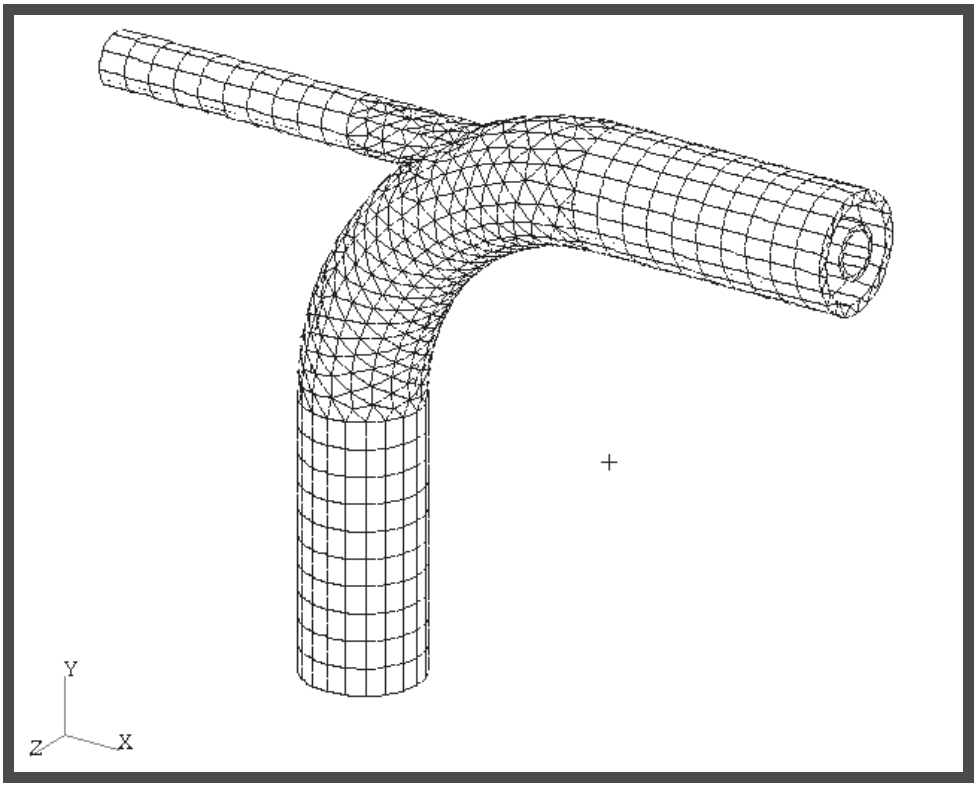
Supplementary Exercise - 5

Pipe Elbow with Intersecting Tube Model



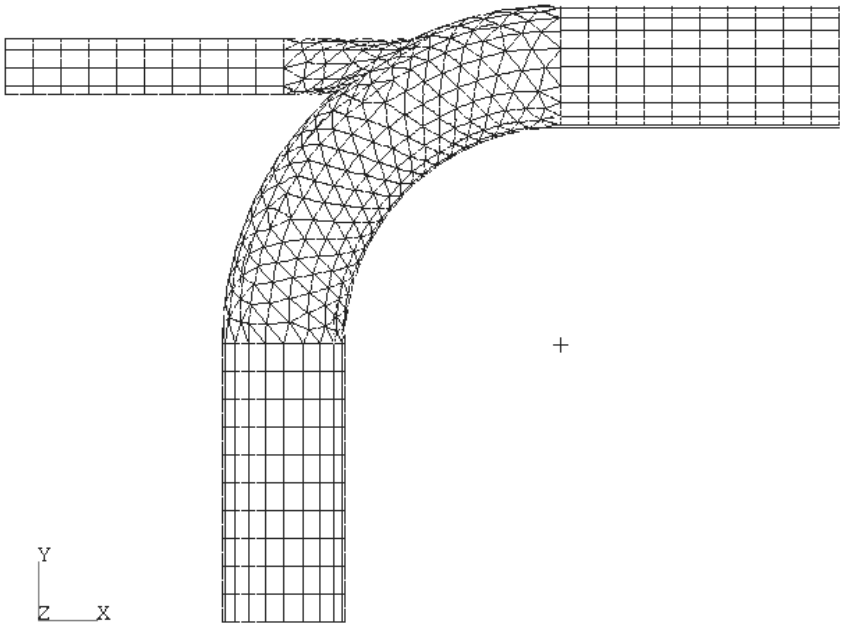
Objective:

- Model an elbow, with a smaller tube piercing it, connected to a straight pipe run.
- Use tet10 and wedge15 elements.

