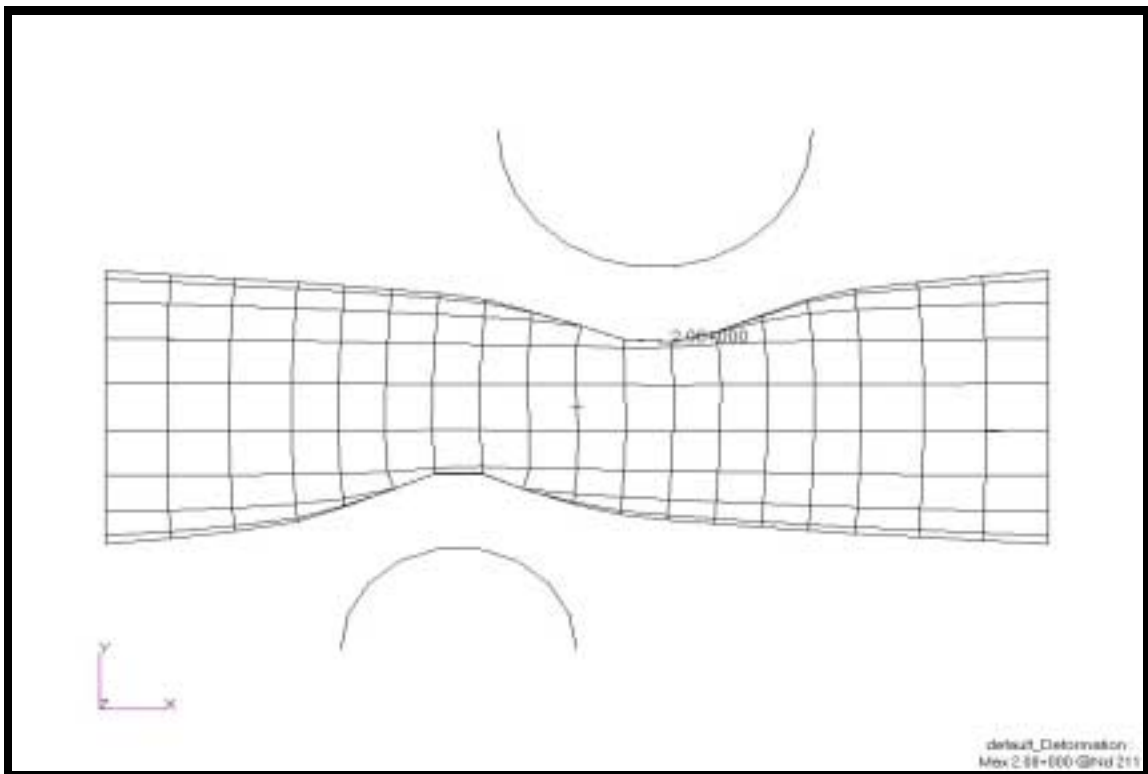


LESSON 12

Crushed Pipe



Objectives:

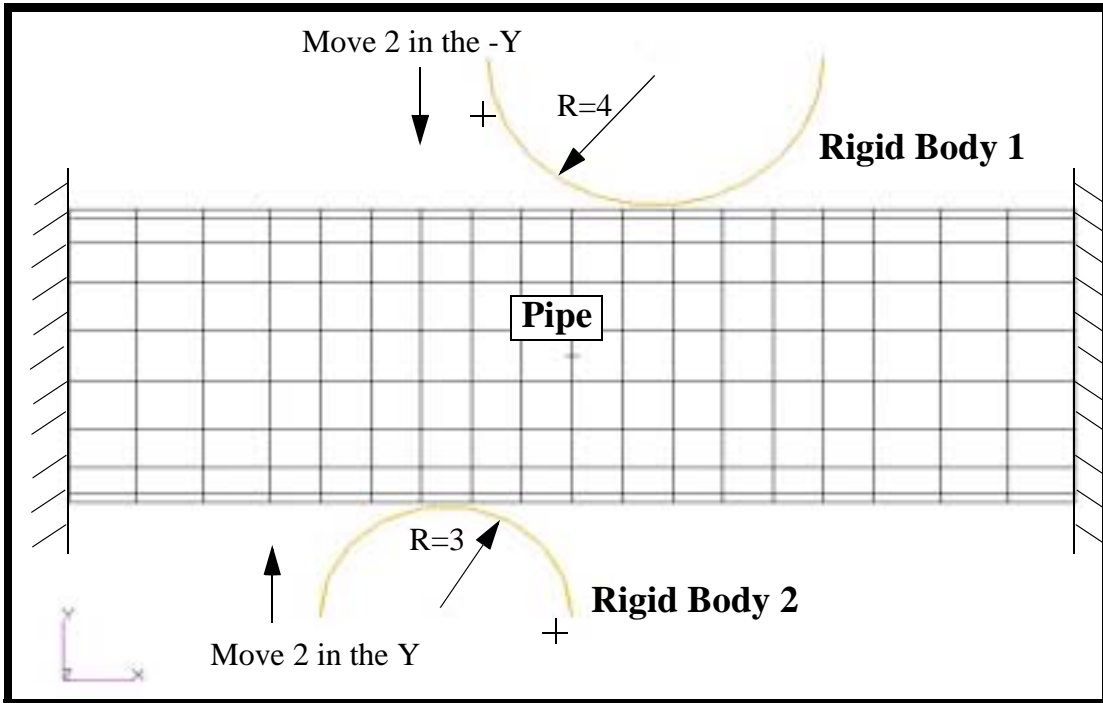
- Large displacement analysis.
- Rigid contact analysis using rigid surfaces.
- Plasticity modeled with perfectly plastic material.



