THE UNIVERSITY OF MICHIGAN

COLLEGE OF ENGINEERING

DEPARTMENT OF ELECTRICAL ENGINEERING & COMPUTER SCIENCE

Radiation Laboratory

Report Title: Examples of Finite Element Mesh Generation using SDRC IDEAS

Report Author: John Zapp

f-56

The Radiation Laboratory
Department of Electrical Engineering
 and Computer Science
The University of Michigan
Ann Arbor, MI 48109-2122

Ann Arbor, Michigan

TECHNICAL REPORT

FOR

NASA Grant NAG-2-541

NASA Technical Monitor: Alex Woo

Institution: The Radiation Laboratory
Department of Electrical Engineering
 and Computer Science
The University of Michigan
Ann Arbor, MI 48109-2122

Period Covered: February 1990 - September 1990

Report Author: John Zapp

Report Title: Examples of Finite Element Mesh Generation using SDRC IDEAS

Principal Investigator: John L. Volakis
Telephone (313)764-0500

Table of Contents

Introduction to IDEAS

IDEAS (Integrated Design Engineering Analysis Software) offers a comprehensive package for Mechanical design engineers. Due to it's multifaceted capabilities, however, it can be manipulated to serve our needs as electrical engineers. IDEAS can be used to perform the following tasks:

- Solid modeling
- System assembly
- Kinematics
- Finite element pre/post processing
- Finite element solution
- System dynamics
- Drafting
- Test data analysis
- Project relational database

Each of IDEAS capabilities is contained in a family. These families, in turn, offer highly interactive, graphics oriented, menu-driven modules. (See figure 1.1).

Family	Module
Solid Modeling (I-DEAS Geomod)	Object Modeling System Assembly Mechanism Solution Excitation Definition Response
Engineering Analysis (I-DEAS Supertab)	Pre/Post Processing Model Solution Optimization Data Loaders Frame Analysis
System Dynamics (I-DEAS Systan)	Component Definition Orthogonality System Assembly Project Documentation Excitation Definition Response Component Loads
Drafting (I-DEAS Geodraw)	– single module –
Test Data Analysis (I-DEAS Tdas)	– single module –

Figure 1.1

1

Pre/Post Processing

As a member of the Engineering Analysis (Supertab) family, Pre/Post Processing allows the user to build finite element models, prepare them for analysis, and display analysis results following the analysis. It is divided into several major areas of functionality, called tasks. (See figure 1.2).
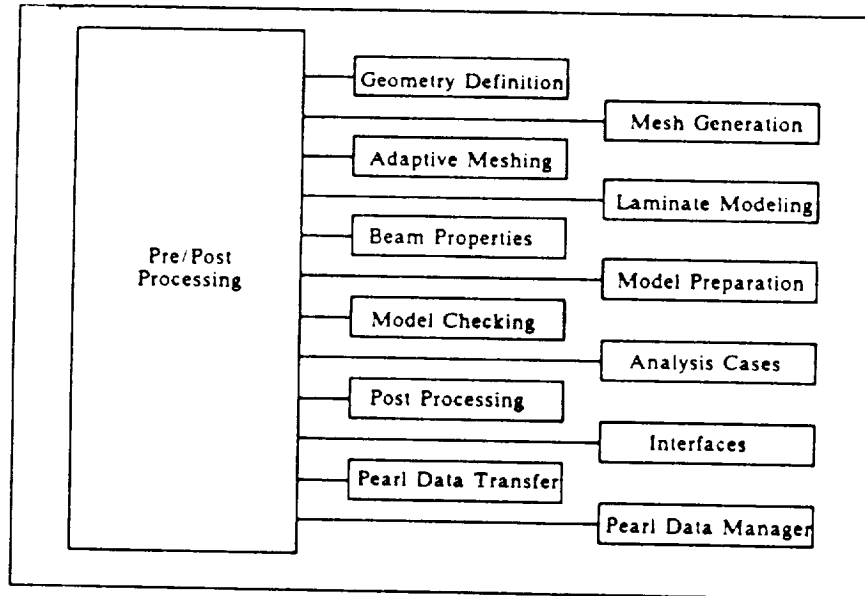
```
Pre/Post
Processing          Geometry Definition
                                        Mesh Generation
                    Adaptive Meshing
                                        Laminate Modeling
                    Beam Properties
                                        Model Preparation
                    Model Checking
                                        Analysis Cases
                    Post Processing
                                        Interfaces
                    Pearl Data Transfer
                                        Pearl Data Manager
```

Figure 1.2

Geometry Definition

Use the Geometry Definition task to create and manipulate points and curves as the basis for mesh generation. After you create geometry, you can access it in the Mesh Generation task and use it to define the criteria that control automatic mesh generation. This task also allows you to transfer solid object geometry from IDEAS Geomod.

Mesh Generation

Use the Mesh Generation task to create mesh areas and mesh volumes that control automatic mesh generation. In mesh area, Supertab generates shell elements, and in mesh volumes, solid elements. In each mesh area or volume, you can specify mapped meshing or free meshing.

Adaptive Meshing

Use the Adaptive Meshing task to redefine mesh areas based on information resulting from a finite element analysis or based on model checking values.

The mesh can be adapted with node movement, element splitting, or a combination of both techniques. Result error approximation techniques are also available in this task.

Model Preparation

Use the Model Preparation task to create nodes, elements, physical property tables, material property tables, coordinate systems, and trace lines. This task also offers the analysis program mask, which allows you to specify the finite element analysis program you plan to use. After you select an analysis program from this menu, Supertab masks all physical and material properties that do not apply to the program you selected.

Model Checking

Use the Model Checking task to verify your model's validity for analysis. For example, you can use this task to check your model for node coincidence, element coincidence, and element distortion. You can also use this task to calculate your model's bandwidth and wavefront.

Post Processing

Use the Post Processing task to display and study the results of an analysis, which exist in the model as analysis datasets. This task can generate displays of deformed geometry, contour plots, arrow plots, criterion plots, XY plots, and beam shear-and-moment diagrams.

Mesh Generation

In Pre/Post Processing, IDEAS offers a complete set of tools for automatic mesh generation, including mapped mesh generation and free mesh generation. Free mesh generation is performed using the Triquamesh algorithm. Both mapped meshing and free meshing can access geometric information in the form of points curves, and surfaces. You can use this geometry to define the mesh generation criteria that Supertab uses in mesh Generation.

To create the geometry to be used in mesh generation, you can use one of two methods. The first method utilizes the solid modeling capabilities of IDEAS using Geomod. Once the geometry is created in Geomod, it can be then transfered to the Pre/Post Processing module for manipulation and mesh generation. The second, and more direct method, is using the Pre/Post Processing's Geometry definition task to create points, curves, and surfaces for your mesh geometry.

In addition to automatic mesh generation, IDEAS offers an extensive variety of ways to create complex meshes of nodes and elements.

Nodes:

Single nodes can be created in several ways, you can:

•Key in coordinates, with the CREATE command.
•Digitize node location, with the DIGITIZE menu.
•Generate nodes between nodes, with the BETWEEN_NODES command.
•Copy nodes to a new location, with the COPY command.
•Generate nodes at point locations, with the ON_POINTS command.
•Generate nodes by reflection existing nodes with the REFLECT command.

Elements:

Elements can be created using the CREATE menu, and one of the following methods:

•Pick nodes to create individual elements with the SINGLE command.
•Copy elements, with the COPY command
•Generate elements by reflecting existing elements, with the REFLECT command.
•Extrude element faces into solid elements, with the EXTRUDE menu.
•Revolve element faces into solid elements with the REVOLVE menu.
•Add a layer of shell elements to a surface of solid elements, with the SURFACE_COATING menu.

To generate nodes and elements, Pre/Post Processing processes the information in a mesh-area. A mesh-area is an entity that is operated on by Pre/Post Processing's mesh generation to produce finite elements. A mesh-area is topological: it represents a geometric configuration that can be mapped to a parameter space. It does not, however, possess any geometry. The figures geometry must be previously define in order to create a mesh-area. This is due to the fact that a mesh-area is a series of curves that define the closed boundaries of a region to be meshed.

Once the geometry of the figure is entered the user must decide which of two types of meshes they require: mapped or free.

A free mesh-area can include from three to 100 curves. This mesh-area can describe fairly complicated regions, which can include interior voids, or holes. The user does, however, have control over quite a few specification of the free mesh create. You can:

•Define the general size of the elements using the
GLOBAL_ELEMENT_SIZE command.
•Choose the number of elements per boundary curve by using the
ELEMENT_PER_CURVE command.
•Define the size of elements at a point using the LOCAL_ELEMENT_SIZE command.

A second type of mesh-area is the mapped mesh area. This is more restrictive than a free mesh-area, because each mapped mesh-area is limited to three or four curves and cannot include interior voids. This curve limitation is not as severe as it might first appear, however, because each curve can be a composite curve. A composite curve is one that you create by merging several simple curves together. By using composite curves, you can model complicated boundaries using only a few curves. Once a mapped mesh area is defined, the users can define how many elements per curve are required. For a body with three sides, all three sides can be defined. For a body with four sides, however, only two adjacent sides can be defined. The computer automatically assigns the other two parallel curves with the same respective values.

Getting Started With I-DEAS

Follow these instructions to enter I-DEAS:

1. From the operating system, type the command for entering I-DEAS and a <CR>. In our case, on the apollo's, this would be:

    % ideas <CR>

2. IDEAS asks what type of terminal you are using. To list supported terminals, type M (for "menu") and a <CR>. Typing M lists the entire terminal menu. For our work we will use the terminal type, *generic apollo.*

    Enter terminal type#   apo   <CR>

3. IDEAS clears the screen and asks you for a Model file name. You can type the name of the Model file you created in a previous IDEAS session, or you can type a new name. The Model file name can be at least 7 characters on any operating system; most systems allow more. Do not include an extension to your Model file name; IDEAS supplies this. You can use upper or lowercase.

    Enter model file name (NONE)#   MODEL1   <CR>

4. If you type a new Model file name, IDEAS asks you to verify creating a new file (rather than opening an existing file). Type YES (or a <CR>) to verify a new Model file. The YES contained in parentheses after the prompt means that YES is the default response. You can accept the default response by typing a <CR> only.

    ```
    Y-YES
    N-NO

    !-BACKUP
    $-ABORT
    New model file OK? (YES)#   <CR>
    ```

5. IDEAS also asks for the model description for a new Model file. The default is set to the Model file name entered previously. Select it by entering <CR> (description can be up to 80 characters long and can include letters, numbers, and symbols. The description appears in the upper left corner of displays (and hard copies.)

    Enter model description (MODEL1)   Example   <CR>

6. The final prompt for a new Model file asks for the system of units you want to use for this Model file. For our Model we will use the default units by entering a <CR>.

    Enter system of units (METRIC_ABS_(SI)#   <CR>

7. IDEAS asks what operating mode you want. Your choices are normal operation or program-file operation. Type <CR> to choose the default response, which is normal operation. Program-file operation allows you to collect commands in a program file, or to run a file of commands you have previously collected.

Enter operation mode (NORMAL_OPERATION)# <CR>

8. IDEAS asks you what "family" you want. First, wait while IDEAS creates a temporary file. The temporary (scratch) file is where all Model file data is stored. You must enter a SAVE command to copy the temporary files to a permanent database. Then, the product family menu appears. For Mesh Generation, choose Engineering Analysis. This can be done using the cursor by clicking the right mouse button on the screen while the arrow or cross hairs are located on EA , or one can type in the mnemonics, in this case EA.

SELECT MENU# EA <CR>

9. IDEAS asks you what module within Engineering Analysis you want. To select a module, type its mnemonic (shown on the left on the menu) or select the menu with the tablet or mouse cursor. We want to enter the Pre-Post Processing Module for mesh generation.

SELECT MENU# P

Wait while IDEAS opens the executable for the module you selected. You are now ready to start!

About the Mesh Tutorials

In the following two tutorials the users can enter the commands in one of two ways. The first is by clicking on the command with the left mouse button when the cursor is located on the command in the menu window. The second method is by typing in the command using its mnemonic symbol. These are displayed in the tutorial in bold, capital print. Note: Only enter what is in bold or capital print. If there is ever confusion with any command entered, there is an appendix with some widely used commands. If the command doesn't appear in the appendix, the user can type HELP (A global command), and a help menu will appear. Continue by clicking on the desired command in either the global menu or current menu. A description of the desired command will appear in the bottom-most right window.

MESH GENERATION

*An example of Mesh Generation using IDEAS

TASK: In this example, we will create the mesh shown in figure 2.10 using the IDEAS Pre-Post Processing module.

PREPARATION: Enter IDEAS, create a model file, enter the Engineering Analysis family, and the Pre-Post Processing Module. (See Getting Started).

STEP #1: GEOMETRY DEFINITION

Our first step in creating a mesh is to define the geometry that we will mesh.

