section, label, dimension, and identify points of interest and problem areas of any given tool. All members of the review meeting will interactively view the same screen at the same time.

- **QuickCompare**
  QuickCompare assists the tool designer in determining the scope and effect of Engineering Changes (ECOs) on the tooling process. QuickCompare mathematically compares the geometrical differences between two sets, graphically marks these differences and documents the changes in a CAD file. Here, the designer updates related components and tooling, while archiving ECOs. The typically long *CAD investigation process* is significantly shortened. QuickCompare ensures that all ECOs have been located, whether or not they were communicated from design.

- **MoldBase 3D**
  MoldBase3D offers an innovative *wizard-based* approach to parametric mold base design. MoldBase3D automatically creates 3D solid (parametric & associative) moldbases, with all components and accessories, from industry-standard catalog suppliers such as HASCO, DME, PCS, FUTABA, DMS, PEDROTTI, RADOURDIN, SIDECO, STRACK and MISUMI. Creation of the assembly and detailed drawings of each plate are automated, complete with 2D and 3D section views, ordinate dimensions, labels, balloons, and an itemized Bill of Materials. This module is fully associative to the mold design and changes are automatically reflected in all stages of the design process.

Cimatron's automated engineering expertise benefits many industries, as competition requires tighter development cycles and efficient fabrication.

Powerful modules within Cimatron^{it} expand your system's capabilities. These may be purchased from your Cimatron representative.

This publication provides a detailed description of the major features of the appropriate Cimatron^{it} application/topic. It is intended to help users in the daily operation of Cimatron^{it}.

A list of Cimatron^{it} documentation, for the current version, is shown on the next page.

Cimatron Documentation

Cimatron[it] documentation comprises Reference Manuals, On-Line Help and Tutorials which together provide a comprehensive guide to Cimatron[it].

The list of Cimatron[it] documentation, for the current version, is as follows:

Cimatron[it] Reference Manuals

Publication	Description	Display Options *
Fundamentals & General Functions	Introduction to the fundamentals of Cimatron[it] and description of the General functions.	**A H**
Modeling	Description of the wireframe and surface Modeling functions.	**A H**
QuickSplit	QuickSplit automates the search and separation of core, cavity and sliders to assist in determining the number of actions required to create a mold.	**A H**
QuickElectrode	QuickElectrode is an EDM electrode design solution used for shortening the electrode process.	**A H**
QuickCompare	QuickCompare mathematically compares the geometrical differences between two models, graphically marks these differences and documents the changes in a CAD file.	**A H**
Drafting	Description of the Drafting functions.	**A H**
Solid Modeling	Solid Modeling functions including Sketcher.	**A H**
MoldBase 3D	Description of the functions associated with the detailed design of mold plates and components. MoldBase3D offers an innovative *wizard-based* approach to parametric mold base design.	**A W**
Numerical Control	Description of the NC functions.	**A H**
Cimatron IMSpost	Cimatron IMSpost is a macro-based system for developing and customizing postprocessors.	**A W**
General Post Processor	General Post Processor (GPP) functions.	**A**
Finite Element Modeling	Description of Finite Element Modeling (FEM) functions.	**A H**
Utilities	Various utilities that may be used with Cimatron[it.] These utilities are either **Internal**, run via the USER function, or **External**, run via the Main Menu.	**A H**
Data Interface Utilities	Description of Cimatron's *comprehensive* data interface utilities; DXF, DWG, IGES, JAMA-IS, VDA, PTC, STEP, SAT, CATIA and UNIGRAPHICS.	**A W**
CimaDEK	Cimatron's specialized Developer's kit, for programming customized functions.	**A**
CimaRender Pro	A photo-realistic rendering package.	**A W**
MPDM: Getting Started **MPDM: Administrator**	Description of how to use Manufacturing Product Data Management to track and organize all files and data associated with a project.	**A W**
Re-Enge	Description of Reverse Engineering design functions.	**A**

Cimatron[it] Tutorials

Design - covers **QuickCompare** and **QuickElectrode**.	**A H**
Drafting - covers the DMS function.	**A H**
NC - covers the differences between versions 11 and 12.	**A H**

* **Legend**:

A	Acrobat PDF
H	HTML
W	Winhelp

❑

Table of Contents

Introduction

About This Manual Intro-1
Typographical Conventions Intro-2

Chapter 1 Cimatron[it] FEM Procedure & Functions

The Procedure 1-1
Pre-processing 1-1
Analysis 1-2
Post-processing 1-2
The Functions - An Overview 1-4
The FEM General Functions 1-5
The FEM Modeling Functions 1-5
The Loads and Properties Functions 1-6
The Post-processing Functions 1-6

Chapter 2 The General FEM Functions

FEMSYS 2-2
F_DISP 2-6
F_FILE 2-16

Chapter 3 The FEM Modeling Functions

ELEM_1D 3-2
ELEM_2D 3-9
ELEM_3D 3-23
F_EDIT 3-32
F_MOVE 3-42

Chapter 4 The Loads and Properties Functions

F_LOAD 4-2
F_PROP 4-16
FLOW ANALYSIS 4-26
VERIFY (FEM) 4-31

Chapter 5 The FEM Post-processor Functions

RESULT 5-2
F_PLOT 5-3
F_WRITE 5-7

Appendix A The FE Method of Structural Analysis

Definition A-1
Finite Elements A-1
Boundary Conditions A-4
Material Properties A-4
Applied Loads A-4
Concentrated Forces A-4
Distributed Loads A-5
Body Loads (Gravity or Inertia) A-5
Thermal Loads A-5
Analysis Results and FEA Program Output A-5
Static Response Problems A-5
Dynamic (Vibrational) Analysis A-6
Forced Dynamic Response Analysis A-6
Applying the Finite Element Method A-6

Appendix B
Glossary of Cimatron^it FEM Terminology

Introduction

We recommend that new users of **Cimatron**^{it} read at least the first three chapters of the **Fundamentals & General Functions Manual**, to acquire a working knowledge of **Cimatron**^{it} software.

As you work with the system, different function overlays will be displayed. At the beginning of many chapters, you will find an overlay diagram which shows the functions described in the chapter and indicates how to access them. Within chapters, functions are listed in alphabetical order. The names of the functions and their options appear in capital letters. At the top of each page a status line tells you which option or sub-option is described on that page.

Although you may only need one particular option within a function, we recommend that you read the description of the interaction for the entire function. Modal parameters, which determine the mode which will be active when the function is executed, are explained in detail for the first option described in each function and briefly in subsequent options.

After you are familiar with the basic system, scan the manual occasionally to discover functions you are not using and to learn how to take full advantage of the power of **Cimatron**^{it} software.

About This Manual

This manual provides explanations of the Finite Element Modeling application.

Chapter 1	**Cimatron**^{it} **FEM Procedure and Functions**.
Chapter 2	**The General FEM Functions**.
Chapter 3	**The FEM Modeling Functions**.
Chapter 4	**The Loads and Properties Functions**.
Chapter 5	**The FEM Post-processor Functions - F_POST**.
Appendix A	**The Finite Element Method of Structural Analysis**.
Appendix B	**Glossary of Cimatron**^{it} **FEM Terminology**.

Typographical Conventions

Throughout this manual, certain conventions have been used to present different types of information.

For each function in the manual the following information is presented (in the order listed below):

Function & Purpose

The name of the function as it appears in the function bank, presented in large bold upper case type, followed by the purpose of the function. For example:

F_DISP

The purpose text, presented in regular type, describes what the function does.

Main Options

Presents the first level of options within each function in table format, as they appear on the screen. For example:

Main Options:

SELECT:	DISPLAY MODES
	DEFINE DISPLAY
	DEFINE COLORS
	SECTION
	HIDDEN LINES

The interaction for each main option is described separately within the function. Where the description begins, the main option name is presented, preceded by the function name and graphic arrows. For example:

F_DISP **>> DEFINE COLORS**

Sub-options

When branching occurs in the interaction of a function, the interaction for each option is described separately. The option names are presented in bold, block letters and the option path is represented by graphic arrows.

Actions (optional)

Summarizes the recommended interaction to be followed in the current function/option, presented in sans serif type. For example:

How To:

1. Specify the type of geometric entity for which finite elements will be generated.
2. Pick curve or surface entities for which finite elements are to be generated.
3. Generate finite elements according to specified conditions.

Interaction

The prompts which tell you how to execute the function are listed in upper case italic letters, and start at the left margin.

Prompt explanations are shown to the right of the prompts. (Some additional notes are also provided in the right column.) For example:

SET ELEM. SIZE & <CR> Press <CR> to set the display size of elements.

Modal Parameters

The modal parameters for each function appear at the end of the function in alphabetic order.

Modal parameters are represented in the text by a filled box as shown below.

Modal parameters marked with an asterisk are system generated and cannot be changed.

Parameter explanations are to the right of the parameters.

■ SHRINK Display elements at a fraction of their full size.

Notes

Provides information to help you avoid problems and achieve accurate results. Each note is preceded by a bullet character for immediate identification. For example:

Notes:

- If the primary factor is a property, all types of element attached to that property will have the same color. All unattached elements will be assigned colors according to element type.

End of Function

The end of a function or section is marked by a box character (❑).

Chapter 1
Cimatron^{it} FEM Procedure & Functions

The Procedure

The FEM application of **Cimatron^{it}** is a tool for analyzing the behavior of physical systems in three stages.

The first is called pre-processing. A FEM, finite element model, which can be analyzed by structural/thermal analysis programs is defined.

In the second stage, with the help of direct links with analysis programs like ANSYS, NASTRAN and others, the behavior of the model under given conditions is analyzed.

Then, in the post-processing stage, the results of the analysis are selectively retrieved and examined. The design is modified, if necessary, and the entire procedure may be repeated until the analysis results show that the design will function properly under the conditions for which it is intended.

The **Cimatron^{it}** FEM application uses the same database, user interface and graphic tools as the **Cimatron^{it}** Modeling and Drafting applications.

Pre-processing

An idealized model of the geometric model, and the associated data which are required for the analysis, are constructed as follows:

- A finite element mesh model is generated from the geometric model for analysis purposes.

  First, a view of the geometric model is defined as a FEMSYS. Its orientation is defined by a finite element model coordinate system. Then, using special FEM application functions, nodes are created at the vertices of 1D, 2D or 3D elements at varying densities on the sides and/or faces of the model.

  Meshes are automatically or interactively generated from these nodes. Additional internal nodes may be specified to refine the finite element model. Finally, several meshes within a given FEMSYS may be combined to produce the complete finite element model.

  Using the tools standard to **Cimatron^{it}**, the finite element model or any part of it can be visually inspected and, if required, immediately modified.

  The following interactive and automatic editing tools make it possible to arrive at the optimal mesh after the initial finite element model(s) has been generated:

  - interactive and automatic joining of meshes
  - merging of meshes of different densities
  - relocation of nodes with the automatic adjustment of the elements connected to those nodes
  - relocation, copying, deletion and rotation of any part of the mesh and elements

- After the finite element model is created, other functions may be used to quickly and efficiently specify supplementary material and geometric properties, boundary and load conditions, and type of analysis.

  Internal properties are defined, such as the inherent properties and supplementary element geometric properties (e.g. cross-sectional properties, thickness etc.)

  External load and constraint conditions are defined such as, supports, nodal forces/moments and temperatures, pressures on sides and/or faces of elements, and body loads on the model as a whole.

- An analysis type is selected, such as structural analysis (static/dynamic) or heat analysis.

The complete finite element model and its supplementary data are stored in the same database as the part file. Revisions to the geometric model are reflected in the FEMSYS and can be applied to the finite element model derived from it.

Analysis

In this stage, the finite element model which was defined in the pre-processor stage is analyzed by applying an external program such as ANSYS or NASTRAN. NASTRAN and ANSYS may be run on the same computers as **Cimatron**^{it}. Both external analysis programs permit either structural or thermal analysis.

Post-processing

Post-processing is performed within the FEMSYS in which the FEM model analyzed was prepared.

In this sub-application of the FEM application, the results of finite element analysis, such as internal loads, stresses, strains, reaction forces, temperatures, temperature gradients and fluxes, can be selectively viewed and/or output.

Each time the analysis is performed, RESULT is run to read the analysis results and to create an internal data file called **out12.dat**. Then, F_WRITE or F_PLOT may be used to describe which results will be viewed and in what form, i.e., written alphanumerically on the screen or in a file, or displayed graphically on the screen.

Among the existing selection possibilities are:

- choosing numeric or graphic output
- color coding of selected output
- superposition of the finite element model on the geometric model
- superposition of the deformed structure on the original one

If results are displayed graphically, displacements of the model and colored contour lines may be produced. Each graphic display is a picture entity which can be saved with the part file and viewed repeatedly. However, it reflects the results at the moment it was created and is not updated when another analysis is performed.

The FEM and geometric models may not be modified from the post-processing sub-application. However, the F_DISP function may be used to control the display parameters such as element size and color, and to display only selected elements.

With the help of the standard tools of the **Cimatron**^{it} system, the results can be checked and critical regions can be isolated.

After checking the results of the analysis, it is easy to correct and change the geometric model or the finite element model. The entire process may be repeated as often as necessary, until the required criteria are met.

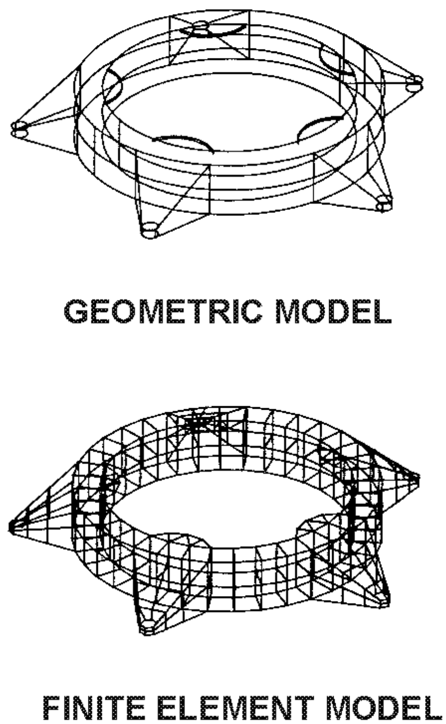


Figure 1-1: Geometric and FInite Element Models

The Functions - An Overview

The FEM application functions are accessed by selecting the FEM application from the main **Cimatron**^{it} applications menu.

The following function overlays show how the FEM functions may be accessed from the application function bank on the upper right side of the screen. In addition, when Overlays I and II are displayed, i.e., during pre-processing, most of the Modeling functions may also be selected. Use <SUBMENU> or <REJECT> to display the appropriate Modeling function overlay.

Refer to the **Cimatron Modeling Manual** for detailed descriptions of each Modeling function and option.

Overlay I	As soon as a FEMSYS is selected, functions in this overlay may be selected.
Overlay II	Select F_POST (in Overlay I) go to FEM post-processing and to display Overlay II.
	Select F_PREP to go back to FEM pre-processing and display Overlay I.

	Overlay I		Overlay II	
	ELEM_1D		**RESULT**	
	ELEM_2D		**F_WRITE**	
	ELEM_3D	<SUBMENU> to Modeling	**F_PLOT**	
	F_EDIT		**F_DISP**	
<REJECT> to Modeling	**F_MOVE**			
	F_DISP		**F_PREP**	F_PREP to Overlay I
	F_PROP			
	F_LOAD			
	F_FILE			
	F_POST	F_POST to Overlay II		
	FEMSYS			

When the LINE, POINT or VERIFY >> DATA >> COORD. functions are used in the FEM application, a FEM node may be indicated like any other point. To do this, first display the POINT menu or the point indication submenu. Press <SUBMENU> and select the last option, FEMNOD. FEMNOD appears in the status area of the screen. You may now <PICK> a node and continue with the function.

The FEM General Functions

FEMSYS

Manage finite element model systems.

- — Define a new finite element model system.
- — Activate, rename or delete and existing FEMSYS.
- — Add entities from the model to an existing FEMSYS.

When you first enter the FEM application, only the FEMSYS function will be displayed in the application function bank. This function defines a 3D view of the model, called a Finite Element Model System or a FEMSYS. Finite elements and meshes will be created relative to the active FEMSYS. After at least one FEMSYS has been defined and named, it may be selected from a menu of existing FEMSYS. Selection of a specific FEMSYS makes it the active one and will result in the display of the remaining FEM functions.

F_DISP

Modify the display of the finite element model and the symbols which represent different properties.

- — Control the display attributes of finite elements.
- — Selectively display finite elements and properties.
- — Display or remove hidden lines.
- — Shrink finite elements for display purposes.

F_FILE

After the finite element model is created and adjusted, F_FILE generates an ASCII output file suitable for input to a specific analysis program.

The General FEM functions are described in detail in Chapter 2.

The FEM Modeling Functions

The FEM element and mesh creation functions are used to automatically create finite elements based on the entities which compose the geometric model. They create 1D, 2D or 3D finite elements. The nature of the part to be analyzed and the type of analysis to be done determine which types of finite elements should be created.

ELEM_1D

Automatically generate 1D finite elements which are straight line segments whose endpoints are nodes.

This function is useful for generating finite elements for structures that can be idealized as an assemblage of linear elements such as beams and spars.

ELEM_2D

Automatically generate meshes of 2D finite elements which are triangles or quadrilaterals whose 3 or 4 vertices are nodes.

This function is useful for generating finite elements for structures to be analyzed using plane strain, plane stress, thin plate bending, and axisymmetric solid idealizations.

ELEM_3D

Automatically generate meshes of 3D hexahedron solid finite elements whose eight vertices are nodes.

In addition, 3D tetrahedron finite elements can be generated interactively using the F_EDIT function described below.

The following two functions offer many options for editing, manipulating and working with all types of finite elements and meshes.

F_MOVE	Translate, scale and rotate finite elements and meshes.
F_EDIT	Create, adjust or delete nodes and elements. Smooth or edit meshes.

These FEM modeling functions are described in detail in Chapter 3.

The Loads and Properties Functions

The functions described below can be used to define material and geometric properties and load conditions, to attach them to specific elements and to verify their status.

F_PROP	Define supplementary material or geometric properties. Attach properties to specific parts of the finite element model.
F_LOAD	Define loads acting upon the finite element model.
VERIFY >> FEM	Verify data relating to finite elements such as: coordinates of nodes, ID numbers, nodes composing each element and the attachment status of supplementary material and/or geometric properties. This option is accessed via the General System function VERIFY in the General System Function Bank.

The functions which are used to work with loads and properties are described in detail in Chapter 4.

The Post-processing Functions

The post-processing functions are used to produce an internal data file from the analysis results and to selectively output those results. Select F_POST from the second overlay of the FEM application menu to make the following FEM post-processor functions available, in addition to F_DISP (See FEM modeling functions.)

RESULT	Read the results of the analysis and create an internal data file.
F_PLOT	Display the selected analysis results on the graphics screen.
F_WRITE	List selected analysis results on the alphanumeric screen or in a file.

When you have finished post processing, select F_PREP to return to Overlay I, or to exit the FEM application.

After post-processing is finished, F_PREP may be selected to leave the F_POST sub-application and return to FEM pre-processing, i.e., the finite element modeling functions, or leave the FEM application.

The special F_POST functions are described in detail in Chapter 5, in the order they are listed above.

How To:

Preprocessing

1. Create property tables for materials using function F_PROD.
2. Create elements and meshes using function ELEM 1D, 2D, and 3D.
3. Apply loads using function F_LOAD.
4. Optional: Edit or move the meshes using function F_EDIT, F_MOVE.

Analysis

5. Send to analysis program ANSYS, NASTRAN, INJECT, C-MOLD, and MOLDEX.

Post Processing

6. Read the analysis results using the function RESULT. This creates the data file out12.dat.
7. Define which results will be viewed and how to display them. Use functions F_WRITE and/or F_PLOT; these functions use the file out12.dat (created above) as input.

❑

Chapter 2
The General FEM Functions

The functions listed below are described in detail in this chapter in alphabetical order.

FEMSYS may be selected at any time in the FEM application. F_DISP and F_FILE may be selected only when a FEMSYS is active.

FEMSYS	Manage finite element model systems.
F_DISP	Modify the FEM display.
F_FILE	Create an ASCII output file.

FEMSYS

Define a new FEMSYS or select, rename or delete an existing one.

The FEMSYS, Finite Element Model coordinate SYStem, is defined by indicating an origin and two points on the active XY plane. The coordinates of the nodes which compose finite elements will be created relative to the FEMSYS.

Note: • ACTIVE is the default mode. To display the option menu, press <REJECT>.

SELECT:	ACTIVE
	RENAME
	DELETE
	APPEND

ACTIVE Define a new FEMSYS or access an existing one.

RENAME Change the name of an existing FEMSYS.

DELETE Delete an existing FEMSYS.

APPEND Include additional entities from the geometric model in the active FEMSYS.

FEMSYS >> ACTIVE

SELECT FEMSYS:	<femsys 1>	<femsys 2>	. . .	<femsys n>

Select an existing FEMSYS or select the blank option to create a new FEMSYS.

If a new FEMSYS is being created, skip to ENTER NEW FEMS.NAME.

If the FEMSYS that was selected does not contain finite elements, select an analysis type.

■ STRUCTURAL ANALYSIS — Heat or strength analysis.

INJECTION MOLDING — Flow analysis.

<CR> TO CONTINUE

Note: • When finite elements have been defined in a FEMSYS, the analysis type may not be changed and these modals will not appear.

PICK WIND. FOR FEMSYS	If multi-windows are displayed, and if the selected FEMSYS does not currently appear in a window, pick the window in which it is to be displayed.
PICK ADDIT.WINDOW	If multi-windows are displayed and the selected FEMSYS already appears in at least one of them, pick another window in which to display it. <EXIT> when finished.
ENTER NEW FEMS.NAME	If the blank option was selected, type up to 6 characters as the name of the new FEMSYS.

If a new FEMSYS is being created from within another FEMSYS, the following appears:

RESTORE MODEL? YES NO

YES Use the orientation and display parameters of the model.

NO Use the orientation and display parameters which are currently on the screen from the last FEMSYS.

Note:

- Display parameters include levels, line attributes and zoom factor.

  Only the orientation and display parameters will be copied from either the last FEMSYS or the model. The entities in the new FEMSYS will be <PICK>ed from the model separately. Entities created in VIEWS, DRAWINGS or another FEMSYS will not be included in this new FEMSYS.

PICK UCS / EXIT	MODEL	———-

If a UCS was picked, FEMSYS will be at its coordinates. If the blank option was selected, continue the definition.

IND. FEMSYS ORIGIN Indicate three points to define coordinates for the FEMSYS.

IND. PNT ON +X AXIS

IND. PNT FOR +Y

ENTIRE MODEL? YES NO

YES CREATE IN PROCESS

The FEMSYS will be created and will include all the entities in the model.

NO PICK ENTITIES & EXIT
CREATE IN PROCESS

After all the entities to be included in the FEMSYS are picked, and <EXIT> is pressed, the FEMSYS will be created.

FEMSYS >> RENAME

RENAME FEMSYS:	<femsys 1>	<femsys 2>	. . .	<femsys n>

Select the FEMSYS which will be assigned a new name.

ENTER NEW FEMS. NAME Type a new name for the FEMSYS just selected and press <CR>.

<REJECT> to return to the previous menu. <EXIT> to leave the option.

FEMSYS >> DELETE

DELETE FEMSYS:	<femsys 1>	<femsys 2>	. . .	<femsys n>

Select a FEMSYS to be deleted. Note that the name of the currently displayed FEMSYS (i.e. <femsys1>) does not appear in the menu and, therefore, cannot be selected for deletion. To delete every FEMSYS of a part, select this DELETE option when no FEMSYS is active.

DELETE <selected FEMSYS>? YES NO

YES DELETION IN PROCESS

Confirm that the selected FEMSYS is to be deleted. The deletion will be performed.

NO Reselect a FEMSYS to be deleted.

<REJECT> to return to the previous menu.

<EXIT> to leave the option.

FEMSYS >> APPEND

SELECT FEMSYS:	<femsys 1>	<femsys 2>	. . .	<femsys n>

The model will be redisplayed and the FEMSYS names will be displayed in the menu. Select names of the FEMSYS to which entities will be added. Press <EXIT> when finished.

PICK ENTITIES & EXIT

<PICK> entities to be included in the selected FEMSYS.

Only entities from the model may be appended to a FEMSYS. Entities which were created in one FEMSYS may not be appended to another FEMSYS.

<REJECT> to return to the previous menu.

<EXIT> to execute and to leave the option. ❑

F_DISP

Control the display attributes of finite elements including the display size, color and selective display of elements and properties, and the removal of hidden lines.

SELECT:	DISPLAY MODES
	DEFINE DISPLAY
	DEFINE COLORS
	SECTION
	HIDDEN LINES

DISPLAY MODES Define the display size of elements and symbols; and define the color display of elements and properties as filled in or not filled in.

DEFINE DISPLAY Set a selection mask to determine which elements and properties will be displayed.

DEFINE COLORS Assign colors for display purposes to elements and properties.

SECTION Display only the tetra elements that intersect with the defined plane.

HIDDEN LINES Remove hidden lines from the display.

F_DISP >> DISPLAY MODES

Define the display size of elements and symbols; and define the color display of elements and properties as filled in or not filled in.

- SHRINK — Display elements at a fraction of their full size.

  FULL SIZE — Display elements at their full size.
- LINES = 0.94 — If SHRINK is selected, enter a value between 0.5 and 1 to set the display size of each type of element relative to its full size.
- TRIANGLES = 0.94
- QUADS = 0.94
- BRICKS = 0.94
- PRISMS = 0.94
- TETRAS = 0.94

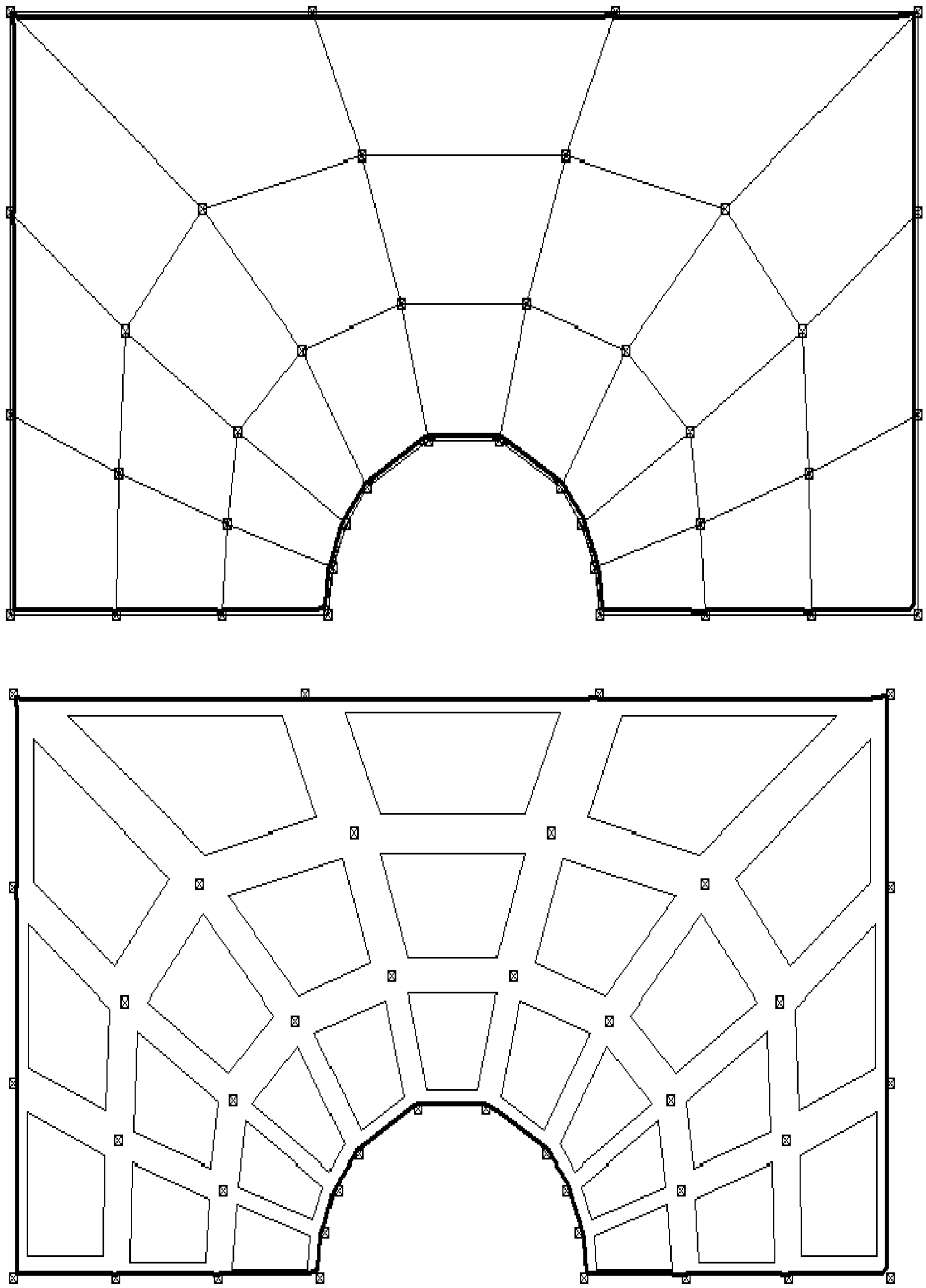
SET ELEM.SIZE & <CR> — Press <CR> to set the display size of elements.