Pipe Elbow

Model Description:

In this exercise you will create a B-rep solid that represents a 90 degree pipe elbow with a straight tube intersecting it. The elbow will be connected to straight pipes. The model will be analyzed to determine it's response for applied pipe internal pressure.



Analysis Code:	MSC/NASTRAN
Element type:	Tet10 and Wedge15
Element Global Edge Length:	0.7

Figure 1-1

Suggested Exercise Steps:

- Create a new database named **`pipe.db`**.
- Change the Tolerance to Default and the Analysis Code to MSC/NASTRAN.
- Create the geometry that represents the elbow and intersecting tube by intersecting and breaking surfaces. From this a B-rep solid is created.
- Tetmesh the B-rep solid that represents the elbow and tube.
- Extrude the free faces of the tetrahedral elements at the ends of the elbow and tube to create the wedge elements that will represent the straight pipe and remainder of the tube.
- Constrain and pressure load the pipe model.
- Define the material and element properties.
- Analyze the model using a linear static simulation.
- Review the results in Patran.

Pipe Elbow

Exercise Procedure:

1. Create a new database and name it **pipe.db**.

File/New...

New Database Name **pipe**

OK

2. Change the *Tolerance* to **Default** and the *Analysis Code* to **MSC/NASTRAN**.

New Model Preference

Tolerance ◆ **Default**

Analysis Code: **MSC/NASTRAN**

OK

3. Create the geometry representing the pipe elbow and intersecting tube.

Create a solid

◆ ***Geometry***

Action: **Create**

Object: **Curve**

Method: **2D Circle**

Circle Radius **2.2**

Construction Plane **Coord 0.1**

Center Point List **[0, 10, 0]**

Apply

Circle Radius **1.8**

Apply

Circle Radius	**1.0**

Apply

Circle Radius	**0.8**

Apply

Create surfaces representing the outside and inside of the elbow.

Action:	**Create**
Object:	**Surface**
Method:	**Revolve**
Total Angle	**90.0**
Curve List	**Curve 1 2**

Apply

Create the surfaces representing the outside and inside of the tube.

Action:	**Create**
Object:	**Surface**
Method:	**Extrude**
Translation Vector	**< -10, 0, 0 >**
Curve List	**Curve 3 4**

Apply

Pipe Elbow

Intersect outside elbow surface with outside tube surface.

Action:	**Create**
Object:	**Curve**
Method:	**Intersect**
Option:	**2 Surface**
Surface 1 List	**Surface 1**
Surface 2 List	**Surface 3**

Apply

Intersect inside elbow surface with outside tube surface.

Surface 1 List	**Surface 2**
Surface 2 List	**Surface 3**

Apply

Break outside elbow surface with outside intersection curve.

Action:	**Edit**
Object:	**Surface**
Method:	**Break**
Option:	**Curve**

☐ ***Delete Original Surface***

Surface List	**Surface 1**
Break Curve List	**Curve 5**

Apply

Break inside elbow surface with inside intersection curve.

Surface List	**Surface 2**
Break Curve List	**Curve 6**

Apply

Break outside tube surface with inside intersection curve.

Surface List	**Surface 3**
Break Curve List	**Curve 6**

Apply

Break outside tube surface with outside intersection curve.

