FEMAP -- Power and Simplicity in PC Modeling

by Enterprise Software Products, Inc.
Harleysville, PA

Enterprise Software Products, Inc. (ESP) has developed **FEMAP**, a powerful interactive PC program for creating finite element models for MSC/NASTRAN, and all versions of MSC/pal.

THE FEMAP APPROACH TO FINITE ELEMENT MODELING

The bulk of the engineering effort in finite element analysis lies in developing a model for the analysis program. ESP has based FEMAP on the philosophy of simplifying your input for constructing a model, while protecting your model data. Using this philosophy, FEMAP has been constructed around a unique menu-driven user-interface, which includes a dual text screen/graphics screen approach for model development. This enables FEMAP to display extensive text information to help you understand a modeling session, while providing separate full-screen color graphics for your plots. An "instant replay" feature is also available to enable you to switch between text and graphics, even during data entry.

THE FEMAP INTERFACE

The FEMAP user-interface is based on a menu tree, with the commands invoked by pressing the corresponding function buttons. The commands produce subsequent menus, or a prompt for data. The upper quarter of the text screen is used for the active menu, related parameters, and current model data. The interactive session is conducted in the remaining portion of the screen, and a scrolling feature is available to enable you to review up to 3 previous screens of text. English-style prompts and verification messages are used to simplify your input, and easy-to-understand error messages are provided to help you to correct input errors. FEMAP also provides an On-Line Calculator which interprets algebraic/trigonometric equations during data input. In addition, FEMAP contains extensive input checking to verify that you are entering data in the required format, and illegal keys are locked out. FEMAP also provides techniques to review all model data, using extensive graphic display features and data listing capabilities. An extensive On-Line Help mode can also be invoked at any time during a modeling session, by pressing **Esc.**

GRAPHICS

The graphics screen is used for plotting all model data, and is invoked at any command level by pressing the **Ins** key. FEMAP switches to graphics mode and constructs a plot of your model, using the active display parameters. All entities can be plotted or switched off, with or without various labels and prefixes. An arbitrary view angle can be selected using the animated rotation command, or the view can be aligned to an XY-plot in any coordinate system. In addition, complete control of plot size and position, entity colors, shading, element style and plot style is available. Models can be plotted with elements as lines or filled in, with size-selectable shrink. Plot style can contain all lines, feature lines only, or hidden lines can be removed. Hidden line plots can be created with nodes, loads, constraints, and labels for each. Subsets of a model can be plotted using Groups, where nodes, elements, coordinate systems, constraints and loads can be selected or eliminated using a variety of options. Plots can be saved on file, replayed from file, or printed. When a plot is completed, FEMAP issues a beep and waits for **enter**, which returns you to the text mode.

MODEL DATA

Nodes, elements, materials, section properties, loads, constraints and coordinate systems can all be created in FEMAP. For accurate geometric representations, all real number operations are performed in double precision. Model size is virtually unlimited, with FEMAP supporting up to 32760 of each entity type, so even very large models can be created and manipulated with ease.

The FEMAP element library contains mass, line, plane and solid elements, with related section and material properties:

FEMAP	MSC/pal	MSC/NASTRAN
Mass (MAS)	MASS	CONM2
Linear Rod (LRD)	----	CROD
Linear Bar (LBR)	BEAM 1	CBAR
Linear Tube (LTB)	BEAM 3	CTUBE
Linear Link (LLK)	LINK	----
Linear Beam (LBM)	BEAM 1	CBEAM
Linear Spring (SPR)	STIFFNESS	CELAS
Linear Damper (DMP)	DAMPER	CDAMP
Linear Curved Beam (LCB)	CURVED BEAM *	CBEND
Linear Shear Panel (LSH)	QUADRILATERAL *	CSHEAR
Linear Plate (LPL)	TRIANGULAR	CTRIA3
	QUADRILATERAL *	CQUAD4
Parabolic Plate (PPL)	----	CTRIA6
	----	CQUAD8
Parabolic Axisym (PAX)	----	CTRIAX6
Linear Solid (LSO)		
4-noded	----	CTETRA
6-noded	----	CPENTA
8-noded	----	CHEXA

* MSC/pal 2 and MSC/pal 2 Version 2

GEOMETRY GENERATION

FEMAP provides a variety of geometry generation capabilities, ranging from direct coordinate input to complete pattern generation. Multi-level coordinate systems, nodes and elements can be used to create even the most complicated model geometry, in rectangular, cylindrical or spherical coordinate systems. Generation algorithms include meshing between corners, meshing arbitrary regions between patterns of existing nodes, and copying or reflecting nodes. You can also scale a mesh in any direction in any coordinate system. Elements can be created by typing in the nodes, by using the graphics cursor to pick nodes, or by generating from existing node patterns. Node patterns can also be generated simultaneously with the elements. If section properties or materials do not exist, FEMAP allows you to create default properties during element creation.