SELECT MENU# TAsk
SELECT MENU# Geometry_Definiton
SELECT MENU# CReate_Wire
SELECT MENU# Point
SELECT MENU# Position
Pick SCREEN_POSITION
Key_IN
Enter local X,Y,Z (0.0,0.0,0.0)# **10,0**
Enter local X,Y,Z (10,0,0.0,0.0)# **0,5**
Enter local X,Y,Z (0.0,5.0,0.0)# **-10,0**
Enter local X,Y,Z (-10.0,0.0,0.0)# **0,-5**
Enter local X,Y,Z (0.0,-5.0,0.0)# **Done**
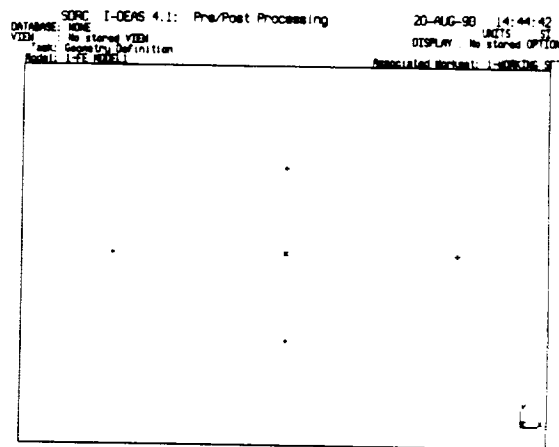
Figure 2.1

Now we have created the boundary points of the ojive, (See figure 2.1), use the autoscaling command AU so we can display on the monitor window the points we have just create. This is a Global command. (For help on Global commands type Help and then Global).

SELECT MENU# **AU**
SELECT MENU# **!**   (back up to Create_wire menu)
SELECT MENU# **Arc**
SELECT MENU# **P3**   (thru_3_points)
Specify start location
Pick VISIBLE

Here we want to use the cursor to select our points in order to create an arc. Select the 10,0 point first.

Specify intermediate location
Pick VISIBLE

Select the 0,5 point with the cursor.

Specify end location
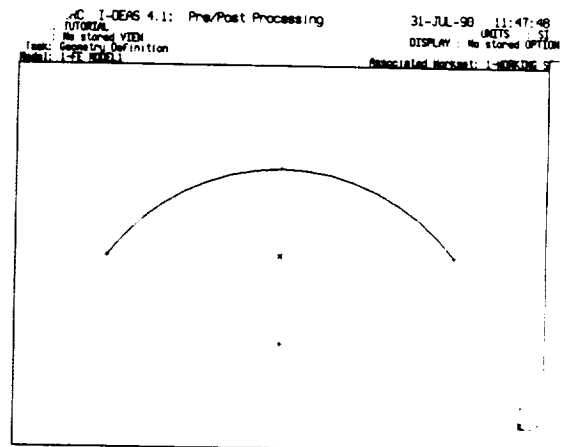Pick VISIBLE

Select the -10,0 point with the cursor

Figure 2.2

9

Now you have created the upper half of the ojive. (See figure 2.2). Repeat this procedure to create the lower half of our desired geometry. Exit the Arc command by entering Done. Next we will define our outer boundary.
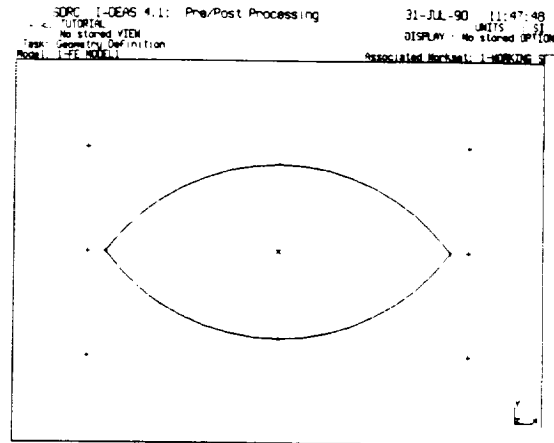
Specify start location
Pick VISIBLE
Done

SELECT MENU# /CReate_Wire
SELECT MENU# Point
SELECT MENU# Position
Pick Screen_Position
Key_In
Enter local X,Y,Z (0.0,0.0,0.0)# **11,6**
Enter local X,Y,Z (11.0,6.0,0.0)# **,0**
Enter local X,Y,Z (11.0,0.0,0.0)# **,-6**
Enter local X,Y,Z (11.0,-6.0,0.0)# **-11**
Enter local X,Y,Z (-11.0,-6.0,0.0)# **,0**
Enter local X,Y,Z (-11.0,0.0,0.0)# **,6**
Enter local X,Y,Z (-11.0,6.0,0.0)# **Done**
SELECT MENU# !   (backup to Create_Wire Menu)
SELECT MENU# Line
SELECT MENU# SIngle_Line

Figure 2.3

(It isn't necessary to enter a coordinate if it is to remain the same as the default value. A comma before the number means that first or X value will remain the same as the previous value.)   (See figure 2.3)

Pick VISIBLE

Select points in pairs, 10,0 to 11,0 first then 11,0 to 11,6; 11,6 to -11,6 and -11,6 to -11,0 and finally -11,0 to -10,0. (See figure 2.4). Continue doing this until we have our inside boundary lines. (See Figure 2.5). Type done when this is completed.
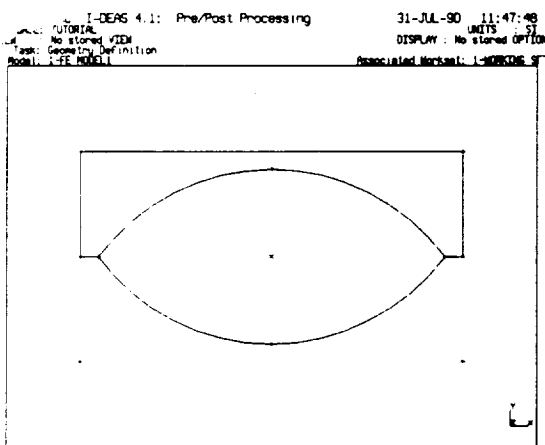
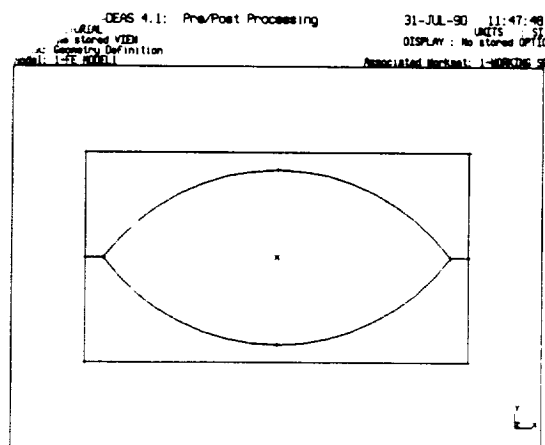We have now completed our geometry.

Figure 2.4   Figure 2.5

STEP #2: MESH GENERATION

The first step in generating a mesh is defining mesh areas from the geometry.

```
SELECT MENU#   /TAsk
SELECT MENU#   ME              (back to main menu, and enter Mesh Generation)
SELECT MENU#   MA              (Mesh  Areas)
SELECT MENU#   CReate
Enter mesh-area start label, inc (1,1)#  <CR>
Ok to use thin shell linear quadrilateral elements#   No
Select element family (THIN_SHELL)#  <CR>
Select element order (LINEAR)#  <CR>
Select element topology (QUADRILATERAL)#  Triangle
Enter PHYSICAL PROPERTY TABLE name or no. (1-PHYSICAL PROPERTY TABLE1)#  <CR>
Ok to use default values to create table? (YES)#  <CR>
Select material type (ISOTROPIC)#  <CR>
Enter MATERIAL PROPERTY TABLE name or no. (1-MATERIAL PROPERTY TABEL1)#  <CR>*
Ok to use default values to create table? (YES)  <CR>
3-100 curves required for mesh-area 1
Pick  curves
```

Now we must select the curves or lines in this case that border our mesh areas by clicking the mouse on them. This will cause the lines to be highlighted. (See figure 2.6). Once you are done selecting the boundaries of the area, enter Done.

```
Select  Curves
Pick  Visible
Done
Ok to pick a surface? (NO)#  <CR>
```

```
3-100 curves needed for mesh area 2
```

Select the curves for mesh area 2, and repeat the previous procedure for mesh areas 2 through 4. (See Figure 2.7). We have now defined the mesh areas. Next we must define the size of the mesh that we require.
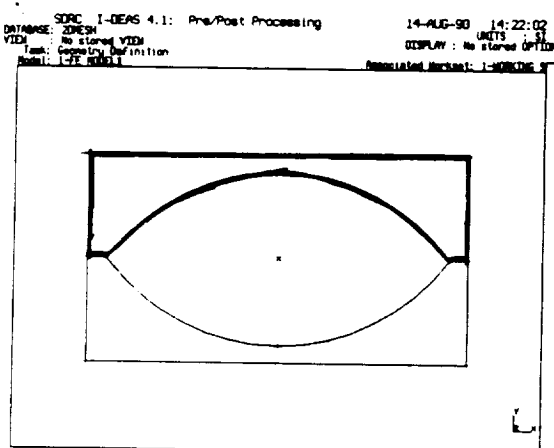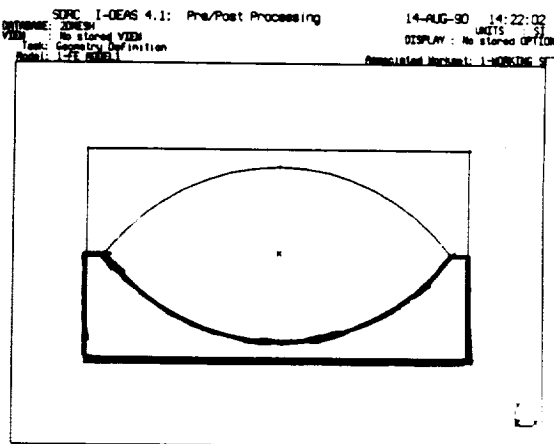

Figure 2.6

Figure 2.7

*See note at end of Tutorial

```
SELECT MENU#  /MEsh_size
SELECT MENU#  Free_Meshing
SELECT MENU#  Global_Element_Size
Enter global element length (0.0254)#   1


SELECT MENU#  /GEnerate_Mesh
Select generation type (SHELLS)#  <CR>
Pick Visible mesh-area
```

Select mesh-areas 1 and 2 by clicking the cursor around the middle of the mesh area. Enter Done when finished. A mesh should appear on the monitor.

Ok to keep nodes and elements? (YES)# <CR>

The mesh is now completed.

STEP #3: FORMATTING DATA OUTPUT

The next step is in getting the data we have just created to a data file we can use. We will create Trace lines so that we know what nodes occur at the boundaries of our geometry.

```
SELECT MENU#  TAsk
SELECT MENU#  MP                     (Model_Preperation)
SELECT MENU#  Trace_Lines
SELECT MENU#  CReate
Enter trace line name or no. (1-TRACE LINE1)#   Outer  Boundary
Pick Visible node
Related_to
Selected related entity type (ELEMENT)#   CUrve
Pick Visible curve
```

Now select the curves that form the outer rectangular boundary by clicking on each of them with the cursor. (See figure 2.8). They should become highlighted. Enter Done when you have finished.

```
Pick Visible curve
Done
Pick Visible node
Done
```

Repeat this procedure for the inner conduction boundary. (See figure 2.9). We are now at our final step, writing out our data file. These files are called universal files. They contain more information in them than we really need. We can, however, control to an extent what we output.
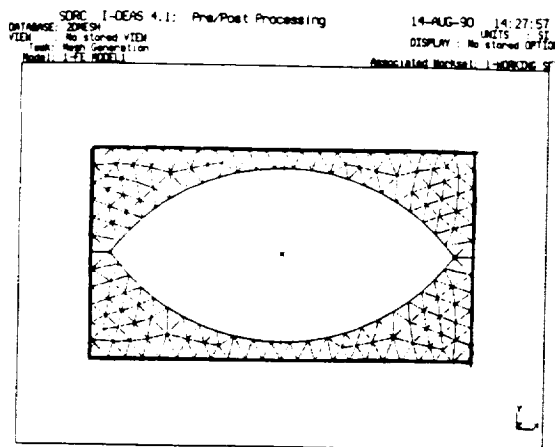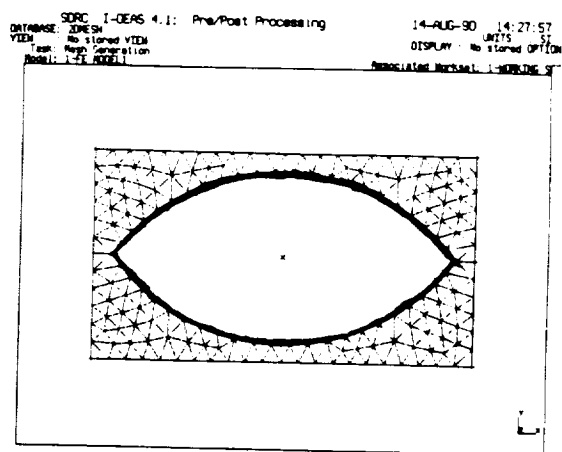
Figure 2.8

Figure 2.9

SELECT MENU#   Universal_File
SELECT MENU#   ENtity_Read/Write
SELECT MENU#   OFf_(all)
SELECT MENU#   FE_entity_data_sw

Enter Finite element entity data switch ON or OFF (ON)#
SELECT MENU#   Trace_line_data_sw

Enter Trace line data switch ON or OFF (ON)#   <CR>
SELECT MENU# !   (backup to Universal_File menu)
SELECT MENU# Write

Enter Universal filename (DEFAULT)#   <CR>
Ok to write new file? (YES)# <CR>

We have now completed our task. To save all of your work on disk enter the SAVE
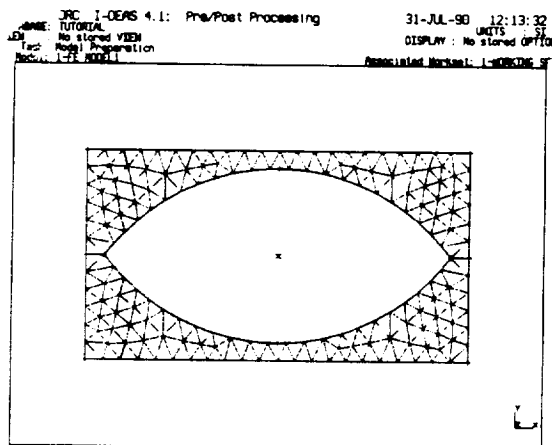command.   To get an output plot of the geometry use the global command HC or hard
copy.

