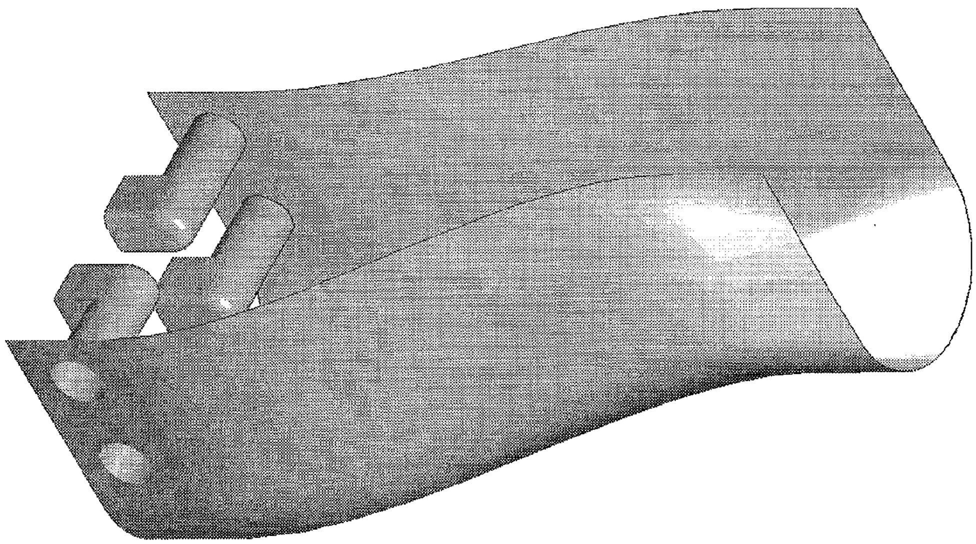
Multi Disciplinary Analysis in MSC/NASTRAN with FEMAP

Mark A. Sherman
Enterprise Software Products, Inc.

FEMAP is a general-purpose finite element modeling and post-processing software package available for Personal Computers running Microsoft Windows 3.1 and Windows NT, DEC Alpha Workstations running Windows NT, and HP, SGI, SUN, and IBM UNIX Workstations. This paper presents the multi-disciplinary analysis of a hot-gas diffuser that was modeled in FEMAP, and analyzed with MSC/NASTRAN, complete with step-by-step modeling descriptions that will provide the reader with a thorough understanding of the capabilities of FEMAP as applied to a combined static, modal, and thermal analysis of the hot-gas diffuser.

Geometry

As with any engineering analysis, this one begins with a description of the part to be modeled. 3-D Geometry suitable for meshing can be created in FEMAP, or imported through IGES or DXF formatted files. In this case the hot-gas diffuser was designed in MicroStation. The geometry was exported in a DXF formatted file to FEMAP.



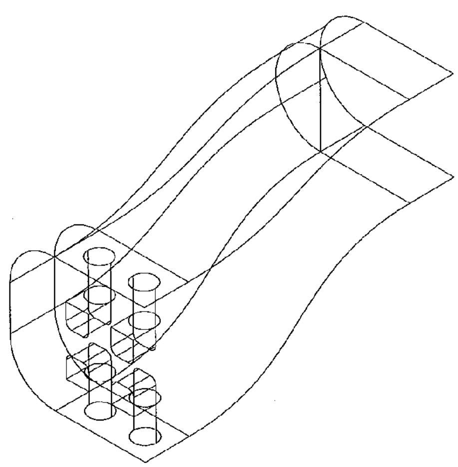
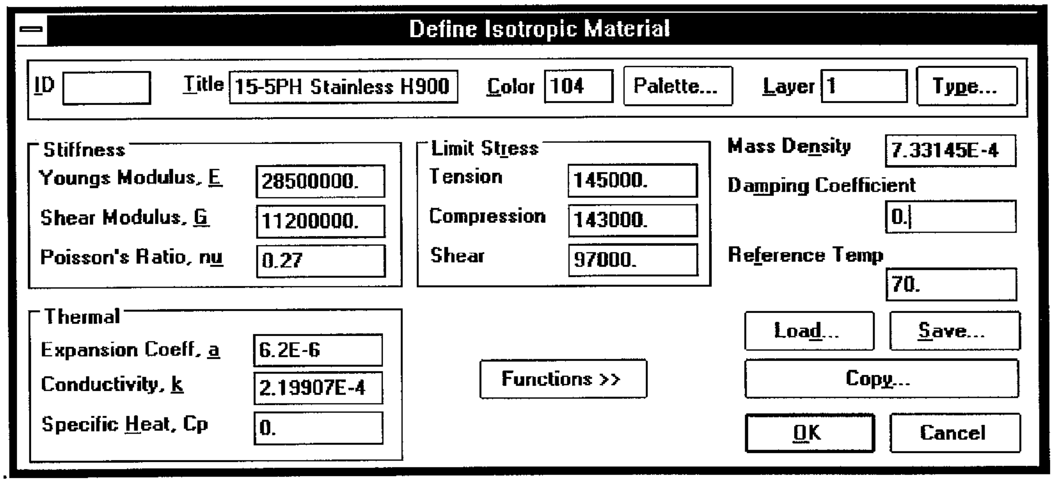


Figure 1 - CAD Geometry as Imported by FEMAP

The hot-gas diffuser is designed to lower the temperature of the inlet air by mixing in cooler air through the eight nozzles. Only half of this structure will be analyzed, using FEMAP's ability to quickly specify symmetry boundary conditions. This analysis will determine the combined effects of pressure and temperate on the stainless steel components. In addition the dynamic characteristics of the assembly will be determined.