LOADS AND CONSTRAINTS

Boundary conditions are defined using Constraint Sets, or as permanent conditions at the node level. Loads are defined using Load Case Sets, and available load types include point loads (forces, moments, enforced displacements, applied accelerations); element pressures; temperatures (node, element and default); and body accelerations (translational, angular and centripetal).

LISTINGS

Listings of all model data are provided in tabular form, with each data field described by an appropriate header. Listings can be directed to the screen, to a printer or to a disk file, and nodes and coordinate systems can be listed in any existing coordinate system. Screen listings of up to 68 lines of data can be reviewed using the scroll-back feature.

DATA INTEGRITY

Many safeguards are also built into FEMAP to protect your model data. These include prevention of overwriting entities which already exist, and of deleting entities referenced by other entities. In addition, FEMAP contains commands to verify model integrity, including free edge plots, checking planarity of node ranges, checking for element warping, and coincident node/element checks.

TRANSLATORS

Integrated read and write translators are included for MSC/NASTRAN and MSC/pal, MSC/pal 2 and MSC/pal 2 Version 2. If you have an MSC/pal version on your IBM PC, you can invoke the solution routine directly from FEMAP. You can then exit FEMAP, and after the MSC/pal run is complete, you can examine the results of the analysis using the "SCREEN" or "VIEW2" programs. When reading existing models, FEMAP will identify any missing entities. As you correct errors, FEMAP can be used iteratively to identify remaining inconsistencies. You can even use FEMAP's graphics capabilities to display incomplete models. This provides a powerful capability to verify or complete your models.

PROGRAM FILES

Although you don't have to be a programmer to use FEMAP, the program file interface allows you to supply your own algorithms and develop custom procedures. These include creating common families of geometries and performing repetitive tasks such as creating nodal coordinates from equations. The FEMAP programming language includes any legitimate FEMAP command, IF and GOTO/LABEL statements for creating branches and loops, arithmetic statements which use on-line calculator expressions, and WAIT/PAUSE to allow user interaction during program file execution.

Program files can also be used to record a modeling session. They can be replayed with or without pause after plots for demonstrations. They can also be edited to vary modeling parameters, and then replayed to create a revised model.

HARDWARE

FEMAP is designed for the IBM PC, XT, AT or 100% compatibles (running MS DOS 2.0 or higher), and supports both Color Graphics (CGA) and Enhanced Graphics (EGA) adapters. FEMAP operates with as little as 512K memory, and a 10 MByte hard disk. In addition, many popular dot matrix and laser printers are supported, for both plots and listings.

PERFORMANCE

FEMAP is **FAST**. It provides features and speed you would expect in a mainframe system, and which you will find amazing on your PC. For example, on an IBM AT (6 MHz), nodes can be generated at the rate of 40-50 nodes per second, while a hidden-line shaded plot for a model with 800 plate elements can be produced within 7 minutes.

Sample Modelling Session -- Impeller

The following session for developing a model of an impeller is used to demonstrate the FEMAP commands. The commands are invoked by pressing the appropriate function button which corresponds to the menu command in the screen menu. FEMAP then echoes the first 3 letters of the command, as shown in boldface. You will drop through the menu tree, until you encounter a command which produces a prompt for input data. Only a few of the many features available in FEMAP are needed to create the model of the impeller.

In this example, node creation features include keying in nodes, copying nodes (in cylindrical coordinates), and using the "Region" command to create the pattern for a single impeller blade between two curved ranges of nodes. Element creation features include Between Corners generation, and copying to existing node patterns.

Physical data for the impeller include the following:

```
Radius of Inner Hub:        Variable from 2.0" to 2.5".
Thickness of Inner Hub:     0.070"
Blade Angle of Twist:       Variable to 30 degrees
Thickness of Blades:        0.036"
Outer Radius of Blades:     4.0"
Number of Blades:           12
Material:                   Aluminum
Flanges on hub:             0.15" x 0.15"
```

The following steps include the menu commands (in boldface), the resulting prompts, and the user responses (also in boldface). Note the relevant information in parentheses and the default responses in square brackets. Figures 1 thru 9 show the model in different stages of development.