Figure 2.10

*Material   Properties

If a different material property than the default value is desired, we can modify IDEAS default table for material properties to meet our requirements. Every mesh-area that we create has a material property table. We can, however, create different tables for each mesh area. A material property table consists of a series of around fourteen different properties such as shear strength, elasticity, etc.. We can however simply reassign these to meet our needs by entering our data, epsilon for example, into an elasticity slot. When it is output into the universal file, the data is output as a series of numbers, each representing a property in the table. The data does not include the property name. Therefore we can read these numbers using our code, assuming we know which ones they are, and use them for analysis.

Example:

Enter MATERIALS PROPERTY TABLE name or no. (1-MATERIAL_PROPERTY_TABLE1)#
MATERIAL#1
Ok to use default values? (YES)# No

From here, enter the property number you would like to assign, and the accompanying value. For example if we needed a mu and an epsilon each with a real and an imaginary part, the first four values of the table would be used.

14

Mesh Generation
* A 3-D Example

Task: In this example we will create a three dimensional geometry using IDEAS Geomod (See figure 3.1). This geometry will be accessed from Solid Modeling and then transfered to Engineering Analysis where it will be modified using the Pre/Post Processing Module. Our final geometry will then be broken into closed loops call mesh-areas. These mesh-areas in turn will be used to create mesh-volumes which will be automatically meshed by the computer.

Preparation: Enter IDEAS, create a model file, and enter IDEAS Solid Modeling family, and Object Modeling module. (See Getting Started. See Solid Modeling tutorial for more information on IDEAS Geomod).

Step #1:   Geometry Definition

SELECT MENU# TAsk
SELECT MENU# PRofile_2D
SELECT MENU# CReate_profile
Pick 2D or 3D point
Key_in
Enter X,Y of point (0.0,0.0)# .5,0
Enter X,Y of point (.5,0)# x
Enter X,Y of point (.5,0)# 1,0
Enter X,Y of point (1,0)# 1,2
Enter X,Y of point (1,2)# 0,2
Enter X,Y of point (0,2)# 0,1
Enter X,Y of point (0,1)# .5,1
Enter X,Y of point (.5,1)# c
SELECT MENU# AU
SELECT MENU# TAsk
SELECT MENU# Object_3D
SELECT MENU# Create
SELECT MENU# Revolve
Enter angle about Y (360.0)# 9 0
minimum radius change = 0.0
Enter translation along Y axis, change in radius (0.0,0.0)# <CR>
Select edge generation option (PERCENT_OF_CHORD_LENGTH)# <CR>
Enter maximum % of chord length deviation (11,0)# <CR>
Enter number of sections (4)# <CR>
Enter OBJECT color name or no. (11-RED)# <CR>
SELECT MENU# STOre
Enter OBJECT name or no. (1-OBJECT1)# <CR>
SELECT MENU# NM   (New model)
SELECT MENU# CF   (Change family)
SELECT MENU# EA   (Engineering Analysis)
SELECT MENU# P   (Pre/Post Processing)
SELECT MENU# TAsk
SELECT MENU# Geometry_definition
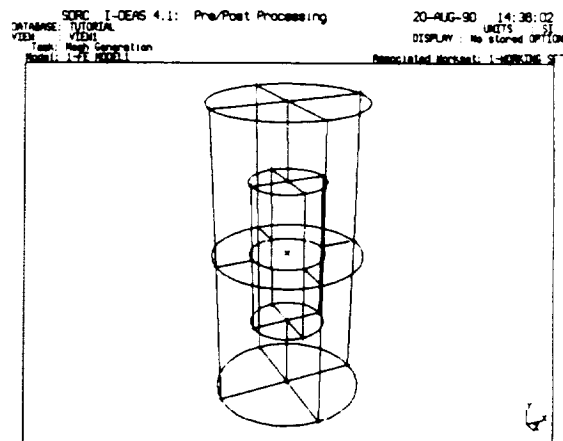SELECT MENU# CReate_wire
SELECT MENU# OB   (From Object)

Figure 3.1

SELECT MENU# Get_object
Enter WORKING_SET name or no. (1-WORKING_SET1)# <CR>
Enter BIN name or no. (1-MAIN)# <CR>
Enter OBJECT name or no. (DIRECTORY)# <CR>
Enter OBJECT name or no.# 1
Enter curve tolerance (0.00032)# <CR>
SELECT MENU# DRaw
SELECT MENU# /View
SELECT MENU# RM
Enter X,Y,Z degrees (0.0,0.0,0.0)# 25,0,0
SELECT MENU# /LAbel
Pick visible point or curve
*   (all)
Pick visible point or curve
Done
SELECT MENU# COpy_and_orient
SELECT MENU# Rotate
Pick Visible point or curve
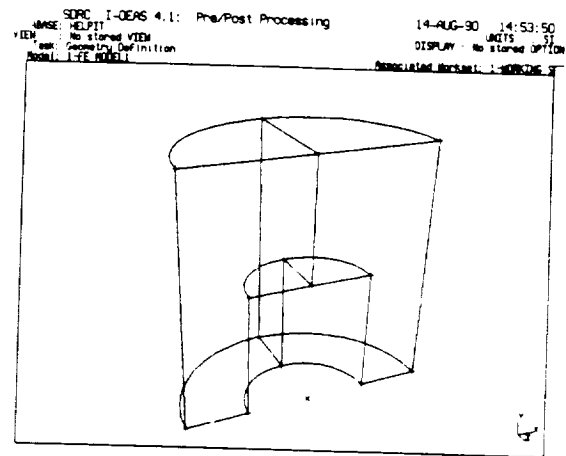Label_range
Enter point or curve label range#

Figure 3.2

Here we will enter a the series of curves C1-C6 and C12-C15. Each one is entered individually. When done type **D** for done.

Pick pivot point
Pick VISIBLE
Key_in
Enter local X,Y,Z (0.0,0.0,0.0)# <CR>
Enter local X,Y,Z rotation (0.0,0.0,0.0)# 0,90,0
Pick visible point or curve
Done
SELECT MENU# AU
SELECT MENU# /View
SELECT MENU# RM
Enter X,Y,Z degrees (0.0,0.0,0.0)# 0,25,0   (See Figure 3.2)
SELECT MENU# /LAbel
Pick visible point or curve
*   (all)
Pick visible point or curve
Done
SELECT MENU# COpy_and_orient
SELECT MENU# Rotate
Pick visible point or curve
*   (all)
Pick pivot point
Pick VISIBLE
Key_in
Enter local X,Y,Z (0.0,0.0,0.0)# <CR>
Enter local rotation (0.0,0.0,0.0)# 0,0,180
Pick visible point or curve
Done
SELECT MENU# AU (See figure 3.3)
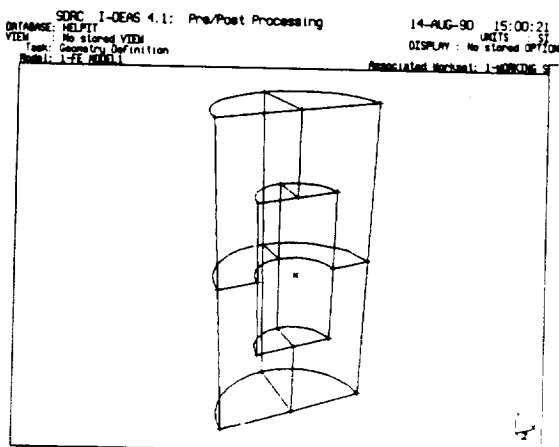SELECT MENU# /DELete

Figure 3.3

Pick visible point or curve
Label_range

Enter point or curve label range#

Here we will enter the series of curves: C1, C7, C12, C15, C16, C19, C20, C25, C45.
Each is entered individually.   Type D for done when this is completed.

Ok to delete 9 entitie(s)? (YES)# <CR>
SELECT MENU# /LAbel
Pick visible point or curve
*   (all)
Pick visible point or curve
Done
SELECT MENU# COpy_and_orient
SELECT MENU# Rotate
Pick visible point or curve
Label_range
Enter point or curve label range#

Here we will enter the series of curves: C3, C17, C14, C18, C4, C13, C5, C36, C2,
C35, C39, C32, C37, C34, C48, C50, C38, C33, C47, C46, C49. As before each is entered
individually and you should type D for done when you have this is complete.

Pick visible pivot point
Pick VISIBLE
Key_in
Enter local X,Y,Z (0.0,0.0,0.0)# <CR>
Enter local X,Y,Z rotation (0.0,0.0,0.0)# 0,180,0
Pick visible point or curve
Done
SELECT MENU# AU

At this point we have completed defining our geometry.   (See figure 3.4).   It is
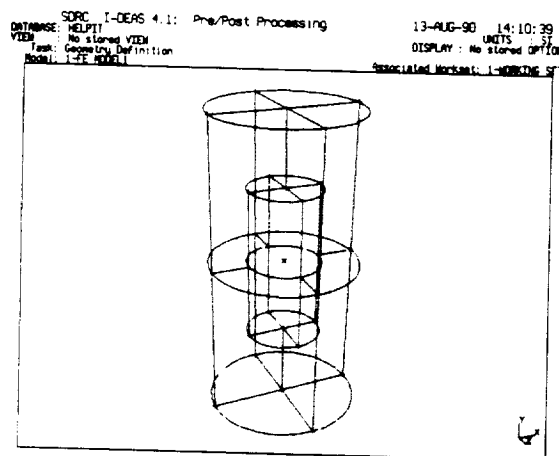recommended that you save your work so far with the SAVE command.

Figure 3.4

17

Step #2:   Mesh Areas

Using the same procedure as in the 2-D mesh example, we will define mesh areas from our geometry. These entities will then be used to define mesh volumes with which the computer will generate a mesh using solid elements. As you can see, our geometry contains eight identical sections or volumes. (See figure 3.4). Our first step is to define the closed looped areas that define each section. There are atotal of 44 meshareas in this geometry. (Note: Some areas are shared by two sections. These areas need to be defined only once.) Use figures 3.5-3.11 to define the first set of mesh areas. Continue using symatry to define all 44 mesh areas. (If you have difficulty selecting a curve, see HELP, SPECIAL_TOPICS, SELECTING_ENTITIES or Appendix A).

```
SELECT MENU# /TAsk
SELECT MENU# MEsh_generation
SELECT MENU# MA          (Mesh  Areas)
SELECT MENU# CReate
Enter mesh-area start label, inc. (1,1)#   <CR>
Ok to use thin shell linear quadrilateral elements#   No
Select element family (THIN_SHELL)#   <CR>
Select element order (LINEAR)#   <CR>
Select element topology (QUADRILATERAL)#   Triangle
```

From here take the defaults until IDEAS ask for curve selection. Use the figures 3.5-3.11 for mesh area curve selection. After each sequence of curves entered for a mesh area, enter D for done.

Ok to pick a surface? (NO)# <CR>
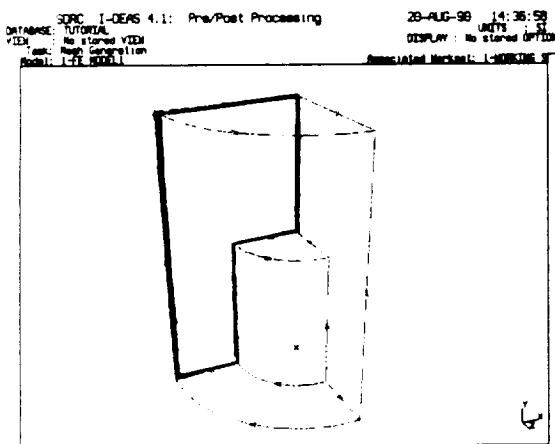
Figure 3.5
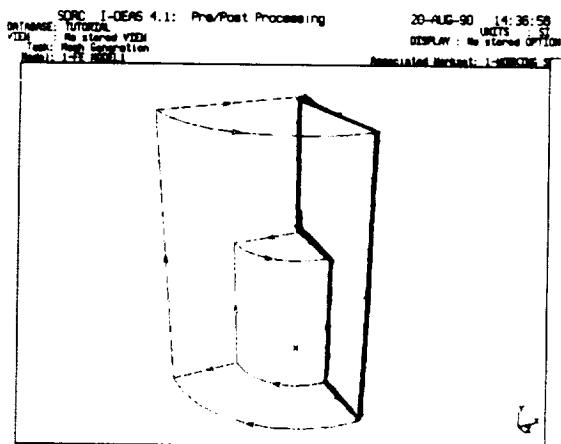
Figure 3.6
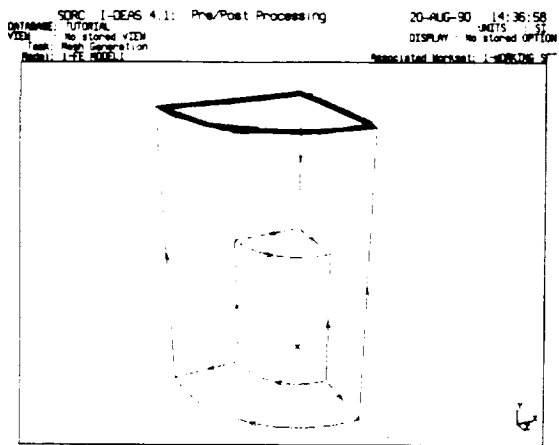
Figure 3.7
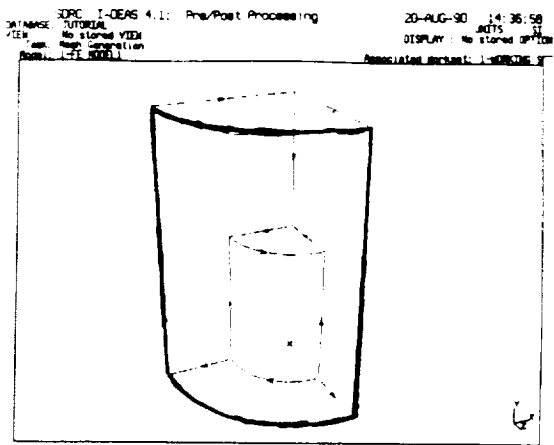
Figure 3.8
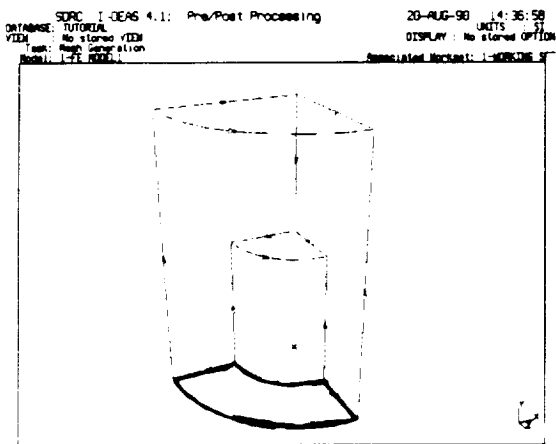
Figure 3.9