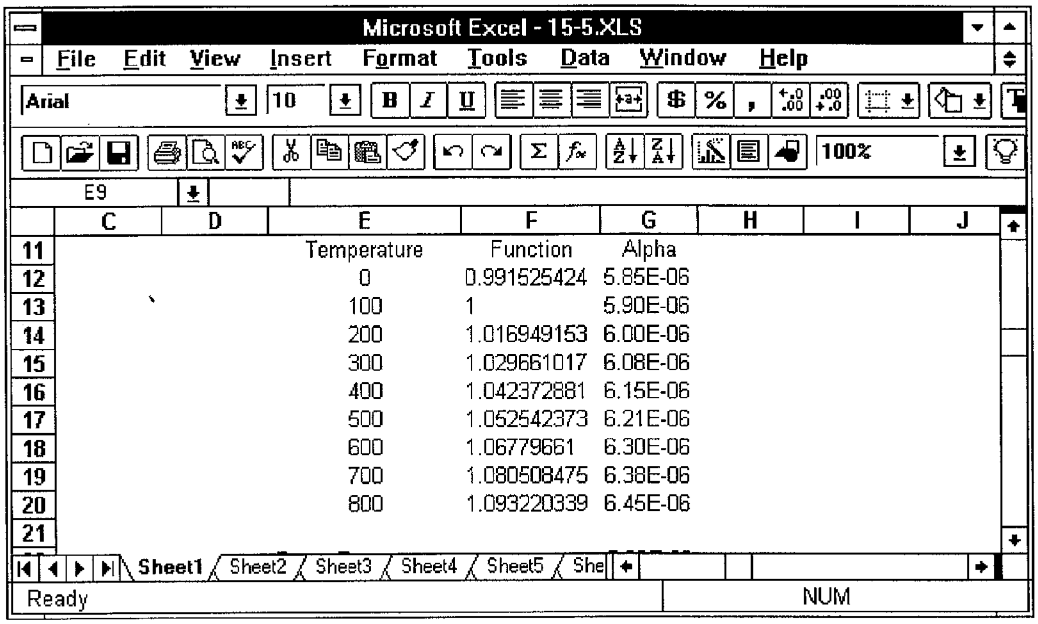
Material Definition

The first step in defining the finite element model will be the creation of properties to be used in the analysis. The part is constructed entirely of 15-5PH Stainless Steel. The default material library that ships with FEMAP contains the appropriate material properties as defined in MIL-HDBK-5.

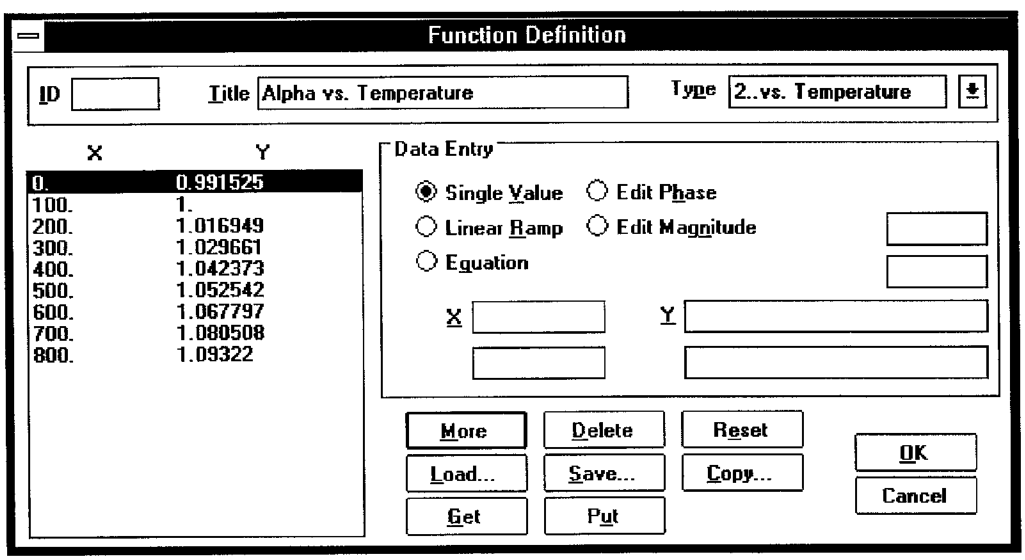


The Function button on the Material Creation Dialog is used to attach time or temperature dependent material properties to the various material properties. For this analysis the temperature dependence of several of the properties will be incorporated.