Step 1: Set the Definition Coordinate System (Cylindrical)

```
MODel NODe PARameter DEF
Enter Definition Coordinate System ID [0] >1      (Cylindrical System)
Active Definition Coord Sys ID is 1.
```

Step 2: Create a row of the inner hub nodes manually (**Figure 1**)

```
<End> CRE
Enter R,Theta,Z (Node 1) [0.,0.,0.] >2,,2                Type in First
Enter R,Theta,Z (Node 2) [2.,0.,2.] >2.05,,1.5            Five Nodes
Enter R,Theta,Z (Node 3) [2.05,0.,1.5] >2.15,,1
Enter R,Theta,Z (Node 4) [2.15,0.,1.] >2.3,,.5
Enter R,Theta,Z (Node 5) [2.3,0.,0.5] >2.5,,0
Enter R,Theta,Z (Node 6) [2.5,0.,0.] ><End>
```

Step 3: Copy nodes to complete the inner hub (**Figure 2**)

```
GENerate COPy
Enter Node ID Range to Copy [5,0,1] >1,5                        Copy Nodes 1-5
Enter Number of Repetitions [1] >24
Confirm Each Creation [N] ?enter
Use Active Parameters [Y] ?enter
Enter Delta Coordinates (CSys 1) [0.,0.,0.] >,15          15-degree delta
120 Nodes Created.
```

Step 4: Create section property for inner hub

```
<Home> PROperty CREate
Enter Material ID (Property 1) [1] >enter
ERROR 55: Material 1 does not Exist.
Ok to Generate Default Material [Y] ?enter
Enter Thickness (up to 4) [1.,0.,0.,0.] >.07
Enter Nonstructural Mass per Unit Area [0.] >enter
Enter Fiber Distances [0.5,-0.5] >.035,-.035
Enter Bending Stiffness Parameter (12I/T**3) [1.] >enter
Enter Transverse Shear/Membrane Thicknesses [0.833] >enter
Section Property 1 Created.
Enter Material ID (Property 2) [1] ><End>
```

Step 5: Generate elements between corners for the inner hub (**Figure 3**)

```
<Home> ELEment GENerate BETween
Enter Number of Corners >4
Ok to Use Existing Nodes [Y] ?enter
Enter Starting Node ID,Inc [1,1] >enter
Enter Number of Nodes on Edge 1-2 (Including Corners) >5
Enter Number of Nodes on Edge 1-4 (Including Corners) >25
Ok to Use Triangular Elements [N] ?enter
Checking 125 Nodes...
Generating...
96 Elements Created.
```

Step 6: Create a row of outer nodes for the first blade (**Figure 4**)

```
<Home> NODe CREate
Enter R,Theta,Z (Node 126) [2.5,0.,0.] >4,-20,2
Enter R,Theta,Z (Node 127) [4.,-20.,2.] >,-10,1.5
Enter R,Theta,Z (Node 128) [4.,-10.,1.5] >,-3,1
Enter R,Theta,Z (Node 129) [4.,-3.,1.] >,2,0.5
Enter R,Theta,Z (Node 130) [4.,2.,0.5] >,7,0
Enter R,Theta,Z (Node 131) [4.,7.,0.] ><End>
```

Step 7: Create remaining nodes for first blade using REGION (**Figure 5**)

```
GENerate REGion
Enter First Node ID Range [130,0,1] >121,125
Enter Second Node ID Range [130,0,1] >126,130
Enter Number of Nodes Between Ends (Include Ends) [3] >5
Enter Nodal Bias Between Ends [1.] >enter
Generating 25 Nodes...
25 Nodes Created.
Ok to Merge Region Definition Ranges [Y] ?N                    Nodes 131-155
```

Step 8: Copy blade nodes for the remaining blades

```
COPy
Enter Node ID Range to Copy [155,0,1] >131,155
Enter Number of Repetitions [1] >11
Confirm Each Creation [N] ?enter
Use Active Parameters [Y] ?enter
Enter Delta Coordinates (CSys 1) [0.,0.,0.] >,30
275 Nodes Created.
```

Step 9: Change active section property ID for elements

```
<Home> ELEment PARameter PROperty
Enter Section Property ID [1] >2
Active Section Property ID is 2.
```