- FIXED SYMBOL SIZE — The symbols for all forces and moments in the FEM will be the system default size.

  PROPORTIONAL SIZE — The symbols for all forces and moments will be a function of the system default size.
- FORCE = 100.000 — If PROPORTIONAL SIZE is selected, enter the FORCE unit represented by the default symbol size.
- MOMENT = 100.000 — If PROPORTIONAL SIZE is selected, enter the MOMENT unit represented by the default symbol size. For example:
  - If FORCE = 100.000, a force of 500 will be represented by a symbol that is 5 times the size of the default symbol.

SET SYMB.SIZE & <CR> — Press <CR> to set the display size of force and moment symbols.

Select the primary factor which will determine the color of each element.



SHRINK LINES=0.7, QUADS=0.8

Figure 2-1: Shrink Lines- Before and After

Note: • If the primary factor is a property, all types of element attached to that property will have the same color. All unattached elements will be assigned colors according to element type.

- ELEMENTS — Elements will be assigned colors only according to their types. (Not available for flow analysis.)

  STRNG.MAT.PROP — Elements of all types attached to the same strength material properties will have the same color. (Not available for flow analysis.)

  GEOMETRIC PROP — Elements of all types attached to the same geometric properties will have the same color.

  HEAT MAT.PROP — Elements of all types attached to the same heat material properties will have the same color. (Not available for flow analysis.)

- NO COLOR FILL — Color only the boundaries of the elements.

  COLOR FILL — Fill the area within the boundaries of the appropriate elements with color.

SET COLOR MODE & <CR>

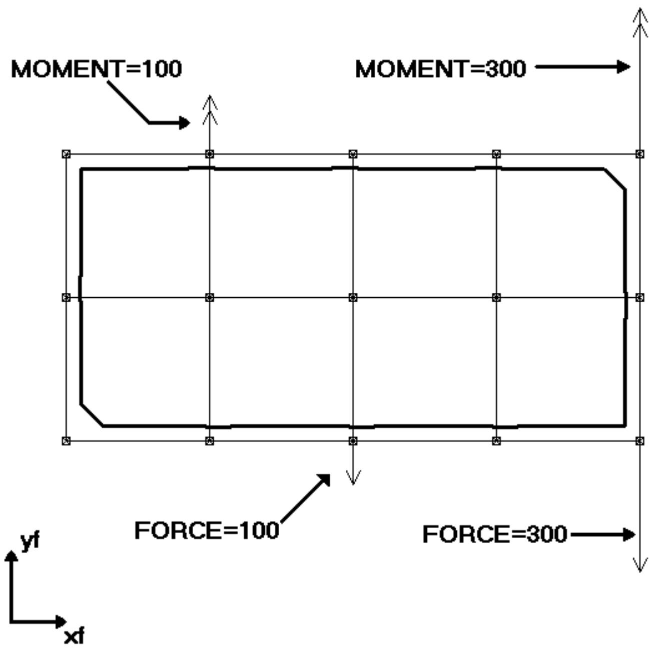


Figure 2-2: FORCE/MOMENT Symbol Size

F_DISP >> DEFINE DISPLAY

Set a selection mask to determine which elements and properties will be displayed.

SET DISP.MASK & EXIT	NODE	SPAR	BEAM	PLATE
	MEMBRN	PL STRS	PL STRN	AXISYM
	BRICK	PRISM	TETRA	MASS
	SPRING	RIGID	CONS-DIS	CONS-ROT
	LD-FORCE	LD-MOMNT	TRACTION	TEMPER
	HT-NDTEM	HEAT FLW	CONVECTN	NODENUM
	ELEMNUM	ALL-ELEM		

or:

SET DISP.MASK & EXIT	NODE	CONIC	STRIP	PLATE	NODENUM
	ELEMNUM	ALL-ELEM			

If flow analysis is specified, the second menu is displayed.

The default display status for all items in the list is lit, i.e. selected for display except for NODES, ALL-ELEM, NODE_NUM, and ELEM_NUM. Those are not displayed by default in order to improve display performance. To change the status of an item, select it. Its status will be reversed.

<EXIT> when finished. <EXIT> a second time to leave the function and adjust the display as specified.

Only items which are lit will be visible.

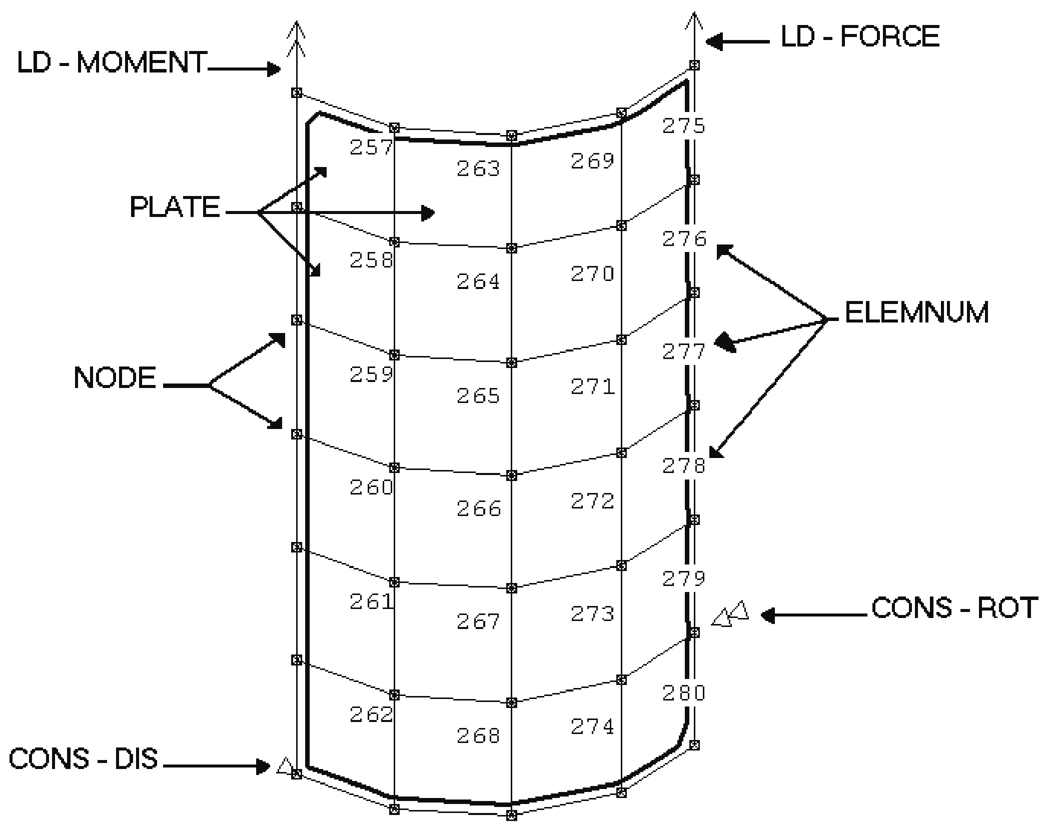


Figure 2-3: DEFINE DISPLAY

F_DISP >> DEFINE COLORS

Assign colors for display purposes to elements and properties.

DEFINE COLOR OF:	ELEMENTS	STRNG MAT.PROPS	GEOMETRIC PROPS	LOADS	HEAT MAT.PROPS

or:

DEFINE COLOR OF:	ELEMENTS	GEOMETRIC PROPS

If flow analysis is specified, the second menu is displayed.

In monochromatic systems, the actual colors cannot be displayed. Nevertheless, the colors of elements and properties can be assigned and changed.

DEFINE COLORS >> ELEMENTS

- NODE
  SPAR
  BEAM
  CONIC
  STRIP
  PLATE
  MEMBRANE
  PL STRESS
  PL STRAIN
  AXISYM
  BRICK
  PRISM
  TETRA
  MASS
  SPRING
  RIGID

Select an element whose color is to be set.

- MAGNTA
  RED
  YELLOW
  BLUE
  CYAN
  PURPLE
  GREEN
  COL 8 . . . COL 15

Select a color for the displayed element.

SET COLORS & <EXIT>

Press <EXIT> to accept the color settings.

DEFINE COLORS > STRNG MAT.PROPS

- <strength material property 1>
  <strength material property 2>
  . . . .
  <strength material property n>
  Select a strength material property whose color is to be set.
- <list of colors as in ELEMENTS> Select a color for the displayed strength material property.

SET COLORS & EXIT Press <EXIT> to accept the color settings.

DEFINE COLORS >> GEOMETRIC PROPS

- <geometric property 1>
  <geometric property 2>
  . . . .
  <geometric property n>
  Select a geometric property whose color is to be set.
- <list of colors as in ELEMENTS> Selects a color for the displayed geometric property.

SET COLORS & EXIT Press <EXIT> to accept the color settings.

DEFINE COLORS >> LOADS

- CONS-DIS
  CONS-ROT
  LD-FORCE
  LD-MOMNT
  TRACTION
  TEMPERAT
  HT-NDTEM
  HEAT FLW
  CONVECTN
  Select a load whose color is to be set.
- <list of colors as in ELEMENTS> Select a color for the displayed load.

SET COLORS & <EXIT> Press <EXIT> to accept the color settings.

DEFINE COLORS >> HEAT MAT.PROPS

- <heat material property 1>
  <heat material property 2>
  . . . .
  <heat material property n>
  Select a heat material property to set its color.
- <list of colors as in ELEMENTS> Select a color for the displayed heat material property.

SET COLORS & <EXIT> Press <EXIT> to accept the color settings.

F_DISP >> SECTION

Display the tetra elements that intersect with the defined plane.

PICK TETRAHED MESH	Pick the blue box (used to indicate tetra mesh).
IND.POINT 1	Indicate a point on the desired plane you want displayed. You can use the sub option (right mouse button - RMB) to choose FEMNOD, and then pick NODE on the mesh (the nodes should be displayed initially with F_DISP / DEFINE DISPLAY / NODE).
IND. PNT FOR +X	Indicate a point for the x axis.
IND. PNT FOR +Y	Indicate a point for the y axis.
POINTS OK? YES NO	**YES** Accept the selection. A red line is displayed. **NO** Select points again.
PICK NEW PLANE POS	Pick on the red axis line to display the tetra elements that intersect with the defined plane.

F_DISP >> HIDDEN LINES

Remove all lines from the display that would be hidden from view by other parts of the finite element model if it were not a wire frame representation.

PICK WINDOW FOR HL — If multi-windows are displayed, pick a window for the hidden line picture. A hidden line picture of all the finite elements displayed will be produced.

Note

- Each time the FEM is rotated this option must be selected to remove the hidden lines.

When hidden lines are removed, some of the element and node numbers will be obscured. Use the immediate access function, R, refresh, to redisplay these numbers.

REPAINT? YES NO — Answer YES to redisplay the FEM model without the hidden lines and without the need for reprocessing.

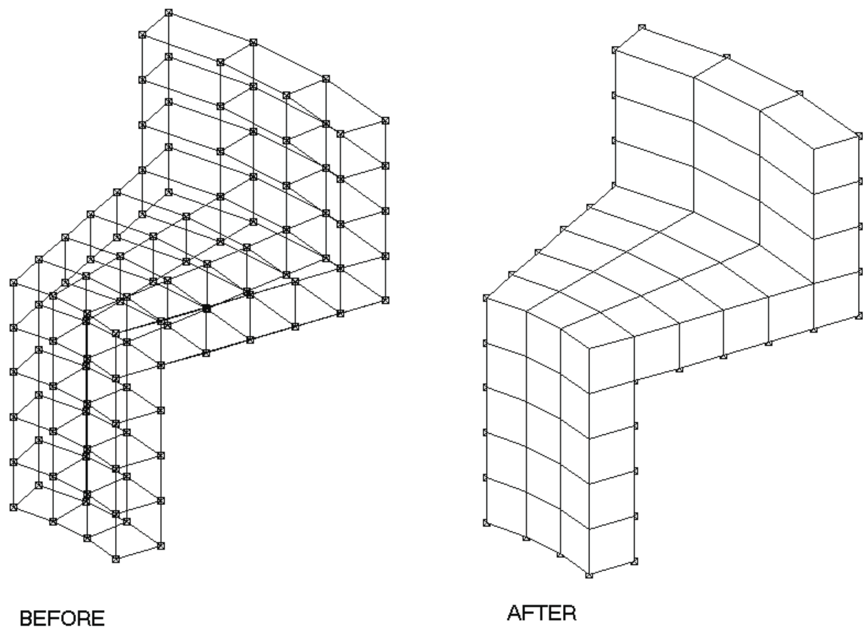


Figure 2-4: HIDDEN LINES o

F_FILE

Create an ASCII file to be used by a specific analysis program.

SELECT ANALYSIS PROG:	ANSYS*
	NASTRAN*

or:

SELECT ANALYSIS PROG:	INJECT**
	C_MOLD**
	MOLDEX**

If flow analysis is specified, the second menu appears.

* Applicable for strength or heat analysis only.

** Applicable for flow analysis only.

Note:

- Geometric and supplementary material properties should be defined for all elements to be analyzed by ANSYS or NASTRAN. The program will display a message if this has not been done and the following question will appear:

  CONTINUE WRITING? YES NO

  If YES is selected, a file will be written without the missing data. The missing data may be defined later and this function rerun, or the file may be edited manually.

F_FILE >> ANSYS

This **Cimatron**[it] version supports ANSYS version 5.2.

Note:

- The following types of ANSYS analysis are available:

  STATIC ANALYSIS computes the displacements, stresses, internal loads and boundary reactions of a structure with prescribed boundary constraints which is acted upon by a given set of applied surface and/or body loads.

  HEAT TRANSFER ANALYSIS computes:

Thermal	Temperature distribution and heat flow rates within a body.
Magnetics	Intensity and flux density of a magnetic field.
Fluid flow	Pressure and flow rates in fluid flow problems.
Electrical	Voltages and current in electrical fields.
Coupled fields	Response due to thermal stress, piezoelectrics, thermal-electric, etc. interaction.

  This analysis type also solves problems such as shaft torsion and pressurized membranes.

  Additional data must be added to the file for other types of analysis.

FILE (NO EXTENSION)	Enter the prefix of the file name to be created. The suffix is automatically **.dat**.
■ STATIC	Static
BUCKLING	Linear Buckling
MODAL	Modal
FULL HARMONIC	Full Harmonic Response
NON LINEAR TRANS.DYN	Nonlinear Transient Dynamic
LINEAR TRANS.DYN	Linear Transient Dynamic
HARMONIC	Reduced Harmonic Response
SUPER ELEMENT	Substructuring
HEAT	Heat Transfer, Magnetics, Field Analyses
SET ANALYS.TYPE & CR	Select the type of ANSYS analysis to be done and press <CR>.
■ OPTIMAL DIRECTION X OPTIMAL DIRECTION Y OPTIMAL DIRECTION Z	Select the most efficient wavefront direction ordering the elements. (Refer to ANSYS documentation for a more detailed description of wavefronts.)
■ DESCENDING ASCENDING	Choose descending or ascending order.
SET WAVEFRNT DIR.&CR	Set modals and press <CR>.

■ CHECK	If this modal is selected, when the analysis program is run, the data will be read and checked, but no calculations will be done.

Note: • A check run is recommended for large analyses.

EXECUTION	If this modal is selected, the data will be analyzed.
SET RUN TYPE & <CR>	Set the appropriate run type and <CR>.
■ ALL ELEMENTS	Include data for all elements in the ASCII file.
DISPLAYED ELEMENTS	Write data describing only visible elements to the ASCII file.
SET WRITE MODE & <CR>	Set the desired write option and press <CR>.
EXECUTING . . .	The output file will be created.

F_FILE >> NASTRAN

This **Cimatron**[it] version supports NASTRAN version 68.

Note: • STATIC and HEAT analyses are supported.

Additional data and commands must be added to the file for other types of analysis.

FILE (NO EXTENSION)	Enter the prefix of the file name to be created. The suffix is automatically **.dat**.
■ STATIC	Static solution 101
HEAT	Heat solution 101
BUCKLING	Buckling solution 105
SET ANALYS.TYPE & CR	Select the type of NASTRAN analysis to be done and press <CR>.
■ ALL ELEMENTS	Include data for all elements in the ASCII file.
DISPLAYED ELEMENTS	Write data describing only visible elements to the ASCII file.
SET WRITE MODE & <CR>	Set the desired write option and press <CR>.
EXECUTING . . .	The output file will be created.

F_FILE >> INJECT / C_MOLD / MOLDEX

ENTER FILE NAME	Type the name of the file that will be created.
■ ALL ELEMENTS	Include data for all elements in the ASCII file.
DISPLAYED ELEMENTS	Write data describing only visible elements to the ASCII file.
SET WRITE MODE & <CR>	Set the desired write option and press <CR>.
EXECUTING . . .	The output file will be created. ❑

Chapter 3
The FEM Modeling Functions

The following functions are described in detail in this chapter in alphabetical order. They may be selected anytime a FEMSYS is active. Press <EXIT> to access the modeling functions.

ELEM_1D	Automatically generate 1D finite elements.
ELEM_2D	Automatically generate meshes of 2D finite elements.
ELEM_3D	Automatically generate meshes of 3D finite elements.
F_EDIT	Create, adjust or delete nodes and elements. Smooth or edit meshes.
F_MOVE	Translate, scale and rotate finite elements and meshes.

ELEM_1D

Automatically generate 1D finite elements, which are straight line segments whose endpoints are nodes.

How To:

1. Specify the type of geometric entity for which finite elements will be generated.
2. Pick curve or surface entities for which finite elements are to be generated.
3. Generate finite elements according to specified conditions.

SELECT:	SINGLE CURVES
	2D CONTOUR
	3D CONTOUR
	SURFACE

SINGLE CURVES Generate strings of 1D finite elements on single curves.

2D CONTOUR Generate strings of 1D finite elements on consecutive connected curves which lie in the same plane.

3D CONTOUR Generate strings of 1D finite elements on consecutive connected 3D curves.

SURFACE Generate strings of 1D finite elements on a bounded surface.

ELEM_1D >> SINGLE CURVES

Note: • Use this option when non-consecutive curves and/or only a portion of a curve(s) is to be represented as finite elements.

- SPAR — Generate a spar type element for strength or heat analysis.

  BEAM — Generate a beam type element for strength or heat analysis.

  CONIC — Generate a conic type element for flow analysis.

  STRIP — Generate a strip type element for flow analysis.

PICK CURVES & EXIT — <PICK> curves which are to be represented as straight line finite elements. <EXIT> when finished.

SELECT	ELEM TYPE	MATERIAL:NONE	HEAT MATERIAL:NONE	GEOMETRIC:NONE

If one or more property tables have been defined, (material, heat and/or geometric) you may select these properties and attach them to specific parts of the finite element model.

- INTERVALS — Divide the curve into a specified number of intervals. The lengths of the intervals may be fixed or variable as defined by the modal RATIO, below.

  LENGTH — Divide the curve into intervals of a fixed length.

Note: • If LENGTH is selected, the number of intervals is calculated by the system. The curve will be divided into segments of equal length. This length will be as close to the specified length as possible.

In both options, nodes will be generated on the curve at the endpoints of each interval.

- NO. OF INTERVALS = 1 — If INTERVALS is selected, enter the desired number of intervals.

Note: • Number of nodes on each curve = intervals + 1.

LENGTH = 10.000 — If LENGTH is selected, enter the desired length on the curve, of the segments in the intervals between the nodes.

- WHOLE — Use the entire curve as specified.

  PART — Use only a part of the curve as specified.

- RATIO = 1.000 — If INTERVALS is selected, enter the ratio between the last and the first line segments which define the curve.

  The curve will be divided into the specified number of intervals. The length of the last divided by the length of the first will equal the specified ratio. The lengths of the remaining intervals will be graduated between the first and the last.

<CR>TO CONTINUE — If the whole curve is to be used, press <CR> to continue.

or:

IND.START POINT
IND.END POINT — If PART of the curve is to be used, identify it by its start and endpoint.

If the WHOLE curve is to be used; and, if there is more than one interval; and, if the ratio is not equal to one; then, two arrows will be displayed and the following will appear:

INDICATE DIRECTION? YES NO

Choose a direction along the curve. If RATIO is not equal to 1, the direction determines how the curve will be divided. RATIO = length of last segment/length of first segment.

Note:

- In Flow Analysis, the above criteria for the appearance of the question:
  INDICATE DIRECTION? YES NO
  do not apply. The question is displayed irrespective of whether the WHOLE or PART of the curve is to be used, the number of intervals or the ratio.

EXECUTING . . . — The straight line finite elements will be generated as specified.

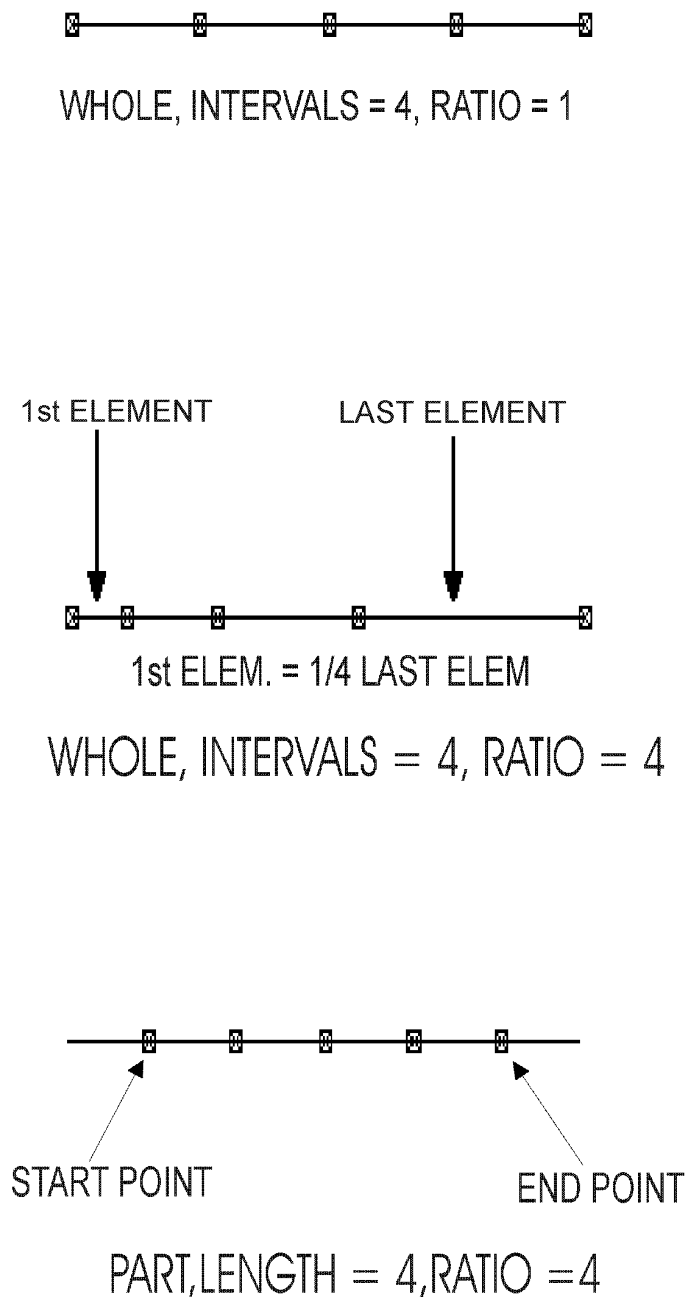


Figure 3-1: Intervals on 1D Elements

ELEM_1D >> 2D CONTOUR (CONSECUTIVE CURVES)
ELEM_1D >> 3D CONTOUR

Notes:

- Use this option to access the curve picking options for defining contours. Definition of the finite elements which represent the curves in the contour will still be done separately for each curve, as in the SINGLE CURVES option.
- When a curve is picked as part of a contour, the whole curve will be used for finite element generation. To use part of a curve, pick it using the SINGLE CURVES option.

SELECT:	OPEN CONTOUR	CLOSED CONTOUR

■ SPAR — Generate a spar type element for strength or heat analysis.

BEAM — Generate a beam type element for strength or heat analysis.

CONIC — Generate a conic type element for flow analysis.

STRIP — Generate a strip type element for flow analysis.

PICK 1ST CURVE
INDICATE DIRECTION? YES NO
PICK 2ND CURVE . . .
. . . PICK NTH CURVE
END CONTOUR O.K.? YES NO

Continue to pick curves as though you were defining an open or closed contour. Refer to the **Fundamentals & General Functions Manual** for a more detailed explanation, if necessary.

Notes:

- Islands may not be defined.
- The picked curves will appear in attention, one at a time. For each curve, set the modal parameters which determine how their finite elements will be generated.
- To divide all the curves in the same way, i.e. using the same modal parameter settings, set the modal parameters for the first curve, press <EXIT> and select YES:

UNIFORM DIVISION? YES NO

For a more detailed explanation of the modal parameters, see SINGLE CURVES option.

SELECT	ELEM TYPE	MATERIAL:NONE	HEAT MATERIAL:NONE	GEOMETRIC:NONE

If one or more property tables have been defined, (material, heat and/or geometric) you may select these properties and attach them to specific parts of the finite element model.

■ INTERVALS — Divide each curve into a specified number of intervals.

LENGTH — Divide each curve into intervals of a fixed length.

■ NO. OF INTERVALS = 1 — If INTERVALS is selected, enter the desired number of intervals.

LENGTH = 10.000 — If LENGTH is selected, enter the desired length on the curve, of the segments in the intervals between the nodes.

■ RATIO = 1.000 — If INTERVALS is selected, enter the ratio between the last and first line segments which define the curve.

<CR> TO CONTINUE	The straight line finite elements will be generated as specified.
EXECUTING . . .	The straight line finite elements will be generated as specified.

ELEM_1D >> SURFACE

Note: • The curves defining the surface will be divided using the parametric definition of the surface.

How To:

1. Specify the type of mesh to be generated.
2. Pick surface(s).
3. Specify mesh generation criteria.
4. Define element size.