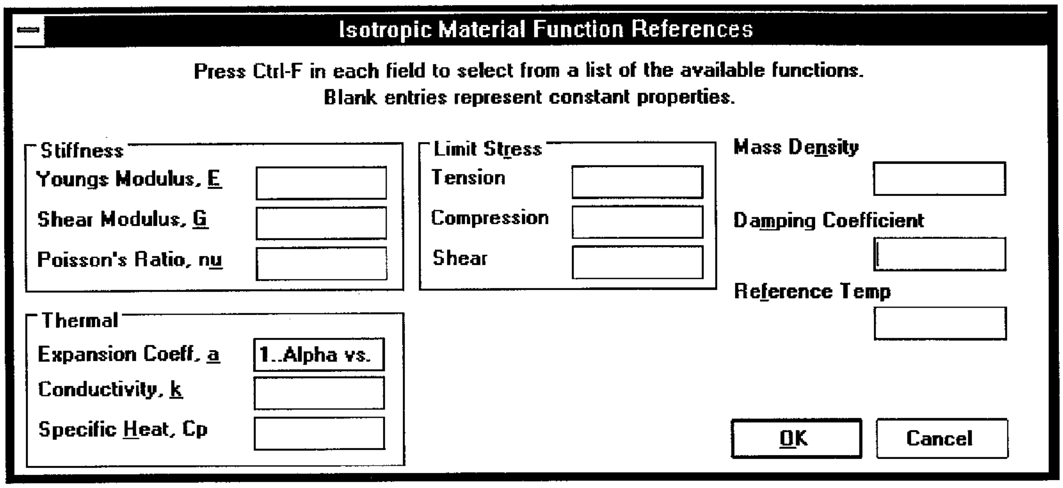
Also from MIL-HDBK-5, the values for 15-5PH H900 Stainless are extracted that relate the change in the Thermal Expansion Coefficient with Temperature. These values are typed into Microsoft Excel, and copied to FEMAP using the Cut



and Paste capabilities of Windows. From Excel, simply highlight the values of interest, and then select Edit Copy from the menu, to copy the values to the Window Clipboard. Switch to FEMAP, selected Create Function from the menu, and the press the Get button, this will get the data from the Clipboard.



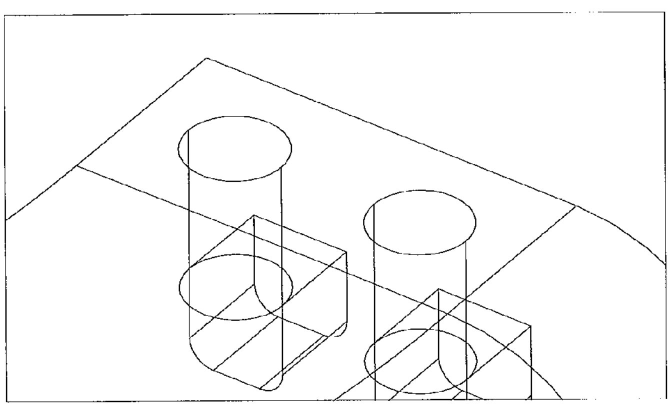
This function can be connected to the Material Thermal Expansion Coefficient by selecting the Function Button in the Create Material Dialog, and then specifying function 1 in the appropriate field.



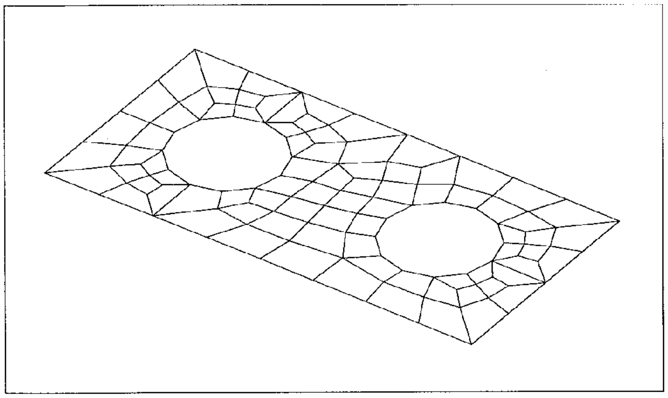
The material definition is now complete.

Modeling

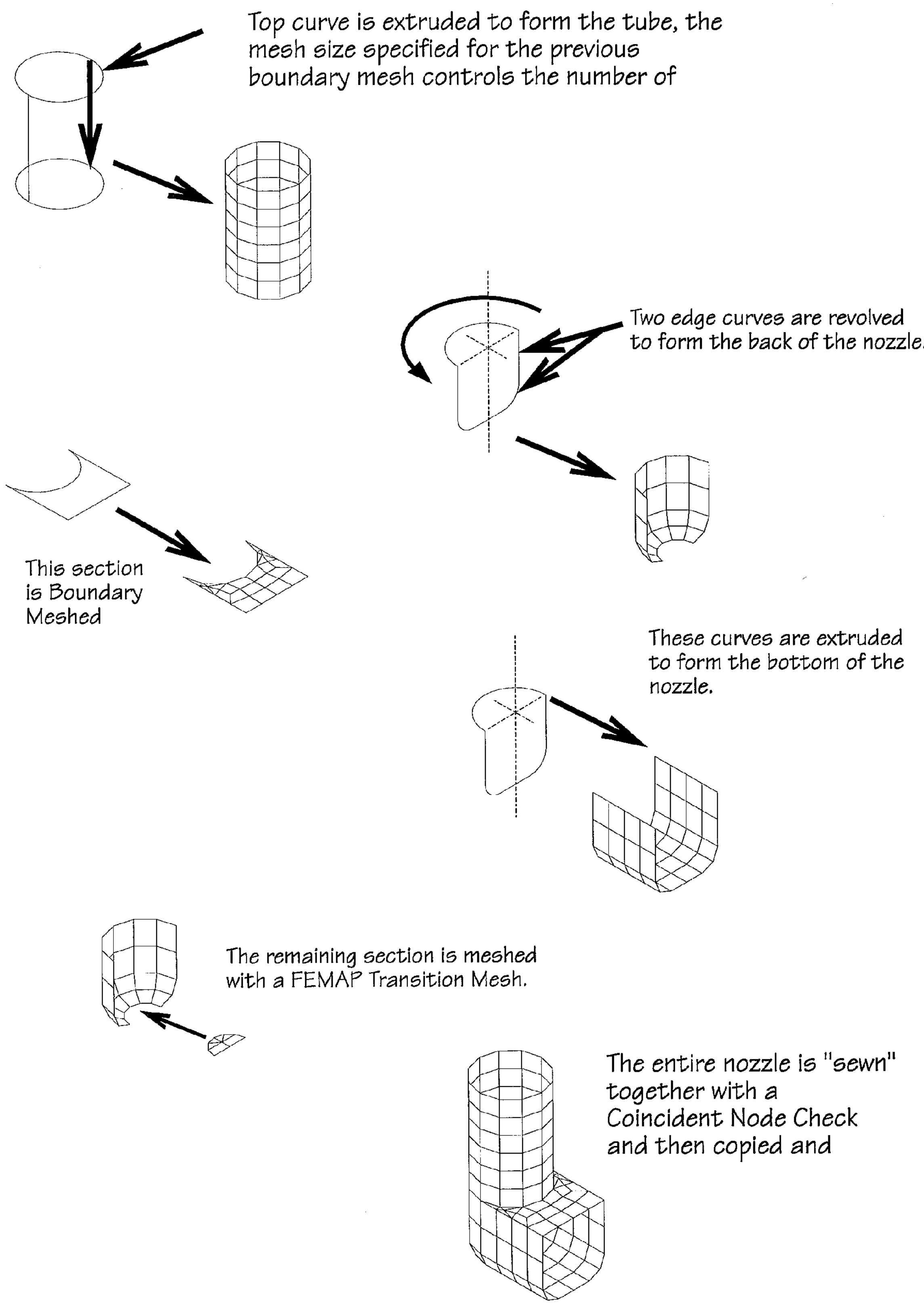
We will begin meshing this part at the most difficult location, the transition between the diffuser body and the nozzles. For this area the FEMAP Boundary Mesher is the best tool for the task. The Boundary Mesher can process an enclosed series of curves, with included holes, and automatically mesh them. Provided the total number of edge elements is even, the Boundary Mesher will generate an all quad mesh.