Step 10: Create Section Property for blades

```
<Home> PROperty CREate
Enter Material ID (Property 2) [1] >enter
Enter Thickness (up to 4) [1.,0.,0.,0.] >.036
Enter Nonstructural Mass per Unit Area [0.] >enter
Enter Fiber Distances [0.5,-0.5] >.018,-.018
Enter Bending Stiffness Parameter (12I/T**3) [1.] >enter
Enter Transverse Shear/Membrane Thicknesses [0.833] >enter
Section Property 2 Created.
Enter Material ID (Property 3) [1] ><End>
```

Step 11: Create elements for first blade (**Figure 6**)

```
<Home> ELEment GENerate BETween
Enter Number of Corners >4
Ok to Use Existing Nodes [Y] ?enter
Enter Starting Node ID,Inc [1,1] >131,1
Enter Number of Nodes on Edge 1-2 (Including Corners) >5
Enter Number of Nodes on Edge 1-4 (Including Corners) >5
Ok to Use Triangular Elements [N] ?enter
Checking 25 Nodes...
Generating...
16 Elements Created.
```

Step 16: Generate beam elements around lower edge of inner hub

Figures 8 (with orientation) and 9 (with node ID)

```
BETween
Enter Number of Corners >2
Ok to Use Existing Nodes [Y] ?enter
Enter Starting Node ID,Inc [1,1] >5,5
Enter Number of Nodes on Edge 1-2 (Including Corners) >25
Checking 25 Nodes...
Generating...
Ok to Use Node ID for Orientation [Y] ?enter
Enter Orientation Node ID [0] >432
Ok to Offset Element From Nodes [N] ?enter
Ok to Enter Element Releases [N] ?enter
24 Elements Created.
```

Step 17: Modify the section property data for the hub flanges

```
<Home> PROperty MODify VALues
Enter ID of Section Property to Modify [3] >enter
Enter Cross Sectional Area (A) [1.] >.0625
Enter Moments of Inertia (I1,I2,I12 or Ivv,Iww,0) [1.,1.,0.] >
    .00033,.00033,0.
Enter Torsional Stiffness (J) [1.] >.00066
Enter Shear Area (0=No Shear Deformation) [0.,0.] >enter
Enter Nonstructural Mass per Unit Length [0.] >enter
Enter Y,Z Fiber Distances [0.,0.] >.125,.125
Section Property 3 Modified.
```

You can also use FEMAP's checking capabilities to identify and merge coincident nodes, check planarity of node ranges, check for warping of elements and to perform free edge plots.

The MSC/pal 2 or MSC/NASTRAN translators can then be employed to create input files for a modes solution for the impeller model.

In addition, loads and boundary conditions could also be applied for static or dynamic analysis.

SUMMARY

Figures 10 thru 15 show other models created using FEMAP. FEMAP contains many powerful features which will enable you to create, display and verify your models, while the process remains simple to use. In addition, FEMAP provides many measures of protection for your model data. Use FEMAP to prepare your models, and in conjunction with MSC/pal 1 or 2 and MSC/NASTRAN, you will be able to model and solve engineering problems in a wide range of technical disciplines. After using FEMAP to build your first model, we are sure that you will agree: FEMAP is power and simplicity in PC modeling.

For further information on FEMAP, call ESP at (215) 256-1829, or write to Enterprise Software Products, Inc., P.O. Box 264, Harleysville, PA 19438.