Surface List	**Surface 10**
Break Curve List	**Curve 5**

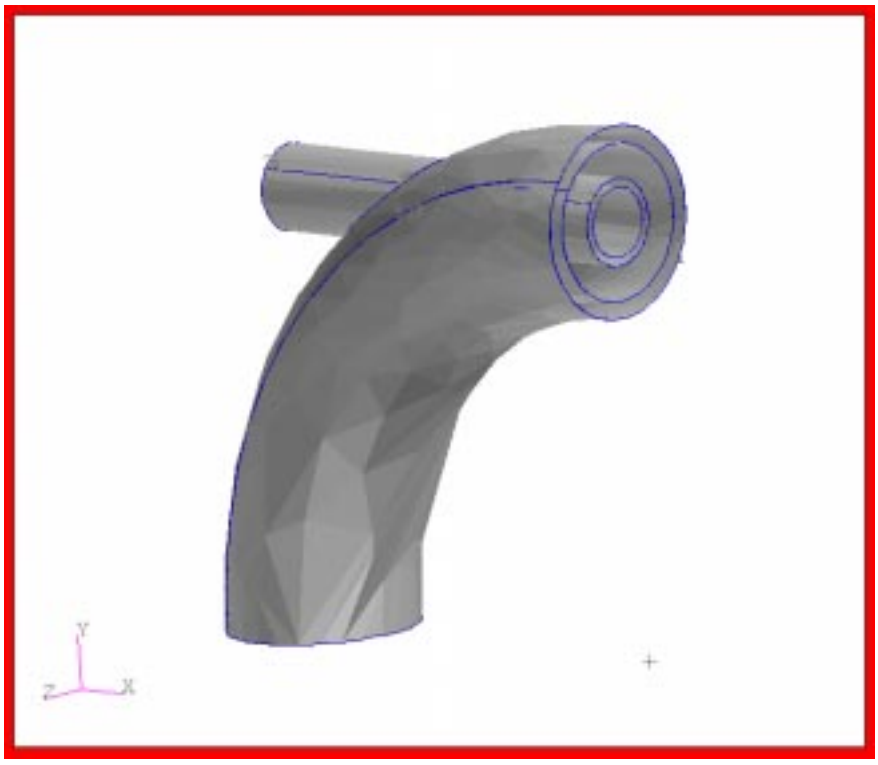
Apply

Delete the eliptical surfaces created by breaking the two elbow surfaces with intersection curves.

Action:	**Delete**
Object:	**Surface**
Surface List	**Surface 6 8**

Apply

The resulting surfaces are shown as below.



Pipe Elbow

Create the surfaces needed to close the ends of the volume.

Action:	**Create**
Object:	**Surface**
Method:	**Curve**
Option:	**2 Curve**
Starting Curve List	**Curve 1**
Ending Curve List	**Curve 2**

Apply

Repeat the last step with curves **3**, **4** and Surfaces **5.3**, **7.3** and **12.2**, **4.2**.

Create a B-rep solid to represent the pipe elbow.

Erase unneeded surfaces. Select the **Plot/Erase** form icon.

Select surfaces 4, 9, 12, 14 and 16, then, erase them.