A global mesh size is set so that all edge curves that do not have their mesh size explicitly defined will generate well shaped elements. The default mesh is set to 0.25 inches. A mesh size of 12 elements is specified at the hole. This causes the automesher to increase the mesh density in the most critical area. The default values for element aspect ratio, minimum interior angles, etc. are used, and the following mesh is generated.

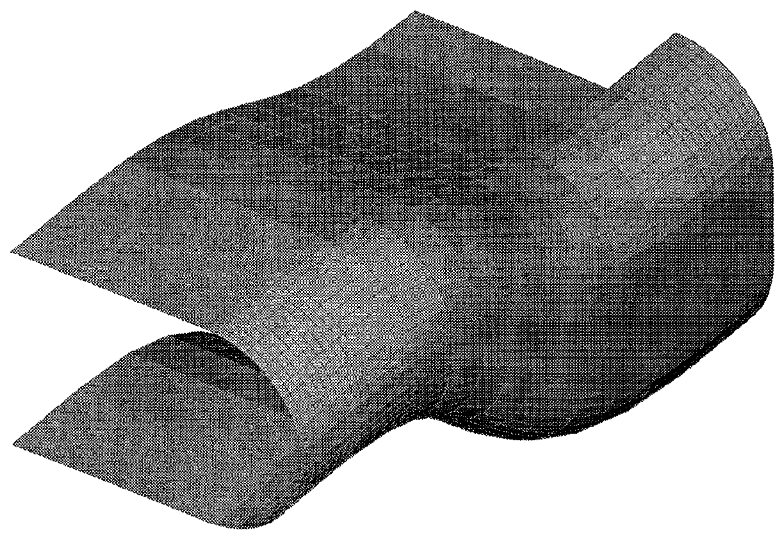


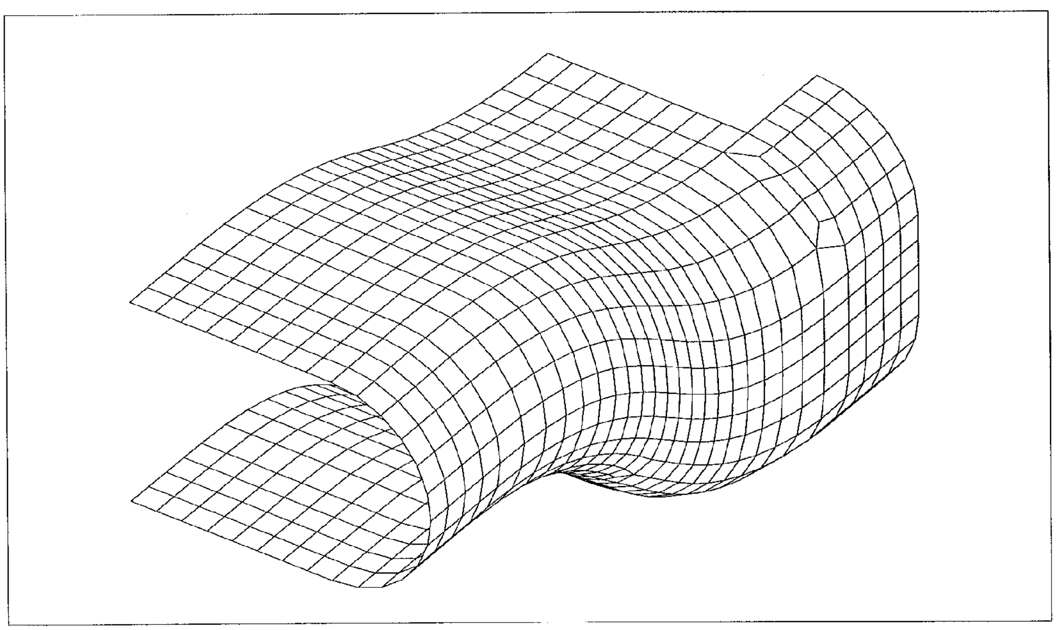
The tubes themselves are quickly generated using a variety of meshing techniques. The following page details the techniques used.