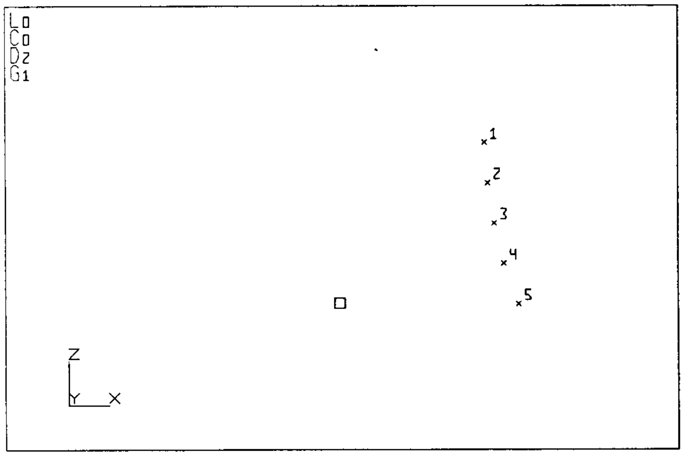

Figure 1

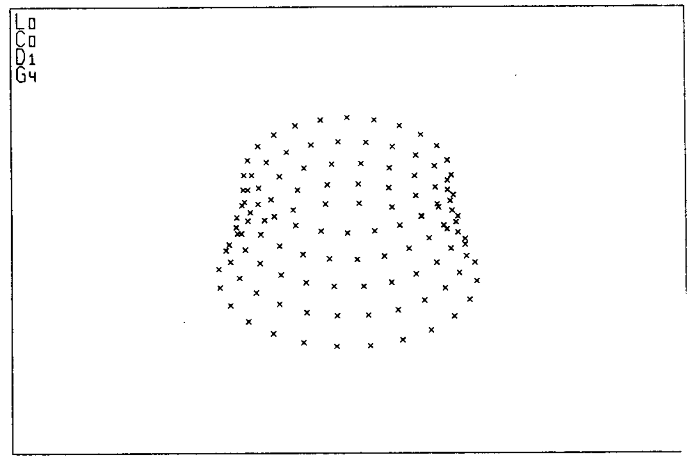

Figure 2

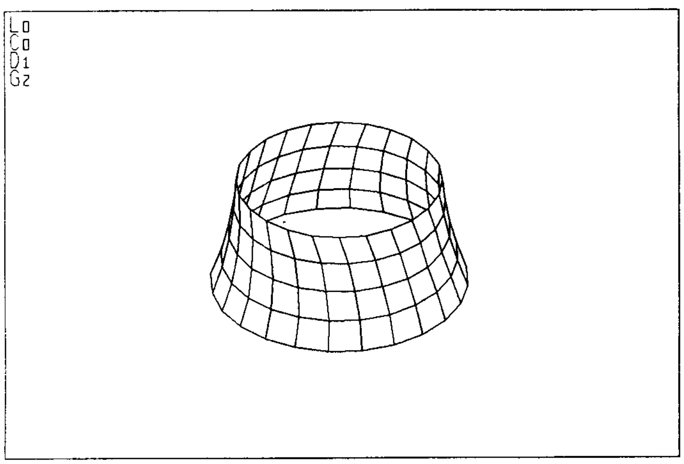

Figure 3

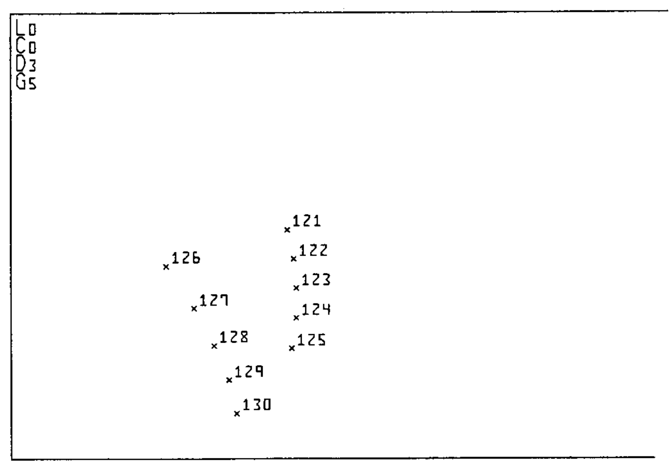

Figure 4

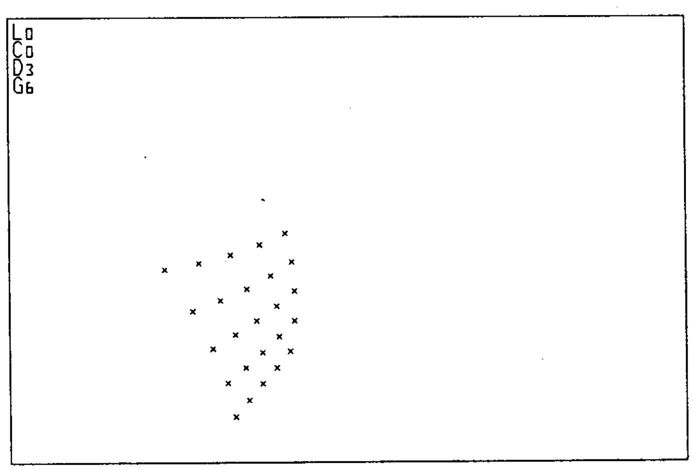

Figure 5

Step 16: Generate beam elements around lower edge of inner hub

Figures 8 (with orientation) and 9 (with node ID)

```
BETween
Enter Number of Corners >2
Ok to Use Existing Nodes [Y] ?enter
Enter Starting Node ID,Inc [1,1] >5,5
Enter Number of Nodes on Edge 1-2 (Including Corners) >25
Checking 25 Nodes...
Generating...
Ok to Use Node ID for Orientation [Y] ?enter
Enter Orientation Node ID [0] >432
Ok to Offset Element From Nodes [N] ?enter
Ok to Enter Element Releases [N] ?enter
24 Elements Created.
```