■ SPAR	Generate a spar type element for strength or heat analysis.
BEAM	Generate a beam type element for strength or heat analysis.
CONIC	Generate a conic type element for flow analysis.
STRIP	Generate a strip type element for flow analysis.

PICK SURFACE	Pick the surface on which the mesh of 1D elements will be created.

Note: • Only non-trimmed surfaces can be picked.

SELECT	ELEM TYPE	MATERIAL:NONE	HEAT MATERIAL:NONE	GEOMETRIC:NONE

If one or more property tables have been defined, (material, heat and/or geometric) you may select these properties and attach them to specific parts of the finite element model.

THIS WAY? YES NO	Accept an orientation for the curves which will define the surface on which nodes will be generated.
■ NO. OF CURVES = 2	Enter the number of curves to be used to define the surface.
■ NO. OF INTERVALS = 1	Enter the number of intervals into which each curve which defines the surface will be divided.

Note: • All the curves will be divided into the same number of intervals.

<CR> TO CONTINUE	Set the modal parameters for the division of the curves of the surface and press <CR>.
EXECUTING . . .	The straight line finite elements will be generated as specified.

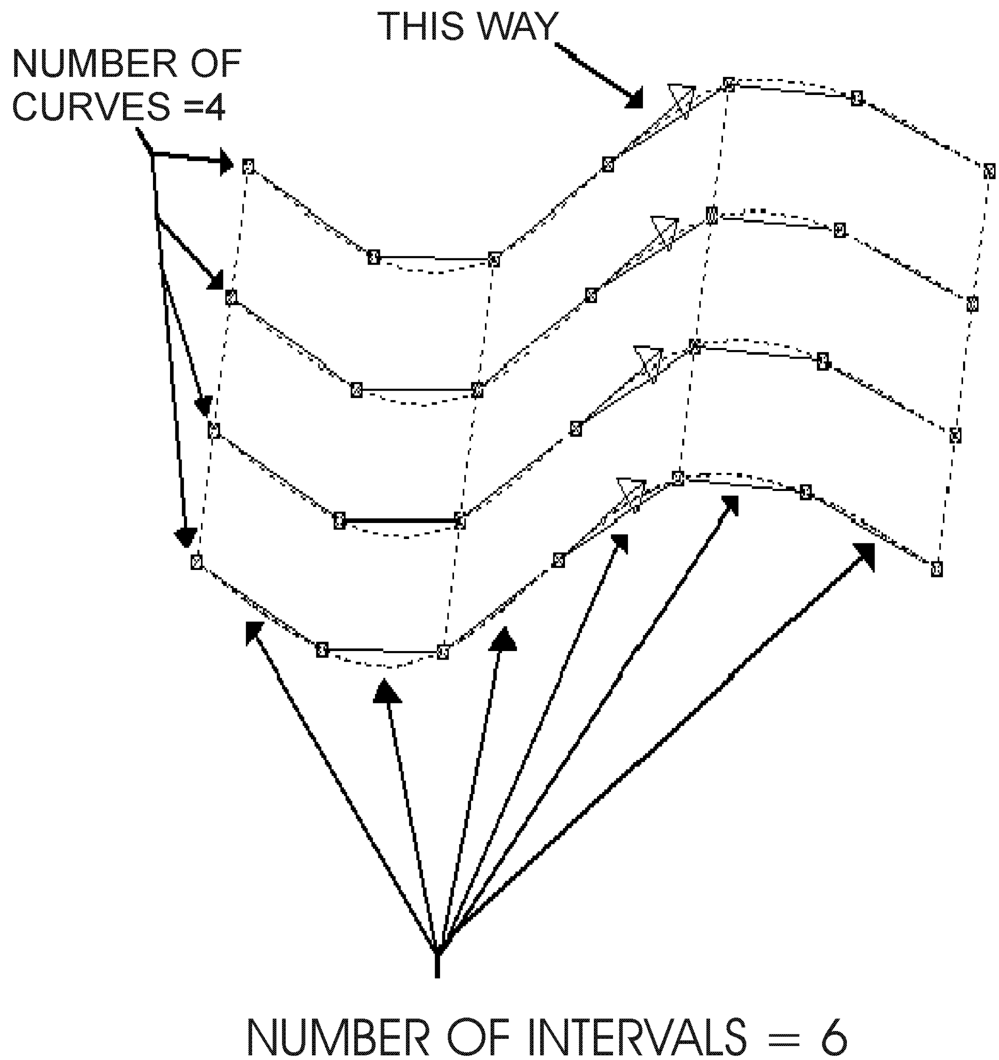


Figure 3-2: Mesh of 1D Elements on a Surface o

ELEM_2D

Automatically generate meshes of 2D finite elements, which are triangles or quadrilaterals whose 3 or 4 vertices are nodes.

This function is useful for generating finite elements for structures to be analyzed using plane stress, plane strain or thin plate bending assumptions.

Create mesh on solid faces. The 2D meshes can be used for closed 3D mesh, after meshing around the body.

How To:

1. Specify the type of mesh which will be generated.
2. Pick boundary curves defining a closed contour or the surface or the face on which a mesh is to be generated.
3. Specify mesh generation criteria:
   - — triangular or quadrilateral elements
   - — how each boundary curve should be divided.

   Generate nodes and mesh.
4. Smooth the mesh of triangular or quadrilateral elements.

Notes:

- 2D finite elements cannot be generated automatically from an open contour.
- If the mesh will be generated from a 3D contour, islands may not be defined.

SELECT:	2D CONTOUR
	3D CONTOUR
	SURFACE
	SOLID

2D CONTOUR Generate a 2D mesh within an area bounded by 2D curves.

3D CONTOUR Generate a 2D mesh on a surface defined by selected 3D curves.

SURFACE Generate a 2D mesh on a surface.

SOLID Generate a 2D mesh on solid face(s).

ELEM_2D >> 2D CONTOUR
ELEM_2D >> 3D CONTOUR

- PLATE
  MEMBRANE
  PL_STRESS
  PL_STRAIN
  AXI SYMMETRIC

Select the type of finite element to be generated based on the curves in this contour. For flow analysis, only plate elements may be generated.

PICK 1ST CURVE
INDICATE DIRECTION? YES NO
PICK 2ND CURVE
. .. PICK NTH CURVE

Continue to pick curves to define a closed contour. Refer to the **Cimatron Fundamentals & General Functions Manual** for a more detailed explanation if necessary.

END CONTOUR O.K.? YES NO

Notes:

- If the mesh will be generated from a 3D contour, do not define islands. Continue with the QUADRILATERALS option.
- SINGLE INDICATION is available for closed 2D contours. In an ISO view, indicate a point on the Z plane.

ISLAND M: 1ST CURVE. . .
. . .ISLAND N: NTH CURVE

Press <EXIT> when you are finished picking curves.

END DEFINITION? YES/NO

Confirm that you are finished picking curves.

If islands were defined, select the TRIANGLES option. If no islands were defined, select either option.

SELECT	ELEM TYPE	MATERIAL:NONE	HEAT MATERIAL:NONE	GEOMETRIC:NONE

If one or more property tables have been defined, (material, heat and/or geometric) you may select these properties and attach them to specific parts of the finite element model.

SELECT MESH TYPE:	TRIANGLES	QUADRILATERALS

TRIANGLES The mesh will be composed of triangular elements.

QUADRILATERALS The mesh will be composed of quadrilateral elements.

2D CONTOUR >> TRIANGLES

Define a 2D mesh of triangular elements.

SELECT MODE:	DIVIDE BOUNDARY
	BY ELEM. SIZE
	BY NUMB. OF ELEM.

DIVIDE BOUNDARY Divide each boundary by interval or length.

BY ELEM. SIZE Define the mesh by element size.

BY NUMB. OF ELEM. Define the mesh by the number of elements.

Notes:

- The curves that were picked will appear in attention, one at a time. For each curve, set the modal parameters which determine how they will be divided to generate the mesh.
- To divide all the curves in the same way, i.e. using the same modal parameter settings, set the modal parameters for the first curve, press <EXIT> and select YES:

UNIFORM DIVISION? YES NO For a more detailed explanation of the modal parameters, see ELEM_1D >> SINGLE CURVES option.

If nodes already exist on the curves, the following will appear for each curve:

USE PREDEFINED NODES? YES NO

YES Use the nodes which already exist on the curve. Skip to ADD INTERNAL NODES.

NO Create nodes as described by modals.

■ INTERVALS Divide each curve into a specified number of intervals.

LENGTH Divide each curve into intervals of a fixed length.

■ NO. OF INTERVALS = 1 If INTERVALS is selected, enter the desired number of intervals.

LENGTH = 10.000 If LENGTH is selected, enter the desired length on the curve, of the segments in the intervals between the nodes.

■ RATIO = 1.000 If INTERVALS is selected, enter the ratio between the last and the first line segments which define the curve.

<CR> TO CONTINUE Set the modal parameters and <CR>.

ADD INTERNAL NODES? YES NO

YES IND. NEW NODE & EXIT
Indicate a node to be added. The default indication option is SCREEN. <EXIT> when finished.

NO Continue.

EXECUTING . . . The triangular finite elements will be generated as specified.

2D CONTOUR/3D CONTOUR >> QUADRILATERALS

Define a 2D mesh of quadrilateral elements.

SELECT MODE	DIVIDE BOUNDARY
	BY ELEM SIZE
	BY NUMB OF ELEM

2D CONTOUR/3D CONTOUR >> QUADRILATERALS >> DIVIDE BOUNDARY

Define a 2D mesh by specifying how each boundary curve should be divided.

Notes:

- Four boundary curves must be specified for the area inside the closed contour to be automatically divided into quadrilaterals.
- Opposing boundaries must contain the same number of nodes.
- Nodes are automatically created at the intersection points between curves.

If the contour contains more than four curves, indicate four endpoints of its curves to divide the outer contour into four boundaries.

CORNERS O.K.? YES NO — Select YES to confirm the indicated corners. Select NO and press <REJECT> to indicate different corners.

IND. 1ST CORNER

IND. 2ND CORNER

IND. 3RD CORNER

IND. 4TH CORNER

DIVIDE THIS BOUNDARY? YES NO — Determine which of two opposing boundaries determines their division. Both boundaries will be divided in the same way. Repeat for the remaining two opposing boundaries.

Notes:

- The curves that were picked above will appear in attention, one at a time. For each curve, set the modal parameters which determine how they will be divided to generate the mesh.
- To divide all the curves in the same way, i.e. using the same modal parameter settings, set the modal parameters for the first curve, press <EXIT> and select YES:

UNIFORM DIVISION? YES NO

- For a more detailed explanation of the modal parameters, see ELEM_1D >> SINGLE CURVES option.

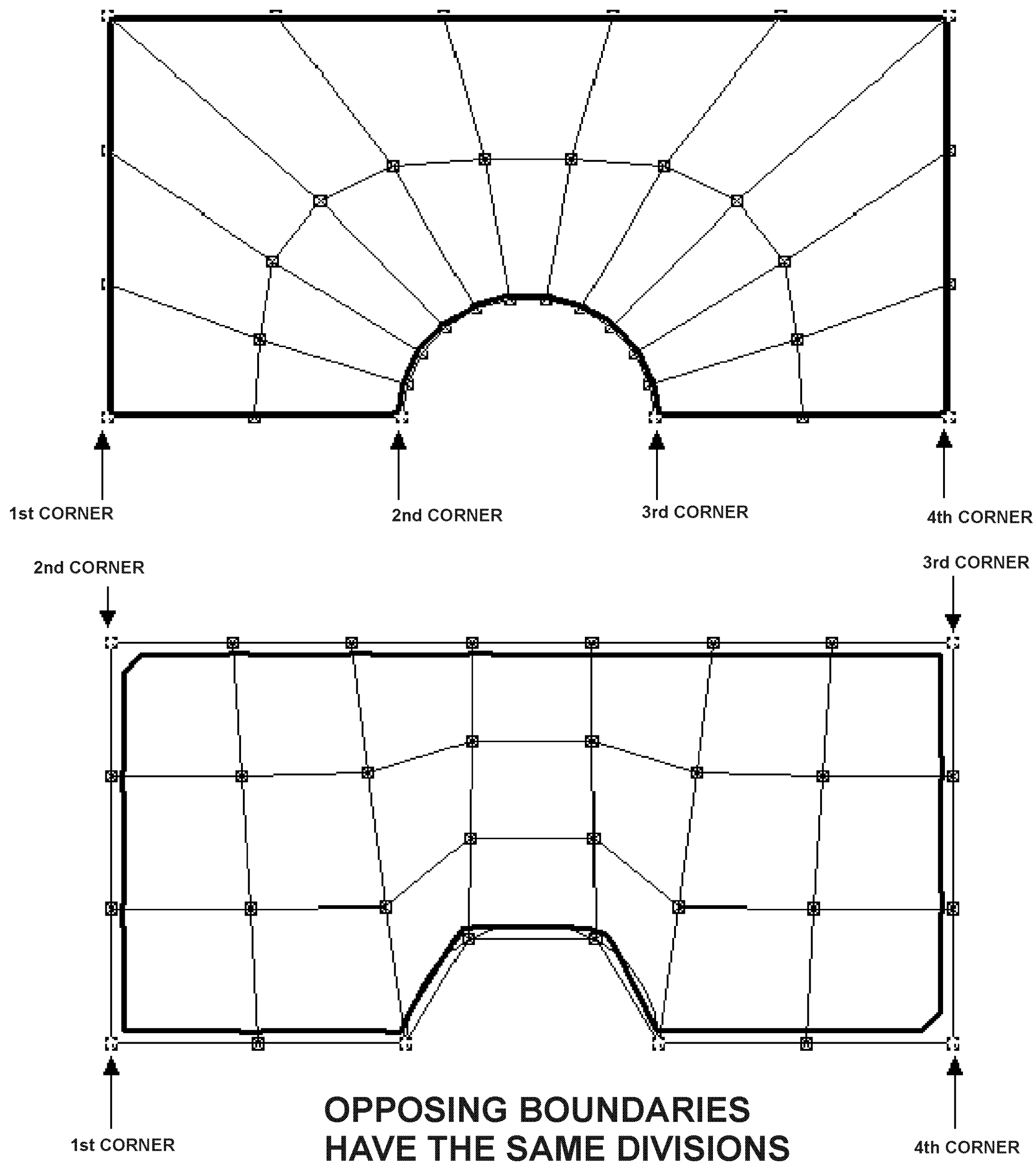


Figure 3-3: 2D CONTOUR >> Indicate Corners for Quadrilateral

If nodes already exist on the curves, the following will appear for each curve:

USE PREDEFINED NODES? YES NO

YES Use the nodes which already exist on the curve. Skip to BOUNDARY NODES O.K.?

NO Create nodes by defining the following parameters:

<CR> TO CONTINUE	INTERVALS	NO. OF INTERVALS = 1	RATIO = 1.000
	OPPOS:SAME DIVISION		
	OPPOS:NEW DIVISION		

- INTERVALS — Divide each curve into a specified number of intervals.

  LENGTH — Divide each curve into intervals of a fixed length.

- NO. OF INTERVALS = 1 — If INTERVALS is selected, enter the desired number of intervals.

  LENGTH = 10.000 — If LENGTH is selected, enter the desired length on the curve, of the segments in the intervals between the nodes.

- RATIO = 1.000 — If INTERVALS is selected, enter the ratio between the last and the first line segments which define the curve.

- OPPOS: SAME DIVISION — Divide opposing boundaries in the same way.

  OPPOS: NEW DIVISION — Define the division of each curve in each boundary separately.

<CR> TO CONTINUE — If OPPOS: SAME DIVISION is selected, press <CR> to accept modals which describe the division of opposing boundaries simultaneously.

If OPPOS: NEW DIVISION is selected, press <CR> to accept modals which described the division of only the curve in attention.

Caution: • Opposing boundaries must contain the same number of nodes.

BOUNDARY NODES O.K.? YES NO

YES The mesh of quadrilateral elements will be created.

NO PICK NODE

Pick a node to be moved that is not at the endpoint of a curve.

IND. NEW LOCATION

Indicate a new location on the same curve and so that it has the same adjacent nodes. The order of the nodes may not be changed. <EXIT> when finished moving nodes.

The mesh of quadrilaterals will be created after the displayed boundary nodes are accepted.

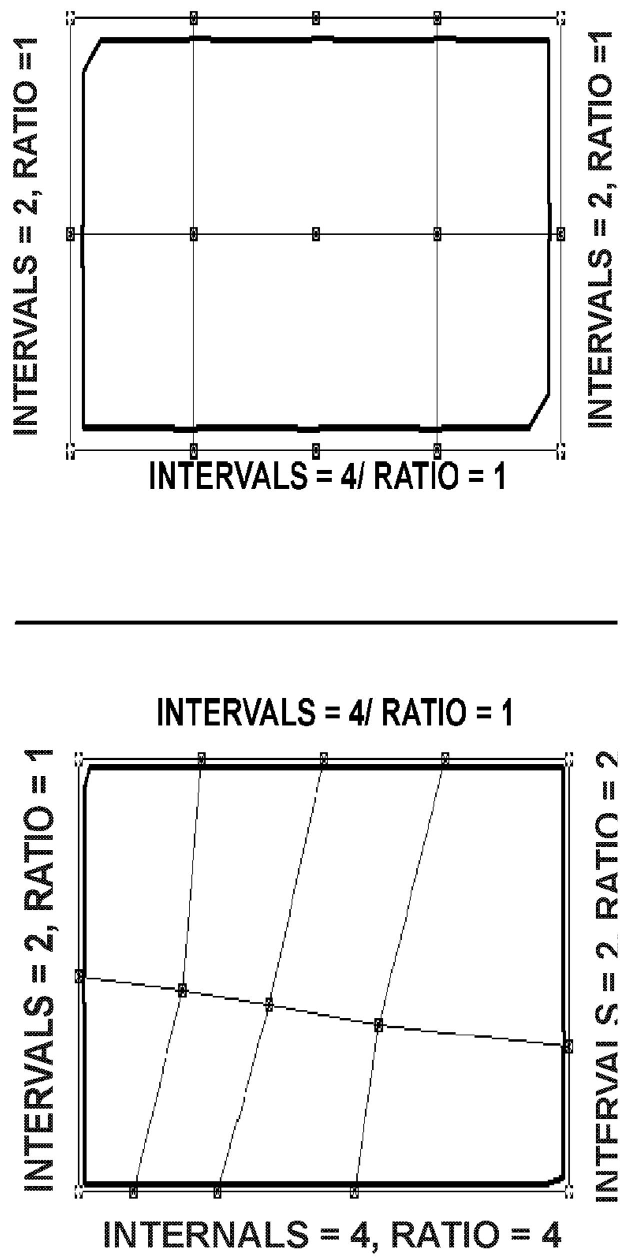


Figure 3-4: 2D CONTOUR >> QUADRILATERALS >> Division of Opposing Sides

SMOOTH & EXIT:	EQUIPOTENTIAL	AVERAGE

EQUIPOTENTIAL Smooth the mesh surface by moving each internal node so that its coordinates are the weighted average of the coordinates of the eight nodes in the adjacent quadrilaterals. The four nodes in the corners which are not adjacent to the internal node, carry greater weight.

AVERAGE Smooth the mesh surface by moving each internal node so that its coordinates are the average of the coordinates of the eight nodes in the adjacent quadrilaterals. These eight nodes all carry the same weight.

<EXIT> when you are finished smoothing the mesh.

2D CONTOUR/3D CONTOUR >> QUADRILATERALS >> BY ELEM SIZE

Define a 2D mesh by specifying the size of quadrilateral elements.

<CR> TO CONTINUE	ELEMENT LONGSIDE = 50.000	LONG/SHORT RATIO = 1.000

- ELEMENT LONGSIDE = 50.000 — The length of the longest side of the quadrilateral element.
- LONG/SHORT RATIO = 1.000 — The aspect ratio between the longest and shortest sides of the quadrilateral boundary. The range of the ratio may be between 1 to 10.

Note:
- The result will be an approximate of the modal value.

2D CONTOUR/3D CONTOUR >> QUADRILATERALS >> BY NUMB OF ELEM

Define a 2D mesh by specifying the number of quadrilateral elements.

<CR> TO CONTINUE	NUMB OF ELEMENTS = 4	

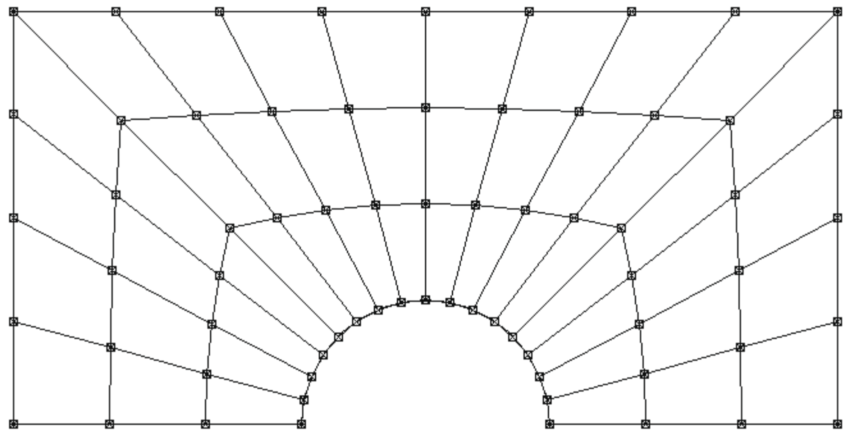
- NUMB OF ELEMENTS — The total number of elements which will be created.

Notes:
- Occasionally, the result will be an approximate of the modal value.
- The aspect ratio is approximated to 1 between the longest and the shortest sides of the quadrilateral element.

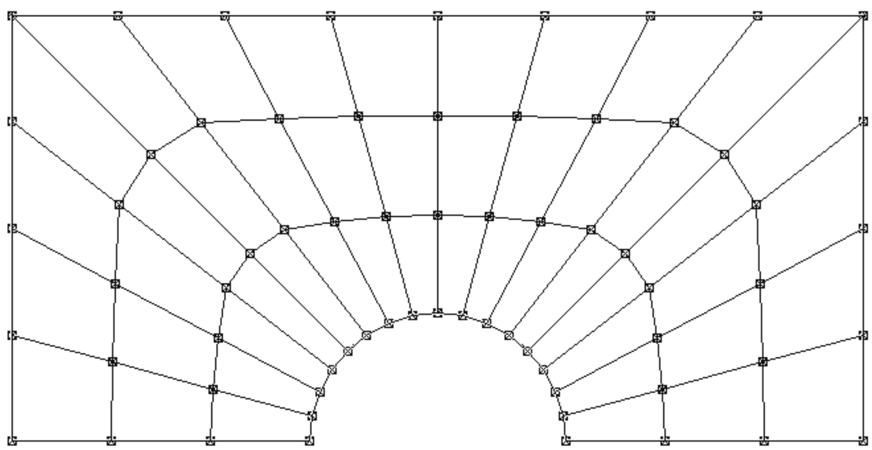
3D CONTOUR

Proceed as for 2D CONTOUR >> QUADRILATERAL.

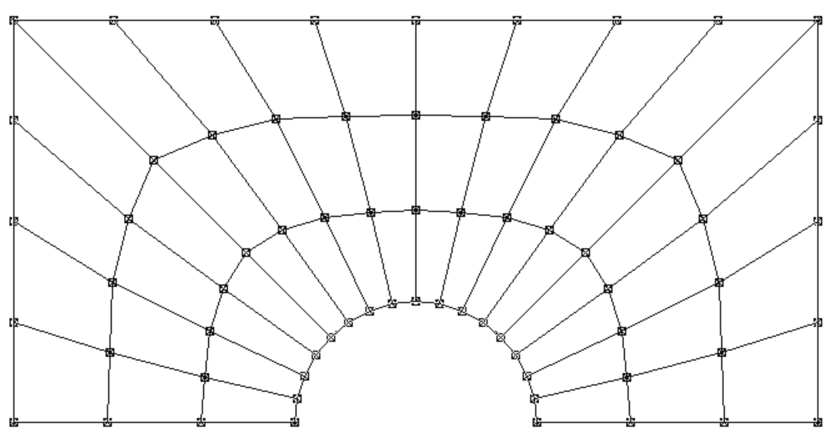
Islands may not be defined for 3D contours.



BEFORE SMOOTH



AFTER - AVERAGE



AFTER - EQUIPOTENTIAL

Figure 3-5: 2D CONTOUR >> QUADRILATERALS >> AVERAGE, EQUIPOTENTIAL

ELEM_2D >> SURFACE

Generate a mesh on a surface model with a user defined element size.

How To:

1. Specify the type of mesh to be generated.
2. Pick surface(s).
3. Specify mesh generation criteria.
4. Define element size.

Note:

- The curves defining the surface will be divided using the parametric definition of the surface.

■ PLATE MEMBRANE PL_STRESS PL_STRAIN AXI SYMETRIC	Select the type of finite element to be generated based on the curves in this contour. For flow analysis, only plate elements may be generated.

SELECT	SINGLE
	CHAIN
	ALL

SINGLE	Create a mesh on a picked surface.
CHAIN	Create a mesh on the first picked surface, and extend the mesh to additional picked neighboring surfaces.
ALL	Create one mesh on all picked surface in one step.

PICK SURFACE: Pick the surface on which a mesh is to be generated.

SELECT	ELEM TYPE	MATERIAL:NONE	HEAT MATERIAL:NONE	GEOMETRIC:NONE

If the file contains properties tables, you may select material, heat and geometric properties.

SELECT MESH TYPE:	TRIANGLES	QUADRILATERALS

TRIANGLES	The mesh will be composed of triangular elements.
QUADRILATERALS	The mesh will be composed of quadrilateral elements.

SURFACE >> SINGLE >> TRIANGLES

Define a 2D mesh of triangular elements.

SELECT MODE:	DIVIDE BOUNDARY
	BY ELEM. SIZE
	BY NUMB. OF ELEM.

DIVIDE BOUNDARY Divide each boundary by interval or length.

BY ELEM. SIZE Define the mesh by element size.

BY NUMB. OF ELEM. Define the mesh by the number of elements.

Note:

- Each boundary of the surface will be indicated by a set of arrows. For each boundary, set the modal parameters which determine how it and the surface will be divided to generate the mesh.

  For a more detailed explanation of the modal parameters, see ELEM_1D >> SINGLE CURVES option.

■ INTERVALS Divide each curve into a specified number of intervals.

■ NO. OF INTERVALS = 1 Enter the desired number of intervals.

■ PARAMETRIC The path is divided parametrically, and not geometrically.

■ RATIO = 1.000 If INTERVALS is selected, enter the ratio between the last and the first line segments which define the boundaries of the surface.

<CR> TO CONTINUE Set the modal parameters for each boundary.

INDICATE DIRECTION? Confirm a direction.

ADD INTERNAL NODES? YES NO
YES Continue to ADD INTERNAL NODES options.
NO Complete interaction and execute.

ADD INTERNAL NODES:	SINGLE	DISP INTERS

SINGLE Add internal nodes on the display curves of the surface by indicating them one at a time.

DISP. INTERS. Add internal nodes automatically, at the intersection points of the display curves of the surface, and indicate more as in the SINGLE option, if necessary.

SURFACE >> SINGLE >> QUADRILATERALS

Define a 2D mesh of quadrilateral elements.

Note:

- When a singular point (ball, cone, etc.) is selected, several triangular elements are automatically created in the singular zone.

SURFACE >> SINGLE >> QUADRILATERALS >> DIVIDE BOUNDARY

Define 2D meshes by defining how each boundary curve should be divided.