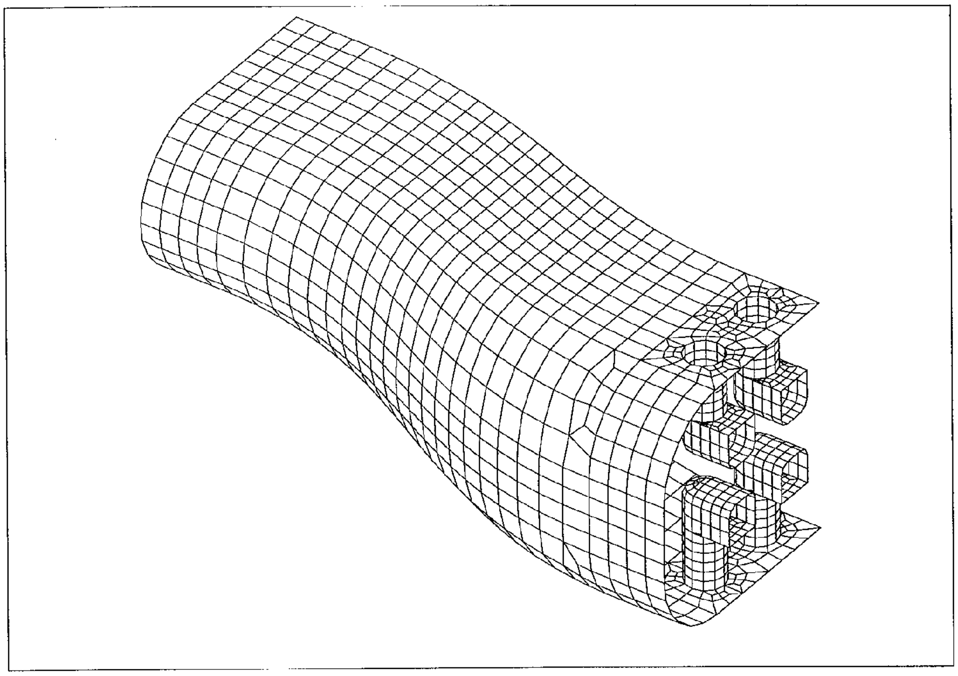
Nozzle Mesh Construction

The remainder of the geometry is used to create FEMAP Surfaces. Once the surfaces have been created, they are all meshed in a single command. FEMAP meshes surfaces in one of two ways, if the specified mesh size for the surface's edge curves is the same on opposite sides, the surface is mapped mesh. If not, the same two-dimensional mesh as that used in the Boundary Mesh in invoked for the surface.

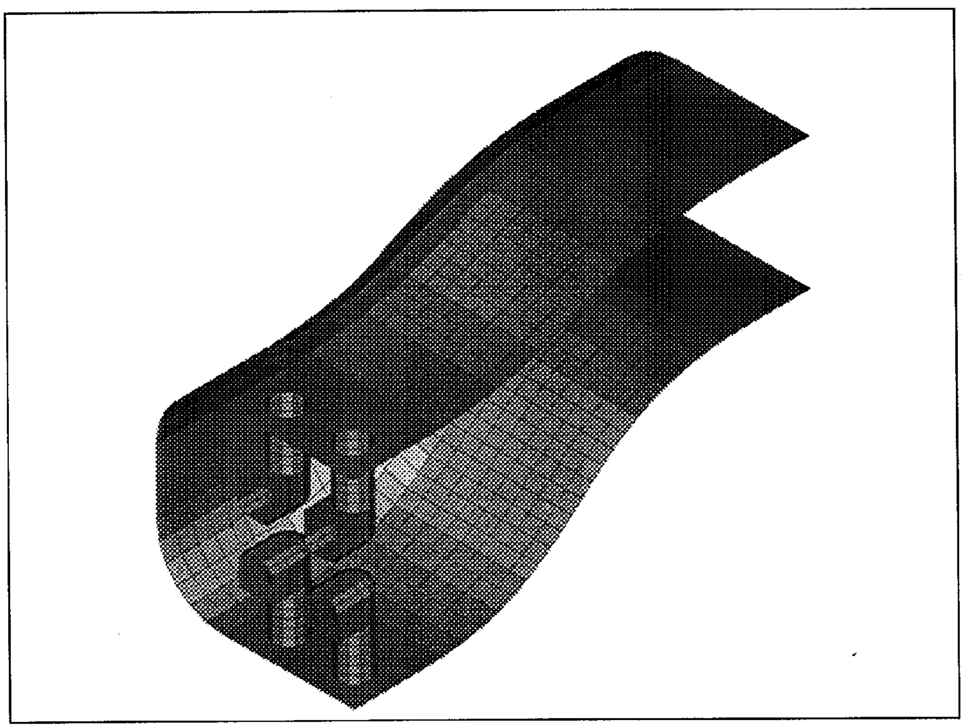




This portion of the model is merged with the nozzle portions with a coincident node check. This completes the meshing portion of the analysis.