Step 17: Modify the section property data for the hub flanges

```
<Home> PROperty MODify VALues
Enter ID of Section Property to Modify [3] >enter
Enter Cross Sectional Area (A) [1.] >.0625
Enter Moments of Inertia (I1,I2,I12 or Ivv,Iww,0) [1.,1.,0.] >
    .00033,.00033,0.
Enter Torsional Stiffness (J) [1.] >.00066
Enter Shear Area (0=No Shear Deformation) [0.,0.] >enter
Enter Nonstructural Mass per Unit Length [0.] >enter
Enter Y,Z Fiber Distances [0.,0.] >.125,.125
Section Property 3 Modified.
```

You can also use FEMAP's checking capabilities to identify and merge coincident nodes, check planarity of node ranges, check for warping of elements and to perform free edge plots.

The MSC/pal 2 or MSC/NASTRAN translators can then be employed to create input files for a modes solution for the impeller model.

In addition, loads and boundary conditions could also be applied for static or dynamic analysis.

SUMMARY

Figures 10 thru 15 show other models created using FEMAP. FEMAP contains many powerful features which will enable you to create, display and verify your models, while the process remains simple to use. In addition, FEMAP provides many measures of protection for your model data. Use FEMAP to prepare your models, and in conjunction with MSC/pal 1 or 2 and MSC/NASTRAN, you will be able to model and solve engineering problems in a wide range of technical disciplines. After using FEMAP to build your first model, we are sure that you will agree: FEMAP is power and simplicity in PC modeling.

For further information on FEMAP, call ESP at (215) 256-1829, or write to Enterprise Software Products, Inc., P.O. Box 264, Harleysville, PA 19438.