DIVIDE THIS PATH? *YES NO*	Determine which of two opposing boundaries determines their division. Both boundaries will be divided in the same way. Repeat for the remaining two opposing boundaries. For a more detailed explanation of the modal parameters, see ELEM_1D >> SINGLE CURVES option.
■ INTERVALS	Divide each curve into a specified number of intervals.
■ NO. OF INTERVALS = 1	If INTERVALS is selected, enter the desired number of intervals.
■ PARAMETRIC	The path is divided parametrically and not geometrically.
■ RATIO = 1.000	If INTERVALS is selected, enter the ratio between the last and the first line segments which define the boundaries of the surface.
<CR> TO CONTINUE	Set the modal parameters for each boundary.

SMOOTH & EXIT:	EQUIPOTENTIAL	AVERAGE

EQUIPOTENTIAL Smooth the mesh surface by moving each internal node so that its coordinates are the weighted average of the coordinates of the eight nodes in the adjacent quadrilaterals. The four nodes in the corners which are not adjacent to the internal node carry greater weight.

AVERAGE Smooth the mesh surface by moving each internal node so that its coordinates are the average of the coordinates of the eight nodes in the adjacent quadrilaterals. These eight nodes all carry the same weight.

<EXIT> when you are finished smoothing the mesh.

SURFACE >> SINGLE >> QUADRILATERALS >> BY ELEM SIZE

Define a 2D mesh by specifying the size of quadrilateral elements.

<CR> TO CONTINUE	ELEMENT LONGSIDE = 50.000	LONG/SHORT RATIO = 1.000

■ ELEMENT LONGSIDE = 50.000	The length of the longest side of the quadrilateral element.
■ LONG/SHORT RATIO = 1.000	The aspect ratio between the longest and shortest sides of the quadrilateral boundary. The range of the ratio may be between 1 to 10.

Note: • The result will be an approximate of the modal value.

SURFACE >> SINGLE >> QUADRILATERALS >> BY NUMB OF ELEM

Define a 2D mesh by specifying the number of quadrilateral elements.

<CR> TO CONTINUE	NUMB OF ELEMENTS = 4	

- NUMB OF ELEMENTS = 4 The total number of elements which will be created.

Notes:

- Occasionally, the result will be an approximate of the modal value.
- The aspect ratio is approximated to 1 between the longest and the shortest sides of the quadrilateral element.

SURFACE >> CHAIN

Create a mesh on the first picked surface, and extend the mesh to additional picked neighboring surfaces.

PICK 1ST SURFACE:	ELEMENT SIZE = 10.0

- ELEMENT SIZE = 10.0 Define the element size, and pick the surface on which a mesh is to be generated.

THIS SURFACE YES / NO Confirm the picked surface.

EXECUTING

PICK NEIGHBOR SURFACE:	ELEMENT SIZE = 8.0

Define the element size and pick the next surface. A different element can be selected. The surfaces have a common boundary.

Press <REJECT> to return one step back.

SURFACE >> ALL

Create one mesh on all picked surfaces in one step.

PICK SURFACE & EXIT	ELEMENT SIZE = 10.0

Enter the element size for all surfaces. Pick the surfaces (the multi-pick Submenu may be used). A mesh is generated for each group of connected surfaces. A surface will not be meshed in cases where an element size is too large.

ELEM_2D >> SOLID

Generate a 2D mesh on solid face(s).

How To:

1. Specify the type of mesh to be generated.
2. Pick face(s).
3. Define element size.

■ PLATE MEMBRANE PL_STRESS PL_STRAIN AXISYMETRIC	Select the type of finite element to be generated.
PICK FACES & EXIT	Press <SUBMENU> to display picking options.

SINGLE / BOX
POLYGON
EXCLUDE
ALL
ENTIRE SOLID

SELECT	ELEMENT SIZE = 10.0

■ ELEMENT SIZE = 10.0	Specify element size.

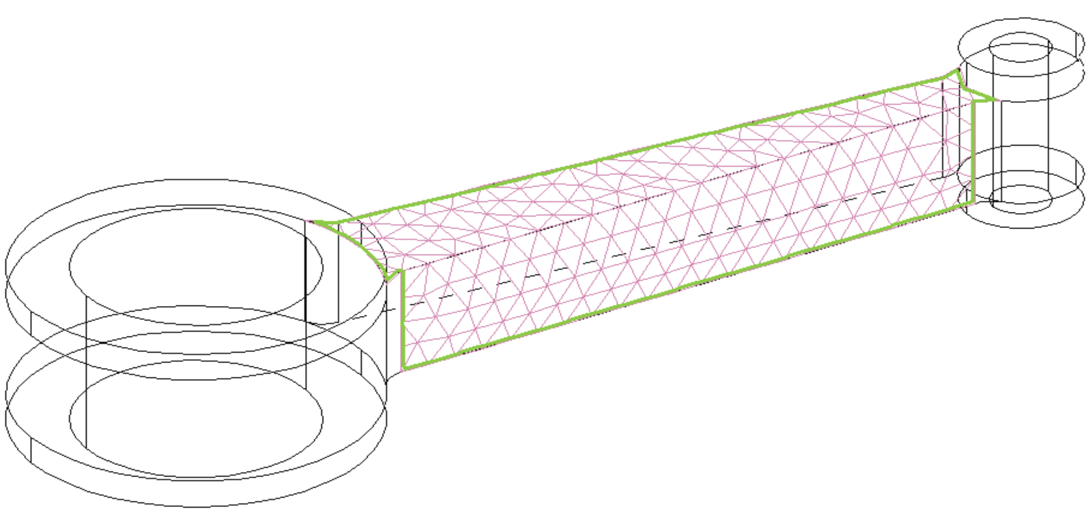


Figure 3-6: Solid 2D Mesh on a Solid Body

❑

ELEM_3D

Automatically generate a mesh of solids, (tetra, brick, or prism), by picking a solid body, by closing a 2D mesh, or by sweeping a 2D mesh.

SELECT	SWEEP MESH
	CLOSED MESH
	SOLID

ELEM_3D >> SWEEP MESH

How To:

1. Pick meshes of 2D elements, or 2D elements, from which the 3D mesh of solids will be created.
2. Define a linear or angular sweep to be performed and the number of intervals to be created.

SELECT:	MESHES
	ELEMENTS

MESHES or ELEMENTS

PICK MESHES & EXIT Pick 2D meshes of elements or 2D elements which are to be swept to create 3D meshes of solids.

or:

PICK ELEMENTS & EXIT

SELECT	ELEM TYPE	MATERIAL:NONE	HEAT MATERIAL:NONE	

If one or more property tables have been defined, (material and/or heat) you may select these properties and attach them to specific parts of the finite element model.

PICK MESHES & EXIT

SELECT OPTION:	LINEAR SWEEP
	ANGULAR SWEEP
	NORMAL TO SURFACE

LINEAR SWEEP Sweep meshes, elements, loads or constraints along a vector which is defined by two points.

ANGULAR SWEEP Sweep meshes, elements, loads or constraints through a given angle about a rotation axis.

NORMAL TO SURFACE Sweep only meshes generated on a surface along a normal to the surface.

MESHES / ELEMENTS >> LINEAR SWEEP

Sweep meshes, elements, loads or constraints along a vector which is defined by two points.

- NO. OF INTERVALS = 1 Enter the number of tiers into which the 3D mesh of solids will be divided.

  Set the remaining modals to sweep, or not sweep, characteristics which are attached to the 2D elements or meshes to be swept.

- NO CONS. SWEEP
  SWEEP CONSTRAINTS
- NO LOADS.SWEEP
  SWEEP LOADS
- NO TRACS. SWEEP
  SWEEP TRACTIONS
- NO TEMPS.SWEEP
  SWEEP TEMPERATURES
- NO NODALTEMP SWEEP
  SWEEP NODAL TEMP
- NO HEATFLOW SWEEP
  SWEEP HEAT FLOW
- NO CONVEC.SWEEP
  SWEEP CONVECTIONS

IND.VECTOR ORIGIN Set the modals and indicate the origin point for the sweep vector.

IND. VECTOR TARGET Indicate the sweep vector's endpoint.

POINT O.K.? *YES NO*	**YES**	The sweep will be executed and the 3D mesh of solids will be created.
	NO	Indicate a different point as the endpoint of the sweep vector.

DELETE THE 2D ELEMS? YES NO

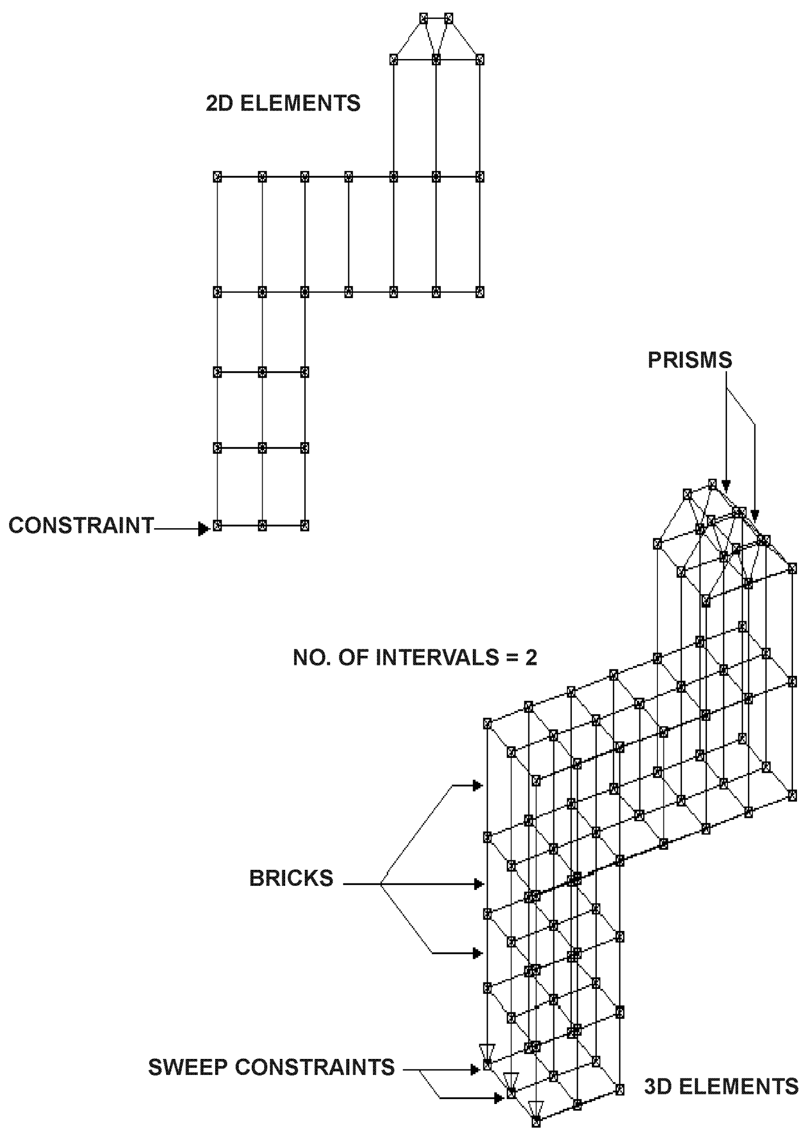


Figure 3-7: Mesh of Solids >> LINEAR SWEEP

MESHES / ELEMENTS >> ANGULAR SWEEP

Sweep meshes or elements through a given angle about a rotation axis.

IND.AXIS ORIGIN	Indicate the origin point of the rotation axis.
IND.AXIS DIRECTION	Indicate a point to define the direction of rotation of the axis.

SELECT:	ANGLE	REF POINTS

ANGLE — Define the angle of rotation of the meshes or elements around the axis, using a modal variable.

REF. POINTS — The angle of rotation of the meshes or elements around the axis will have its vertex on the axis. The angle is defined by two indicated points which are projected onto a plane perpendicular to the rotation axis.

MESHES / ELEMENTS >> ANGULAR SWEEP >> ANGLE

- NO. OF INTERVALS = 1 — Enter the number of tiers into which the 3D mesh of solids will be divided.
- ANGLE = 30.00 — Enter the rotation angle in degrees.

Set the following modals to sweep, or not sweep, characteristics which are attached to the 2D elements or meshes to be swept.

- NO CONS. SWEEP
  SWEEP CONSTRAINTS
- NO LOADS.SWEEP
  SWEEP LOADS
- NO TRACS. SWEEP
  SWEEP TRACTIONS
- NO TEMPS. SWEEP
  SWEEP TEMPERATURES
- NO NODALTEMP SWEEP
  SWEEP NODAL TEMP
- NO HEATFLOW SWEEP
  SWEEP HEAT FLOW
- NO CONVEC.SWEEP
  SWEEP CONVECTIONS

<CR> TO CONTINUE	Press <CR> to accept the displayed modals and continue.
DELETE THE 2D ELEMS? YES NO	Answer YES to delete the original 2D elements or meshes.

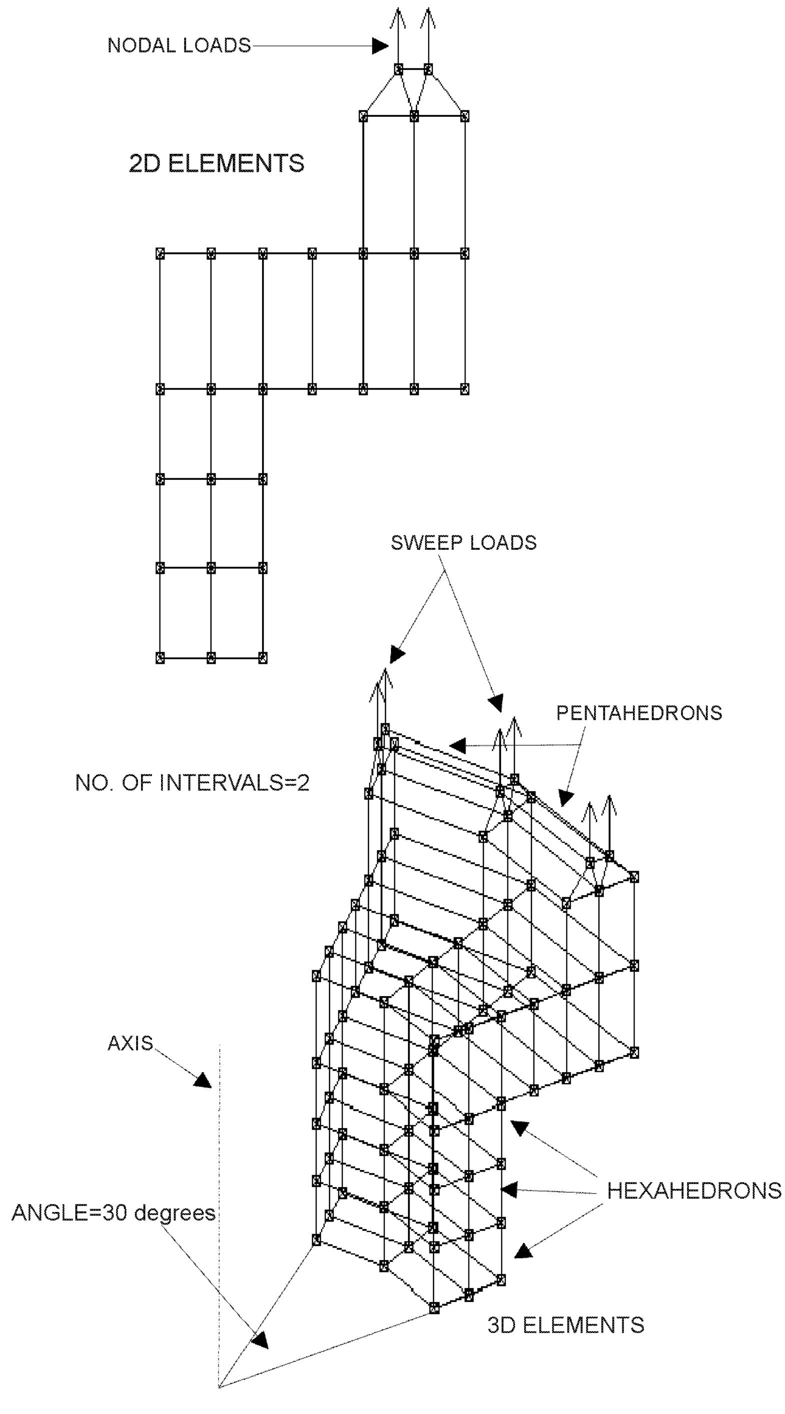


Figure 3-8: Mesh of Solids >> ANGULAR SWEEP

MESHES / ELEMENTS >> ANGULAR SWEEP >> REF POINTS

Modals same as LINEAR SWEEP.

IND. ANGLE START PNT Indicate a point which defines the start of the rotation angle.

IND. ANGLE TARGET PNT Indicate a point which defines the end of the rotation angle.

Note:
- The vertex of the rotation angle is at the origin of the rotation axis.

POINT O.K.?
YES NO

YES The sweep will be executed and the mesh of solids will be generated.

NO Indicate the angle target point again.

DELETE THE 2D ELEMS
YES NO

Answer YES to delete the original 2D elements or meshes.

MESHES >> NORMAL TO SURFACE

Sweep only meshes generated on a surface along a normal to the surface.

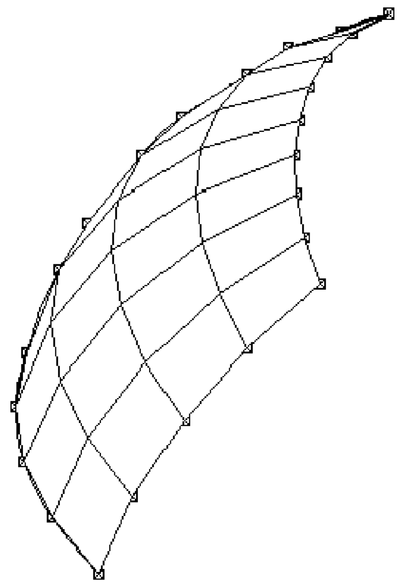
- NO. OF INTERVALS = 1 — Enter the number of tiers into which the 3D mesh of solids will be divided.
- TOTAL THICKNESS=5.00 — Set the thickness of the 3D mesh.

Set the remaining modals to sweep, or not sweep, characteristics which are attached to the 2D elements or meshes to be swept.

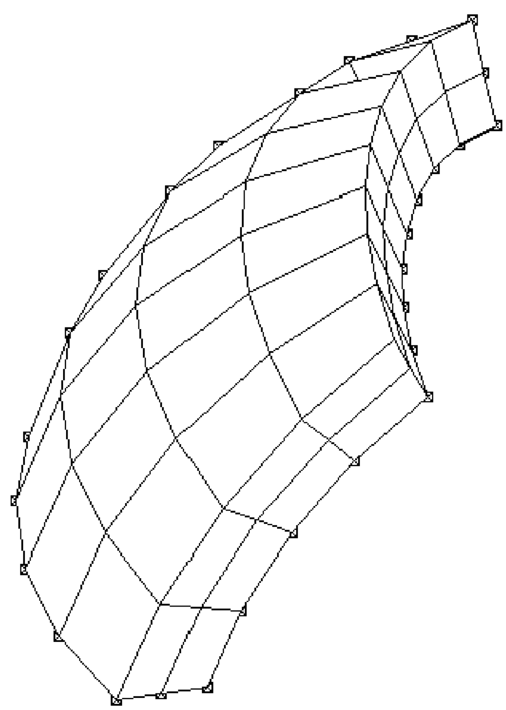
- NO CONS. SWEEP
  SWEEP CONSTRAINTS
- NO LOADS.SWEEP
  SWEEP LOADS
- NO TRACS. SWEEP
  SWEEP TRACTIONS
- NO TEMPS.SWEEP
  SWEEP TEMPERATURES
- NO NODALTEMP SWEEP
  SWEEP NODAL TEMP
- NO HEATFLOW SWEEP
  SWEEP HEAT FLOW
- NO CONVEC.SWEEP
  SWEEP CONVECTIONS

SELECT DIRECTION Indicate the sweep direction.

DELETE THE 2D MESH.? YES NO



2D MESH GENERATED ON A SURFACE



RESULT OF NORMAL TO SURFACE SWEEP

Figure 3-9: MESHES/ELEMENTS >> NORMAL TO SURFACE

ELEM_3D >> CLOSED MESH

Generate a tetra mesh from the 2D mesh on the outer envelope.

How To:

1. Pick CLOSE MESH. A closed mesh mark with magenta contour box will appear.
2. Define element size and internal points.
3. Choose whether to delete the 2D mesh.

PICK MESH	Select a closed mesh (A closed mesh is enclosed by a magenta colored box).
SELECT ELEMENT SIZE	Specify element size. The system default is based on calculating the outer size mesh.
INTERNAL POINTS? YES / NO	Create internal points. If the object is thin, then the internal points can be avoided.
DELETE THE 2D MESH?	Delete the 2D mesh that was used for constructing the 3D mesh.
YES / NO	**YES** Delete the 2D mesh. **NO** Return to the previous frame.

ELEM_3D >> SOLID

Generate a tetra mesh from a solid object.

How To:

1. Define internal points. The internal points can be avoided if the object is thin.
2. Define the element size.

Note: • You do not have to pick faces.

INTERNAL POINTS? YES / NO	**YES**	Create internal points. If the object is thin, then the internal points can be avoided.
	NO	
SELECT ELEMENT SIZE		
Specify element size.	The system default is based on calculating the outer size mesh.	

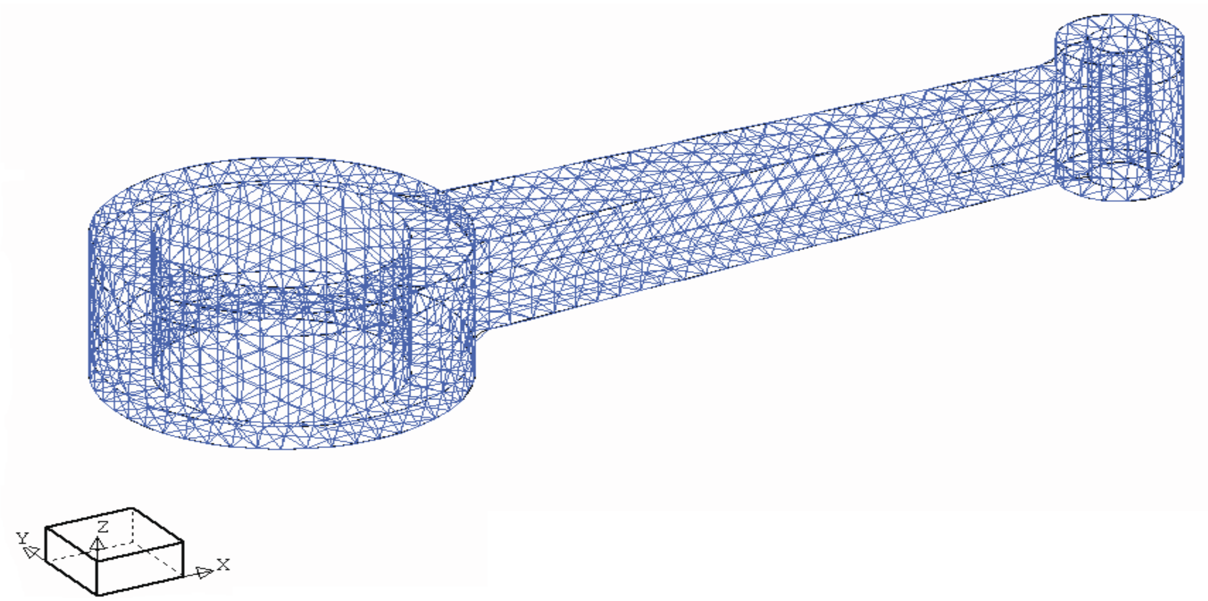


Figure 3-10: 3D Solid Mesh on a Solid Body

F_EDIT

Change, create or delete nodes, elements and meshes.

SELECT:	NODE
	ELEMENT
	MESH
	Z-DIRECTION
	DELETE 3D ELEMENTS

NODE Create manually, delete or move nodes.

ELEMENT Create manually, delete or move elements.

MESH Edit, delete or merge meshes.

Z-DIRECTION Determine the Z axis direction of 2D elements.

DELETE 3D ELEMENTS Deletes all 3D elements created in one session.

F_EDIT >> NODE

SELECT:	CREATE	DELETE	MOVE	MERGE

NODE >> CREATE

IND.POINT — At each point that is indicated a node will be created. <EXIT> when finished creating nodes.

NODE >> DELETE

PICK NODES & EXIT — <PICK> the nodes to be deleted. <EXIT> when finished. The picked nodes will be deleted.

Warning: • Deletion of a node causes deletion of all elements in which that node was used.

Note: • A node that belongs to a mesh cannot be deleted.

NODE >> MOVE

PICK NODE — <PICK> a node to be moved.

IND.TARGET POINT — Indicate a new location for the node. Continue to indicate new locations. When it is located properly, press <EXIT>.

<PICK> another node to be moved or <EXIT> again.

NODE >> MERGE

PICK NODES & EXIT — <PICK> the nodes to be merged and press <EXIT>.

OR

Press <SUBMENU> to display the multi-pick menu.

SINGLE
BOX

Use this menu to select a method of picking the nodes. Press <EXIT> when finished.

■ TOLERANCE = 0.001 — Enter the maximum deviation that will be tolerated.

<PICK> another node(s) to be merged or <EXIT> again.

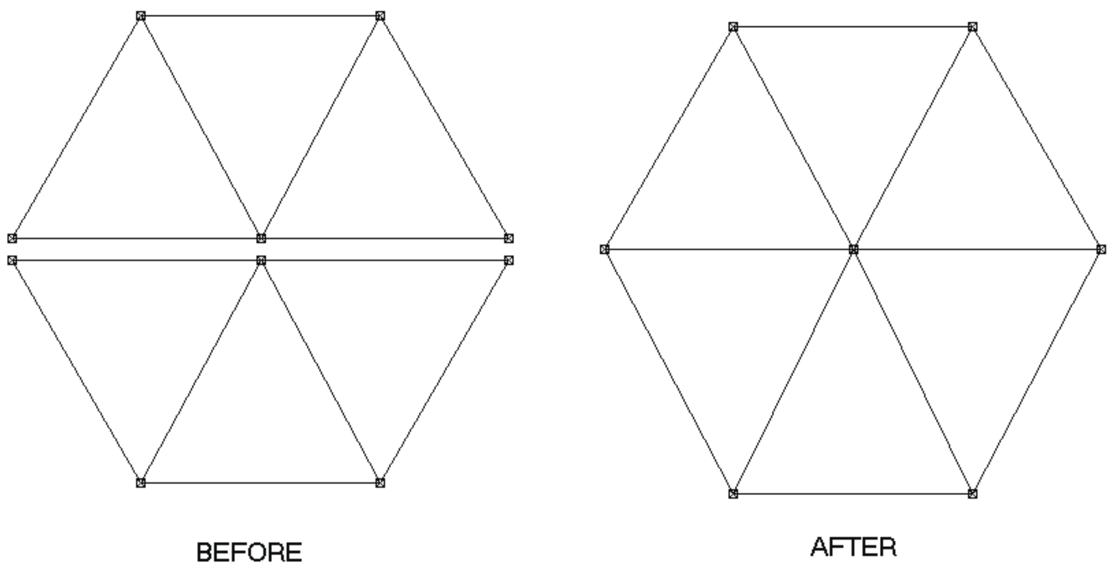


Figure 3-11: NODE Merge

F_EDIT >> ELEMENT

SELECT:	CREATE	DELETE	CHANGE TYPE

ELEMENT >> CREATE

SELECT:	LINEAR	TRIANGLE	QUAD.	BRICK	PRISM	TETRA
	MASS	SPRING	RIGID			

ELEMENT >> CREATE >> LINEAR

■ SPAR	Generate a spar type element for strength or heat analysis.
BEAM	Generate a beam type element for strength or heat analysis.
CONIC	Generate a conic type element for flow analysis.
STRIP	Generate a strip type element for flow analysis.
■ SINGLE	Generate a single linear element.
CONTINUOUS	Generate a continuous series of linear elements.