Figure 3.10

Figure 3.11

19
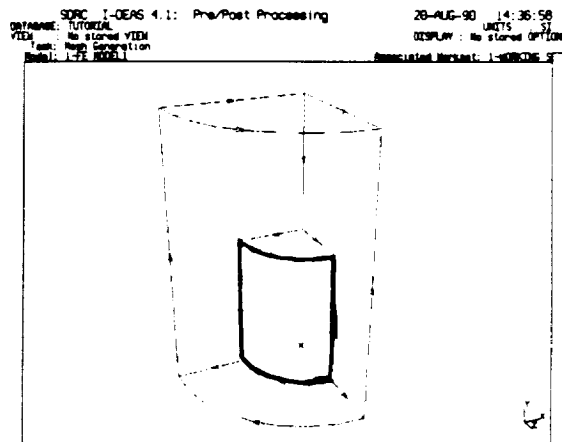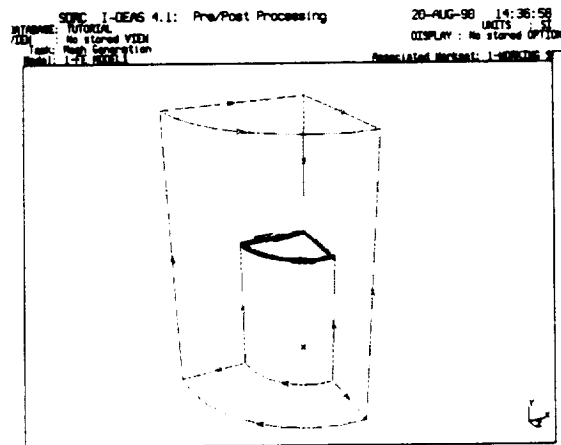
Step #3: Mesh Volumes

Once you have created all of the required mesh areas, you are ready to create mesh volumes.

SELECT MENU# MV
SELECT MENU# CReate
Enter mesh-volume start label, inc. (1,1)# <CR>
Ok to use thin shell linear quadrilateral elements# No

From here take the defaults until IDEAS ask for mesh-area selection. Use figure 3.12 to select the mesh-areas in the first volume. There will be a total of eight mesh-volumes, each symetric to the one shown in figure 3.12.
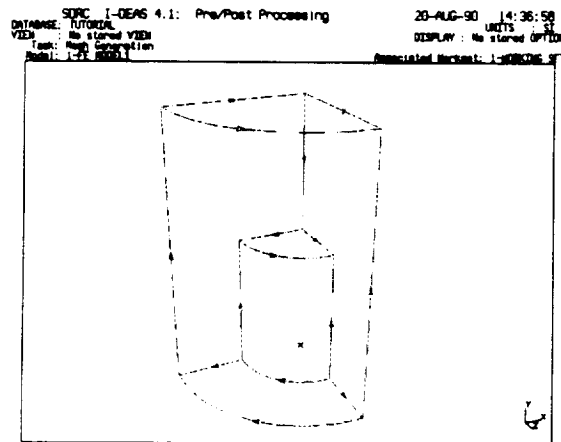
Figure 3.12

Step #4:   Mesh Generation

Our next step is to define the size of our mesh.   Once this is completed, a mesh with solid elements can be produced.

SELECT MENU# /MEsh_size
SELECT MENU# Free_meshing
SELECT MENU# Global_element_size
Enter Global element size (.02542)# .5
SELECT MENU# /GEenerate_mesh
Select Generation type (SHELLS)# Solids
Select mesh-volume
Pick Visible
Label_range

Here enter mesh-volumes 1-8 by number.   When done enter D for done.

SELECT OPTION# Done_with_preview
Ok to continue processing? (YES)# <CR>
Ok to keep nodes and elements? (YES)# <CR>

You have now completed your 3-D mesh.   Enter the SAVE command to store your work.

APPENDIX A

Widely Used Global Commands

SA-SAVE: The SAVE command converts the temporary files initialy created by IDEAS when the user enters the program, to permanent files.

EX-EXIT: The EXIT command allows the user to exit the IDEAS program. All changes since the last SAVE command will be lost.

ZM-ZOOM: The ZOOM command is used to scale the display so that a rectangular area of the screen, which you define, fills the entire screen.

HC-HARD COPY: The HARD COPY command converts the display window to a bitmap file which is written onto the current working directory. This file may be then printed using the *prf* command once the user is back in the UNIX operation system.

AU-AUTOSCALE: The AUTOSCALE command scales the display so that the entire model fits on the screen.

HE-HELP: The HELP command may be used to find information about any command in the current or global directories. A Special Topics section is also included in the help menu, which contains descriptions on selecting entities, and other topics.

Creating Entities

To create an entity such as a node or element, you use the CREATE command in the menu that is named for the entity you want to create. For example, to create a node, use the NODE menu. In the NODE menu, you will find the CREATE command. A CREATE command is offered by every entity's menu.

After you create an entity, you can modify its attributes, using the MODIFY command offered by the entity's menu. For example, after you create a node, you can modify its coordinates or its color. A MODIFY command appears in every entity's menu.

Besides modifying an entity, you can also delete it, using the DELETE command in the entity's menu. For example, the NODE menu offers a DELETE command. Other entity management commands are offered in each entity's menu.

Selecting Entities for Modeling Operations

As you create and work with entities in your model, many commands require you to select the entities on which you want to perform modeling operations. For example, one way to select a group of nodes to modify is to select the nodes associated with select elements. To let you select these nodes, a menu appears that offers this option for selecting nodes.

The options in this menu depend on the kind of entities being selected:

● Entities other than element free edges and free faces

● Element free edges

● Element free faces

Selecting Entities Other Than Element Free Edges and Free Faces

The following menu appears when you're prompted to pick entities for an operation (unless the prompt is to pick free edges or free faces of elements):

```
V-VISIBLE
L-LABEL_RANGE
G-GROUP
A-ATTRIBUTE
S-SCREEN_AREA
E-ENCLOSED_VOLUME
R-RELATED_TO
O-HIGHLIGHT
Y-DRAW
Z-CHANGE_VIEW
*-ALL
D-DONE
!-BACKUP
$-ABORT
```

Pick visible (entity)

You can select the entities to be modified by first choosing the method you want to use to select them. After you choose a selection method, the program prompts you to enter the necessary information. By entering this information, you select one or more entities to be processed. Next, you can select more entities, using either the same selection method or a different one.

The following list describes each method:

● V-VISIBLE

This option prompts "Pick visible (entity)". Use the terminal's screen picking device to pick entities visible on the screen.

24

- **LABEL_RANGE**

  This option prompts "Enter (entity) label range". Enter the label of the entity you want to select. To select a series of entities, enter the labels of the first and last entities in the series plus an increment, if you need an increment other than one. An increment may be positive negative. For example, you can enter 8,2,-6, selecting the entities labeled 8 and 2.

- **GROUP**

  This option prompts "Enter group name or no.". Enter the name or number of a stored group (including a group from the old stack), the processed group, or the current group. *T7 means that you are selecting all the entities of the requested type in the group.* For example if you are picking elements, all elements in the selected group are selected.

- **ATTRIBUTE**

  This option prompts "Choose attribute". Choose an attribute from the menu that accompar the prompt. These are attributes of the entity you want to select. This means you are select all of these entities that share the specified attribute.

- **SCREEN_AREA**

  This option automatically selects all the visible entities that appear inside a screen area that you specify. To define a screen area, you can choose one of four methods:

  D -DIAGONAL_POINTS

  This method prompts for two screen points. These two points will define the corners of a rectangle on the screen. All the entities visible in that rectangle are selected. When you de the second point of the rectangle, you have the opportunity to select or reject the entities which cross the area boundary. If you select P-PART_INSIDE, which appears at the promp for the second point, those entities partially inside the area are selected.

  IP-INSIDE_POLYGON / OP-OUTSIDE_POLYGON

  INSIDE_POLYGON and OUTSIDE_POLYGON prompt for 3 to 50 screen points. These points connected to define a polygon. All the entities visible and inside/outside of that polygon at selected. When you define each point of the polygon, you have the opportunity to select c reject the entities that cross the polygon boundary. If you select P-PART_INSIDE, those entities partially inside the polygon are selected.

  IC-INSIDE_CIRCLE / OC-OUTSIDE_CIRCLE

  INSIDE_CIRCLE and OUTSIDE_CIRCLE prompt for a point that defines a circle center and point to define the circle radius. All the entities visible inside/outside of the circle are selected. When you define the radius of the circle, you have the opportunity to select or reject the entities that cross the screen boundary. If P-PART_INSIDE is selected, those ent partially inside the circle are selected.

  S -SIDE_OF_LINE

  This method prompts for two points that define a line across the screen. After picking tw points, you are prompted to pick which side of the line from which to select entities. All entities visible on the side of the line you pick are selected. When you define the side of line to use, you have the opportunity to select or reject the entities that cross the line. If pick S-SELECT_ON_LINE, those entities that cross the line are selected.

> ═══════ **NOTE** ═══════
>
> If dynamic viewing is on, don't rotate or zoom the viewport after you begin defining a screen area (for example, by using dials).

```
╔═══════════════════════════════════════════╗
║  ═══════════ NOTE ═══════════              ║
║  If you're using SCREEN_AREA to pick        ║
║  face pressures, the screen area must       ║
║  include the entire element on which the    ║
║  face pressure exists. If the entire        ║
║  element is not within the screen area,     ║
║  the face pressure will not be picked.      ║
╚═══════════════════════════════════════════╝
```

- ENCLOSED_VOLUME

  This option automatically selects all entities inside a volume that you define, with respect to model coordinates. *All entities in the volume are selected regardless of whether they are visible on the screen.*

  You can choose from three methods to define a volume, listed below. After defining the volume, you are prompted "Ok to include (entities) which cross volume boundary?". This lets you include or reject the entities partially inside the volume.

  D -DIAGONAL_POINTS

  This method prompts you to pick two model locations. These two points define the diagonals of a volume. If the volume was defined in a cartesian coordinate system, the volume will be rectangular. If the volume was defined in a cylindrical coordinate system, the volume will be cylindrical. If the volume was defined in a spherical coordinate system, the volume will be spherical.

  CE-CENTER_&_DELTAS

  This method prompts you to pick a volume center point and enter the length of the volume in each of the three coordinate directions.

  CO-COORDINATE_MIN_&_MAX

  This method lets you define a volume by entering the minimum and maximum dimensions of the volume in each of the three coordinate directions.

- RELATED_TO

  This option prompts "Pick related entity type". Choose a related entity from the menu that accompanies the prompt. This means that you selecting all of these entities related to the entities that you specify. For example, if you are selecting elements, you could pick nodes, so that you effectively select all elements related to the nodes you pick.

- HIGHLIGHT

  This option lets you highlight the entities which you have already selected. For example, if you select 50 nodes by a label range and select 0-HIGHLIGHT, these 50 nodes are highlighted on the screen.

- DRAW

  This option is the same as the global DRAW command. If the current display was generated by a SKETCH_GROUP command, this option repeats the SKETCH_GROUP command.

- CHANGE_VIEW

  This option gives you direct access to the VIEW menu. If the AUTO_DRAW switch is on, the display is automatically redrawn.

- ALL

  This option selects all occurrences of this entity type. For example, you can select all elements in a model.

- DONE

  This option terminates the entity selection process.

Setting Defaults for Points

For points, you can set defaults for the following attributes:

Label: number assigned to the next point created.

Increment of label: increment used to compute the label of the next point created.

Color: color of the next point created.

Symbol: symbol used to display the next point created.

Generation coordinate system: the coordinate system used to generate points based on a point's location.

To set defaults for points, use the DEFAULTS_POINT command in the CREATE_WIRE.POINT menu.

Setting Defaults for Curves

For curves, you can set defaults for the following attributes:

Label: number assigned to the next curve created.