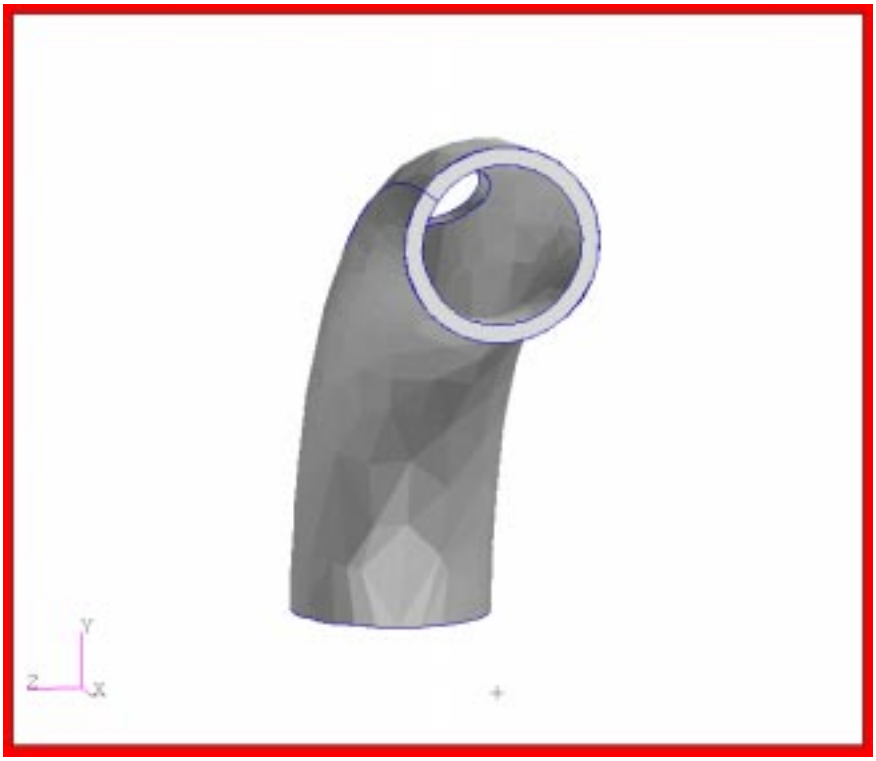
Action:	**Create**
Object:	**Solid**
Method:	**B-Rep**

☐ ***Delete Original Surface***

Surface List	**Surface 5 7 11 13 15**

Apply

Shade and rotate the solid to see what it looks like.



Create a B-rep solid to represent the tube. Plot and erase surfaces as needed. In the **Plot/Erase** form select an additional surface (the one at the hole in the pipe elbow), **Surface 11**. Erase everything from the viewport, then, plot the surfaces in the **Plot/Erase** form list box Selected Entities.

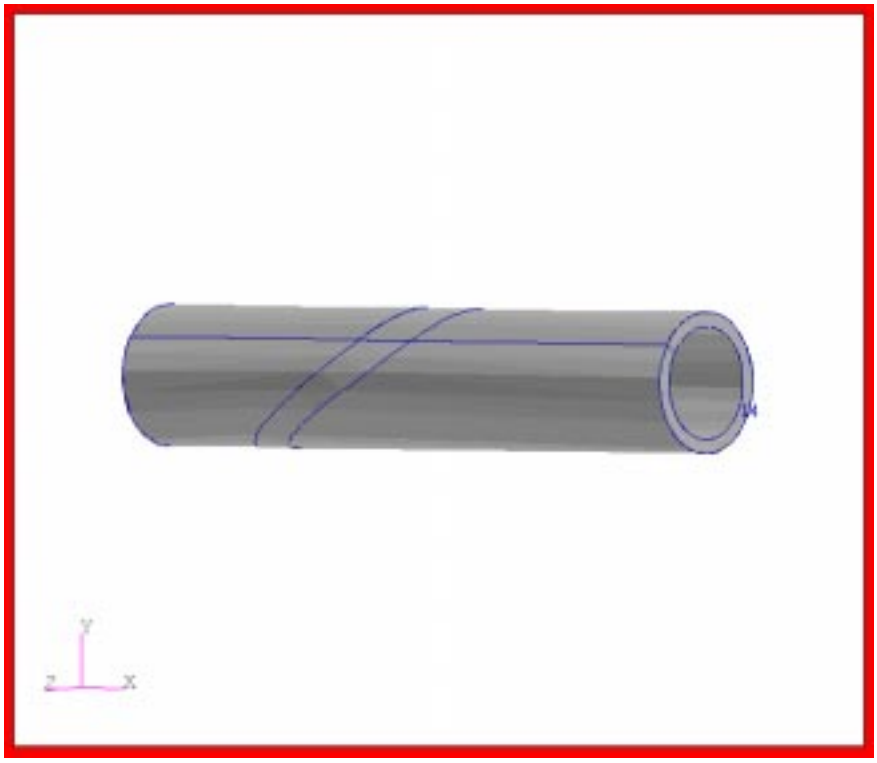
Action:	**Create**
Object:	**Solid**
Method:	**B-Rep**