PICK 1ST NODE
PICK 2ND NODE

Continue to pick nodes to create additional linear elements. <EXIT> when finished.

If CONTINUOUS is selected, a new line element will be generated each time a new 2ND NODE is picked.

ELEMENT >> CREATE >> TRIANGLE

■ PLATE
MEMBRANE
PL STRESS
PL STRAIN
AXISYMMETRIC

Select the type of triangle element to be created based on the three nodes which will be picked.

PICK 1ST NODE
PICK 2ND NODE
PICK 3RD NODE

<PICK> the nodes of the triangle element to be created. Continue to pick nodes to create additional triangle elements. <EXIT> when finished.

ELEMENT >> CREATE >> QUAD

■ PLATE
MEMBRANE
PL STRESS
PL STRAIN
AXISYMMETRIC

Select the type of quadrilateral element to be created based on four nodes which will be picked. If flow analysis is specified, only elements may be created.

PICK 1ST NODE
PICK 2ND NODE

PICK 3RD NODE *PICK 4TH NODE*	<PICK> the nodes of the quadrilateral element to be created. Continue to pick nodes to create additional quadrilateral elements. <EXIT> when finished.

ELEMENT >> CREATE >> BRICK

PICK NODE 1 *PICK NODE 2* . . *PICK NODE 8*	<PICK> the nodes of the brick element to be created. Continue to pick nodes to create additional brick elements. <EXIT> when finished.

ELEMENT >> CREATE >> PRISM

PICK NODE 1 *PICK NODE 2* . . *PICK NODE 6*	<PICK> the nodes of the prism element to be created. Continue to pick nodes to create additional prism elements. <EXIT> when finished.

ELEMENT >> CREATE >> TETRA

PICK NODE 1 *PICK NODE 2* *PICK NODE 3* *PICK NODE 4*	<PICK> the nodes of the tetra element to be created. Continue to pick nodes to create additional tetra elements. <EXIT> when finished.

ELEMENT >> CREATE >> MASS

PICK NODE FOR MASS	<PICK> the node of the mass element to be created. <EXIT> when finished.

ELEMENT >> CREATE >> SPRING

■ SINGLE	Generate a single spring element.
CONTINUOUS	Generate a continuous series of spring elements.
PICK 1ST NODE *PICK 2ND NODE*	Continue to pick nodes to create additional spring elements. <EXIT> when finished. If CONTINUOUS is selected, a new spring element will be generated each time a new 2ND NODE is picked.

ELEMENT >> CREATE >> RIGID

PICK MASTER NODE	Pick an existing NODE which will be the independent node.
■ DX,DY,DZ,RX,RY,RX	Component numbers of the dependent degrees (degrees of freedom). The default is 6 degrees of freedom.
PICK SLAVES & EXIT	Pick slave node(s) and <EXIT>.

Notes:

- A slave may belong to more than one master only if the dependent degrees for each master are different.
- A constrained node may only be a slave if the dependent degrees are different from those of the constraints.

Create Prism Element from Picked Nodes

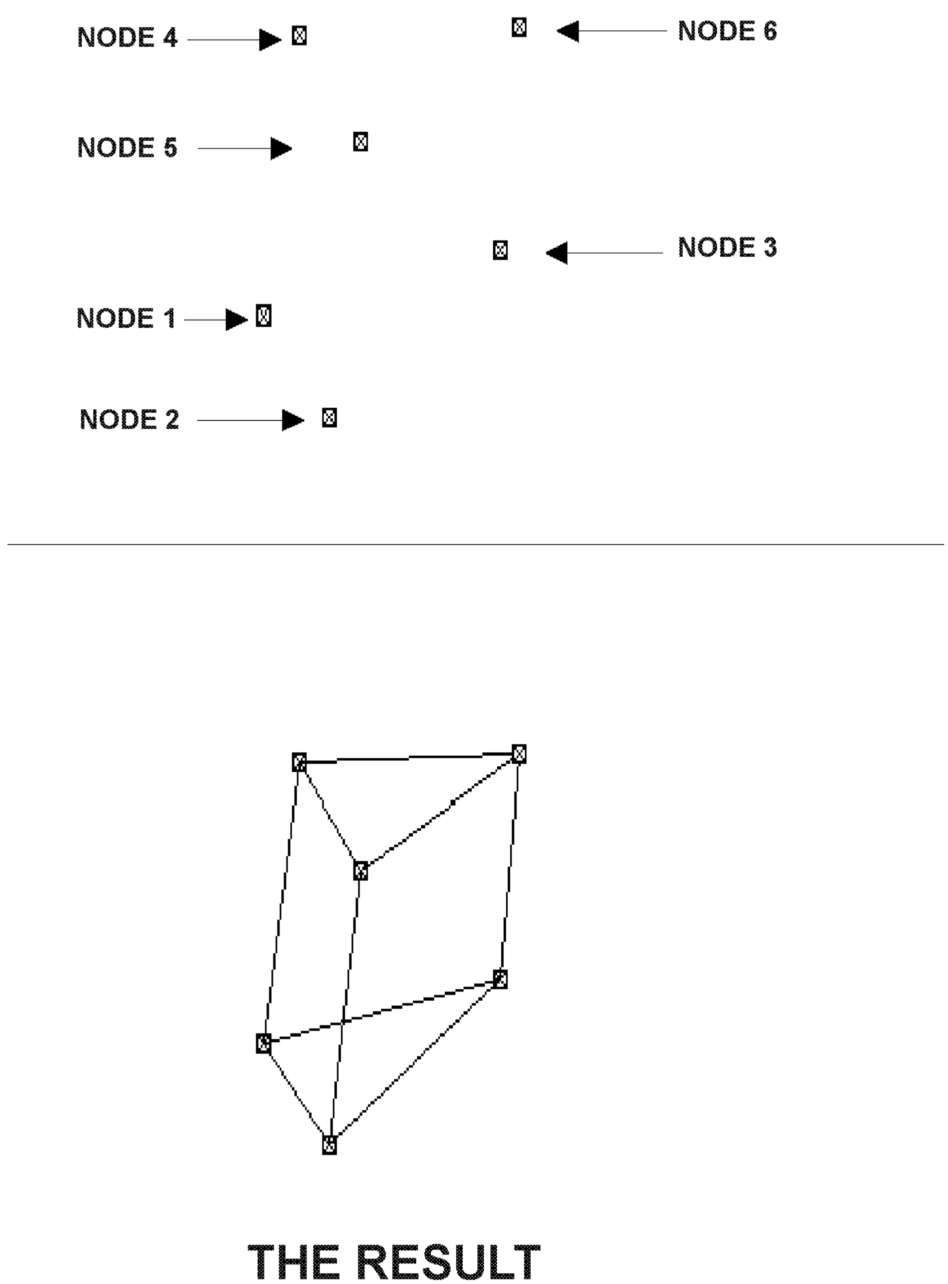


Figure 3-12: PRISM Element

ELEMENT >> DELETE

PICK ELEMENTS & EXIT <PICK> all the elements to be deleted. <EXIT> when finished. The picked elements will be deleted.

ELEMENT >> CHANGE TYPE

Note:

- You may change the type of 1D or 2D elements. You may not change the type of 3D elements.

Be careful to choose a type that is appropriate for the element you pick, i.e. SPAR or BEAM for 1D elements; PLATE, MEMBRAN, PL STRS, PL STRN, or AXISYMM for 2D elements.

■ SPAR
BEAM
RIGID
PLATE
MEMBRAN
PL STRS
PL STRN
AXISYMM

Set the modal to the desired type.

or:

CONIC
STRIP
PLATE

PICK ELEMENTS & EXIT <PICK> the elements whose types are to be changed. <EXIT> when finished. The types will be changed as specified.

F_EDIT >> MESH

Edit, delete or merge meshes.

MESH DELETE	EXPLODE	MESH MERGE

MESH >> MESH DELETE

PICK MESHES & EXIT <PICK> all meshes to be deleted. <EXIT> when finished. The picked meshes will be deleted.

DELETE NODES YES / NO Choose to delete the nodes or not.

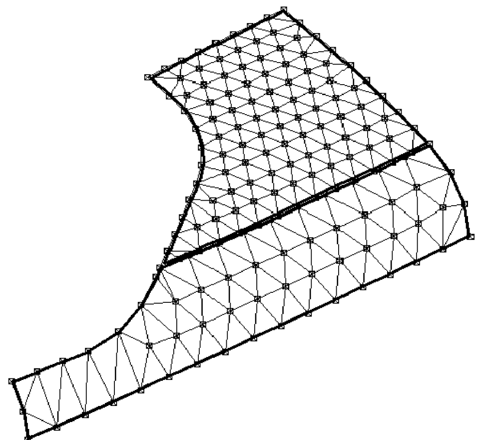
MESH >> EXPLODE

PICK MESHES & EXIT <PICK> all meshes to be exploded. <EXIT> when finished. The picked meshes will be exploded into elements.

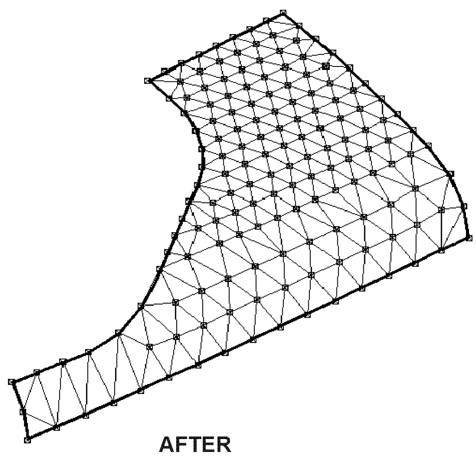
MESH >> MESH MERGE

PICK 1ST MESH

PICK 2ND MESH The two meshes will merge into one mesh.



BEFORE



AFTER

Figure 3-13: MESH MERGE

F_EDIT >> Z-DIRECTION

Determine the Z axis direction of elements.

MESHES
ELEMENTS

PICK ENTITIES & EXIT

FILL	ARROW	FLIP	APPLY

FILL	Assign a color to an element. The color will be according to the Z axis direction of the element.
ARROW	Assign an arrow symbol to an element in the direction of its Z axis.
FLIP	Change the direction of the element.
APPLY	Update the database with the changes.

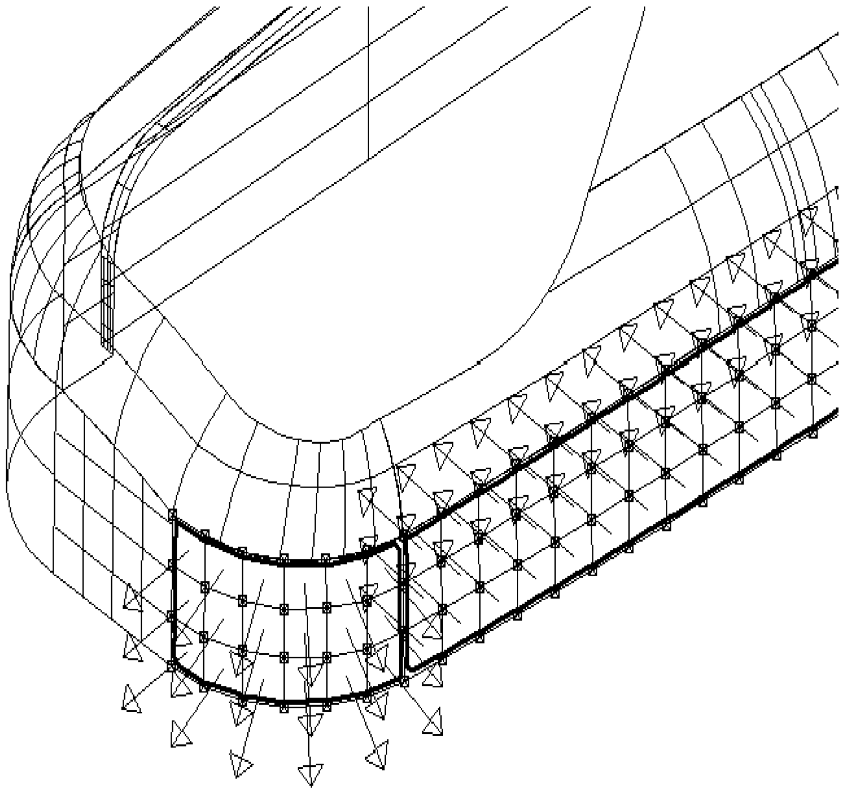


Figure 3-14: ARROW

F_EDIT >> DELETE 3D ELEMENTS

Deletes all 3D elements created in one session.

PICK ELEMENTS & EXIT <PICK> 3D elements to be deleted. <REJECT> to cancel picking.

Note:

- Only elements that were created with angular mesh can be deleted.

❑

F_MOVE

Translate, scale and rotate finite elements and finite element meshes.
All options of this function allow you to MOVE or COPY the selected elements or meshes. If MOVE is selected, the original elements or meshes are transformed. If COPY is selected, the originals remain untouched and a copy of them is created, translated, rotated and scaled as specified.

Refer to the Modeling function MOVE, **Cimatron^{it} Modeling Manual** for a more detailed description of the options and modal parameters, if necessary.

How To:

1. Pick mesh or element.
2. Choose copy / move.
3. Define how to accomplish it.

MESHES
ELEMENTS

F_MOVE >> MESHES
F_MOVE >> ELEMENTS

PICK MESHES & EXIT

or:

PICK ELEMENTS & EXIT

■ COPY	Copy the picked originals.
MOVE	Move the picked originals.
	If COPY is selected:
■ AS ORIGINAL	Assign only the level of the original meshes or elements to the copies.
AS ACTIVE	Assign the active level only to the copies.

■ SAME PROPERTIES	Attach the supplementary material and geometrical properties of the original to the copies.
NO PROPERTIES	Do not attach any properties to the copies.
<CR> TO CONTINUE	Set the modals and press <CR>.

POINT-ANGLE
DELTA
ROT AXIS
REF.POINTS
MIRROR
UCS TO UCS

POINT-ANGLE	Rotate elements or meshes through a given angle in the work plane, scale them and translate them to a given point.
DELTA	Translate elements or meshes by a given vector.
ROT.AXIS	Rotate elements or meshes through a given angle about a given axis.
REF.POINTS	Relocate and scale elements or meshes relative to a reference coordinate system.
MIRROR	Relocate elements or meshes relative to a mirror plane (reflection). The plane of the reflection is defined by three points.
UCS TO UCS	Move/copy elements or meshes between coordinate systems.

MESHES/ELEMENTS >> POINT ANGLE

Rotate elements or meshes through a given angle in the work plane, scale them and translate them to a given point.

■ ANGLE = 30.00	Rotation angle in degrees.
■ SCALE = 1.000	The distance between the reference point and every point on the "moved" elements will be multiplied by this factor.
■ COUNT = 1	Number of successive transformations.
■ DEFINE PLN	Select this modal to define an active coordinate system and a temporary work plane. See Defining a Plane Using DEFINE PLN in the **Cimatron**[it] Modeling Manual for a detailed explanation.
IND.REFERENCE POINT	Indicate the first endpoint of the vector of translation.
IND.POSITIONING PNT	Indicate the direction point of the vector.
EXECUTING . . .	The defined MOVE/COPY will be performed.

MESHES/ELEMENTS >> DELTA

Translate elements or meshes by a given vector

■ DX = 0.000	Define a coordinate system relative to which the move/copy will be performed and enter the distance to translate the elements or meshes along the X, Y, and Z axes.
■ DY = 0.000	
■ DZ = 0.000	
■ CARTESIAN CYLINDRICAL SPHERICAL	
■ MODEL UCS UCS LIST FEMSYS	
■ UCS NAME = . . .	
<CR> OR PICK UCS	See the **Cimatron**[it] **Modeling Manual** for a detailed explanation of the modals, if necessary.
■ COUNT = 1	If COPY is selected, enter the number of successive transformations to execute.
<CR> TO CONTINUE	Set the modals and press <CR>.
EXECUTING . . .	The defined MOVE/COPY will be performed.

MESHES/ELEMENTS >> ROT.AXIS

Rotate elements or meshes through a given angle about a given axis.

IND. AXIS ORIGIN — Indicate the origin of the rotation axis.

IND. AXIS DIR./EXIT — Indicate a direction point for the rotation axis.

Note:
- To define a rotation axis which is perpendicular to the work plane, indicate only one point and press <EXIT>.

■ ANGLE = 30.000 — Rotation angle in degrees.

■ COUNT = 1 — Number of successive transformations.

<CR> TO CONTINUE — Press <CR> to accept the displayed modals and continue.

EXECUTING . . . — The defined MOVE/COPY will be performed.

MESHES/ELEMENTS >> REF.POINTS

Relocate and scale elements or meshes relative to a reference coordinate system.

Note:
- This option moves/copies meshes or elements from one coordinate system to another, performing both translation and rotation. If only one point is defined for the receiving coordinate system, only translation will be performed.

IND. 1ST ORIGIN — Indicate an origin of a reference coordinate system for the meshes or elements to be moved.

■ SCALE = 1.000 — The distance between the reference point and every point on the “moved” entities will be multiplied by this factor.

IND 1ST +X AXIS/EXIT — Indicate a second point on the +X axis or <EXIT> to move/copy only the origin without changing the orientation.

IND 1ST +Y/EXIT — Indicate a third point in the direction of the +Y axis or <EXIT> to perform a 2D move/copy.

IND. 2ND ORIGIN — Indicate an origin for the moved or copied meshes or elements.

IND. 2ND +X AXIS/EXIT — Indicate a point on the +X axis of the moved or copied meshes or elements if one was indicated for the reference system. <EXIT> to move only the origin without changing the orientation.

IND. 2ND + Y/EXIT — Indicate a point in the direction of the +Y axis of the moved or copied meshes or elements if one was indicated for the reference system. <EXIT> to perform a 2D move or copy.

EXECUTING . . . — The defined move/copy will be performed.

MESHES/ELEMENTS >> MIRROR

Relocate elements or meshes relative to a mirror plane (reflection). The plane of the reflection is defined by three points. Copy or move entities by mirroring them onto a given plane.

ACTIVE UCS XY PLANE? YES NO

YES Use the XY plane of the active UCS.

NO One of the following options to define a different active work plane temporarily.

DEFINE PLANE:	UCS	PL FACE
	CURVES	VECTOR
	3 POINTS	COEFF
	PARALLEL	DISPLAY

See Defining a Plane Using DEFINE PLN in the **Cimatron**^{it} Modeling Manual for a detailed explanation of the options.

EXECUTING . . . The defined MOVE/COPY will be performed.

MESHES/ELEMENTS >> UCS TO UCS

Move/copy elements or meshes between coordinate systems.

- SCALE = 1.000 — Enter a factor by which to multiply the display size of the entities.

PIC START UCS Pick the UCS from which the entities will be copied or moved.

PICK DESTINATION UCS Pick the UCS to which the entities will be copied or moved.

UCS O.K.? YES NO

YES Skip to EXECUTING.

NO Pick another destination UCS.

EXECUTING . . . The defined move or copy will be performed each time a destination UCS is picked. <EXIT> when finished. ❑

Chapter 4
The Loads and Properties Functions

The following functions are described in detail in this chapter in alphabetical order. They may be selected anytime a FEMSYS is active. <PICK> the VERIFY function in the General Systems Function Bank, and <PICK> the FEM option to access the FEM VERIFY options. Press <SUBMENU> to access the modeling functions.

F_LOAD	Define loads acting upon the finite element model and its boundary constraints.
F_PROP	Define and attach supplementary material or geometric properties.
VERIFY >> FEM	Verify the coordinates of loads, ID numbers of elements, the nodes composing each element and the attachment status of supplementary material and/or geometric properties.

F_LOAD

Define loads affecting the finite element model.

SELECT:	STRENGTH	HEAT

Notes:

- If a load has already been attached to a node or element, use the UPDATE option to change it. This also applies to the case in which only the X component was already defined. To define the Y or Z component, use the UPDATE option.
- Modal parameters defining loads are expressed in the active coordinate system.

F_LOAD >> STRENGTH

Define the forced displacements, in the active coordinate system, for each degree-of-freedom at nodes. A value of 0 indicates no movement.

SELECT:	NODAL CONSTRAINTS
	NODAL LOADS
	NODAL TEMP
	TRACTIONS
	BODY FORCE

STRENGTH >> NODAL CONSTRAINTS

SELECT:	ATTACH	DETACH	UPDATE

STRENGTH >> NODAL CONSTRAINTS >> ATTACH

Enter a value in each of the following modals to define the forced displacement, in the active coordinate system, for each degree-of-freedom at nodes. A value of 0 indicates no movement.

- DX = ... Nodal displacement in the X direction.
- DY = ... Nodal displacement in the Y direction.
- DZ = ... Nodal displacement in the Z direction.
- RX = ... Nodal rotation in the X direction.
- RY = ... Nodal rotation in the Y direction.
- RZ = ... Nodal rotation in the Z direction.

<CR> TO CONTINUE

PICK NODES & EXIT Small triangular symbols will be displayed at the picked nodes, aligned with the appropriate axes of the FEMSYS, to show that nodal constraints are attached.

Single triangles indicate displacement. Double triangles in a row indicate rotation. Three triangles in a row indicate displacement and rotation along the same axis. See Figure 4-2 on page 4-15.

Specify more nodal constraints or <EXIT>.

STRENGTH >> NODAL CONSTRAINTS >> DETACH

PICK NODES & EXIT All nodal constraints attached to the picked nodes will be removed.

STRENGTH >> NODAL CONSTRAINTS >> UPDATE

PICK NODE & EXIT — For each node picked, redefine the nodal constraint as described in the ATTACH option.

STRENGTH >> NODAL LOADS

Define the FORCE and MOMENT that will be attached to the nodes.

SELECT:	ATTACH	DETACH	UPDATE

STRENGTH >> NODAL LOADS >> ATTACH

SELECT:	DEFINE VECTOR	USE AXES

DEFINE VECTOR — The FORCE and MOMENT will be concentrated in the direction defined by a vector.

USE AXES — The FORCE and MOMENT will be defined in terms of their three orthogonal components, in the directions of the axes of the active coordinate system.

STRENGTH >> NODAL LOADS >> ATTACH >> DEFINE VECTOR

IND. 1ST POINT — Indicate two points to define a direction for the FORCE and MOMENT.

IND. 2ND POINT

- FORCE = 0.000 — Enter the nodal FORCE.
- MOMENT = 0.000 — Enter the nodal MOMENT.

<CR> TO CONTINUE

PICK NODES & EXIT — An arrow(s) will indicate the force and moment at each of the picked nodes, concentrated in the direction indicated by the previously defined vector.

One arrowhead —> indicates a force value is attached. Two arrowheads —>> indicate a MOMENT value is attached. Three arrowheads —>>> indicate that both are attached in the same direction. See Figure 4-2 on page 4-15.

Specify more nodal loads to be attached or <EXIT>.

STRENGTH >> NODAL LOADS >> ATTACH >> USE AXES

- FX = ... X component of the nodal FORCE.
- FY = ... Y component of the nodal FORCE.
- FZ = ... Z component of the nodal FORCE.
- MX = ... X component of the nodal MOMENT.
- MY = ... Y component of the nodal MOMENT.
- MZ = ... Z component of the nodal MOMENT.

<CR> TO CONTINUE

PICK NODES & EXIT Arrows will indicate the attached loads, distributed from each of the picked nodes in the directions indicated by the axes of the active coordinate system.

One arrowhead —> indicates a force value is attached. Two arrowheads —>> indicate a MOMENT value is attached. Three arrowheads —>>> indicate that both are attached in the same direction. See Figure 4-2 on page 4-15.

Specify more nodal loads to be attached or <EXIT>.

STRENGTH >> NODAL LOADS >> DETACH

PICK NODES & EXIT All nodal loads attached to the picked nodes will be removed.

STRENGTH >> NODAL LOADS >> UPDATE

PICK NODE & EXIT For each node picked, redefine the nodal load as described in the ATTACH option.

STRENGTH >> NODAL TEMP

Specify nodal temperatures that will be attached to the nodes.

SELECT:	ATTACH	DETACH	UPDATE

STRENGTH >> NODAL TEMP >> ATTACH

- TEMPERATURE = 0.000 Enter the temperature at the nodes to be picked.

<CR> TO CONTINUE

PICK NODES & EXIT A T will appear next to each node picked to indicate that it is attached to a STRENGTH nodal temperature (as opposed to a HEAT nodal temperature which is represented by a T and a triangle). See Figure 4-2 on page 4-15.

Specify more nodal temperatures to be attached or <EXIT>.

STRENGTH >> NODAL TEMP >> DETACH

PICK NODES & EXIT All nodal temperatures attached to the picked nodes will be removed.

STRENGTH >> NODAL TEMP >> UPDATE

PICK NODE & EXIT For each node picked, redefine the nodal temperature as described in the ATTACH option.

STRENGTH >> TRACTIONS

Define the traction (distributed load) to be attached (FORCE/LENGTH for running loads, FORCE/AREA for pressure).

SELECT:	ATTACH	DETACH	UPDATE	2D MESH PRESSURE

Note: • Select the 2D MESH PRESSURE option to attach all the elements of a mesh of 2D elements at the same time.

STRENGTH >> TRACTIONS >> ATTACH

■ TRACTION = 0.100 — Enter the traction (distributed load) to be attached. (FORCE/LENGTH for running loads, or FORCE/AREA for pressure.)

PICK ELEMENTS & EXIT — Pick 2D or 3D elements to be attached to the specified traction.

Note: • To attach tractions to elements of a mesh of 3D elements, use the procedure for a mesh of 2D elements. Then, sweep the tractions with the mesh.

THIS SIDE? YES NO — Accept a face or boundary edge of each element on which the traction is acting.

A >— symbol will point to the affected face or boundary edge of each element.

STRENGTH >> TRACTIONS >> DETACH

PICK ELEMENTS & EXIT — All tractions attached to the picked elements will be removed.

STRENGTH >> TRACTIONS >> UPDATE

PICK ELEMENT — For each element picked, redefine the traction by setting the modal parameters.

■ TRACTION = 0.100 — Enter the new traction.

■ * SIDE = . . . — The number of the side affected will be displayed here.

<CR> TO CONTINUE — <CR> after each change. <PICK> another element whose traction is to be updated. <EXIT> when finished updating tractions.

* Modals marked with an asterisk * are for information purposes only and may not be modified.