Increment of label: increment used to compute the label of the next curve created.

Color: color of the next curve created.

Line style: type of representation used to display the next curve created (solid or dashed).

Line weight: thickness of line used to display the next curve created.

Order: the order of the polynomial used to compute the next spline created.

Start symbol: the symbol displayed at the beginning of the next curve created.

End symbol: the symbol displayed at the end of the next curve created.

Center symbol: the symbol displayed at the center of the next circle or arc created.

Defining points symbol: the symbol displayed at the location of each point used to define the next spline created. The default is NONE.

Line length: the method used to define the length of a line. Line length can be:

Implied: Line length is set to the value appropriate for the creation method you're using. For some creation methods, this may mean the line length is infinite.

From/To: You must specify both end points of the line. The starting point of the line is found by a normal projection from the first point you select (during line creation), to the new line. The ending point is found by a normal projection from the second point you select, to the new line.

Infinite: unbounded lines are created.

Merge option: how existing curves are handled when you merge curves (delete the old curves, or retain the old curves).

Split option: how existing curves are handled when you split curves (delete the old curves, or retain the old curves).

To set defaults for curves, use the DEFAULTS_CURVE command in the CREATE_WIRE menu. If you use the RESET_DEFAULTS command, all default attributes revert to their original status, as when you first opened the new Model file.

Creating Points

A point is a location in 3D space. Each point is defined by the following attributes:
- Number (label)
- Symbol used to display the point
- Three coordinates that define its location with respect to the work plane active when the point was created
- Color

Each point has a unique definition, because no two points in a working set can have the same number.

To create points, use the POINT command in the CREATE_WIRE menu. It offers a menu that lets you pick the locations where you want to create points. It offers a menu that lets you choose how you want to create points:

POSITION

Create points at user-define locations. POSITION offers a menu that lets you pick the locations where you want to create points. I-DEAS offers several options for selecting point locations. For an explanation of these options, see "Using the Entity Selection Menu" in this chapter.

BETWEEN_POINTS

Create points by interpolating between existing points. First, select a set of one or more points; this is called the first master set. Then, select a corresponding second master set of points; for each point in the first set, you select a corresponding point in the second set. Next, enter the number of points you want to generate between each pair of points. Finally, enter a starting label and increment for the points to be generated, and select the coordinate system you want to use to generate the points.

ON_SURFACE

Create points that lie on a surface. First, select the surface on which you want to create points. Then, select the option you want to use for defining point locations:

CURSOR_LOCATION

At screen locations on the displayed surface. If the surface is visible in the display, select any location on the surface. At each location, a point is created by projecting the screen location to the displayed surface.

PARAMETRIC_VALUES

At parametric coordinates on the surface. Enter the parametric coordinates *(s,t)* of the point. Pre/Post Processing creates a point at the surface location defined by the coordinates.

SPECIFY_XYZ

At the XYZ coordinates of the point. Enter the XYZ coordinates of the point. Pre/Post Processing creates a point at the surface location *nearest* the XYZ coordinates that you enter.

ON_CURVE

Create points that lie on a curve. First, select the curve on which you want to create points. Next, select the locations on the curve at which you want the points to exist.

ON_NODE

Create points that are coincident with nodes. Select the nodes on which you want to create points.

REFLECT

Create points by reflecting the locations of existing points about a plane. First, select the points that you want to reflect. Then, enter a starting label and increment for the points to be generated. Next, define the plane about which you want to reflect the points. Finally, enter the distance from the plane within which you don't want to reflect points. For example, if you don't want to reflect points that lie on the plane, you could enter a value of 0.001 (assuming other points are less than 0.001 units from the plane).

28

COPY_USING_CS

*Create points by copying the locations of existing points and translating the new points. First,
select the points you want to copy. Then, enter the number of copies you want to create.
Next, enter a starting label and increment for the new points. Finally, select the coordinate
system you want to use the generate the points and enter a delta value for each of the three
coordinate directions. For example, to copy a point to a new location that's four units away in
the x direction, select the cartesian coordinate system and enter a delta value of x=4.*

Digitizing Points

You can digitize point locations from a tablet. To digitize points, use the DIGITIZE menu,
located in the CREATE_WIRE menu. For more information about digitizing, see "Digitizing Nodes
and Points" on page 3-12.

Creating Lines

A line is a straight curve. You can create two types of lines: bounded and unbounded. A
bounded line has a definite length, from one set of coordinates to another set of coordinates. An
unbounded line has infinite length.

Each line is defined by the following attributes:

- Number (label)
- Bounded flag
- Length (if bounded)
- Linestyle
- Color
- A starting coordinate and ending coordinate, for a bounded line, or two coordinates through
  which the line passes, for an unbounded line

Each line has a unique definition, because no two lines in a working set can have the same
number.

For most line creation commands, the type of lines created depends on the "line-length mode" in
which you're working. You can set the creation mode by using the CURVE_DEFAULTS menu, or
you can use the Q-LINE_LENGTH option in the menu offered when you select a line creation
method.

```
================ NOTE ================
The type of lines created (bounded or
unbounded) usually depends on the
mode in which you're creating lines.
```

To create lines, use the LINE command in the CREATE_WIRE menu. It offers the following ways
to create lines:

POINT_TO_POINT

*Bounded lines, connected end-to-end. Select the starting point; then, select the ending point of
the first line. Next, select the ending point of the next line. The ending point of the previous
line becomes the starting point of this line. Continue creating lines by selecting the next
ending point. For this command, all lines are bounded. You can start a detached line by
selecting NEW_LINE_START from the menu offered when you select POINT_TO_POINT.*

29

SINGLE_LINE

Bounded lines. Select the starting point and ending point for each line you want to create.

X_DIRECTION

Lines parallel to the x axis of the active work plane, passing through a location. Select the location through which the line must pass. For this command, the implied mode works like the infinite mode.

Y_DIRECTION

Lines parallel to the y axis of the active work plane, passing through a location. Select the location through which the line must pass. For this command, the implied mode works like the infinite mode.

Z_DIRECTION

Lines parallel to the z axis of the active work plane, passing through a location. Select the location through which the line must pass. For this command, the implied mode works like the infinite mode.

VECTOR_POINT

Lines parallel to a vector. Select the location through which the line must pass; then, define a vector to which you want the line to be parallel. For this command, the implied mode works like the infinite mode.

If you use this command to create lines in the infinite mode, you are prompted to select start and end locations through which the line should pass.

PARALLEL_TO_LINE_POINT

Lines parallel to an existing line, through a point. Select the line to which you want the new line to be parallel; then, select a location through which the line must pass.

If you use this command to create lines in the implied mode, the length of each new line is equal to the length of the selected line.

If you use this command to create lines in the infinite mode, you are prompted to select start and end locations through which the line should pass.

PERPENDICULAR_TO_LINE_POINT

Lines perpendicular to an existing line, through a point. Select the line to which you want the new line to be perpendicular; then, select a location through which the line must pass.

If you use this command to create lines in the implied mode, the length of each new line is equal to the length of the selected line.

If you use this command to create lines in the infinite mode, you are prompted to select start and end locations through which the line should pass.

PARALLEL_TO_LINE_AT_DISTANCE

Lines parallel to an existing line, separated by a user-specified distance. Select the line to which you want the new line to be parallel; then, key-in the distance from the original line to the new line. To specify the position of the new line, you can use either of two methods:

VECTOR_METHOD

You can define a vector from the selected line to the new line. With this method, the new line and the selected line can be on different planes.

SIDE_ON_WORK_PLANE

You can select on which side of the selected line the new line will be created. With this method, the new line is on the same plane as the selected line.

If you use this command to create lines in the implied mode, the length of each new line is equal to the length of the selected line.

If you use this command to create lines in the infinite mode, you are prompted to select start and end locations through which the line should pass.

TANGENT_THRU_POINT

Lines tangent to any arc or circle, passing through a user-specified location. Select the arc or circle to which you want the line to be tangent; then, select the location through which the line must pass.

At this point, one or two infinite tangent lines are drawn. You can select either of the lines, but not both. The line you select is kept, while the other line is discarded.

If you use this command to create lines in the implied mode, the length of each new line passes from the first location you select (the *through* location) to the *tangent* location.

If you use this command to create lines in the infinite mode, you are prompted to select start and end locations through which the line should pass.

TANGENT_TO_2_ARCS

Lines tangent to any two arcs or circles, or a circle and an arc. Select the two arcs or circles to which you want the line to be tangent. Up to four infinite, tangent lines are drawn, from which you can select one that you want to keep.

If you use this command to create lines in the implied mode, the length of each new line passes from a location tangent to the first arc or circle, to a tangent location on the second arc or circle.

If you use this command to create lines in the infinite mode, you are prompted to select start and end locations through which the line should pass.

Creating Circles

A circle is a closed arc. Each circle is defined by the following attributes:

- Number (label)
- Linestyle
- Color
- Coordinates of center, with respect to the active work plane
- Coordinates of one location through which the circle passes, with respect to the work plane active when the circle was created
- Unit normal
- Radius

Each circle has a unique definition, because no two circles in a working set can have the same number.

To create circles, use the CIRCLE command in the CREATE_WIRE menu. It offers the following ways to create circles:
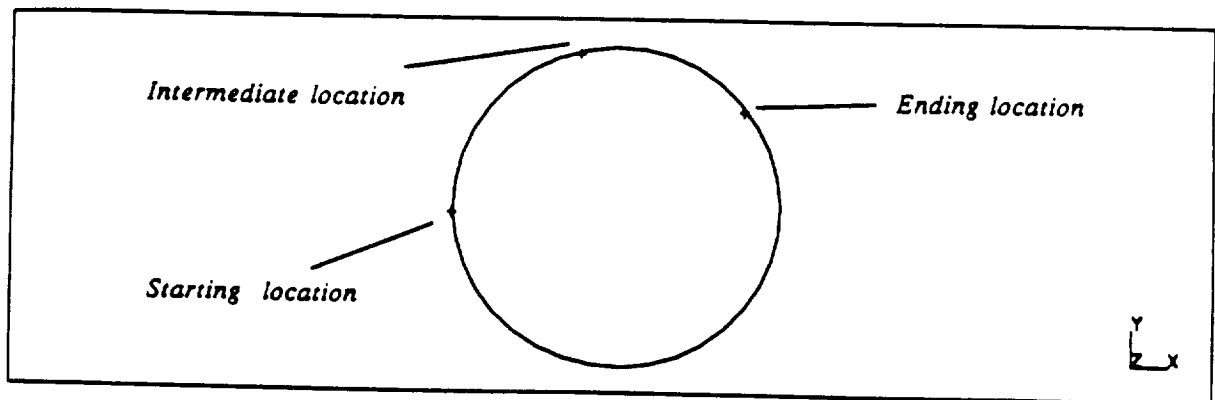
CENTER_RADIUS

Single circle. Select a point and enter a radius. The circle is created in the active work plane (local or global), or in a plane parallel to the active work plane.
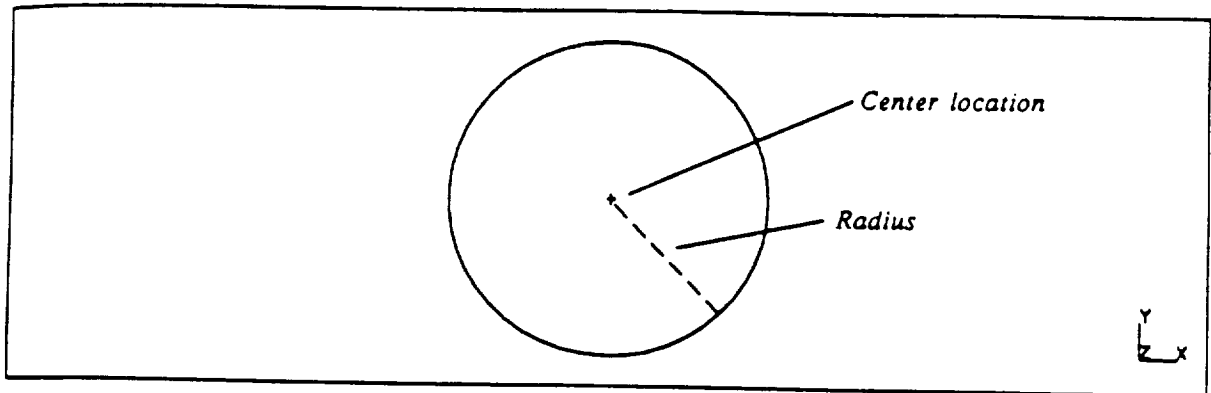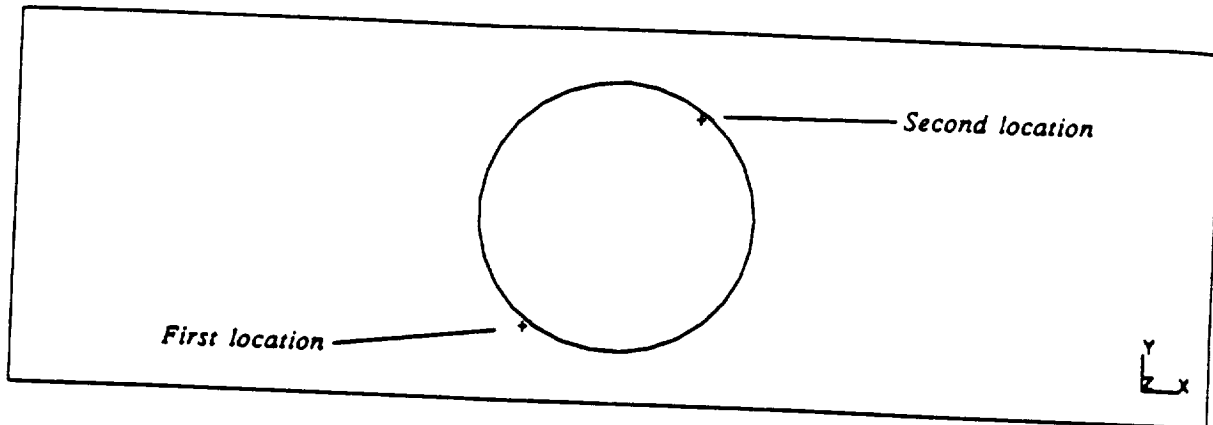