☐ ***Delete Original Surface***

Surface List	**Surface 4 9 11 12 14 16**

Apply

Pipe Elbow



4. Now create the tetrahedral and wedge meshes for the model. Mesh the two B-rep solids with **tet10** elements.

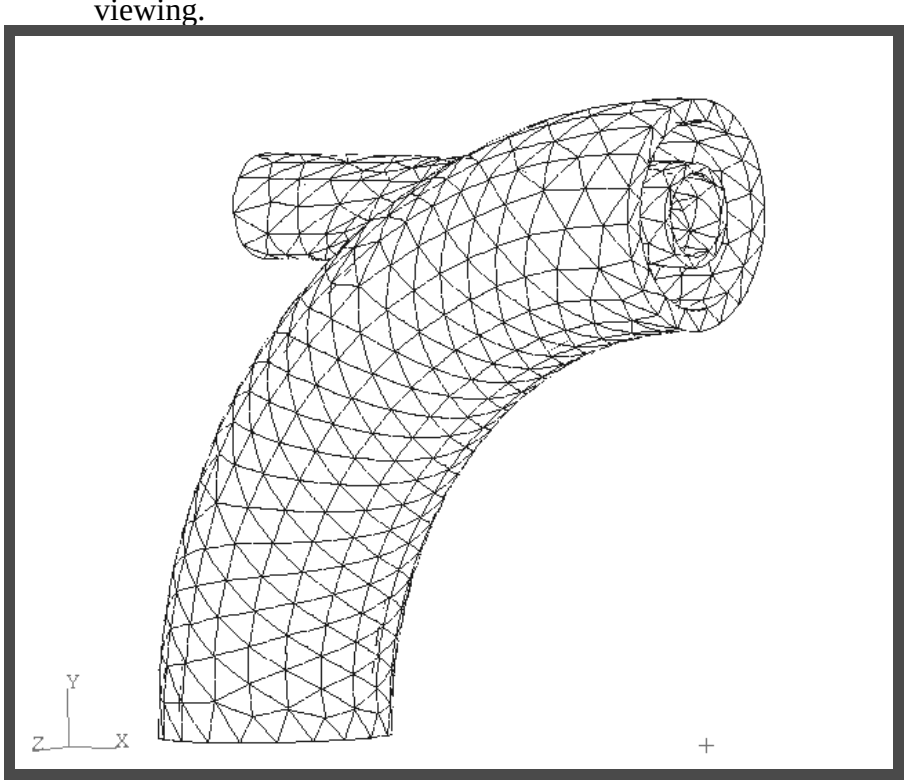
Mesh the model

◆ ***Finite Elements***

Action:	**Create**
Object:	**Mesh**
Type:	**Solid**
Global Edge Length	**0.7**
Mesher:	◆ ***Tet Mesh***
Element Topology	**Tet10**
Input List	**Solid 1 2**

Apply

Erase the geometry and display the meshes using hidden line viewing.



Create the **wedge15** elements to represent the straight pipe runs (connect to the ends of the elbow) and the remainder of the tube.

Action:	**Sweep**
Object:	**Element**
Method:	**Extrude**
Mesh Control...	
◆ ***Number of Elements***	**10**
OK	
Direction Vector	**< 1 0 0 >**
Extrude Distance	**10**
Base Entity List	**Select free faces of tet10's at one end of the pipe elbow**
Apply	