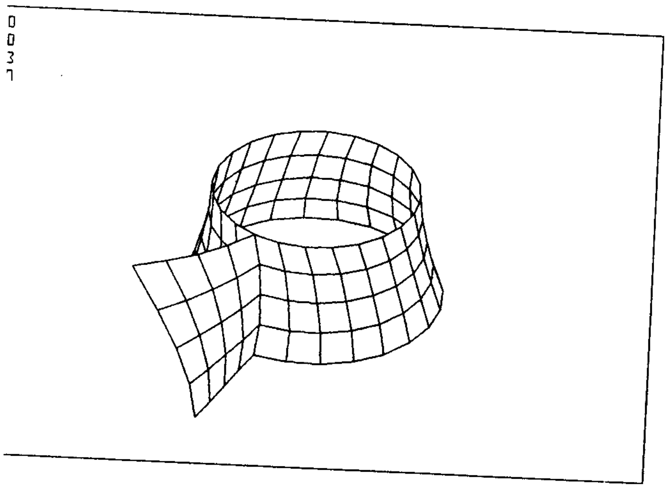

Figure 6

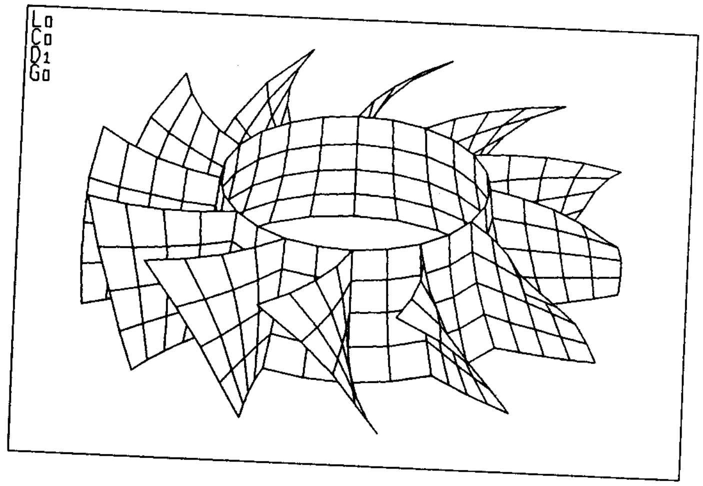

Figure 7

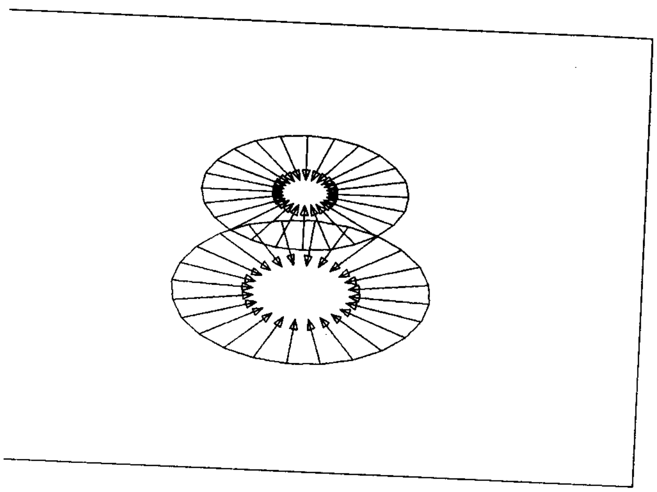

Figure 8

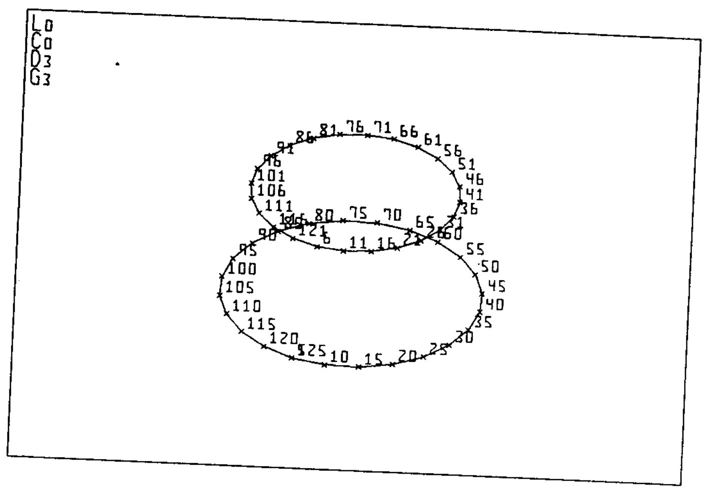

Figure 9

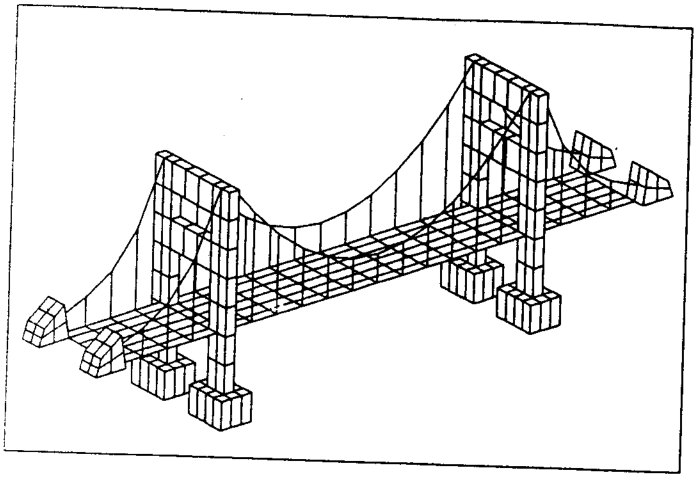

Figure 10