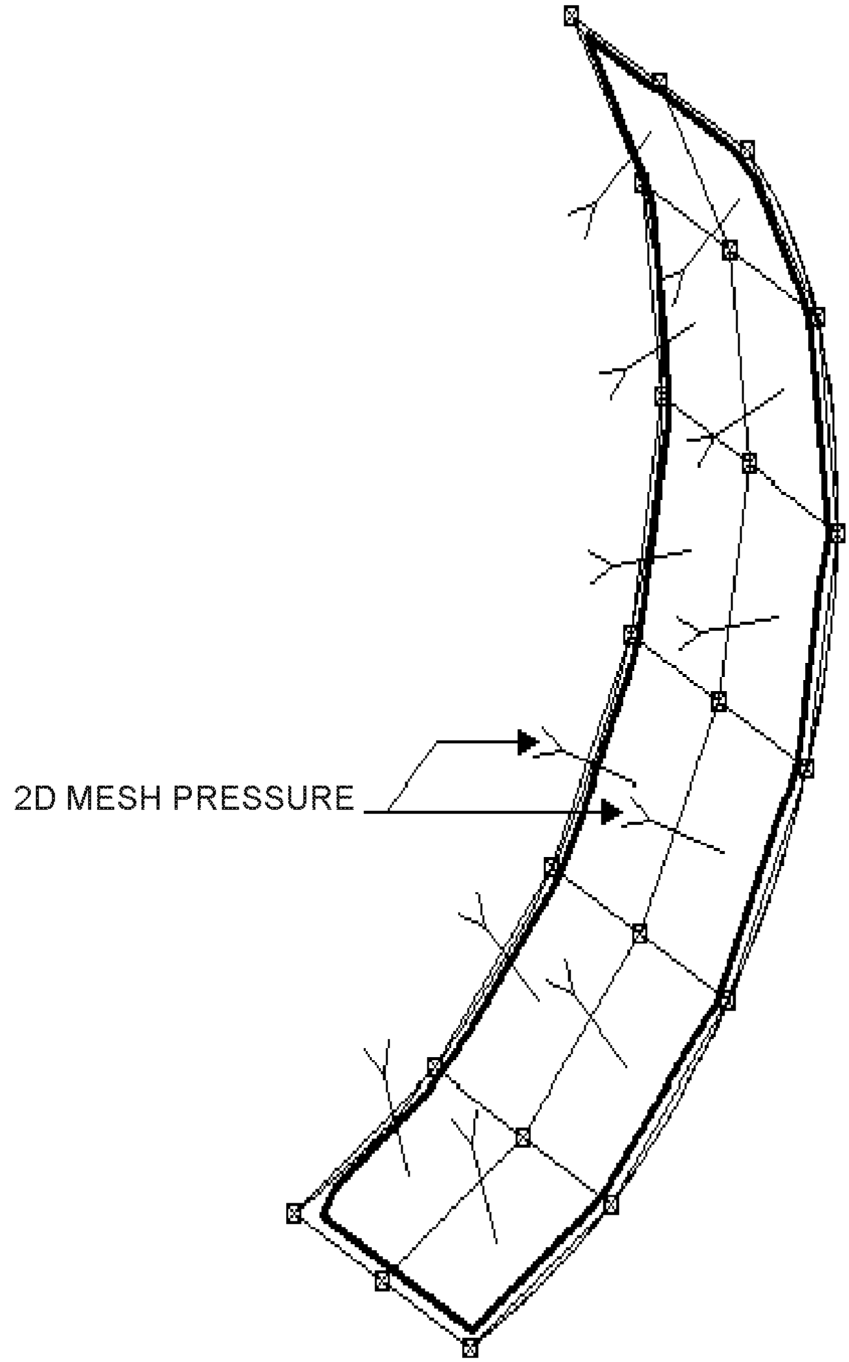


Figure 4-1: TRACTIONS

STRENGTH >> TRACTIONS >> 2D MESH PRESSURE

■ TRACTION = 0.1000 — Enter the traction (distributed load) to be attached.

PICK MESH — Pick a mesh of 2D elements to be attached to the specified traction.

THIS MESH? YES NO — Select YES to accept the mesh which is in attention.

THIS SIDE? YES NO — Select YES to accept the side of the mesh indicated by arrows as the one on which the traction is acting.

A fork symbol (>—) will point to the affected side of each element in the mesh.

STRENGTH >> BODY FORCE

Note: • Body force vectors are defined in the active coordinate system.

SELECT:	GRAVITATION	ANG.VELOCITY	ANG.ACCELERATION

■ GRAVITATION — The value of the gravitational or other linear acceleration acting upon the entire structure (length/time^2).

■ ANG. VELOCITY — The value of the angular velocity (centrifugal body force) acting upon the entire structure (angle in radians/time).

■ ANG. ACCELERATION — The value of the angular acceleration acting upon the entire body (angle in radians/time^2).

STRENGTH >> BODY FORCE >> GRAVITATION

■ GRAV = 0.000 — The value of the gravitational, or other linear acceleration acting on the entire structure (length/time^2).

■ AX = 0.000 — The X component of the gravitational or other linear acceleration vector.

■ AY = 0.000 — The Y component of the gravitational or other linear acceleration vector.

■ AZ = 1.000 — The Z component of the gravitational or other linear acceleration vector.

<CR> TO CONTINUE — Define the vector along which gravity acts on the entire body. <CR> when finished.

STRENGTH >> BODY FORCE >> ANG.VELOCITY

■ VELOC = 0.000	Enter the angular velocity (centrifugal body force) acting on the entire structure (angle in radians/time).
■ OMEGAX = 0.000	X component of the angular velocity vector.
■ OMEGAY = 0.000	Y component of the angular velocity vector.
■ OMEGAZ = 1.000	Z component of the angular velocity vector.
<CR> TO CONTINUE	Define the vector along which angular velocity acts on the entire body. <CR> when finished.

STRENGTH >> BODY FORCE >> ANG.ACCELERATION

■ ACCEL= 0.000	Enter the angular acceleration acting on the entire body (angle in radians/time^2).
■ ALPHAX = 0.00	X component of the angular acceleration vector.
■ ALPHAY= 0.00	Y component of the angular acceleration vector.
■ ALPHAZ = 0.00	Z component of the angular acceleration vector.
<CR> TO CONTINUE	Define the vector along which angular acceleration acts on the entire body. <CR> when finished.

F_LOAD >> HEAT

Define the heat loads to where the modal will be attached.

SELECT:	NODAL TEMP
	HEAT FLOW
	CONVECTIONS

HEAT >> NODAL TEMP

Attach, detach or update nodal temperature constraints which are required for heat analysis, to/from nodes.

SELECT:	ATTACH	DETACH	UPDATE

HEAT >> NODAL TEMP >> ATTACH

- NODAL TEMP = 0.000 — Enter the nodal temperature to be attached.

<CR> TO CONTINUE

PICK NODES & EXIT — A triangle and a T (Δ T) will appear next to each node picked to indicate that it is attached to a HEAT nodal temperature (as opposed to a STRENGTH nodal temperature which is represented by just a T). See Figure 4-2 on page 4-15.

Specify more nodal temperatures to be attached or <EXIT>.

HEAT >> NODAL TEMP >> DETACH

PICK NODES & EXIT — All nodal temperatures attached to the picked nodes will be removed.

HEAT >> NODAL TEMP >> UPDATE

PICK NODE & EXIT — For each node picked, redefine the nodal temperature as described in the ATTACH option.

HEAT >> HEAT FLOW

Specify HEAT FLOW definition values to be attached.

SELECT:	ATTACH	DETACH	UPDATE

HEAT >> HEAT FLOW >> ATTACH

- HEAT FLOW = 0.000 — Enter the HEAT FLOW to be attached.

<CR> TO CONTINUE

PICK NODES & EXIT — An arrow and a T (—>T) will appear next to each node picked to indicate that it is attached to a nodal HEAT FLOW.

Specify more HEAT FLOW definition values to be attached or <EXIT>.

HEAT >> HEAT FLOW >> DETACH

PICK NODES & EXIT — All heat flow values attached to the picked nodes will be removed.

HEAT >> HEAT FLOW >> UPDATE

PICK NODES & EXIT — For each node picked, redefine the heat flow as described in the ATTACH option.

HEAT >> CONVECTIONS

Specify convection definition values to be attached.

SELECT:	ATTACH	DETACH	UPDATE	2D MESH PRESSURE

Note: • Select the 2D MESH PRESSURE option to attach all the elements of a mesh of 2D elements at the same time.

HEAT >> CONVECTIONS >> ATTACH

- FILM COEFF = 0.000 — Enter the algebraic value of the pressure on the element.
- FLUID TEMP + 0.000 — Enter the bulk temperature of fluid adjacent to the convective film.
- AREA FACTOR = 0.000

PICK ELEMENTS & EXIT — Pick 2D elements to be attached to values which define convections on element faces.

THIS SIDE? YES NO — Accept the face or boundary edge of each element on which these pressures are acting.

A T and a fork symbol (T>—) together will point to the affected face or boundary edge of each element. See Figure 4-2 on page 4-15.

Specify more convection definition values to be attached or <EXIT>.

HEAT >> CONVECTIONS >> DETACH

PICK ELEMENTS & EXIT — Pick elements to be detached and <EXIT>. All convection definition values attached to the picked elements, will be removed.

HEAT >> CONVECTIONS >> UPDATE

PICK ELEMENTS — For each element picked, redefine the convection as described in the ATTACH option.

HEAT >> CONVECTIONS >> 2D MESH PRESSURE

- FILM COEFF = 0.000 — Enter the algebraic value of the pressure.
- FLUID TEMP = 0.000 — Enter the bulk temperature of fluid adjacent to the convective film.

PICK MESH	Pick a mesh of 2D elements to be attached to the defined pressures.
THIS MESH? YES NO	Select YES to accept the mesh which is in attention.
THIS SIDE? YES NO	Select YES to accept the side of the mesh indicated by an arrow as the one on which the pressures are acting.
	A T plus a fork symbol (T >-—) will point to the affected side of each element in the mesh.

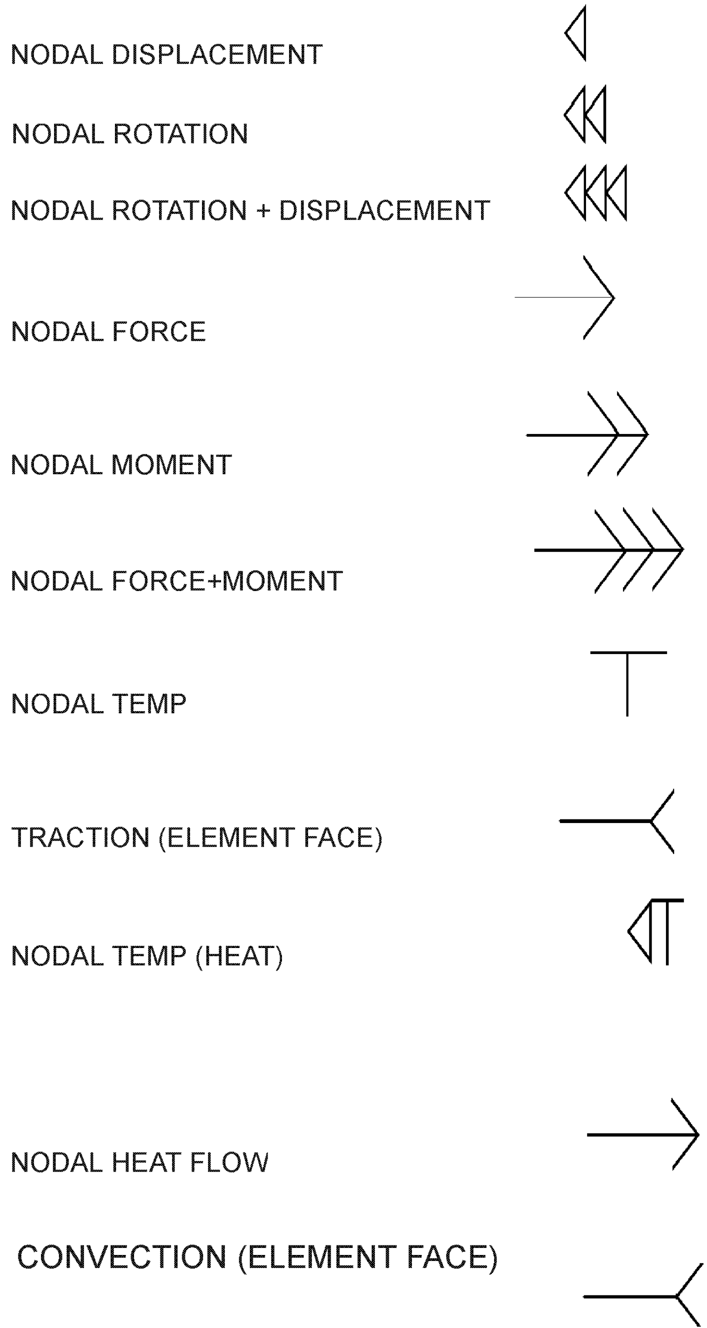


Figure 4-2: Load Symbols o

F_PROP

Define material and geometric properties and attach them to specific parts of the finite element model.

If you are working in FLOW ANALYSIS, see the section on FLOW ANALYSIS.

SELECT PROPERTY TYPE:	MATERIAL
	GEOMETRIC

If MATERIAL is selected, specify STRENGTH or HEAT.

SELECT MAT. PROP. TYPE	STRENGTH	HEAT

F_PROP >> MATERIAL (STRENGTH / HEAT) or F_PROP >> GEOMETRIC

SELECT OPTION:	TABLE
	ATTACH
	DETACH
	VERIFY

TABLE Work with a table of properties.

ATTACH Assign a property to specific elements.

DETACH Disassociate the properties from specific elements.

VERIFY Check the status of a property.

MATERIAL / GEOMETRIC >> TABLE

Create material properties for strength analysis.

SELECT:	CRE-UPD	DELETE	RENAME	REPORT

CRE-UPD Create or update an entry in the table of properties.

DELETE Delete an entry from the table of properties.

RENAME Rename an entry in the table of properties.

REPORT Produce the contents of a table of properties on the screen or in a file for all properties of the specified type.

MATERIAL / GEOMETRIC >> TABLE >> CRE-UPD

SELECT:	INTERNAL	CATALOG

MATERIAL(STRENGTH) >> TABLE >> CRE-UPD >> INTERNAL

SELECT MATERIAL NAME:	<material 1>	<material 2>	. . .	<material n>	<—->

Select the blank space to create new material properties for strength analysis or select a name from the list to update the properties of an existing material.

Note:
- If material is isotropic do *not* enter values for EYY, EZZ, NU-YZ, NU-XZ, ALPHAY, ALPHAZ, GXY, GYZ and GZX.

- EXX = 0.000 — Enter the Young's modulus of elasticity values for orthotropic material.
- EYY = ...
- EZZ = ...
- NU-XY = ... — Poisson ratio. Relates X strain to Y stress.
- NU-YZ = ... — Poisson ratio. Relates Y strain to Z stress.
- NU-XZ = ... — Poisson ratio. Relates X strain to Z stress.
- GXY = ... — The XY direction of the Shear (or Bulk) modulus. (Force/ Area).
- GYZ = ... — YZ direction, Shear (or Bulk) modulus.
- GZX = ... — ZX direction, Shear (or Bulk) modulus.
- ALPHAX = ... — X direction of the thermal expansion coefficient.
- ALPHAY = ... — Y direction of the thermal expansion coefficient.
- ALPHAZ = ... — Z direction of the thermal expansion coefficient.
- TENSION, COMPRES SHEAR — Stress limits for tension, compression, and shear are optionally supplied. They are used to compute margins of safety in certain elements, and have no effect on the computational procedures.

■ DENSITY = ... The density of the material.

<CR> TO CONTINUE Press <CR> to accept the values and to process. Select another material or <EXIT>.

MATERIAL(HEAT) >> TABLE >> CRE-UPD >> INTERNAL

SELECT MATERIAL NAME:	<material 1>	<material 2>	. . .	<material n>	<—->

Select the blank space to create new material properties for heat analysis or select a name from the list to update the properties of an existing material.

■ KXX = 0.000 Coefficient of thermal conductivity in the X direction. (HEAT * LENGTH / TIME * AREA * DEGREE)

■ KYY = ... Coefficient of thermal conductivity in the Y direction.

■ KZZ = ... Coefficient of thermal conductivity in the Z direction.

■ DENSITY = ... The density of the material.

■ CH = ... Specific heat. (HEAT / MASS * DEGREE).

<CR> TO CONTINUE Press <CR> to accept the modal parameters and to process.

Select another material or <EXIT>.

GEOMETRIC >> TABLE >> CRE-UPD >> INTERNAL

SELECT ELEMENT TYPE:	SPAR	BEAM	PLATE	MEMBRANE	PL STRESS
	MASS	SPRING			

GEOMETRIC >> TABLE >> CRE-UPD >> INTERNAL >> SPAR

SELECT GEO. PROPERTY:	<spar 1>	<spar 2>	. . .	<spar n>	<—->

Select the blank space to create new geometric properties for analysis of spars or select a name from the list to update an existing one.

■ AREA = 0.000 Enter the cross-section area which defines the spar.

<CR> TO CONTINUE Press <CR> to accept the modal parameters and to process.

GEOMETRIC >> TABLE >> CRE-UPD >> INTERNAL >> BEAM

SELECT GEO. PROPERTY:	<beam 1>	<beam 2>	. . .	<beam n>	<—->

Select the blank space to create new geometric properties for analysis of beams or select a name from the list to update existing ones.

Note: • X_1, Y_1 and Z_1 = local (beam) coordinate system.

- AREA = 0.000 — Enter the cross-section area which defines the beam.
- IZ = ... — Enter the cross-sectional moment of inertia about the Z_1 axis.
- IY = ... — Enter the cross-sectional moment of inertia about the Y_1 axis.
- IX = ... — Enter the cross-sectional moment of inertia about the X_1 axis.
- THICKZ = ... — Enter the cross-sectional thickness of the beam along the Z_1 axis.
- THICKY = ... — Enter the cross-sectional thickness of the beam along the Y_1 axis.

Notes:

- If a third node will not be defined, the thickness will be measured in the FEMSYS.
- If a third node is defined, the beam lies on the X1 axis of the beam coordinate system.

<CR> TO CONTINUE — Press <CR> to accept the modal parameters and to process.

A 3rd node may be defined for a beam element to define the X_1Y_1 plane of the beam coordinate system on which it lies.

If no 3rd node is defined for the beam, the following appears:

DEFINE 3RD NODE?
YES NO

YES IND. POINT

As soon as the point is indicated the property will be created. The coordinates of the third node will be included in the table of geometric properties.

NO The properties will be created.

or: If a third node already exists for the beam, the following appears:

SELECT:	REDEFINE 3RD NODE	DELETE 3RD NODE

Select the appropriate option to delete the node or redefine it as described above.

GEOMETRIC >> TABLE >> CRE-UPD >> INTERNAL >> PLATE or MEMBRANE

SELECT GEO. PROPERTY:	<plate 1>	<plate 2>	. . .	<plate n>	<—->

Select the blank space to create new geometric properties for analysis of conic elements or select a name from the list to update existing ones.

GEOMETRIC >> TABLE >> CRE-UPD >> INTERNAL >> PLATE or MEMBRANE

SELECT GEO. PROPERTY:	<plate/mem 1>	<plate/mem 2>	. . .	<plate/mem n>	<—->

Select the blank space to create new geometric properties for analysis of plates or membranes or select a name from the list to update existing ones.

Note:

- To define constant thickness throughout the plate or membrane, do not enter values for THICK 2, THICK 3 or THICK 4.

- ■ THICK1 = 0.000 — Enter the thickness at the first node.
- ■ THICK2 = ... — Enter the thickness at the second node.
- ■ THICK3 = ... — Enter the thickness at the third node.
- ■ THICK4 = ... — Enter the thickness at the fourth node.

<CR> TO CONTINUE — Press <CR> to accept the modal parameters and to process.

GEOMETRIC >> TABLE >> CRE-UPD >> INTERNAL >> PL STRESS

SELECT GEO. PROPERTY:	<pl stress 1>	<pl stress 2>	. . .	<pl stress n>	<—->

Select the blank space to create a new geometric property for analysis of planar stress or select a name from the list to update an existing one.

- ■ THICK = 0.000 — Enter the thickness.

<CR> TO CONTINUE — Press <CR> to accept the modal parameters and to process.

GEOMETRIC >> TABLE >> CRE-UPD >> INTERNAL >> MASS

SELECT GEO. PROPERTY:	<mass 1>	<mass 2>	. . .	<mass n>	<—->

Select the blank space to create a new geometric property for mass analysis or select a name from the list to update an existing one.

- ■ MASS = 0.000 — Enter the mass.

<CR> TO CONTINUE — Press <CR> to accept the modal parameters and to process.

GEOMETRIC >> TABLE >> CRE-UPD >> INTERNAL >> SPRING

SELECT GEO. PROPERTY:	<spring 1>	<spring 2>	. . .	<spring n>	<—->

Select the blank space to create a new geometric property for spring analysis or select a name from the list to update an existing one.

- K = 0.000 — Enter the spring constant.
- AXIS-X
AXIS-Y
AXIS-Z
ROT-X
ROT-Y
ROT-Z — Enter the translations in the X, Y and Z directions, or the rotations about the X, Y and Z axes.

<CR> TO CONTINUE — Press <CR> to accept the modal parameters and to process.

MATERIAL / GEOMETRIC >> TABLE >> CRE-UPD >> CATALOG

Import entries from a property table of properties which is in another part file.

ENTER PART FILE NAME — Enter the name of the part file containing the entries to be imported. <CR> to display all the files in this directory. Directory names are highlighted.

SELECT MATERIAL & EXIT	<menu of material or geometric properties in source file>

or:

SELECT GEO. PROP & EXIT	<menu of material or geometric properties in source file>

Select a material or geometric property to be imported.

ENTER LOCAL NAME — Type the name the imported material or geometric property will have in the current file.

<EXIT> when finished.

MATERIAL / GEOMETRIC >> TABLE >> DELETE

Delete a property.

SELECT MATERIAL & EXIT	<menu of existing materials or geometric properties>

The deletion will be performed as soon as the material or geometric property to be deleted is selected.

or:

SELECT GEO. PROPERTY	<list of existing geometric properties>

If elements are attached to the material or geometric property to be deleted, confirm the deletion.

DELETE? YES NO — Continue to select materials or geometric properties to be deleted. <EXIT> when finished.

MATERIAL / GEOMETRIC >> TABLE >> RENAME

Rename a material or geometric property.

SELECT MATERIAL & EXIT	<menu of existing materials or geometric properties>

or:

SELECT GEO. PROPERTY	<list of existing geometric properties>

ENTER NEW NAME Type a new name for the selected material or geometric property and press <CR>, or press <CR> without typing a name to use the same name. Continue to select properties to be renamed. <EXIT> when finished.

MATERIAL / GEOMETRIC >> TABLE >> REPORT

Produce the contents of property tables of properties on the screen or in a file.

SELECT:	SCREEN	FILE

SCREEN Display the tables of values for the selected property type in the alphanumeric window.

FILE Create a file containing the table of values for the selected property type.

MATERIAL / GEOMETRIC >> TABLE >> REPORT >> SCREEN

The table will be displayed when this option is selected.

MATERIAL / GEOMETRIC >> TABLE >> REPORT >> FILE

ENTER FILE NAME Enter a valid file name. The file will be created, if necessary, and the table will be written to that file.

Example 1

TABLE OF MATERIALS

NAME	EXX	EYY	EZZ	NU-XY	NU-YZ	NU-XZ	GXY
	GYZ	GZX	ALPHAX	ALPHAY	ALPHAZ	TENSION *	COMPRES *
	SHEAR *	DENSITY *					
MAT1	30000000.0						
MAT2	30000000.0						
MAT4	21000.0			0.30			0.00

* Double precision modals where the selected value may be between 1.E±19.

Example 2

TABLE OF GEOMETRIC PROPERTIES

SPARS

NAME	AREA
GEO_1	0.15
GEO_2	0.25

BEAMS

NAME	AREA	IZ	IY	IX	THICKZ	THICKY	PTX	PTY
DD	12.00	13.00					9.93	
FF	11.00	22.00					10.86	

PLATES

NAME	THICK1	THICK2	THICK3	THICK4
GEO	1.00			

MEMBRANES

NAME	THICK1	THICK2	THICK3	THICK4

PL STRESS

NAME	THICK

MATERIAL / GEOMETRIC >> ATTACH

Assign a selected material or geometric property to specific elements.

MATERIAL

SELECT ELEMENT TYPE:	SPAR	BEAM	PLATE	MEMBRANE	PL STRESS
	PL STRAIN	AXISYM	BRICK	PRISM	TETRA

GEOMETRIC

SELECT ELEMENT TYPE:	SPAR	BEAM	PLATE	MEMBRANE	PL STRESS
	MASS	SPRING			

If no element of the selected type exists, the selection will be ignored. The elements having the selected type will be displayed in bold lines.

PICK ELEMENTS & EXIT Press <SUBMENU> to display picking options.

SINGLE / BOX	POLYGON	EXCLUDE	ALL

The picked elements will be displayed in attention. <REJECT> to unpick the last element. <EXIT> when finished.

SELECT ELEMENT TYPE Continue to select element types to be attached to selected properties or <EXIT>.

SELECT GEO. PROPERTY

SELECT GEO. PROPERTY	<geo. prop 1>	<geo. prop 2>	. . .	<geo. prop n>	<—->

or:

SELECT MATERIAL NAME:	<material 1>	<material 2>	. . .	<material n>	<—->

Select the material or geometric property to be attached.

MATERIAL / GEOMETRIC >> DETACH

Disassociate picked elements from the properties attached to them.

SELECT:	PICKED	BY TYPE

MATERIAL / GEOMETRIC >> DETACH >> PICKED

Detach all elements from the properties attached to them.

PICK ELEMENTS & EXIT All elements picked will be displayed in attention. <EXIT> to detach them from the properties to which they are attached. <EXIT> when finished.

MATERIAL / GEOMETRIC >> DETACH >> BY TYPE

SELECT ELEMENT TYPE:	SPAR	BEAM	PLATE	MEMBRANE	PL STRESS
	PL STRAIN	AXISYM	BRICK	PRISM	TETRA

If no element of the selected type exists, the selection will be ignored. Elements having the selected type will be displayed in bold.

PICK ELEMENTS & EXIT Press <SUBMENU> to display picking options.

The picked elements will be displayed in attention. <REJECT> to unpick the last element. <EXIT> when finished.

MATERIAL / GEOMETRIC >> VERIFY

Check the attached/detached status of a property or element.

SELECT:	PROPERTY	ELEMENT

MATERIAL / GEOMETRIC >> VERIFY >> PROPERTY

SELECT ELEMENT TYPE:	SPAR	BEAM	PLATE	MEMBRANE	PL STRESS
	PL STRAIN	AXISYM	BRICK	PRISM	TETRA

If GEOMETRIC properties are being verified, select an element type.

SELECT MATERIAL NAME:	<material 1>	<material 2>	. . .	<material n>	<—->

or:

SELECT GEO. PROPERTY	<geo. prop 1>	<geo. prop 2>	. . .	<geo. prop n>	<—->

All elements attached to the material or geometric property will be displayed in attention.

MATERIAL / GEOMETRIC >> VERIFY >> ELEMENT

PICK ELEMENT

<PICK> the element to be verified. The element will be displayed in attention. A message will be displayed in the prompt area for each element picked.

Example:

PLATE : NO MATERIAL PROPERTY ATTACHED
PLATE : MATERIAL PROPERTY IS <name>

<CR> TO CONTINUE

Press <CR> to verify another element. <EXIT> when finished.

FLOW ANALYSIS

In FLOW ANALYSIS, we are only dealing with properties.

F_PROP

SELECT OPTION:	TABLE
	ATTACH
	DETACH
	VERIFY

TABLE Work with a table of properties.

ATTACH Assign a property to specific elements.

DETACH Disassociate the properties from specific elements.

VERIFY Check the status of a property.

TABLE

SELECT:	CRE-UPD	DELETE	RENAME	REPORT

CRE-UPD Create or update an entry in the table of properties.

DELETE Delete an entry from the table of properties.

RENAME Rename an entry in the table of properties.

REPORT Produce the contents of a table of properties on the screen or in a file for all properties of the specified type.

TABLE >> CRE-UPD

SELECT:	INTERNAL	CATALOG

TABLE >> CRE-UPD >> INTERNAL

SELECT ELEMENT TYPE:	CONIC	STRIP	PLATE		

If flow analysis is specified, the second menu is displayed.

Select the type of element for which properties will be defined.