Pipe Elbow

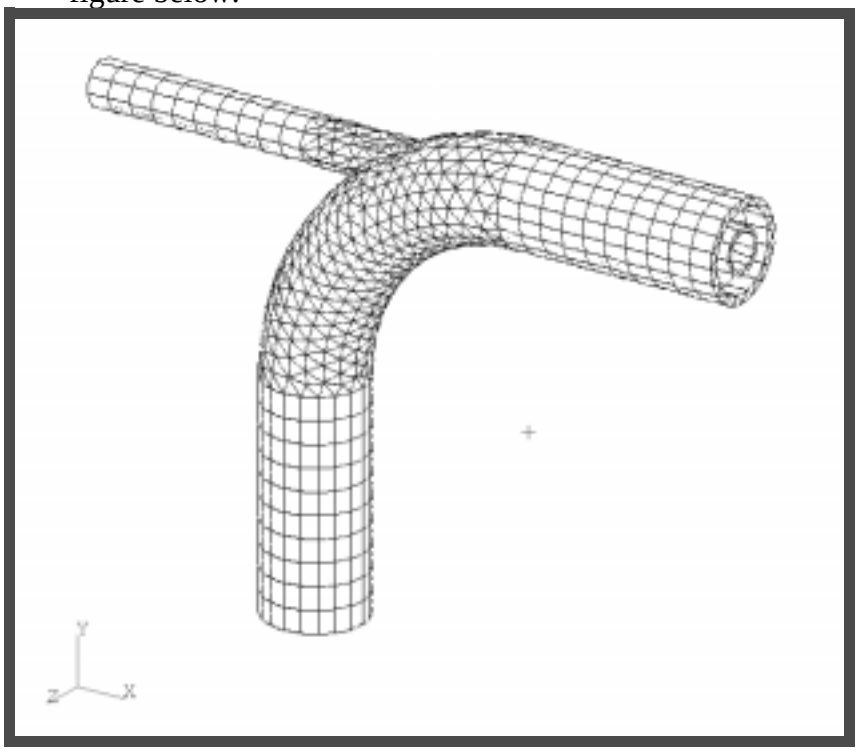
To create the remaining **wedge15** elements use the Direction Vector <0 -1 0> or <-1 0 0>. The other parameter values remain unchanged.

In assembling a pipe elbow with an intersecting tube it is possible to either create a single forging or drill a hole through the elbow, insert the tube and weld it to the elbow. Depending on how the elbow tube assembly is created will determine how the equivalencing must be done. For this problem it is assumed that a forging is used, so the equivalencing can be done using all the nodes in the model, as opposed to just equivalencing where fillet welds would have been.

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**

Apply

Your finite element model should look like the one shown in the figure below.



Create Loads and BCs

5. Create Loads and Boundary Conditions. Specify pressure loading inside the elbow and straight pipe runs, and outside the tube.

◆ ***Loads/BCs***

Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
New Set Name	**elbow_pressure**
Target Element Type:	**3D**

Input Data...

Pressure	**50**

OK

Select Application Region...

Select Solid Faces	**Solid 1.2 2.2**

Add

OK

Apply

New Set Name	**pipe_pressure**

Input Data...

Pressure	**50**

OK

Select Application Region...

◆ ***FEM***

Select 3D Element Faces	**Select wedge15 free faces that make-up the interior of the straight pipe runs and exterior of the portion of the tube inside the piping.**

Add

OK

Apply

Specify pressure loading inside the tube.

Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
New Set Name	**wedge_tube_pressure**

Input Data...

Pressure	**10**

OK

Select Application Region...

Select 3D Element Faces	**Select wedge15 free faces that make up the interior of the tube**

Add

OK

Apply

New Set Name	**solid_tube_pressure**

Input Data...

Pressure	**10**

OK

Select Application Region...