Model Description:

In this exercise, a model of a cylindrical pipe is modeled as being crushed between rigid bodies.

The model is created using 2D thick shell elements to model the pipe. All three of the model entities will be covered with 2D Contact elements in order to model the contact between the pipe and the rigid bodies.

**Data for Exercise:**

Pipe Dimensions:	D = 8 Length = 24 Thickness = 0.4
Pipe Material:	Steel, E = 30E6 $\nu = 0.3$ Yield Stress = 36,000
Mesh-	Rigid Body 1 and 2: 3D analytical Pipe: S4R elements with IRS4 elements to contact rigid bodies

Exercise Procedure:

1. Open a new database. Name it **crush.db**.

File/New ...

Database Name:

crush.db

OK

Change the *Analysis Preference* to **MSC.Marc**.

Analysis Code:

MSC.Marc

OK

2. Create a new group **rigid**.

Group/Create...

New Group Name:

rigid

■ **Make Current**

Group Contents:

Add Entity Selection

Apply

Cancel

3. Create the model geometry.

■ **Geometry**

Action:

Create

Object:

Point

Method:

XYZ

Points Coordinates List:

[-3, -7.1, 4.5]

Apply

Create a second point by changing the *Point Coordinates List* to **[0, -7.1, 4.5]**.

Apply

4. Sweep point to make a curve for the lower rigid surface.

First, turn on the entity labels using the following toolbar icon:

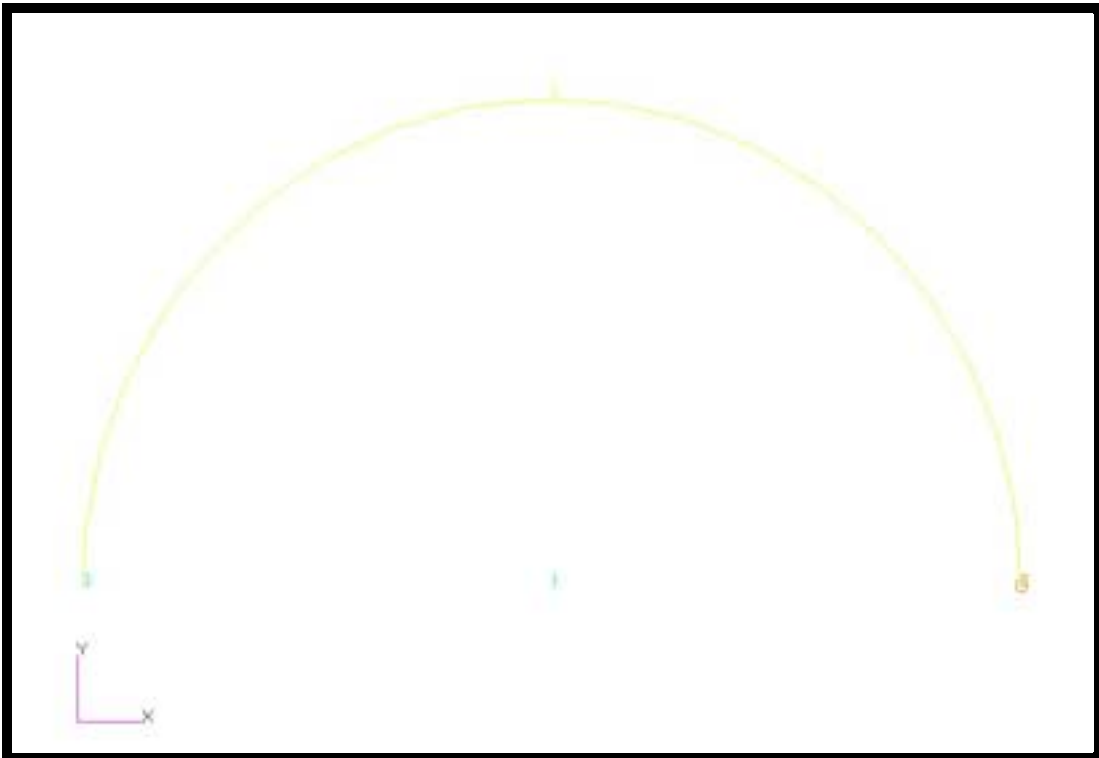


Show Labels

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Curve"/>
<i>Method:</i>	<input type="text" value="Revolve"/>
<i>Axis:</i>	<input type="text" value="{Point 1[X1 Y1 5.0]}"/>
<i>Total Angle:</i>	<input type="text" value="180"/>
<i>Point List:</i>	<input type="text" value="Point 2"/>
<input type="button" value="Apply"/>	

The screen should now display the points and curves as shown in Figure 12.1:

Figure 12.1 - Curve for lower rigid surface



5. Create the geometric points for the upper rigid surface

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Point"/>
<i>Method:</i>	<input type="text" value="XYZ"/>
<i>Points Coordinates List:</i>	<input type="text" value="[2, 8.1, 4.5]"/>
<input type="button" value="Apply"/>	

This will create **Point 4**

In a similar manner, create **Point 5** using the **Create, Point, XYZ** option. The Point Coordinates will be [6, 8.1, 4.5].