TABLE >> CRE-UPD >> INTERNAL >> CONIC

SELECT PROPERTY:	<conic 1>	<conic 2>	. . .	<conic n>	<—->

Select the blank space to create new properties for analysis of conic elements or select a name from the list to update existing ones.

- D1 = 0.000 — Diameter at 1st end of the element.
- D2 = 0.0001 — Diameter at 2nd end of the element.
- TMOULD = 0.001 — Temperature of the mold.

Specify the element usage:

NONE
COLD RUNNER
HOT RUNNER
MELT CHANNEL
GAS CHANNEL
COOLANT CHANNEL

TABLE >> CRE-UPD >> INTERNAL >> STRIP

SELECT PROPERTY:	<strip 1>	<strip 2>	. . .	<strip n>	<—->

Select the blank space to create new properties for analysis of strip elements or select a name from the list to update existing ones.

- THICKZ = 0.000 — Thickness of the strip along the Z axis.
- THICKY = 0.001 — Thickness of the strip along the Y axis.
- TMOULD = 0.001 — Temperature of the mold.

Specify the element usage:

NONE
COLD RUNNER
HOT RUNNER
MELT CHANNEL
GAS CHANNEL
COOLANT CHANNEL

TABLE >> CRE-UPD >> INTERNAL >> PLATE

SELECT PROPERTY:	plate 1>	<plate 2>	. . .	<plate n>	<—->

Select the blank space to create new properties for analysis of plates, or select a name from the list to update existing ones.

- THICK1 = 0.000 — Enter the thickness of the plate.
- TMOULD = 0.001 — Temperature of the mold.

Specify the element usage:

PART FACE /
MOLD FACE

TABLE >> CRE-UPD >> CATALOG

Import entries from a property table of properties which is in another part file.

ENTER PART FILE NAME Enter the name of the part file containing the entries to be imported. <CR> to display all the files in this directory. Directory names are highlighted.

SELECT PROP & EXIT	<menu of properties in source file>

Select a property to be imported.

ENTER LOCAL NAME Type the name the imported property will have in the current file.

<EXIT> when finished.

TABLE >> DELETE

Delete a property.

SELECT ELEMENT TYPE:	CONIC	STRIP	PLATE		

SELECT PROPERTY	<list of existing properties>

If elements are attached to the property to be deleted, confirm the deletion.

Continue to select properties to be deleted. <EXIT> when finished.

TABLE >> RENAME

Rename a property.

SELECT ELEMENT TYPE:	CONIC	STRIP	PLATE		

SELECT PROPERTY	<list of existing properties>

ENTER NEW NAME Type a new name for the selected property and press <CR>, or press <CR> without typing a name to use the same name. Continue to select properties to be renamed. <EXIT> when finished.

TABLE >> REPORT

Produce the contents of property tables of properties on the screen or in a file.

SELECT:	SCREEN	FILE

SCREEN Display the tables of values for the selected property type in the alphanumeric window.

FILE Create a file containing the table of values for the selected property type.

TABLE >> REPORT >> SCREEN

The table will be displayed when this option is selected.

TABLE >> REPORT >> FILE

ENTER FILE NAME Enter a valid file name. The file will be created, if necessary, and the table will be written to that file.

ATTACH

Assign a selected property to specific elements.

SELECT ELEMENT TYPE:	NODE	CONIC	STRIP	PLATE

If NODE is selected, assign a property (polymer entrance, gas entrance, or coolant entrance) to a specific node.

If no element of the selected type exists, the selection will be ignored. Elements having the selected type will be displayed in bold.

SELECT ELEMENT TYPE Select element types to be attached to selected properties or <EXIT>.

PICK ELEMENTS & EXIT Press <SUBMENU> to display picking options.

SINGLE / BOX	POLYGON	EXCLUDE	ALL

The picked elements will be displayed in attention. <REJECT> to unpick the last element. <EXIT> when finished.

SELECT PROPERTY

SELECT PROPERTY	<list of existing properties>

DETACH

Disassociate picked elements from the properties attached to them.

SELECT:	PICKED	BY TYPE

DETACH >> PICKED

Detach all elements from the properties attached to them.

PICK ELEMENTS & EXIT All elements picked will be displayed in attention. <EXIT> to detach them from the properties to which they are attached. <EXIT> when finished.

DETACH >> BY TYPE

SELECT ELEMENT TYPE:	NODE	CONIC	STRIP	PLATE

If no element of the selected type exists, the selection will be ignored. Elements having the selected type will be displayed in bold.

PICK ELEMENTS & EXIT Press <SUBMENU> to display picking options.

The picked elements will be displayed in attention. <REJECT> to unpick the last element. <EXIT> when finished.

VERIFY

Check the attached/detached status of a property or element.

SELECT	ELEMENT	PROPERTY

VERIFY >> ELEMENT

PICK ELEMENT <PICK> the element to be verified. The element will be displayed in attention. A message will be displayed in the prompt area for each element picked.

Example:

PLATE : NO MATERIAL PROPERTY ATTACHED
PLATE : MATERIAL PROPERTY IS <name>

<CR> TO CONTINUE Press <CR> to verify another element. <EXIT> when finished.

VERIFY >> PROPERTY

SELECT ELEMENT TYPE:	CONIC	STRIP	PLATE

SELECT PROPERTY	<prop 1>	<prop 2>	. . .	<prop n>	<—->

All elements attached to the property will be displayed in attention. ❑

VERIFY (FEM)

Verify data relating to finite elements.

The following data may be verified using this function:

- the coordinates of nodes, if nodes are selected
- the ID numbers of all elements selected
- a list of the nodes composing each element
- the names of the materials and/or geometric properties associated with each element
- Analysis type
- Total number of nodes and elements

Notes:

- The “on/off” status of each element is retained only for the current session of the function. Exiting from the function will return all elements to the default status of “on.”
- This option is accessed via the VERIFY function in the General Systems Function Bank.

SELECT	**BY PICK**
	BY NUMBER
	LIST
	ALL

BY PICK Display information on picked node or element.

BY NUMBER Node or element is highlighted.

LIST A list of all nodes and selected elements with their properties.

ALL Analysis type and total number of nodes and elements.

VERIFY >> BY PICK

Display information on picked node or element.

PICK NODE OR ELEMENT	NODE
	ELEMENT

- NODE Number and coordinates of the selected node.
- ELEMENT Shape, type and a list of nodes composing the selected element.

VERIFY >> BY NUMBER

Node or element is highlighted.

ENTER NUMBER:	NODE	NODE NO. =
	ELEMENT	ELEMENT NO. =

Enter the number of the node/element that you wish to highlight.

VERIFY >> LIST

A list of all nodes and selected elements with their properties.

SELECT ELEMENTS:	NODES	SPARS	BEAMS	PLATES	MEMBRN	PLSTRS
	PLSTRN	AXISYM	BRICKS	PRISMS	TETRAS	MASS
	SPRING	RIGID				

or:

SELECT ELEMENTS:	NODES	CONICS	STRIPS	PLATES

If flow analysis was specified, the second menu is displayed.

Select an element type to reverse its current status. All the elements you want to verify should be "on." Elements which are "off" will have the same background as the graphics portion of the screen. <EXIT> when finished.

If flow analysis is specified, skip to SELECT PROPS?

SELECT MATERIALS? YES NO

YES SELECT MATERIALS <list of materials> <UNDEFI>

Select materials from the list to verify their status for all of the selected elements. Select the last one, <UNDEFI>, to verify selected elements which have no material properties attached to them. <EXIT> when finished.

NO Do not verify any material properties.

SELECT PROPS? YES NO

YES SEL. GEO. PROPERTIES <list of sets of geometric properties> <UNDEFI>

Select sets of geometric properties from the list to verify their status for all of the selected elements. Select the last one, <UNDEFI>, to verify selected elements which have no geometric properties attached to them. <EXIT> when finished.

NO Do not verify any geometric properties.

SELECT	MARK BY ATTENTION	WRITE ELEMENTS

Select an output mode for the data being verified. <EXIT> when finished.

ALL TYPES >> MARK BY ATTENTION

- * NO.OF ELEMENTS= . . . This number of elements meet the specified conditions. They will be displayed in attention.

Note: • Modals marked with an asterisk may not be modified.

<CR> TO CONTINUE Press <CR> to remove the elements from attention and to return to:
SELECT: MARK BY ATTENTION WRITE ELEMENTS.

ALL TYPES >> WRITE ELEMENTS

Note: • The output list of the selected elements will include the names of the materials and geometric properties attached to each (or undefined).

WRITE ON:	SCREEN	FILE

ALL TYPES >> WRITE ELEMENTS >> SCREEN

The system will beep and the specified elements will be listed on the alphanumeric screen/window.

ALL TYPES >> WRITE ELEMENTS >> FILE

ENTER FILE NAME Enter a valid file name. The list will be written to the file.

Example 1

————- QUERY LIST :——————————- (9) ELEMENTS.

1.	PLATE:	ID = 257	MAT = MAT4	GEO = GEO3	NODS =	1	11	12	2
2.	PLATE:	ID = 258	MAT = MAT4	GEO = GEO3	NODS =	2	12	13	3
3.	PLATE:	ID = 259	MAT = MAT4	GEO = GEO3	NODS =	3	13	14	4
4.	PLATE:	ID = 260	MAT = MAT4	GEO = GEO3	NODS =	4	14	15	5
5.	PLATE:	ID = 261	MAT = MAT4	GEO = GEO3	NODS =	5	15	16	6
6.	PLATE:	ID = 262	MAT = MAT4	GEO = GEO3	NODS =	6	16	17	7
7.	PLATE:	ID = 263	MAT = MAT4	GEO = GEO3	NODS =	7	17	18	8
8.	PLATE:	ID = 264	MAT = MAT4	GEO = GEO3	NODS =	8	18	19	9
9.	PLATE:	ID = 265	MAT = MAT4	GEO = GEO3	NODS =	9	19	20	10

Example 2

—————— QUERY LIST : ————————-(29) ELEMENTS.

1.	NODE	1	X = 10.000	Y = 0.000	Z = 0.000				
2.	NODE	2	X = 20.000	Y = 0.000	Z = 0.000				
3.	NODE	3	X = 30.000	Y = 0.000	Z = 0.000				
4.	NODE	4	X = 40.000	Y = 0.000	Z = 0.000				
5.	NODE	5	X = 50.000	Y = 0.000	Z = 0.000				
6.	NODE	6	X = 60.000	Y = 0.000	Z = 0.000				
7.	NODE	7	X = 70.000	Y = 0.000	Z = 0.000				
8.	NODE	8	X = 80.000	Y = 0.000	Z = 0.000				
9.	NODE	9	X = 90.000	Y = 0.000	Z = 0.000				
10.	NODE	10	X = 100.000	Y = 0.000	Z = 0.000				
11.	NODE	11	X = 10.000	Y = 10.000	Z = 0.000				
12.	NODE	12	X = 20.000	Y = 10.000	Z = 0.000				
13.	NODE	13	X = 30.000	Y = 10.000	Z = 0.000				
14.	NODE	14	X = 40.000	Y = 10.000	Z = 0.000				
15.	NODE	15	X = 50.000	Y = 10.000	Z = 0.000				
16.	NODE	16	X = 60.000	Y = 10.000	Z = 0.000				
17.	NODE	17	X = 70.000	Y = 10.000	Z = 0.000				
18.	NODE	18	X = 80.000	Y = 10.000	Z = 0.000				
19.	NODE	19	X = 90.000	Y = 10.000	Z = 0.000				
20.	NODE	20	X = 100.000	Y = 10.000	Z = 0.000				
21.	PLATE:	ID = 257	MAT = ———-	GEO = ———-	NODS =	1	11	12	2
22.	PLATE:	ID = 258	MAT = ———-	GEO = ———-	NODS =	2	12	13	3
23.	PLATE:	ID = 259	MAT = ———-	GEO = ———-	NODS =	3	13	14	4
24.	PLATE:	ID = 260	MAT = ———-	GEO = ———-	NODS =	4	14	15	5
25.	PLATE:	ID = 261	MAT = ———-	GEO = ———-	NODS =	5	15	16	6
26.	PLATE:	ID = 262	MAT = ———-	GEO = ———-	NODS =	6	16	17	7
27.	PLATE:	ID = 263	MAT = ———-	GEO = ———-	NODS =	7	17	18	8
28.	PLATE:	ID = 264	MAT = ———-	GEO = ———-	NODS =	8	18	19	9
29.	PLATE:	ID = 265	MAT = ———-	GEO = ———-	NODS =	9	19	20	10

VERIFY >> ALL

Analysis type and total number of nodes and elements.

SELECT	ALL ELEMENTS
	DISPLAYED ELEMENTS

- ALL ELEMENTS Information of all elements in the FEMSYS.
- DISPLAYED ELEMENTS Information of displayed elements. ❑

Chapter 5
The FEM Post-processor Functions

F_POST is a sub-application of the FEM application. It is used to view the results of finite element analysis. The F_POST sub-application is accessed from the second overlay of the FEM application menu. Selection of F_POST makes the following FEM post-processor functions available, in addition to F_DISP (See Chapter 3, The FEM Modeling Functions.).

jon

RESULT	Read the results of the analysis and create an internal data file.
F_PLOT	Display the selected analysis results on the graphics screen.
F_WRITE	List selected analysis results on the alphanumeric screen or in a file.

The special F_POST functions will be described in detail in this chapter in the order they are listed above.

F_POST functions are run within the FEMSYS in which the FEM model analyzed was prepared.

Each time the analysis is performed, RESULT must be run to read the analysis results and to create an internal data file called **out12.dat**.

Then, F_WRITE or F_PLOT may be used to describe which results will be viewed and in what form, i.e., written alphanumerically on the screen or in a file, or displayed graphically on the screen.

If results are displayed graphically, displacements of the model and colored contour lines may be produced. Each graphic display is a picture entity which can be saved with the part file and viewed repeatedly. However, it reflects the results at the moment it was created and is not updated when another analysis is performed.

The FEM and geometric models may not be modified when post-processing is being done. However, the F_DISP function may be used to control the display parameters such as element size and color, and to display only selected elements.

The FILE and EXIT General System functions may not be accessed during post-processing. Select F_PREP to return to the pre-processor to access these functions.

After post-processing is finished, F_PREP may be selected to leave the F_POST sub-application and return to FEM pre-processing, i.e. the finite element modeling functions, or leave the FEM application.

RESULT

Read the results of the analysis and create an internal data file.

Notes:

- If pictures of analysis results which were run earlier already exist, they will not be updated. Either delete them, change their names or work in a different directory.
- Each time this function is run it writes in the file named **out12.dat**. To save the contents of a data file produced by an earlier run of this function, rename it before running RESULT.

SELECT ANALYSIS:	ANSYS
	NASTRAN
	INJECT_3
	OTHERS

OTHERS is not yet implemented.

ENTER FILE NAME Type the name of the file containing the results of the analysis. Ex: **file12.dat** for ANSYS analysis.

EXECUTING The internal data file, named **out12.dat**, will be created. ❑

F_PLOT

Display the selected analysis results graphically, on the screen and create a picture which may be recalled.

Notes:

- One picture may be saved for each of the analysis selection criteria. The system assigned name for each picture is provided in brackets next to the parameter which it represents. This name appears in the DELETE-PIC option. Access this option to check if a specific picture already exists.
- If a picture with the same parameters (i.e., same displacement and scale; same stress or load component, number of contours and display mode) was defined for earlier analysis results, it will be displayed when those parameters are specified again, using those analysis results. Delete it before specifying selection criteria to create a picture based on the new analysis results.

  It takes a long time to produce a picture for the first time. Pictures defined previously, are displayed much faster.
- Pictures are created based on the contents of the file produced by the last run of the RESULT function (**out12.dat**) and without regard to the current part file or display.
- To display pictures which reflect the latest analysis results, delete obsolete pictures with DELETE-PIC before defining new ones and rerun RESULT. FEM analysis pictures may not be deleted with the DELETE function.
- Use F_DISP to control the display of the undistorted finite element model.
- The picture will be displayed in all windows, based on the model in the active window when the function was executed. If the model is rotated or changed in any way, the picture will not be updated automatically.

SELECT:	DISPLACEMENTS
	CONTOURS
	DELETE-PIC

DISPLACEMENTS Create a picture of the displaced model.

CONTOURS Draw iso-parametric lines based on the selected parameters.

DELETE-PIC Delete pictures.

F_PLOT >> DISPLACEMENTS

Create a picture of the displaced model.

- WITH ELEMENTS — The displaced model will be displayed, subdivided into elements.

  WITHOUT ELEMENTS — Only the outer boundaries of the displaced model will be displayed. Not implemented yet.

- SCALE = 1.000 — The displacements will be magnified by this scale factor to make them easier to view.

SELECT DISPLAY MODE — The picture of the displaced model will be displayed as specified.

F_PLOT >> CONTOURS

Draw iso-parametric lines based on the selected parameters.

SELECT:	ALL
	BY ELEMENT TYPES
	BY MATERIAL PROPERT
	BY GEOMETRICAL PROP
	BY RESULTS VALUES

Select the criteria to be included in the display.

SELECT:	STRESSES	PRINCIPAL STRESSES	INTERNAL LOADS

Note:

- If the REGULAR DISPLAY modal is selected, the suffix of the name of the picture will be **reg**. If DISPLACED DISPLAY is selected, the suffix of the name of the picture will be **dic**.

CONTOURS >> STRESSES

SELECT STRESSES:	SXX	SYY	SZZ	SXY	SXZ	SYZ

Note: • When reading Inject3 results, the following table is displayed:

SELECT:	SHEAR_RATE	SHEAR_STRESS	FRONT_TEMPERATURE
	TIME_TO_SOLIDIFICAT		

The selected stress component will be constant at every point on each displayed contour.

A picture may be displayed for each stress component by selecting it and setting the following modal parameters.

<CR> TO CONTINUE	NO. OF CONTOURS = 10	REGULAR DISPLAY	
		DISPLACED DISPLAY	SCALE = 100.000

- NO. OF CONTOURS = 10 Enter the number of contours to be used to show the variations in the selected parameter. There may be between 3 and 15 contours.

Note: • On color displays, the number of display colors will equal NO.OF CONTOURS + 1.

- REGULAR DISPLAY Display the contours superimposed on the finite element model.

  DISPLACED DISPLAY Display the contours superimposed on the displaced model.

- SCALE = 100.000 If DISPLACED DISPLAY is selected, enter a scale value by which displacements will be multiplied to make them easier to view.

<CR> TO CONTINUE The defined picture will be created, if necessary, and displayed.

Note: • Creation of a new picture takes considerably more time than displaying an existing one.

CONTOURS >> PRINCIPAL STRESSES

SEL. PRINCIP STRESS:	SIG1	SIG2	SIG3	SIGE	SIGT

The selected principal stress will be constant at every point on each displayed contour.

Display pictures by setting the modals as for STRESSES.

Note:

- SIG1, SIG2, and SIG3 are principal stresses where SIG1 > SIG2 > SIG3.

  SIGE is the equivalent stress:

❑

F_WRITE

List selected analysis results on the screen or in a file.

Note:

- F_WRITE uses the file output by the RESULT function. To ensure that the most recent analysis results are used, run RESULT before accessing F_WRITE.

SELECT:	NODES
	ELEMENTS

NODES The average of the analysis results at each node in each of its adjacent elements is listed.

ELEMENTS Analysis results are listed by element for every node in the element.

NODES and ELEMENTS

SELECT:	SCREEN	FILE

SCREEN Display the selected analysis results on the alphanumeric screen.

FILE Write the selected analysis results in a file.

NODES >> SCREEN and FILE

If FILE is selected, enter the name of the file to which the selected results will be written.

ENTER FILE NAME

SELECT:	DISPLACEMENTS	REACTIONS	STRESSES
	PRINCIPAL STRESSES	INTERNAL LOADS	

Selected the type of results to be listed.

ELEMENTS >> SCREEN and FILE

If FILE is selected, enter the name of the file to which the selected results will be written.

ENTER FILE NAME

SELECT:	ALL
	BY ELEMENT TYPES
	BY MATERIAL PROPERT
	BY GEOMETRICAL PROP
	BY RESULTS VALUES

Select the criteria to be included in the analysis results.

SELECT:	STRESSES	PRINCIPAL STRESSES	INTERNAL LOADS

Select the type of results to be listed for the above criteria.

NODES >> SCREEN and FIL >> DISPLACEMENTS

SELECT DISPLACEMENTS:	DX	DY	DZ	RX	RY	RZ

Select any combination of displacements to be included in the output. Selected displacements will be lit. <EXIT> when finished.

The output will be sent.

NODES >> SCREEN and FILE >> REACTIONS

SELECT REACTIONS:	FX	FY	FZ	MX	MY	MZ

Note:

- When reading Inject3 results, the following table is displayed:

SELECT:	PRESSURE	MAX_TEMPERATURE	FLOWFRONT
	SOLIDIFICATION	VOL_SHRINKAGE	REMAINING_COOLTIME

Select any combination of reaction forces to be included in the output. Selected reaction forces will be lit. <EXIT> when finished.

The output will be sent.

NODES and ELEMENTS >> SCREEN and FILE >> STRESSES

SELECT STRESSES:	SXX	SYY	SZZ	SXY	SXZ	SYZ

Note: • When reading Inject3 results, the following table is displayed:

SELECT:	SHEAR_RATE	SHEAR_STRESS	FRONT_TEMPERATURE
	TIME_TO_SOLIDIFICAT		

Select any combination of stresses to be included in the output. Selected stresses will be lit. <EXIT> when finished.

The output will be sent.

NODE >> SCREEN and FILE >> PRINCIPAL STRESSES

SEL. PRINCIP STRESS:	SIG1	SIG2	SIG3	SIGI	SIGE

ELEMENT >> SCREEN and FILE >> PRINCIPAL STRESSES

SEL. PRINCIP STRESS:	SIG1	SIG2	SIG3	SIGE	SIGT

Select any combination of principal stresses to be included in the output. Selected principal stresses will be lit. <EXIT> when finished.

The output will be sent.

NODES and ELEMENTS >> SCREEN and FILE >> INTERNAL LOADS

SEL. INTERNAL LOADS:	FX	FY	FZ	MX	MY	MZ

Note: • When reading Inject3 results, the following table is displayed:

SELECT:	PRESSURE	MAX_TEMPERATURE	FLOWFRONT
	SOLIDIFICATION	VOL_SHRINKAGE	REMAINING_COOLTIME

Select any combination of internal loads to be included in the output. Selected internal loads will be lit. <EXIT> when finished.

The output will be sent. ❑

Appendix A
The FE Method of Structural Analysis

Definition

The Finite Element Method is a generic mathematical numeric technique used to analyze the behavior of physical systems which can be represented by the general equations of continuum mechanics. This technique is most commonly used to analyze static/dynamic elastic structural behavior and heat conduction within a body. Therefore, only these applications are considered throughout this manual. Nevertheless, the pre/post-processors described are equally applicable to the other disciplines of continuum mechanics.

The Finite Element Method of structural analysis is a technique which dramatically streamlines the static and dynamic analysis of complex elastic structures. Analyses which were formerly only theoretically possible, are both possible and practical using this method. Its unique strength lays in its remarkable capability to predict accurate structural behavior for even the most complex structural designs exhibiting:

- arbitrarily complex geometry
- arbitrarily complex boundary conditions
- non-homogeneous material properties
- linear/nonlinear elastic behavior
- arbitrarily complex applied loading

Finite Elements

The Finite Element Method is based upon the initial idealization of the designed structure as an assemblage of a finite number of basic structural building blocks or “elements”. These elements are connected physically in a continuous manner, at their vertices (nodes) and along their common boundaries.

Each of these basic “finite element” building blocks is a simple 1D, 2D or 3D parametric geometric shape. It is assumed to have constant, or simply varying, geometric and material properties. As a result, for each element for which the geometric and material properties are input, an “elemental stiffness matrix” may be automatically computed by the FEA, Finite Element Analysis, program. The elemental stiffness matrix represents the essential elastic behavior of a specific element.

Using a systematic approach, the program automatically “merges” all the individually compiled elemental stiffness matrices into one large “structural stiffness matrix”. This structural stiffness matrix represents the elastic behavior of the entire structure relative to the displacement and rotational degrees-of-freedom at the nodes of the structure.

For example, if the structure were idealized as an assemblage of 1000 finite elements containing a total of 1200 nodes; and if at most of the nodes there were 6 allowed degrees-of-freedom (three displacement components plus three rotational components); then, the size of the structural stiffness matrix would be 1200 x 6 = 7200 by 7200.

$$[\,K\,] = 7200 \text{ rows, each containing } 7200 \text{ numbers}$$
$$7200 \times 7200$$

or:

$$\begin{bmatrix} K1,1 & K1,2 & K1,3 & \cdots\cdots & K1,7200 \\ K2,1 & K2,2 & K2,3 & \cdots\cdots & K2,7200 \\ . & . & & & ,\,, \\ . & . & & & ,\,, \\ K7200,1 & K7200,2 & K7200,3 & \cdots\cdots & K7200,7200 \end{bmatrix}$$

This structural stiffness matrix serves primarily as a coefficient matrix in the set of 7200 linear equations with 7200 unknowns.

$$\underset{7200 \times 7200}{[\quad K \quad]}\ \underset{7200 \times 1}{\{\quad u \quad\}} = \underset{7200 \times 1}{\{\quad F \quad\}}$$

In this example, the {u} column matrix represents the 7200 unknown displacement and rotation components at the 1200 nodes of the structure. The {F} column matrix represents the up to 7200 concentrated applied force and moment components at the respective nodal degrees-of-freedom of the structure.

The following basic element types are supported by most Finite Element Analysis programs.

Element Type		Element Description
1.	Rod/Bar	Axial elastic behavior only; constant section properties.
2.	Torsion Bar	Torsional elastic behavior only; constant section properties.
3.	Beam	Pure bending only; in any plane; constant section properties.
4.	General Beam	Superposition of rod + torsion + beam.
5.	Triangular Membrane	In-plane elastic behavior only; constant thickness; constant /linearly-varying strain assumptions. (Used in plane strain/stress problems.)
6.	General Quadrilateral Membrane	In-plane elastic behavior only; constant thickness; constant /linearly-varying strain assumptions. (Used in plane strain/stress problems.)
7.	Triangular Plate	Thin plate (constant thickness) out-of-plane bending only.
8.	Quadrilateral Plate	Thin plate (constant thickness) out-of-plane bending only.
9.	Tetrahedron	Four-faced solid polyhedron, (four triangular faces).
10.	Pentahedron	Five-faced solid polyhedron, (two opposing triangular faces, connected by three quadrilateral faces).
11.	Hexahedron	Six-faced solid polyhedron, (two opposing quadrilateral faces, connected by four quadrilateral faces).
12.	Axi-symmetric Triangular Ring	Axi-symmetric solid ring element, represented by its triangular cross-section in cylindrical coordinates.
13.	Axi-symmetric Quadrilateral Ring	Axi-symmetric solid ring element, represented by its quadrilateral cross-section in cylindrical coordinates.

No.	Element Name	Element Type	Nodes	Degrees-of-Freedom	
				Translational	Rotational
1.	Rod/Bar	Line	2	2	0
2.	Torsion Bar	Line	2	0	2
3.	Beam	Line	2	4	4
4.	General Beam	Line	2	6	6
5.	Triangular Membrane	Area	3	6	0
6.	General Quadrilateral Membrane	Area	4	8	0
7.	Triangular Plate	Area	3	3	6
8.	Quadrilateral Plate	Area	4	4	8
9.	Tetrahedron	Volume	4	12	0
10.	Pentahedron	Volume	6	18	0
11.	Hexahedron	Volume	8	24	0
12.	Axi-symmetric Triangular Ring	Area (=Solid)	3	6	0
13.	Axi-symmetric Quadrilateral Ring	Area (=Solid)	4	8	0

Additional “advanced technology” elements, such as “ISOPARAMETRIC” or “SUPER” elements may be available in a program library. These special elements provide for a more accurate geometric fit of the structure to be analyzed. As a result, equally accurate results can be produced with a smaller number of larger elements of these types.