◆ ***Geometry***

Select Solid Faces	**Solid 2.1**

Add

OK

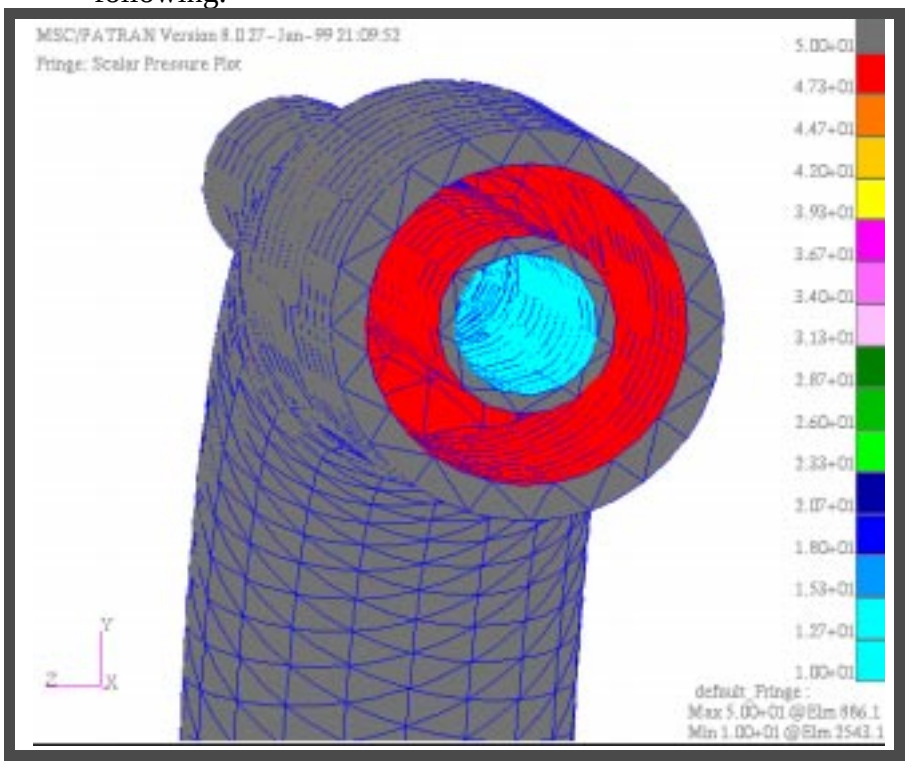
Apply

Action:	**Plot Contours**
Object:	**Pressure**
Existing Sets:	**elbow_pressure** **pipe_pressure** **wedge_tube_pressure** **solid_tube_pressure**
Select Data Variable:	**Pressure**
Select Groups:	**default_group**

Apply

Reset Graphics

The contours (fringe) for the applied pressure should look like the following.



Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**

Pipe Elbow

New Set Name **clamped_edges**

Input Data...

Translations **< 0 0 0 >**

OK

Select Application Region...

◆ *FEM*

Select Nodes **Select nodes at the ends of the pipe/tube (two ends each)**

OK

Apply

Action: **Create**

Object: **Force**

Type: **Nodal**

New Set Name **load**

Input Data...

Force **< 100 0 0 >**

OK

Select Application Region...

◆ *FEM*

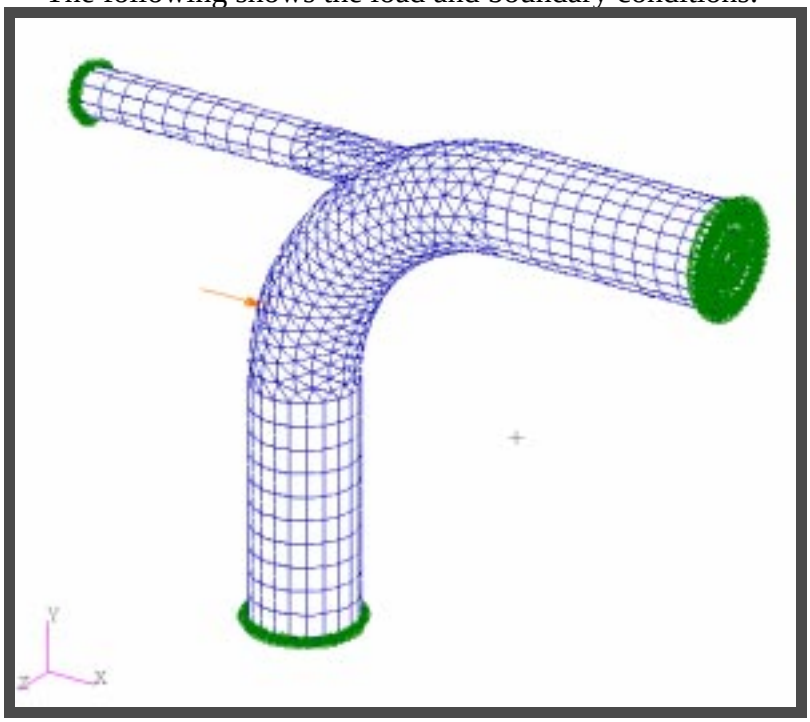
Select Nodes **Select a node similar to the one selected on the following diagram**

Add

OK

Apply

The following shows the load and boundary conditions.