Finite Element Model -Hidden Line Plot



Finite Element Model - Hidden Line with Shading

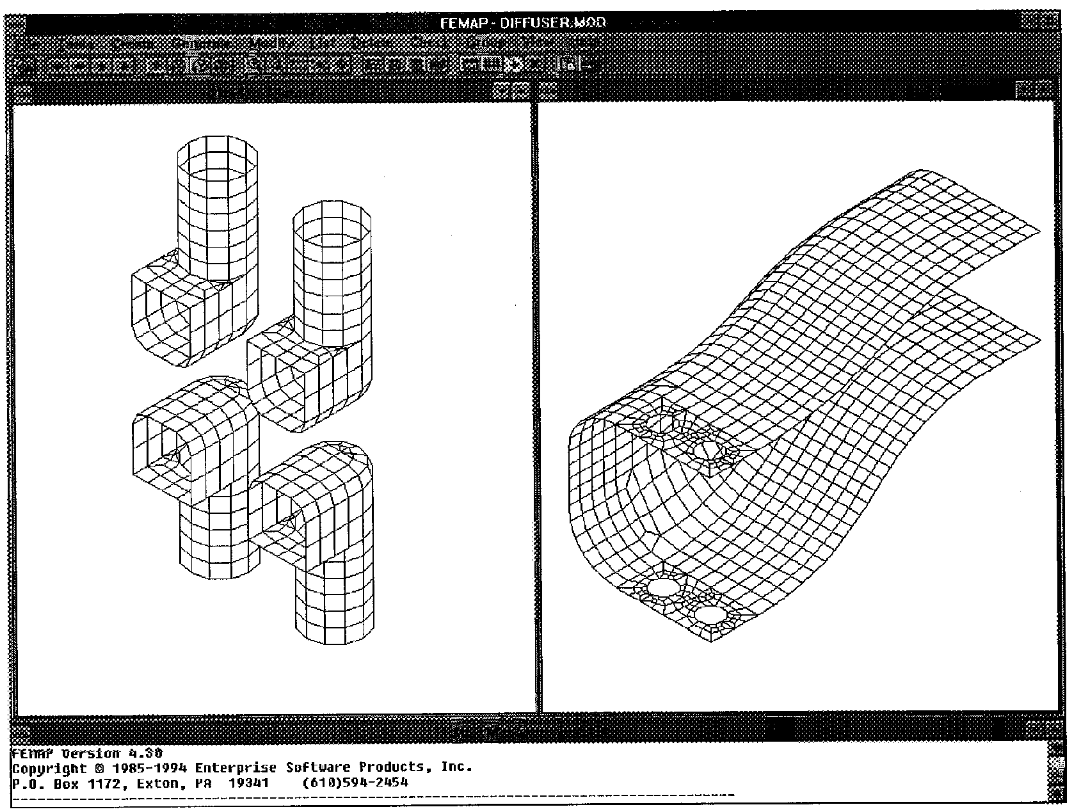
Loads and Boundary Conditions

Thermal Loading

The first step in this analysis will be to determine the temperature distribution in the diffuser. The loading is defined as follows:

- 300°F air flows through the diffuser
- 70°F air is injected through the nozzles

To make placement of loads as easy as possible, the Group capability of FEMAP is used. Two groups are created, one represents the nozzles, and the other the body of the diffuser. The nozzles are placed into a group first. Using multiple views, elements are added and removed with three box picks to include only the nozzles in the active group. To create the group that represents everything but the nozzles, the FEMAP Group Operations Not Command is used to create a new group that contains every element not in the nozzle group.

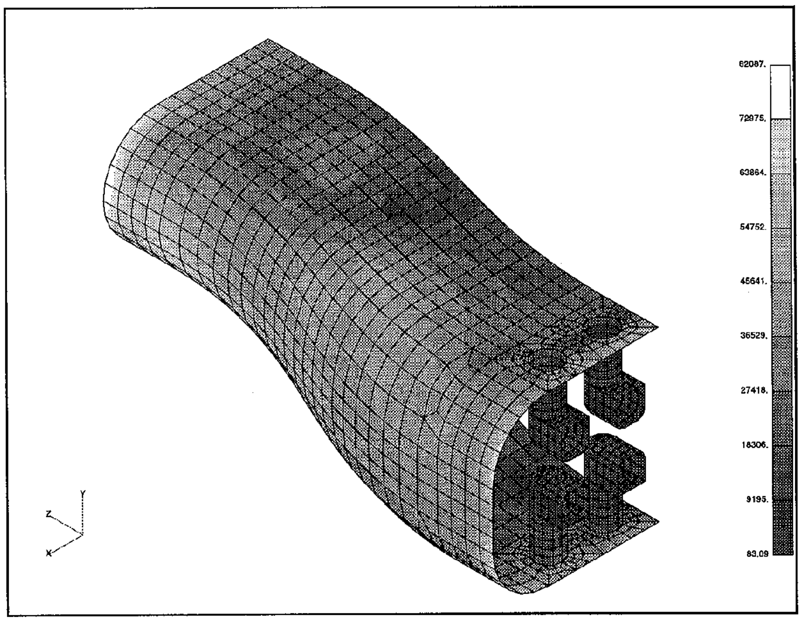


The above loading conditions are applied to the model using the two groups.