Boundary Conditions

Before a structure can be analyzed so that its behavior under applied loads may be predicted, it must be constrained to prevent rigid body motion as a result of load application. In the finite element idealized model of the structure, the boundary constraints are imposed by constraining motion in the direction of the appropriate degrees-of-freedom of the relevant nodes. Usually, this means constraining such motion to zero. However, the required nodal displacement/rotation component may also be constrained to a prescribed force value. Sometimes such constraints may also exhibit prescribed elastic characteristics.

Material Properties

The Finite Element Method may be used to analyze structures composed of materials whose characteristics vary non-uniformly within the structure. In regular metallic type structures, each finite element is usually assumed to exhibit isotropic behavior, that is; the element material is assumed to possess equal elastic properties in all directions. However, non-isotropic elemental material characteristics may be defined as well.

For example, an orthotropic material which exhibits different elastic properties with respect to different orthogonal directions may be specified.

In some Finite Element Analysis programs general anisotropic material matrices are permitted. Thus, a complex structure made of “general” composite materials may also be analyzed.

Applied Loads

When acted upon by a set of applied loads, an elastic structure undergoes deformation (each point within the structure experiences a displacement and/or rotation). Consequently, each point within the structure is put under stress, and reaction forces are produced at the boundary constraints

Concentrated Forces

These are discrete forces which act upon a given point on the boundary of the structure and in a given direction. In the finite element method, concentrated forces are applied at the appropriate node point of the structure and in the appropriate direction (using its 3 orthogonal force components). [units = force]

Distributed Loads

- **Running Load** - A continuously varying (running) load, which acts along a line or curve, and is measured in force/length units, where F=F(s). [force per unit running segment length]
- **Pressure** - A continuously varying normal distributed load (or area shear load) which acts over a closed planar boundary face or spatial surface. [units = force/length squared (force per unit area)]

Body Loads (Gravity or Inertia)

Continuously distributed kinetic "volumetric" forces which are generated within each internal mass point of the structure as a result of:

- gravity (g) loads
- linear and/or curvilinear accelerated motion (including centrifugal and Coriolis forces).

Thermal Loads

In general, these are internal structural loads generated as a result of the existence of a variable temperature environment and/or materials exhibiting different thermal expansion properties.

Analysis Results and FEA Program Output

After all the relevant data has been input into the Finite Element Analysis program the following calculations are performed.

Static Response Problems

First, the program computes the numerical value of the individual elemental stiffness matrices.

Then the program transforms the elemental stiffness matrices relative to a common global coordinate system and merges them together to generate an assembled structural stiffness matrix. This enforces geometric continuity at the nodes connecting the various elements and at their common boundaries.

The structural stiffness matrix represents the coefficients of the final set of n equations in n unknowns (n = total number of nodal degrees-of-freedom of the structure).

$$\underset{n \times 1}{\{\mathbf{F}\}} = \underset{n \times n}{[\mathbf{K}]} \; \underset{n \times 1}{\{\mathbf{u}\}}$$

Applied forces at nodal degrees-of-freedom

Structural stiffness matrix

Resulting linear and/or rotational displacements at respective nodal degrees-of-freedom

The calculated displacements {u} are used to compute the resulting reactive forces and/or moments at the applied nodal boundary constraints, and the resulting internal loads and/or stresses within all the elements (or at their nodes).

The stress at any point of a structure is represented by a 3 x 3 symmetric matrix, whose terms are a function of the chosen orientation. The unique orientation for which all non-diagonal stress terms become zero is the "principal direction", and the 3 non-zero diagonal terms ($\sum$ xx, $\sum$ yy, $\sum$ zz) are the "principal stresses".

The most popular stress criteria for analysis are:

- Von-Mises stress criterion
- Tresca condition

Dynamic (Vibrational) Analysis

The program outputs the requested natural frequencies of the structure (measured in CPS) and, the corresponding mode shapes (measured as relative linear and/or rotational displacements at selected nodal degrees-of-freedom.

Forced Dynamic Response Analysis

The program outputs displacements, internal loads and boundary reactions, and element stresses, all as a function of time.

Applying the Finite Element Method

Historically, the F.E. model was prepared for Finite Element Analysis, without the help of a FEM pre-processing package such as **Cimatron**[it] FEM, and input into a FEA program. The manual procedure was as follows:

After the finite element model is conceived, it is manually constructed directly on the engineering drawing(s) of the structure to be analyzed using the standard tools of the draftsman, i.e. pencil, eraser, ruler, etc.

- A "global" coordinate system is chosen and drawn.
- The Finite Element Model is constructed based on the design of the structure to be analyzed.
- The numerous idealized finite elements are drawn in detail directly on the engineering drawing(s).

The following data is then "lifted" off the drawing, copied onto formatted data sheets and coded for input into a FEA program.

- **Node Data** - Each node is given a unique integer number which identifies it. For each node the X, Y and Z coordinates are specified and listed.
- **Element Data** - Each element is given a unique element integer number which identifies it. Type and vertex node numbers must be specified for each element, as well as its associated geometric section properties and material constants.

- **Boundary Constraint Data** - All constrained nodal degrees-of-freedom must be specified together with the associated constraint type for each (e.g. total constraint/forced displacement).
- **Load Data**
  - All nodal degrees-of-freedom, at which applied concentrated forces/moments exist, must be identified and the values specified.
  - All boundaries upon which distributed loads act must be identified and the respective magnitudes specified (including thermal environments).
  - Body load environments and magnitudes must be specified.

Manual idealization of the FEM and preparation of the required input is a very labor-intensive, frustrating and error-prone process. This is especially true when the structure has many elements and degrees-of-freedom, and when 3D geometry and solid elements are used.

Preparation of a FEM, Finite Element Model, for a 5 minute computerized FE analysis, commonly requires a 5 week input preparation prelude, when done manually as described above. In addition, a simple error in input may produce incorrect FE analysis results which cannot be easily verified. As a result, the structure may be incorrectly designed and manufactured.

The computerized FE Method pre/post-processing approach provided by **Cimatron**^{it} FEM, eliminates all these problems and can provide for as much as a 50:1 decrease in the time required for correct FEA data preparation and analysis results.

The computerized post-processor can selectively display the analysis results in the following graphic forms:

- 2D and 3D graphic illustration of displaced structure (vs. original structure)
- Contour plots displaying lines of constant stress (These may be color-coded to indicate the areas of stress which are of interest).

These graphic displays allow the designer to see deformations and stress patterns without having to analyze, in frustrating detail, an immense volume of numeric data. Potentially weak (or overly strong) spots that need to be reinforced (or reduced) can be instantly detected. The required modifications can then be rapidly made to the geometry, constraints and/or materials, and the modified structure can be reanalyzed.

The generation of the proper model is usually an iterative process. A “coarse” model with relatively few simple elements is created first. It is then refined, as often as required by the ensuing analysis results, with more elements in the critical areas. This process is repeated until sufficient accuracy is achieved. ❑

Appendix B
Glossary of Cimatron^{it} FEM Terminology

AXISYMMETRIC

If a 3D system (geometry and loading) can be generated by revolving a 2D section 360 degrees about an axis, the system is said to be axisymmetric.

AXISYMMETRIC SOLIDS

Bodies of revolution whose finite elements are circular rings with either triangular or quadrilateral section shapes.

BEAM

A type of one-dimensional finite element represented graphically as a straight line. It has tension-compression and bending capability.

BOUNDARY CONSTRAINTS (CONDITIONS)

A loaded body or structure is free to experience unlimited rigid body motion unless some supports or constraints are imposed that will ensure the equilibrium of the loads. These constraints are the boundary conditions.

BRICK

See HEXAHEDRON.

BULK MODULUS

Also SHEAR MODULUS. See ELASTIC MATERIAL CONSTANTS.

DEGREES-OF-FREEDOM

At a node, they are the independent orthogonal translational and rotational directions allowed at that node. For example: the full set of degrees-of-freedom at a node of a finite element structural model include 3 orthogonal displacement components, plus 3 orthogonal rotational components (= 6 degrees-of-freedom).

The number of degrees-of-freedom of a finite element model is the total of all independent degrees-of-freedom at all the nodes of the model. Those degrees-of-freedom at which constraints exist are called the boundary constrained degrees-of-freedoms of the structural model.

DISCRETIZATION or Finite Element Idealization

The process of subdividing a (structural) continuum into an assemblage of discrete finite elements which are geometrically connected at the element vertices and at their boundary edges. (See also FINITE ELEMENT.)

DISTRIBUTED LOAD

See LOAD.

DOF (Also D.O.F.)

See DEGREES-OF-FREEDOM.

EIGENVALUE

A mathematical term used in buckling and structural frequency analysis which represents buckling load OR natural frequency.

ELASTIC MATERIAL CONSTANTS

A tensile load F acting on a uniform bar specimen causes elongation of the bar by an amount of STRAIN:

$$\varepsilon = \Delta L/L$$

where L is the original length of the bar and ΔL is the change in length due to the application of the tensile force F.

The tensile STRESS, σ, measured at any point of the bar, is

$$\sigma = F/A$$

where A is the constant cross-sectional area of the bar (i.e. is measured in force per unit area.)

The tensile load F on the bar also causes a lateral contraction. In the linear elastic range, this lateral strain, ε_{lat}, is proportional to the longitudinal strain, ε, and is expressed as:

$$\varepsilon_{lat} = -\upsilon\varepsilon$$

where the constant of proportionality, υ, is called POISSON'S RATIO and the negative sign denotes contraction.

The stress, σ, at a point in the bar, is linearly proportional to the strain, ε, and is expressed as:

$$\sigma = E_{\varepsilon}$$

where the constant of proportionality E is called Young's Modulus of Elasticity.

E and υ are in fact the fundamental material constants characterizing the general three-dimensional behavior of linearly elastic, homogeneous, isotropic materials.

A third (alternative) material constant which is usually used in shear and 3D problems, is called the BULK or SHEAR MODULUS, G, and is related to and E as follows:

$$G = E/(2(1+\upsilon))$$

ELEMENT, FINITE

See FINITE ELEMENT.

FEMSYS

A 3D view of the geometric model with a unique coordinate system, relative to which finite elements and meshes are generated.

FINITE ELEMENT

A synthetic discrete element which is produced in the process of the finite element idealization of a continuous body. Each finite element has a distinct generic geometric shape containing vertices (node points), and each exhibits independent discrete physical properties. (See also DISCRETIZATION.)

FINITE ELEMENT MODEL

A discrete idealization of a real system. Nodes, elements and boundary conditions are used to describe the model. (See also FINITE ELEMENT and DISCRETIZATION.)

GEOMETRIC PROPERTIES

Intrinsic shape properties of the finite structural elements of the model, such as cross-sectional area, moment-of-inertia and thickness.

HEAT CONDUCTION

See HEAT TRANSFER ANALYSIS.

HEAT TRANSFER (THERMAL) ANALYSIS

A type of analysis which finds the temperature distribution and heat flow rates within a body. Also called Heat Conduction Analysis.

HEXAHEDRON

A type of 3D solid element formed by sweeping a general quadrilateral (via translation and/or rotation), and connecting its vertices by straight lines. The hexahedron is composed of 6 quadrilateral faces (4 connecting and 2 opposing), and bounded by 8 vertices and 12 edges.

A rectangular BRICK normally implies a kind of hexahedron which is formed by a linear sweep only, of a rectangular base. In **Cimatron**^{it} FEM, the word BRICK is used in its general sense to mean HEXAHEDRON.

ISOTROPIC

Identical material properties in all directions.

LINEAR BUCKLING ANALYSIS

A type of analysis which finds the critical load factor and buckled mode shape of a linear structure.

LINEAR STRUCTURE

See explanation of NONLINEAR STRUCTURE.

LOAD

An applied environmental condition which produces stresses within a structure and causes it to deform.

Examples of Loads

- **Point (discrete) loads**
  Forces and/or moments concentrated at specific points on the boundaries of a structure. (A moment is a rotational force which is expressed in units of force * length.)
- **Traction loads**
  Forces and/or moments distributed over external boundaries of the structure.
- **Body loads**
  Forces acting on each and every molecule of the body (e.g. gravity or inertia load).

- **Thermal loads**
  Thermal environment which produces stresses and deformations in the structure.

All loads acting on a real structure must be converted into an equivalent set of discrete forces acting at the nodal degrees-of-freedom of the finite element model of the structure, unless this conversion is done automatically by the specific F.E.Analysis program to be used.

MATERIAL PROPERTIES

Intrinsic physical elastic properties of the structural finite elements of the model, such as: Modulus of Elasticity, Shear (Bulk) Modulus, Poisson's Ratio, Coefficient of Thermal Expansion, etc.

MEMBRANE

A type of two-dimensional thin finite element, represented geometrically as a triangle or quadrilateral, which is capable of sustaining in-plane loads only (i.e. no bending). Usually used for plane stress or plane strain analysis.

MESH

A collection of 2D or 3D finite elements which share at least one common, non-overlapping edge and which was created automatically with ELEM_2D or ELEM_3D.

MODAL ANALYSIS

A type of analysis which finds the natural frequencies and mode shapes of a linear structure.

MODULUS OF ELASTICITY

See YOUNG'S MODULUS OF ELASTICITY.

MOMENT

See LOAD.

NODAL FORCES

Forces and moments which are concentrated at the nodes of a finite element model.

NODE (Also called a Joint)

A point on the boundary of a finite element, usually at a vertex.

For example:

- a triangular finite element usually has 3 nodes, at its 3 vertices;
- a solid tetrahedron finite element usually has 4 nodes at its 4 vertices.

See also FINITE ELEMENT and DISCRETIZATION.

NONLINEAR STRUCTURE

A NONLINEAR STRUCTURE is a structure which is assumed to possess one or both of the following characteristics:

- NONLINEAR GEOMETRY
  A geometrically nonlinear structure is a structure for which small displacement theory may not be assumed, i.e. the deflections and rotations resulting from the loading are not very small.
- PHYSICAL NONLINEARITY
  A physically nonlinear structure is a structure composed of materials for which a linear stress-stain relationship may not be assumed. (See also MATERIAL CONSTANTS.)

ORTHOTROPIC MATERIAL

A material which possesses different material properties in different orthogonal directions.

Examples:

- The Cartesian coordinate directions: Ex, Ey, Ez.
- The Cylindrical coordinate directions: $E\theta$, $E\phi$, Ez

PENTAHEDRON

A type of 3D solid element formed by sweeping a triangle (via translation and/or rotation) and connecting its vertices by straight lines. See Figure 3-8 on page 3-27. The pentahedron is composed of 3 connected quadrilateral faces and 2 opposing triangular faces, and is bounded by 6 vertices and 9 edges.

A triangular PRISM normally implies a kind of pentahedron which is formed by a linear sweep only, of the triangular base. See Figure 3-7 on page 3-25. In **Cimatron**[it] FEM, the word PRISM is used in its general sense to mean PENTAHEDRON.

PLANE STRAIN (or PL STRAIN)

Plane strain is an assumption made in the theory of Elasticity when the body under consideration is long, and its cross-sectional geometry and loading do not vary significantly longitudinally. In this case, any thin slice of the body (cut perpendicular to the longitudinal direction), acted upon by its proportional share of the load, approximately represents the behavior of any other parallel thin slice of the body; and the longitudinal strain component at any point within the body is assumed to be zero. Thus, the 3D solid problem is reduced to a simple 2D problem, which may be represented by thin membrane finite elements. See Figure B-1.

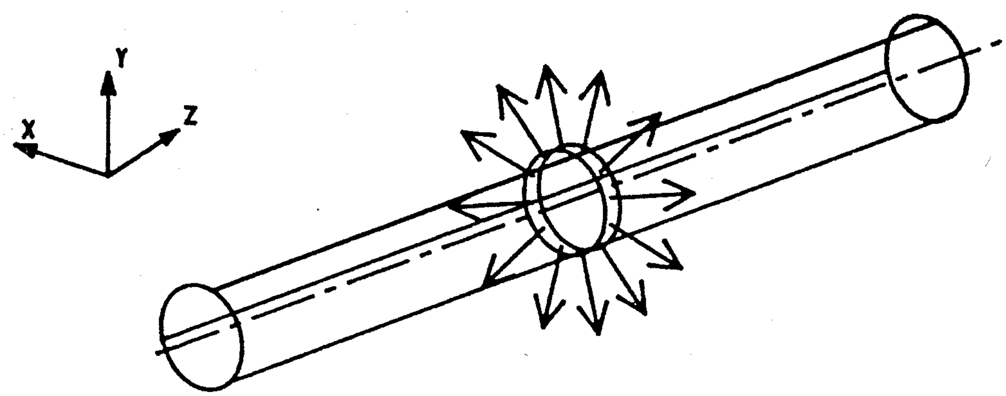


Figure B-1: Plane Strain

PLANE STRESS (or PL STRESS)

Plane stress is an assumption made in the theory of Elasticity when the body under consideration is, in fact, a thin membrane, with no out-of-plane loads acting upon it. Therefore, the assumption is made that the stress component which is normal to the plane of the membrane at any point within the membrane, is zero. Such a structure is represented by 2D membrane finite elements.

PLATE

A type of 2D thin finite element, represented geometrically as a triangle or quadrilateral, which is capable of sustaining out-of-plane loading. Usually used for thin plate and shell bending analysis.

POISSON RATIO

See ELASTIC MATERIAL CONSTANTS.

PRISM

See PENTAHEDRON.

SHEAR MODULUS

Also BULK MODULUS. See ELASTIC MATERIAL CONSTANTS.

SHELL

A thin shell is a 3D structure shaped like a doubly curved surface and made of a thin material. It can sustain both in-plane and out-of-plane loads. It may be idealized either by doubly curved thin shell finite elements (not available in **Cimatron**^{it} FEM), or planar 2D plate and membrane elements.

Examples of shells are:

- thin circular cylindrical shell
- thin truncated conical shell
- thin surface of revolution shell

SHRINK

A function which displays a picture of the finite element model such that all the finite elements are reduced in size visually by a given factor, while maintaining the original scale and node position of the finite element model. See Figure 2-1 on page 2-8: Shrink Lines - Before and After.

SPAR

A type of one-dimensional finite element represented geometrically as a straight line, which is capable of sustaining axial loads only.

STATIC ANALYSIS

A type of structural analysis which finds displacements, stresses, strains and forces in a structure subject to applied loads and boundary constraints.

STRAIN

Percentage of relative displacement. See also ELASTIC MATERIAL CONSTANTS.

STRESS

Unit force per area. See also ELASTIC MATERIAL CONSTANTS.

SUBSTRUCTURE ANALYSIS

See SUPER-ELEMENT ANALYSIS.

SUPER-ELEMENT ANALYSIS

A type of analysis which generates a super-element by reducing the assembled element matrices to a subset of master degrees-of-freedom. This super-element may, in general (e.g. ANSYS), be used in any other analysis type.

SUPPLEMENTARY GEOMETRIC PROPERTIES

See GEOMETRIC PROPERTIES.

TETRA(HEDRON)

A type of 3D solid finite element composed of 4 connected triangular faces and bounded by 4 vertices and 6 edges. See Figure B-2.

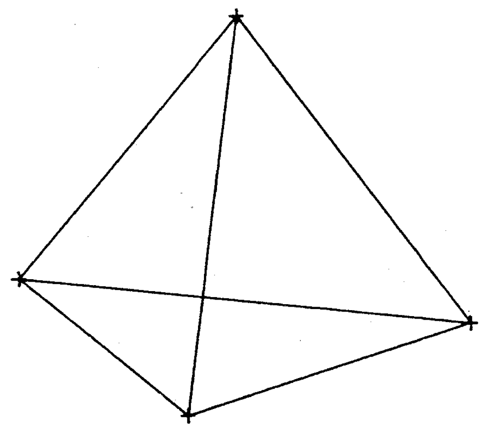


Figure B-2: Tetrahedron

TRACTION

See LOAD.

YOUNG'S MODULUS OF ELASTICITY

See ELASTIC MATERIAL CONSTANTS. ❑

Index

!

2D CONTOUR option 3-6, 3-10
See also ELEM_1D
See also ELEM_2D
3D CONTOUR option 3-6, 3-10
See also ELEM_1D
See also ELEM_2D

A

ACTIVE option 2-2
See also FEMSYS
Analysis 1-2
See also FEM Procedure
See Heat Transfer
See Linear Buckling
See Modal
See Static
See Substructure
See Super-Element
See Thermal
ANSYS option 2-17
See also F_FILE
APPEND option 2-5
See also FEMSYS
Applied Loads A-4
See also FE Method-Structural Analysis
Applying FE Method A-6 TO A-7
See also FE Method-Structural Analysis
Axisymmetric B-1
Axisymmetric Solids B-1

B

Beam B-1
Body Loads B-3
See also Load
Body Loads (Gravity or Inertia) A-5
Boundary Conditions A-4
See also FE Method-Structural Analysis
Boundary Constraints B-1
Brick
See Hexahedron

C

C_MOLD option 2-18
See also F_FILE
Concentrated Forces A-4
Conditions B-1
CONTOURS option 5-4
See also F_PLOT

D

DEFINE COLORS option 2-12
See also F_DISP
DEFINE DISPLAY option 2-10
See also F_DISP
Definition of FEM A-1
See also FE Method-Structural Analysis
Degrees-of-Freedom B-1
DELETE 3D ELEMENTS option 3-41
See also F_EDIT
DELETE option 2-4
See also FEMSYS
Discretization B-1
DISPLAY MODES option 2-7
See also F_DISP
Distributed Load
See Load
Distributed Loads A-5
DOF
See Degrees-of-Freedom
Dynamic (Vibrational) Analysis A-6

E

Eigenvalue B-2
Elastic Material Constants B-2
ELEM_1D modeling function 3-2 to 3-8
- 2D CONTOUR option 3-6
- SINGLE CURVES option 3-3
- SURFACE option 3-7

ELEM_1D modeling option
- 3D CONTOUR option 3-6

ELEM_2D modeling function 3-9 to 3-22
- 2D CONTOUR option 3-10
- 3D CONTOUR option 3-10
- SURFACE option 3-18

ELEM_3D modeling function 3-23 to 3-31
- ELEMENTS option 3-23
- MESHES option 3-23

ELEMENT option 3-35
- See also F_EDIT

Element, Finite B-2
ELEMENTS option 3-23, 3-42, 5-7
- See also ELEM_3D
- See also F_MOVE
- See also F_WRITE

F

F_DISP general function 2-6 to 2-15
- DEFINE COLORS option 2-12
- DEFINE DISPLAY option 2-10
- DISPLAY MODES option 2-7
- HIDDEN LINES option 2-15

F_EDIT modeling function 3-32 to 3-41
- DELETE 3D ELEMENTS option 3-41
- ELEMENT option 3-35
- MESH option 3-39
- NODE option 3-33

F_FILE general function 2-16 to 2-18
- ANSYS option 2-17
- C_MOLD option 2-18
- INJECT_3 option 2-18
- NASTRAN option 2-18

F_LOAD loads/prop function 4-2 to 4-15
- HEAT option 4-11
- STRENGTH option 4-3

F_MOVE modeling function 3-42 to 3-46
- ELEMENTS option 3-42
- MESHES option 3-42

F_PLOT post-proc function 5-3 to 5-6
- CONTOURS option 5-4

F_PROP loads/prop function 4-16 to 4-30
- GEOMETRIC option 4-16
- MATERIAL option 4-16

F_WRITE post-proc function 5-7 to 5-9
- ELEMENTS option 5-7
- NODES option 5-7

FE Method-Structural Analysis A-1
- Applied Loads A-4
- Applying FE Method A-6 TO A-7
- Boundary Conditions A-4
- Definition A-1
- Finite Elements A-1 TO A-3
- Material Properties A-4
- Results & FEA Output A-5

FEM Procedure 1-1 to 1-3
- Analysis 1-2
- Post-processing 1-2
- Pre-processing 1-1

FEMSYS B-2
FEMSYS general function 2-2 to 2-5
- ACTIVE option 2-2
- APPEND option 2-5
- DELETE option 2-4
- RENAME option 2-4

Finite Element B-2
Finite Element Model B-3
Finite Elements A-1 TO A-3
- See also FE Method-Structural Analysis

Forced Dynamic Response Analysis A-6

Functions-Overview 1-4 to 1-7
- General Functions 1-5
- Loads & Properties Functions 1-6
- Modeling Functions 1-5
- Post-processing Functions 1-6

G

General Functions-Overview 1-5
- See also Functions-Overview

GEOMETRIC option 4-16
- See also F_PROP

Geometric Properties B-3
Glossary B-1
Gravity A-5

H

Heat Conduction
- See Heat Transfer Analysis

HEAT option 4-11
- See also F_LOAD

Heat Transfer Analysis B-3
Hexahedron B-3
HIDDEN LINES option 2-15
- See also F_DISP

I

Inertia A-5
INJECT_3 option 2-18
- See also F_FILE

Isotropic B-3

J

Joint (Node) B-4

L

Linear Buckling Analysis B-3
Linear Structure
- See Nonlinear Structure

Load B-3
- Body B-3
- Point (discrete) B-3
- Thermal B-4
- Traction B-3

Loads & Properties Functions-Overview 1-6
- See also Functions-Overview

M

MATERIAL option 4-16
- See also F_PROP

Material Properties A-4, B-4
- See also FE Method-Structural Analysis

Membrane B-4
Mesh B-4
MESH option 3-39
- See also F_EDIT

MESHES option 3-23, 3-42
- See also ELEM_3D
- See also F_MOVE

Modal Analysis B-4
Modeling Functions-Overview 1-5
- See also Functions-Overview

Modulus (Bulk/Shear)
- See Elastic Material Constants

Modulus of Elasticity
- See Young's

Moment
- See Load

N

NASTRAN option 2-18
- See also F_FILE

Nodal Forces B-4
Node (Joint) B-4
NODE option 3-33
See also F_EDIT
NODES option 5-7
See also F_WRITE
Nonlinear Structure B-4

O

Orthotropic Material B-5

P

Pentahedron B-5
Plane Strain (PL Strain) B-6
Plane Stress (PL Stress) B-6
Plate B-6
Point (discrete) Loads B-3
See also Load
Poisson Ratio
See Elastic Material Constants
Post-processing 1-2
See also FEM Procedure
Post-processing Functions-Overview 1-6
See also Functions-Overview
Pre-processing 1-1
See also FEM Procedure
Prism
See Pentahedron
Properties
See Geometric
See Material
See Supplementary Geometric

R

RENAME option 2-4
See also FEMSYS
RESULT post-proc function 5-2
Results & FEA Output A-5
See also FE Method-Structural Analysis

S

Shell B-7
Shrink B-7
SINGLE CURVES option 3-3
See also ELEM_1D
Spar B-7
Static Analysis B-7
Static Response Problems A-5
STRENGTH option 4-3
See also F_LOAD
Stress B-7
Substructure Analysis
See Super-Element Analysis
Super-Element Analysis B-7
Supplementary Geometric Properties
See Geometric Properties
SURFACE option 3-7, 3-18
See also ELEM_1D
See also ELEM_2D

T

Tetra(hedron) B-8
Thermal Analysis B-3
Thermal Loads A-5, B-4
See also Load
Traction
See Load
Traction Loads B-3
See also Load

V

VERIFY loads/prop function 4-31 to 4-35

Y

Young's Modulus of Elasticity
See Elastic Material Constants