Create Material

6. Create a material for aluminum called **alum**.

◆ ***Materials***

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**alum**
Input Properties...	
Elastic Modulus	**10e6**
Poisson Ratio	**0.3**
Density	**0.000259**

Apply

Cancel

7. Define element properties called **pipe**.

Create Property

◆ ***Properties***

Action:	**Create**
Object:	**3D**
Type:	**Solid**
Property Set Name	**pipe**
Input Properties...	
Material name	**m:alum**
OK	
Select Members	**Select all elements**
Add	
Apply	

8. Analyze the model

◆ ***Analysis***

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Apply	

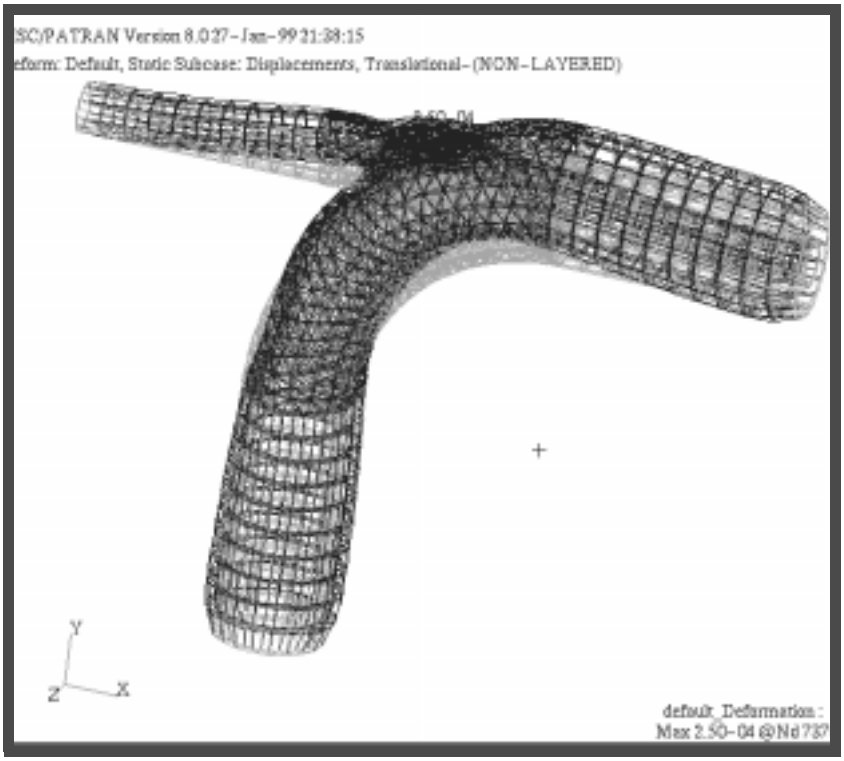
9. Read in the results.

Action:	**Read Output2**
Object:	**Result Entities**
Method:	**Translate**
Select Results File...	
Selected Results File:	**pipe.op2**
OK	
Apply	

10. Display the displaced shape of the model using Results. Because the elbow forging in this problem used a sharp transition from the elbow to tube and because the applied load was at a single node, only displacements should be used in studying the results.

◆ ***Results***

Action:	**Create**
Object:	**Quick Plot**
Select Result Cases:	**Default, Static Subcase**
Select Deformation Result:	**Displacements, Translational**
Apply	



11. To complete the exercise, close the database.

File/Quit...