Post-Processing Thermal Analyses - Preparing for Stress Analysis

The results of the MSC/NASTRAN thermal analysis were read into FEMAP. A color contour of the nodal temperatures is reviewed to verify that the analysis went as planned. The results appear satisfactory for the applied loading. The main goal of this analysis, to combine the results of the thermal analysis with the internal pressure loading is quickly accomplished in FEMAP. The Create Load From Output Command is used to convert the nodal temperature output to Nodal Temperatures for a MSC/NASTRAN Static Analysis.

A static analysis using the nodal temperatures was run, the results of that analysis are shown in the following contour:



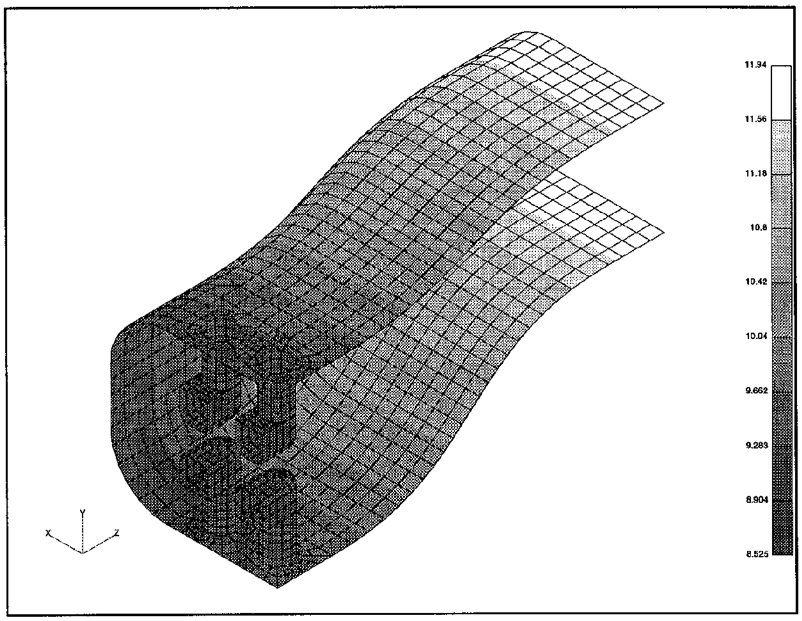
Thermal Stress Results - Von Mises Stresses

Pressure Loading

In addition to the thermal loading, the internal pressure must be applied to complete this analysis. The internal pressure was defined as 3 psi at the inlet, and dropping off at .1 psi per inch through the diffuser. To create this loading in FEMAP, a Coordinate System is defined at the diffuser inlet, with its X-Axis running in the flow direction. The following equation is used to create the pressure load:

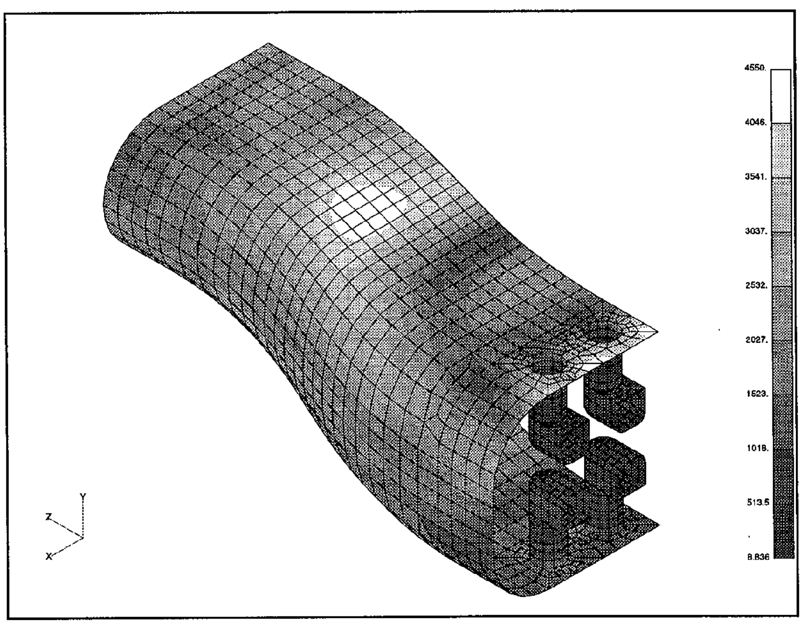
$$P = 12.0 - (xel(i) \cdot 0.5)$$

where xel(i) represent the x-value of each element being considered in the active coordinate system. To ensure that the pressure distribution is correct, the Create Output From Load command is used. This command will take the pressure loading, and convert it to elemental output for contouring, making it possible to view the pressure distribution as a contour.