THRU_3_POINTS

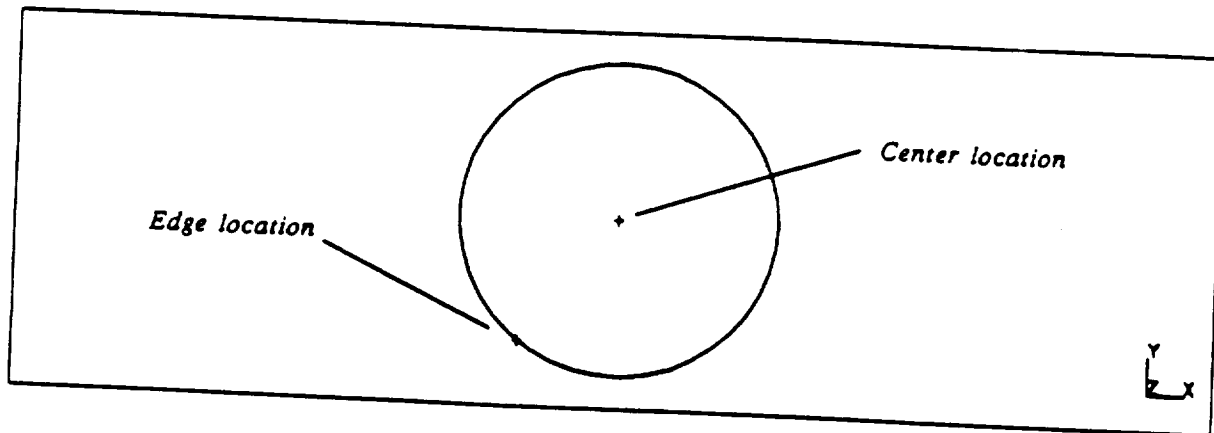*Single circle.* Select three points through which the circle must pass.

THRU_2_POINTS

Single circle. Select two locations through which the circle must pass. If both locations lie in the active work plane, the circle is created in the active work plane. If either or both locations don't lie in the active work plane, the locations are used to compute a center location and compute a radius. The circle is oriented in the plane parallel to the active work plane, passing through a center location.

First location

Second location

CENTER_EDGE

Single circle. Select a center location and a location through which the circle must pass. If both locations lie in the active work plane, the circle is created in the active work plane. If either or both locations don't lie in the active work plane, the locations are used to compute a radius. The circle is oriented in the plane parallel to the active work plane, passing through a center location.

Edge location

Center location

33

CENTER_2_POINTS

Single circle. Select a center location and two locations through which the circle must pass. If the "through" locations are not equidistant from the center location, the radius is determined by the first "through" location.
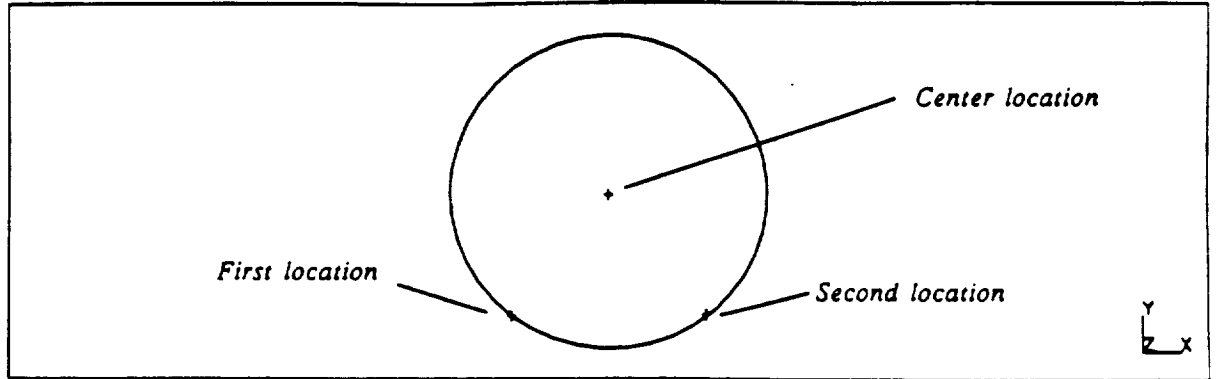
CONC_CIR_CENTER

Multiple concentric circles. Select the common center location, then key-in the radius for each concentric circle. The circles are created in the active work plane.
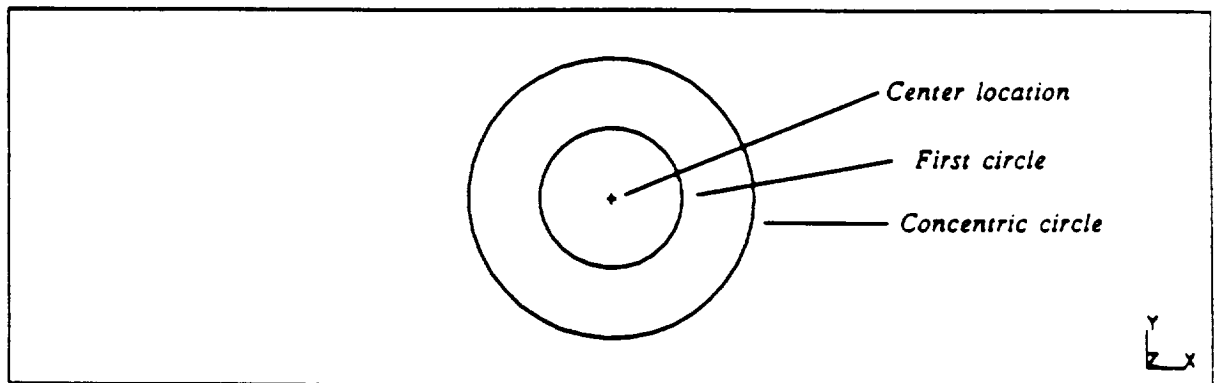
CONC_CIR_TO_ARC

Multiple concentric circles, about an existing circle or arc. Select an arc or circle; then, key-in the radius for each concentric circle. The circles are created in the active work plane. Their common center location is the center of the selected arc or circle.

34
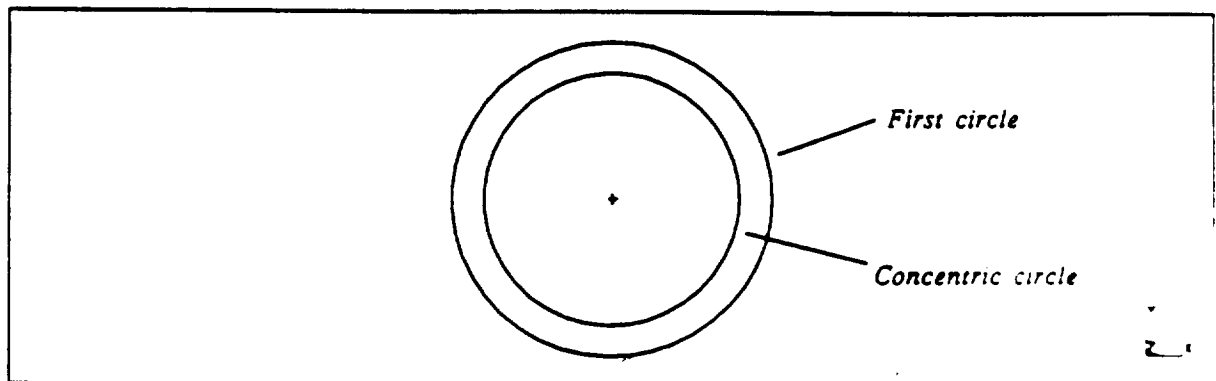
TANGENT_THRU_POINT

Single circle tangent to a curve. Select a curve; then, select a location through which the circle must pass. The tangent location is determined by a normal projection from the "through" location. These two locations are used to compute the center location and the radius. The circle is created in the plane parallel to the active work plane, passing through the center location.

Location through which circle passes

Curve to which circle is tangent
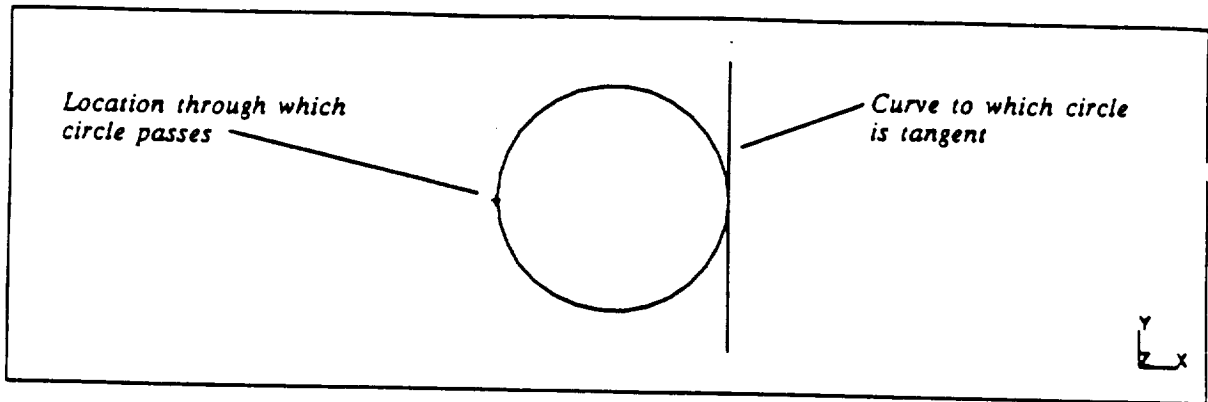
TAN_TO_2_CURVES

Single circle tangent to two curves. Select two curves; then, select a point on one of the curves, to which you want the circle to be tangent. If more than one tangent circle can be created, you can select which circle you want to keep.

First circle

Second circle

Location of tangency

35
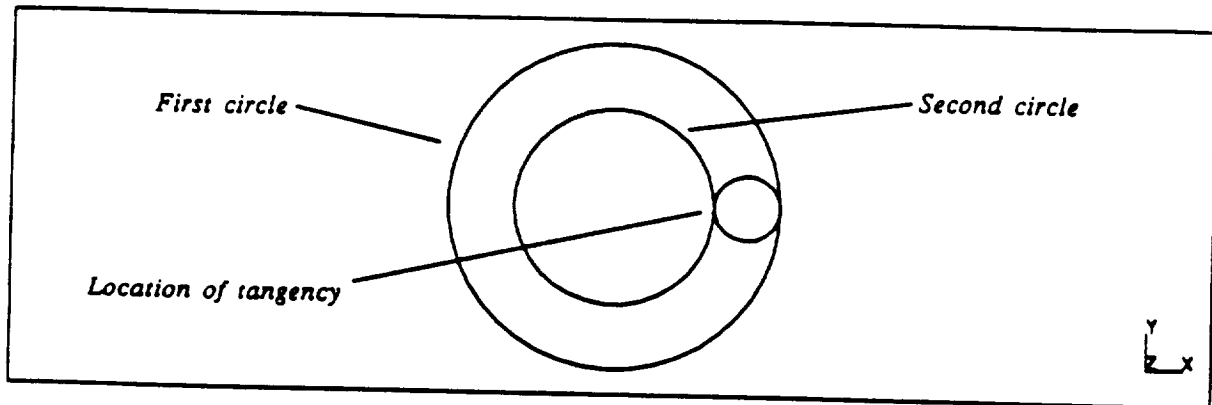
Creating Arcs

An arc is an open curve with a constant radius. Each arc is defined by the following attributes:

- Number (label)
- Linestyle
- Color
- Coordinates of center, with respect to the active work plane
- Coordinates of the starting location of the arc, with respect to the work plane active when the arc was created
- Coordinates of the ending location of the arc, with respect to the work plane active when the arc was created
- Unit normal
- Radius

Each arc has a unique definition, because no two arcs in a working set can have the same number.

To create arcs, use the ARC command in the CREATE_WIRE menu. It offers the following ways to create arcs:

GENERAL_ARC

General arc. Select the center of the arc. The result is an arc with a curvature of 45 degrees and a radius of one unit (exception: if the system of units is MM, the radius is 10 units).

A menu lets you modify this arc before the arc creation process is complete.

THRU_3_POINTS

Single arc. Select three locations through which the arc must pass.