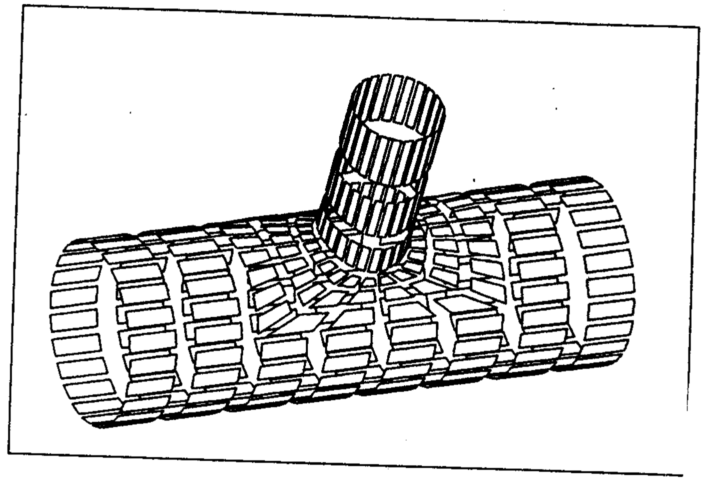

Figure 11

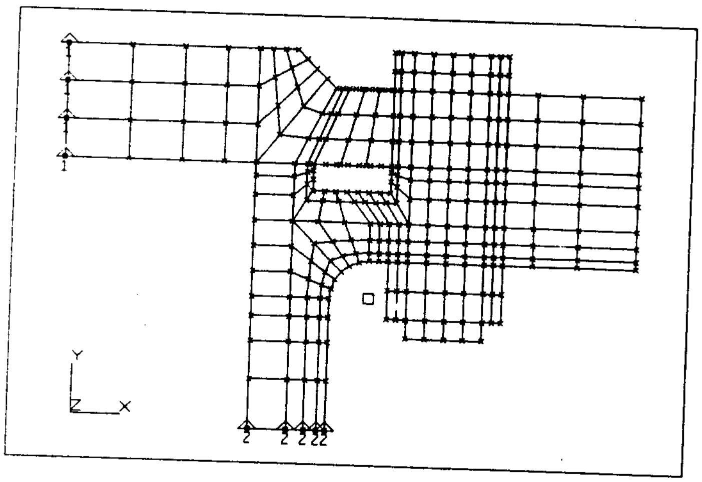

Figure 12

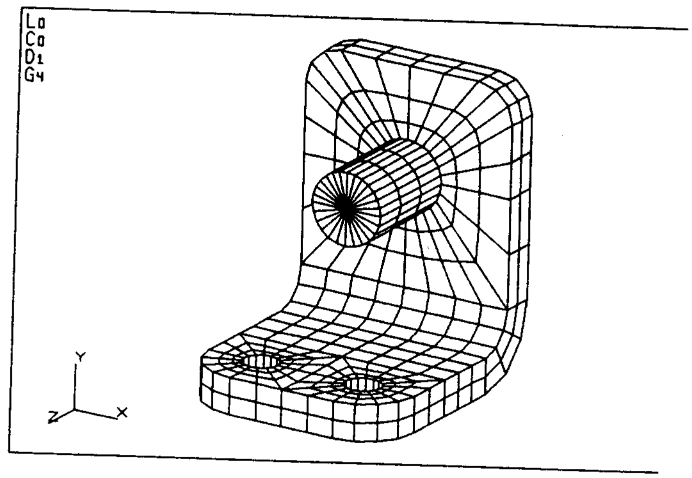

Figure 13

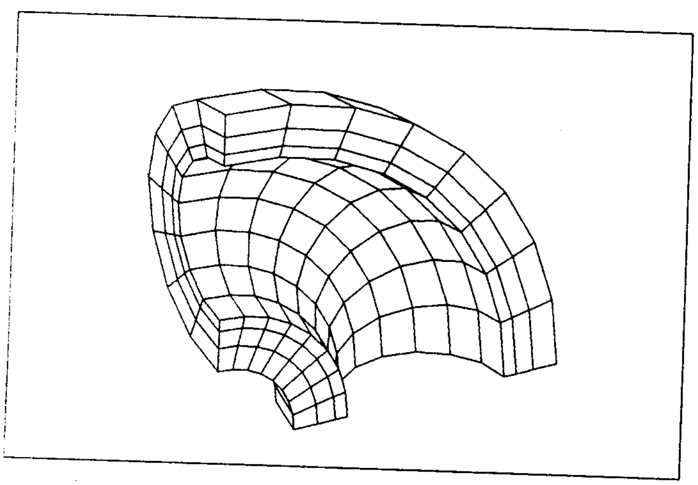

Figure 14

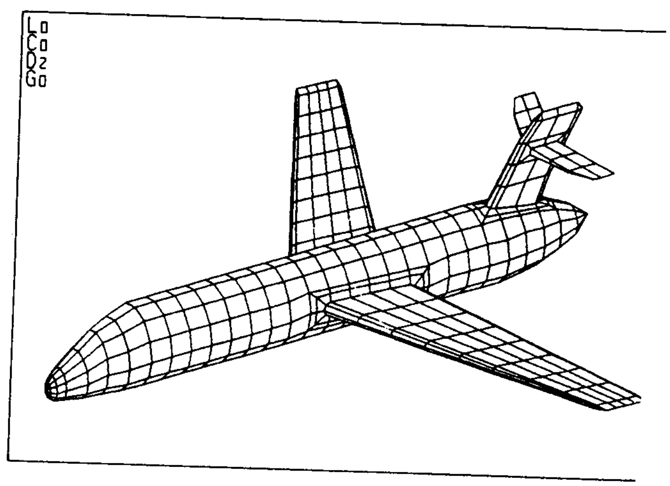

Figure 15