Pressure Distribution

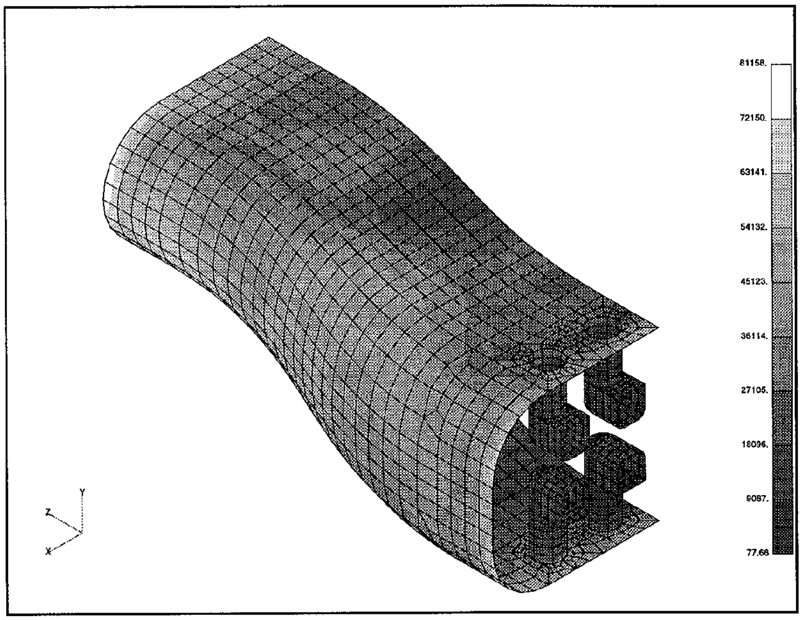
The stress results of this analysis are as follows:



Von Mises Stresses - Pressure Loading

Load Case Combination

FEMAP is also used to combine the static and thermal stressed output cases. The linear combination is also capable of accepting multiplication factors and can combine any number of output sets into a new one. In this case the two output sets are combined, each with a factor of 1.0. FEMAP will only combine entities that can be linearly combined, output such as Von Mises Stresses will not be combined, but will be recalculated by FEMAP from the combined component stresses.



Combined Loading - Von Mises Stress

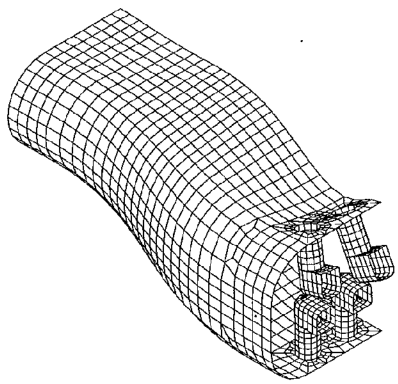
It is interesting to note that the pressure loading actually relieves some of the thermal stress at the inlet opening.

Modal Analysis

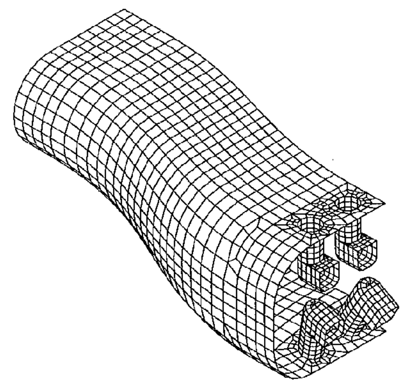
As a final part of this analysis, the same FEA model will be analyzed for its normal modes. When creating input decks for MSC/NASTRAN or any other FEA solver, FEMAP makes it possible to specify what type of analysis will be performed, and collect the necessary data to correctly set up the analysis deck. In the case of MSC/NASTRAN and Normal Modes Analysis, FEMAP makes it possible for the user to select which solution method is used, and to specify other required input parameters.

Modal Analysis Results

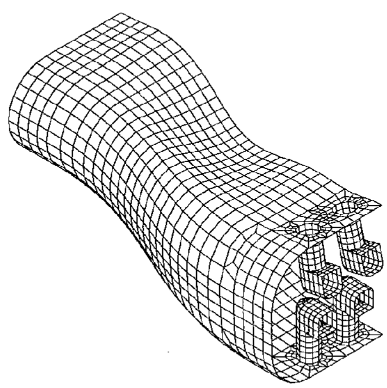
Mode 1	1137.342 Hz
Mode 2	1180.596 Hz
Mode 3	1334.096 Hz
Mode 4	1449.218 Hz
Mode 5	1575.011 Hz
Mode 6	1603.583 Hz
Mode 7	1857.237 Hz
Mode 8	1908.948 Hz
Mode 9	2226.667 Hz
Mode 10	2241.78 Hz



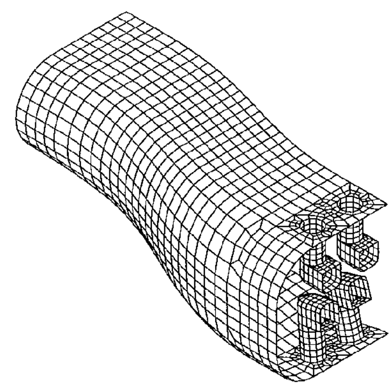
Mode 1



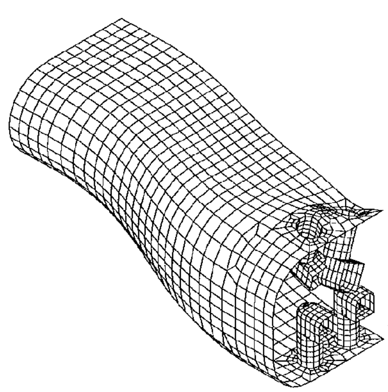
Mode 2



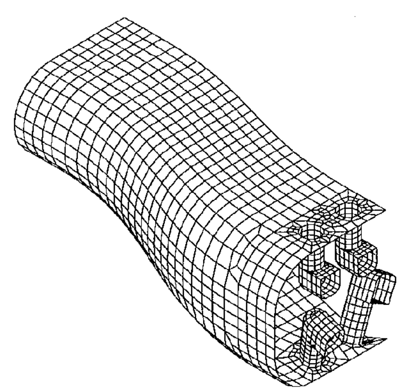
Mode 4



Mode 3



Mode 5



Mode 6

MODE SHAPES 1 THROUGH 6