36
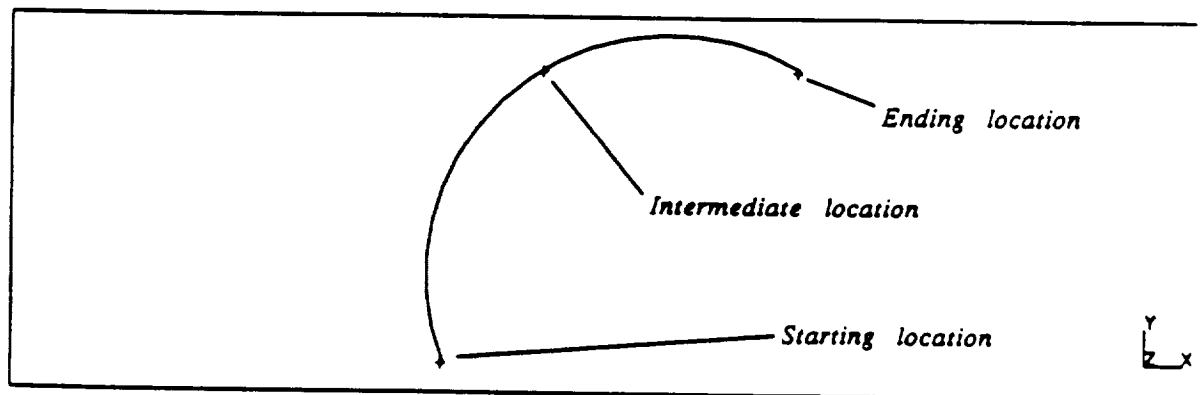
RADIUS_START_END

Single arc. Enter the radius of the arc; then, select two locations through which the arc must pass. Finally, select the approximate center of the arc you want to place the center of the arc. The resulting arc is constructed in a counter clockwise fashion, relative to the normal vector.

Side of arc for center

Ending location

Starting location

Y
Z X
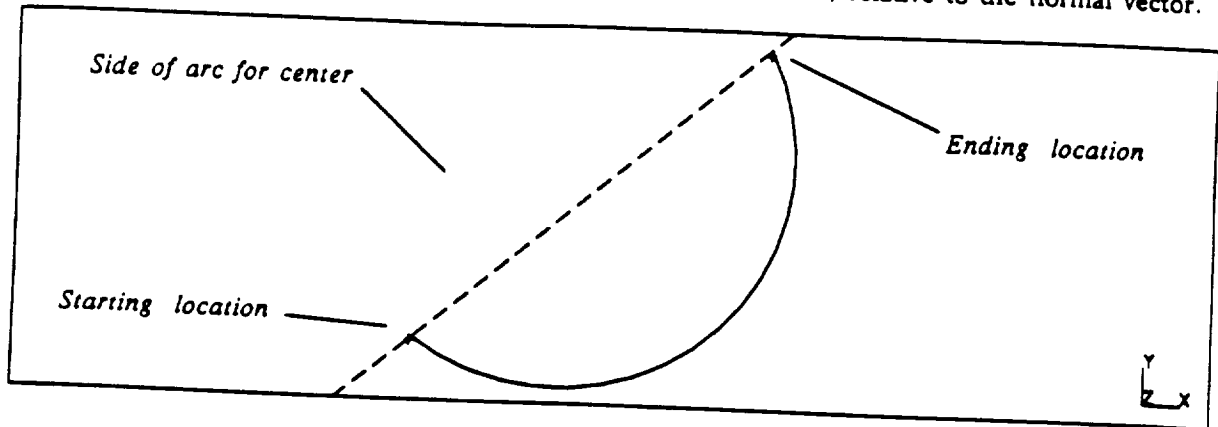
CENTER_START_END

Single arc. Select the location of the center of the arc; then, select the starting point and ending point of the arc. The resulting arc has a curvature that is less than or equal to 180 degrees. If the start and end locations are not equidistant from the center location, the radius is equal to the distance from the center to the start location.

Center location

Starting location

Ending location

Arc

Y
Z X
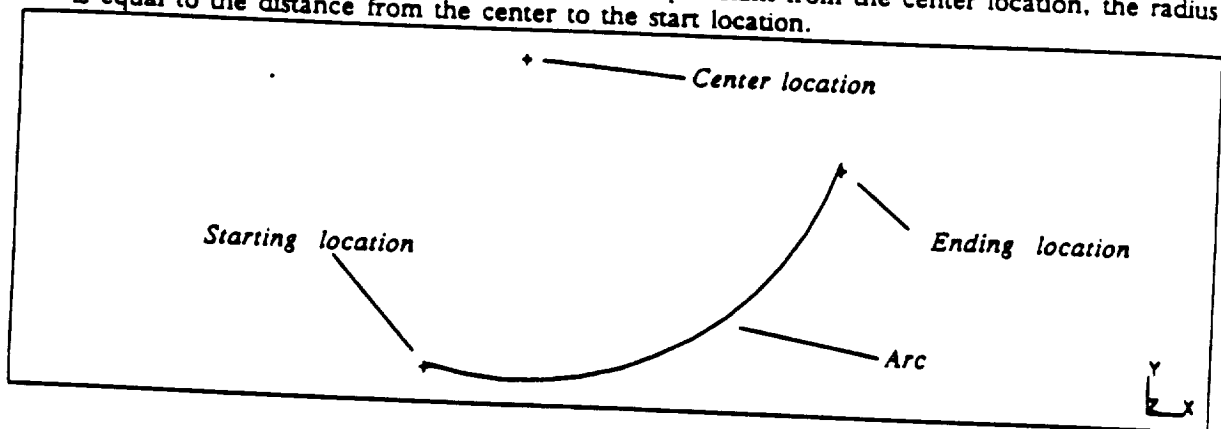
CONC_ARC_TO_ARC

Single concentric arc. Select the arc to which you want the new arc to be concentric; then, enter a radius for each new arc. To create the new arc, the center point, plane normal, and sweep angle of the existing arc are used.
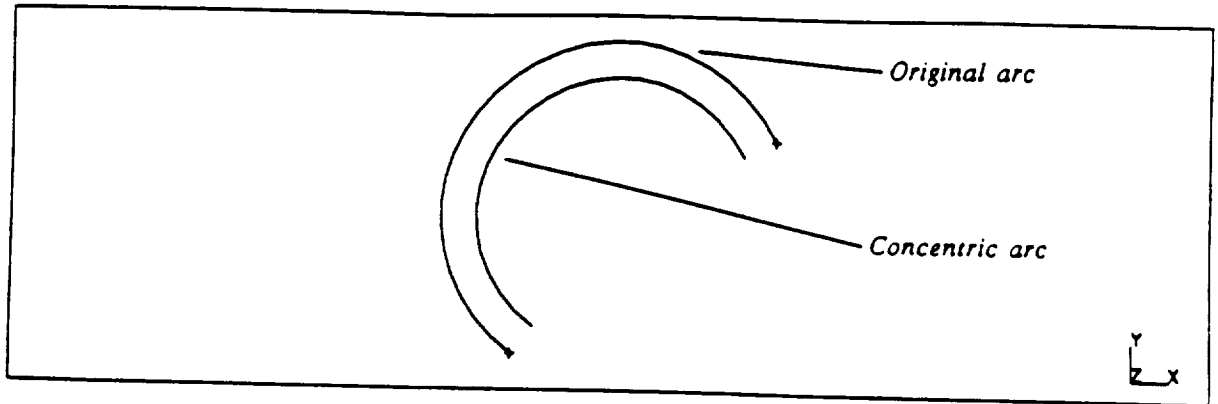
Creating Splines

A **spline** is a closed or open curve whose shaped is defined by a higher order polynomial. The order of the polynomial is set in the CURVE_DEFAULTS menu, located in the CREATE_WIRE menu. Its value can be from three to eight.

Each spline is defined by the following attributes:

- Number (label)
- Linestyle
- Color
- Coordinates of each location through which the spline passes, with respect to the work plane active when the spline was created

Each spline has a unique definition, because no two splines in a working set can have the same number.

To create splines, use the SPLINE command in the CREATE_WIRE menu. It lets you select the locations through which the spline must pass. To create a spline, you can select from three to 251 defining locations.

38
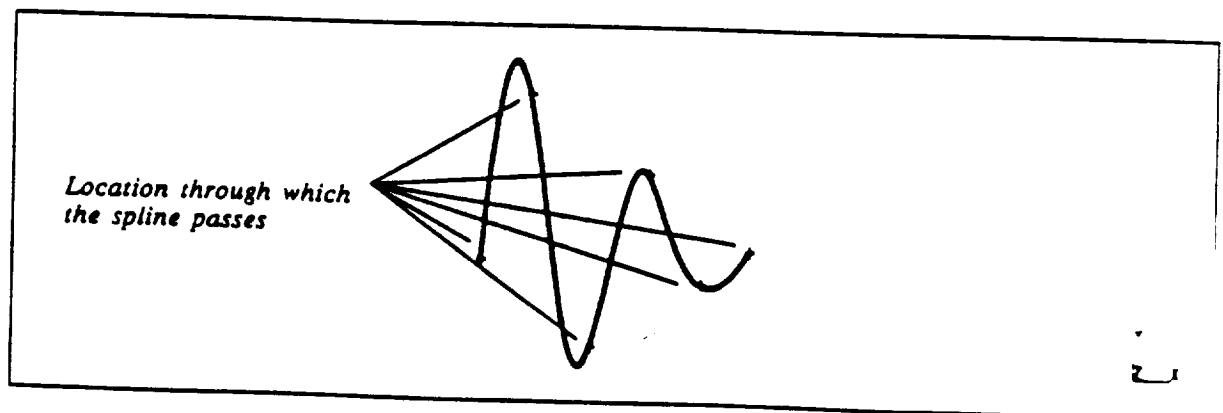
Thin Shell

A thin shell element is shown in Figure 8-13. A thin shell element is available with linear, parabolic, or cubic order. Its topology may be either triangular or quadrilateral.

Figure 8-13. Thin Shell Element

A thin shell element may have the physical properties shown in Table 8-18. For a listing of *material* properties, see the Appendix.

To create a thin shell element, pick the nodes required to define the element's connectivity, in the sequence shown in Figure 8-13.

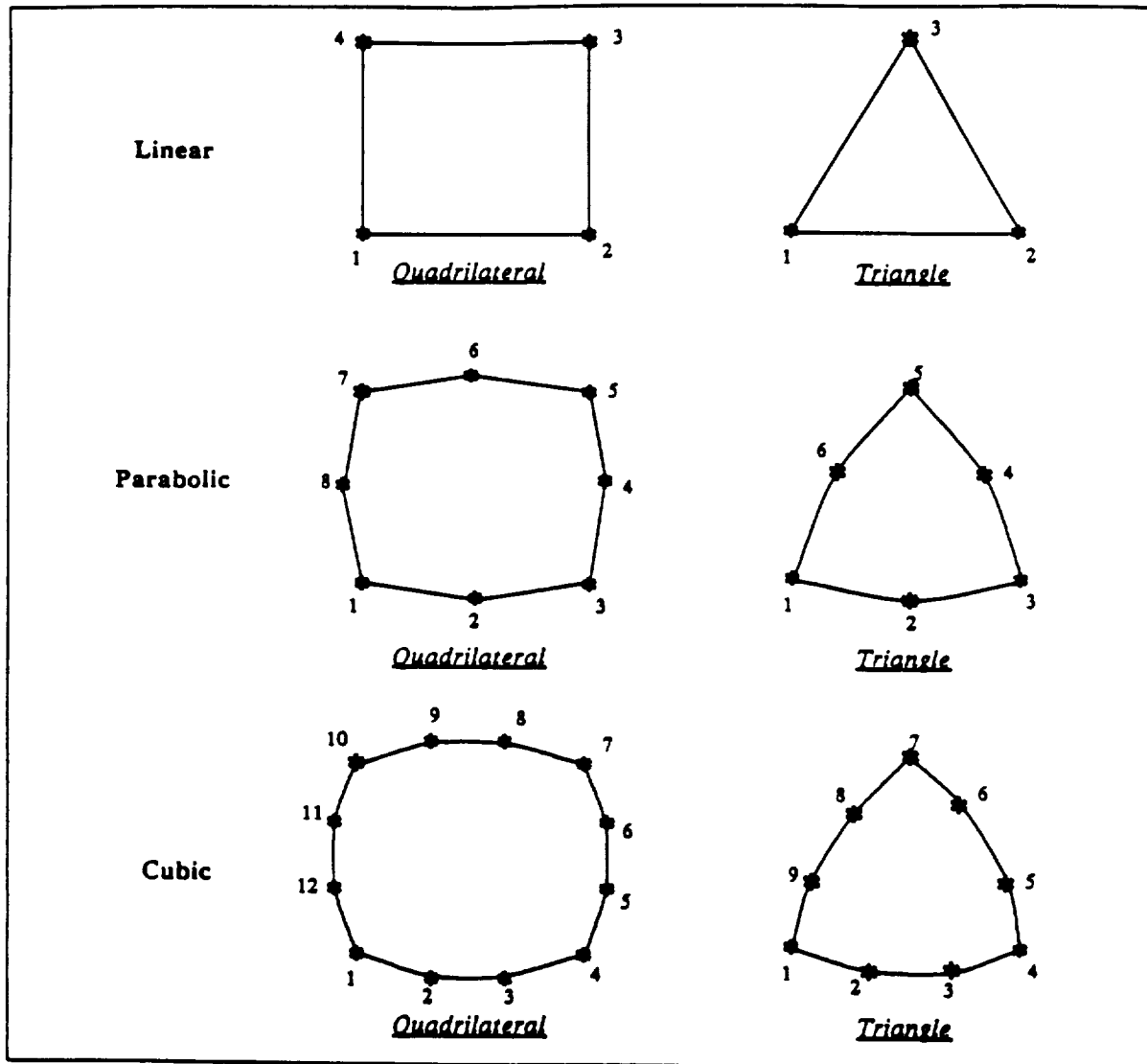|				
498	Cross-Sections		499	Joint
499	Joint		500	System Assembly Status
500	System Assembly Status		501	Component – Header
501	Component – Header		502	System Header
502	System Header		503	System Hierarchy Nodes
503	System Hierarchy Nodes		504	System Instance Orientations
504	System Instance Orientations		505	System – Animations
505	System – Animations		509	Geomod – Auxiliary Function Data
509	Geomod – Auxiliary Function Data		511	System – Rigid Bodies
511	System – Rigid Bodies		512	System Markers
512	System Markers		520	Object – Points
520	Object – Points		800	Working Set Header
800	Working Set Header		801	Working Set Curves
801	Working Set Curves		802	Working Set Points
802	Working Set Points			

I-DEAS Supertab Datasets:

Written

15	Nodes
18	Coordinate Systems
55	Data at Nodes
56	Data at Elements
57	Data at Nodes on Elements
71	Elements
82	Trace Lines
92	Surfaces
705	Mesh Areas
706	Mesh Volumes
731	Physical Properties
735	Node/Element – Geometry Associativity
743	Supertab Points
744	Supertab Curves
747	Material Properties
748	Associated Element Data
749	Model Header
752	Permanent Groups
753	Case Sets
754	Constraint Sets
755	Restraint Sets
756	Load Sets
757	DOF Sets
759	Ply Properties
760	Laminate Definitions
761	Global Element Length
800	Working Set Header
801	Working Set Curves
802	Working Set Points

Read

15	Nodes
18	Coordinate Systems
55	Data at Nodes
56	Data at Elements
57	Data at Nodes on Elements
71	Elements
82	Trace Lines
92	Surfaces
705	Mesh Areas
706	Mesh Volumes
731	Physical Properties
735	Node/Element – Geometry Associativity
743	Supertab Points
744	Supertab Curves
747	Material Properties
748	Associated Element Data
749	Model Header
752	Permanent Groups
753	Case Sets
754	Constraint Sets
755	Restraint Sets
756	Load Sets
757	DOF Sets
759	Ply Properties
760	Laminate Definitions
761	Global Element Length
800	Working Set Headers
801	Working Set Curves
802	Working Set Points

Chapter 27. Current Universal File Datasets

The following datasets are the current datasets which are processed by I-DEAS.

```
Universal Dataset
Type: 15
Name: Nodes
Status: Current
Written by: I-DEAS Level 4     Frame
Written by: I-DEAS Level 4     Supertab
Written by: I-DEAS Level 4     Tdas
Written by: I-DEAS Level 4     Systan
Read    by: I-DEAS Level 4     Supertab
Read    by: I-DEAS Level 4     Systan
Read    by: I-DEAS Level 4     Tdas
Written by: I-DEAS Level 3     Frame
Written by: I-DEAS Level 3     Supertab
Written by: I-DEAS Level 3     Systan
Read    by: I-DEAS Level 3     Supertab
Read    by: I-DEAS Level 3     Tdas
Read    by: I-DEAS Level 3     Systan
Written by: I-DEAS Level 2.5 Supertab
Written by: I-DEAS Level 2.5 Systan
Read    by: I-DEAS Level 2.5 Supertab
Revision date: 30-AUG-1987
```

```
Record 1:          FORMAT(4I10,1P3E13.5)
                   Field 1        -- node label
                   Field 2        -- definition coordinate system number
                   Field 3        -- displacement coordinate system number
                   Field 4        -- color
                   Fields 5-7     -- 3-Dimensional coordinates of node
                                     in the definition system
```

Record 1 is repeated for each node in the model.

Example:

```
    -1
    15
         1        0        0        8  0.00000E+00  0.00000E+00   0.00000E+00
         2        0        0        8  5.00000E-01  0.00000E+00  -5.00000E-02
         .
         .
         .
       100        0        0        8  1.20000E+01  1.20000E+01  -4.50000E+00
    -1
```

Dataset 71

Universal Dataset
Type: 71
Name: <u>Elements</u>
Status: Current
Written by: I-DEAS Level 4   Frame
Written by: I-DEAS Level 4   Supertab
Written by: I-DEAS Level 4   Tdas
Read   by: I-DEAS Level 4   Frame
Read   by: I-DEAS Level 4   Supertab
Read   by: I-DEAS Level 4   Systan
Read   by: I-DEAS Level 4   Tdas
Written by: I-DEAS Level 3   Frame
Written by: I-DEAS Level 3   Supertab
Read   by: I-DEAS Level 3   Supertab
Read   by: I-DEAS Level 3   Systan
Written by: I-DEAS Level 2.5 Supertab
Read   by: I-DEAS Level 2.5 Supertab
Revision date: 19-NOV-1987

Record 1:   FORMAT(7I10)
                  Field 1   -- element label
                  Field 2   -- fe graphical description id
                  Field 3   -- fe descriptor id
                  Field 4   -- physical property table number
                  Field 5   -- material property table number
                  Field 6   -- color
                  Field 7   -- number of nodes on element

Record 2:   FORMAT(8I10)
                  Fields 1-n   -- node labels defining element

Records 1 and 2 are repeated for each element in the model.

Example:

```
    -1
    71
         1        19         1         1         1         8         8
        11        12        13        16        21        20        19        15
         2        12         2         2         1         8        16
        31        32        33        34        35        36        37        38
        39        40        41        42        43        44        45        46
         .
         .
         .
       124        19         1         1         1         8         8
         9        10        11        15        19        18        17        14
    -1
```

Universal Dataset
Type: 82
Name: <u>Trace Lines</u>
Status: Current
Written by: I-DEAS Level 4   Supertab
Written by: I-DEAS Level 4   Systan
Written by: I-DEAS Level 4   Tdas
Read   by: I-DEAS Level 4   Supertab
Read   by: I-DEAS Level 4   Systan
Read   by: I-DEAS Level 4   Tdas
Written by: I-DEAS Level 3   Systan
Read   by: I-DEAS Level 3   Systan
Read   by: I-DEAS Level 3   Tdas
Written by: I-DEAS Level 2.5 Systan
Revision date: 19-NOV-1987

Record 1:	FORMAT(3I10)	
	Field 1	-- trace line number
	Field 2	-- number of nodes defining trace line (maximum of 250)
	Field 3	-- color
Record 2:	FORMAT (80A1)	
	Field 1	-- Identification Line
Record 3:	FORMAT (8I10)	
	Field 1	-- nodes defining trace line
		= > 0 draw line to node
		= 0 move to node (a move to the first node is implied)

Notes:
1.) MODAL-PLUS node numbers must not exceed 8000.
2.) Identification line may not be blank.
3.) Systan only uses the first 60 characters of the identification text.
4.) MODAL-PLUS does not support trace lines longer than 125 nodes.
5.) Supertab only uses the first 40 characters of the identification line for a name.
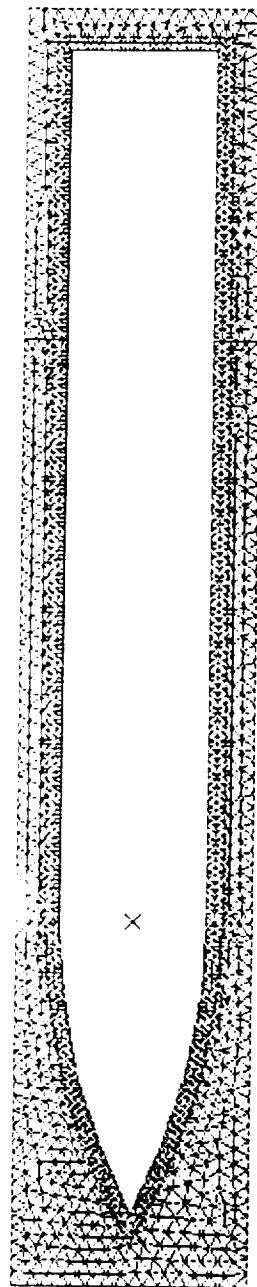
Example:

```
    -1
    82
         1
TRACE LINE1          5          1
         1
    -1             2          3          4          5
```

APPENDIX B

SDRC   I-DEAS 4.1:   Pre/Post Processing   7-AUG-90   11:14:(

DATABASE: MISSILE   UNITS :
VIEW : No stored VIEW   DISPLAY : No stored OP
   Task: Model Preparation
   Model: 1-FE MODEL1

                        Associated Workset: 1-WORKING

SDRC I-DEAS 4.1: Pre/Post Processing   6-JUL-90 14:40:

DATABASE: CYLO   UNITS :
VIEW   : No stored VIEW   DISPLAY : No stored OP
  Task: Mesh Generation
  Model: 1-FE MODEL1   Associated Workset: 1-WORKING

46

SDRC I-DEAS 4.1: Pre/Post Processing   1-AUG-90 16:09:.

DATABASE: QUAD MESH   UNITS :
VIEW : No stored VIEW   DISPLAY : No stored OP
    Task: Mesh Generation
Model: 1-FE MODEL1   Associated Workset: 1-WORKING

47

Pre/Post Processing

DATABASE: OGIVE3D
VIEW : No stored VIEW
  Task: Mesh Generation
  Model: 1-FE MODEL1

48

SDRC I-DEAS 4.1: Pre/Post Processing

26-JUL-90 10:41:.
UNITS :
DISPLAY : No stored OP

Associated Workset: 1-WORKING

DATABASE: WINGER
VIEW : VIEW1
Task: Model Checking
Model: 1-FE MODEL1

49

50

SDRC I-DEAS 4.1: Pre/Post Processing   29-JUN-90 11:20:.

DATABASE: 3DMESH#2   UNITS :
VIEW   : No stored VIEW   DISPLAY : No stored OP
  Task: Mesh Generation
 Model: 1-FE MODEL1
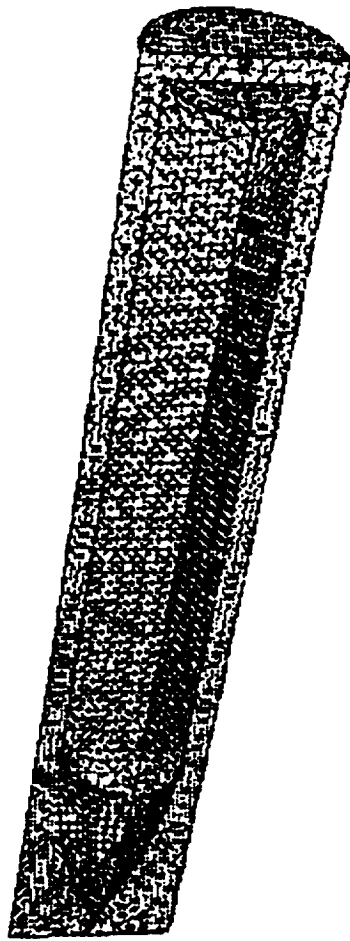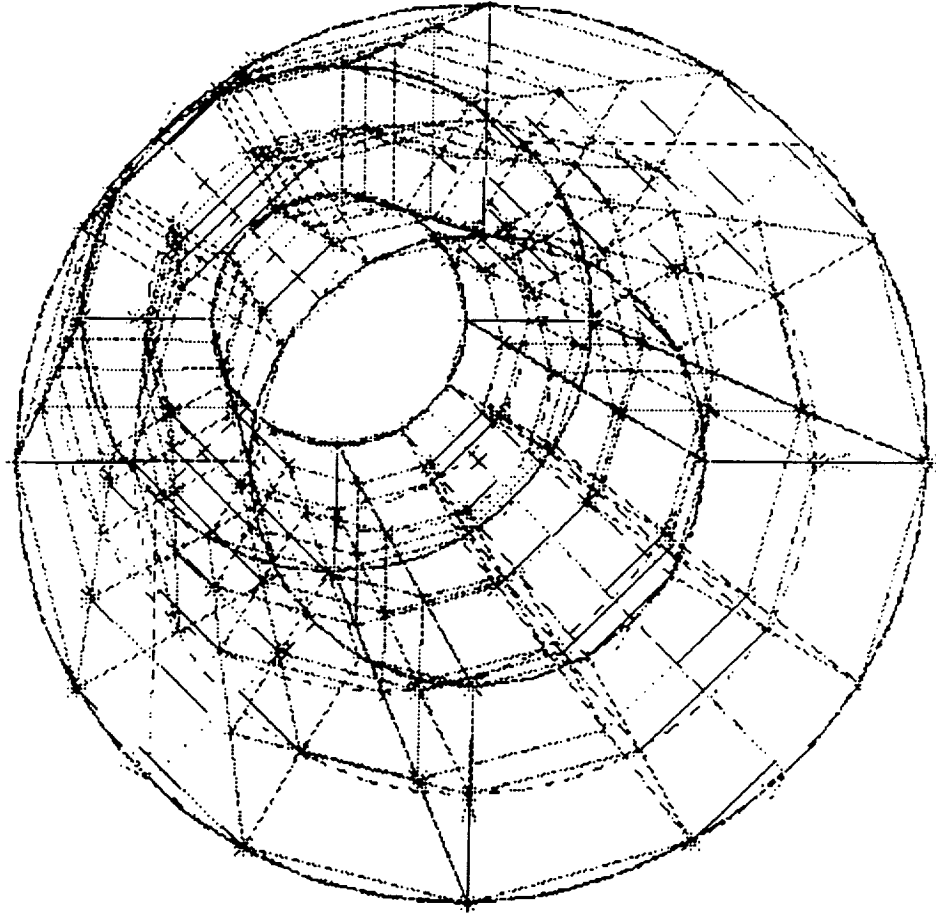
                        Associated Workset: 1-WORKING

SDRC I-DEAS 4.1: Pre/Post Processing   9-JUL-90 14:37:
                                                          UNITS :
                                              DISPLAY : No stored OP
                                        Associated Workset: 2-WORKING

DATABASE: 3DMESH#2
VIEW : No stored VIEW
    Task: Model Checking
  Model: 2-FE MODEL2