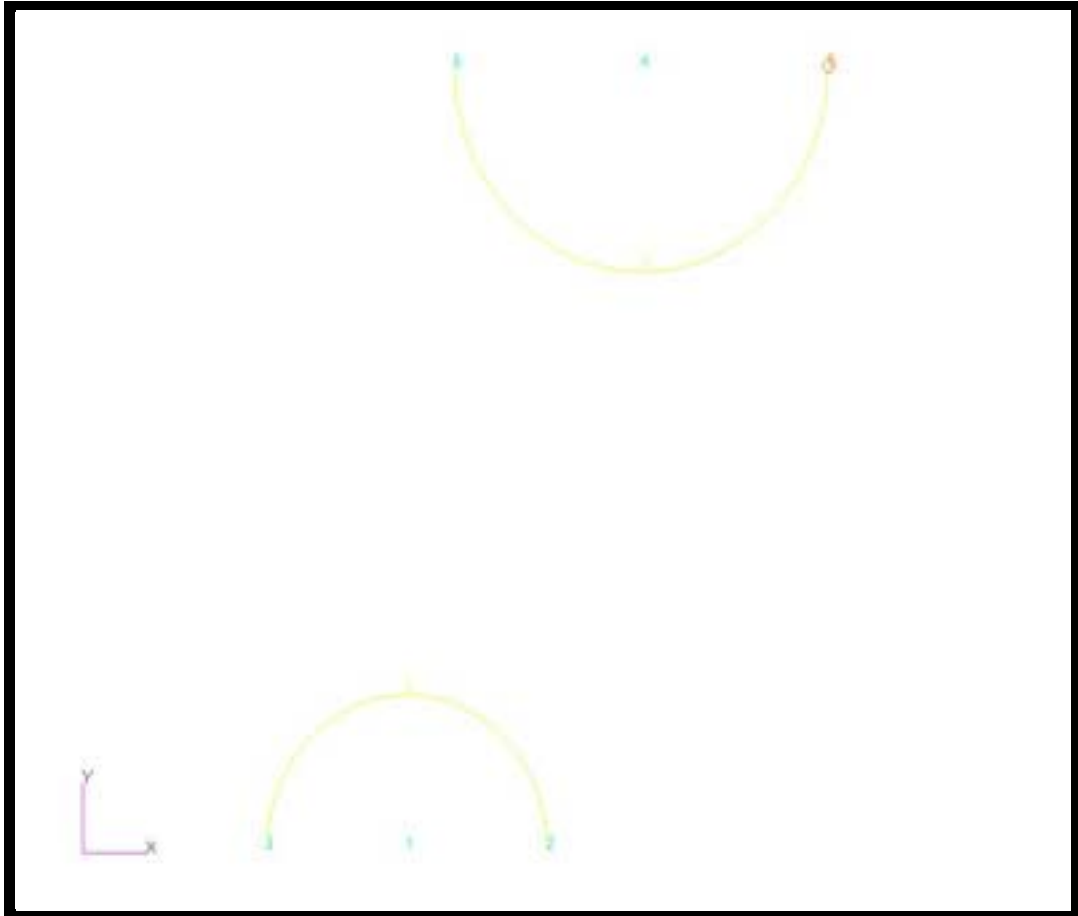
6. Create the curves for the upper rigid surface

Sweep **Point 5** into an arc using **Point 4** as the axis

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Curve"/>
<i>Method:</i>	<input type="text" value="Revolve"/>
<i>Axis:</i>	<input type="text" value="{Point 4[X4 Y4 5]}"/>
<i>Total Angle:</i>	<input type="text" value="-180"/>
<i>Point List:</i>	<input type="text" value="Point 5"/>
<input type="button" value="Apply"/>	

This ends the definition of the geometry of the upper and lower rigid bodies. The screen should now show the points and curves displayed in Figure 12.2:

Figure 12.2 - Curves to form rigid surfaces



7. Create a new group **pipe** and the geometry for the pipe.

Group/Create*New Group Name:***pipe****■ Make Current***Group Contents:***Add Entity Selection****Apply****Cancel**

First, change to the isometric view by clicking on the **Iso1 View** icon in the toolbar.



Iso 1 View

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Point"/>
<i>Method:</i>	<input type="text" value="XYZ"/>
<i>Points Coordinates List:</i>	<input type="text" value="[-12 0 0]"/>
<input type="button" value="Apply"/>	

In a similar manner, using the **Create, Point, XYZ** method to create **Point 8** by changing the *Point Coordinates List* to **[-12 4 0]**.

Sweep **Point 8** into a circle

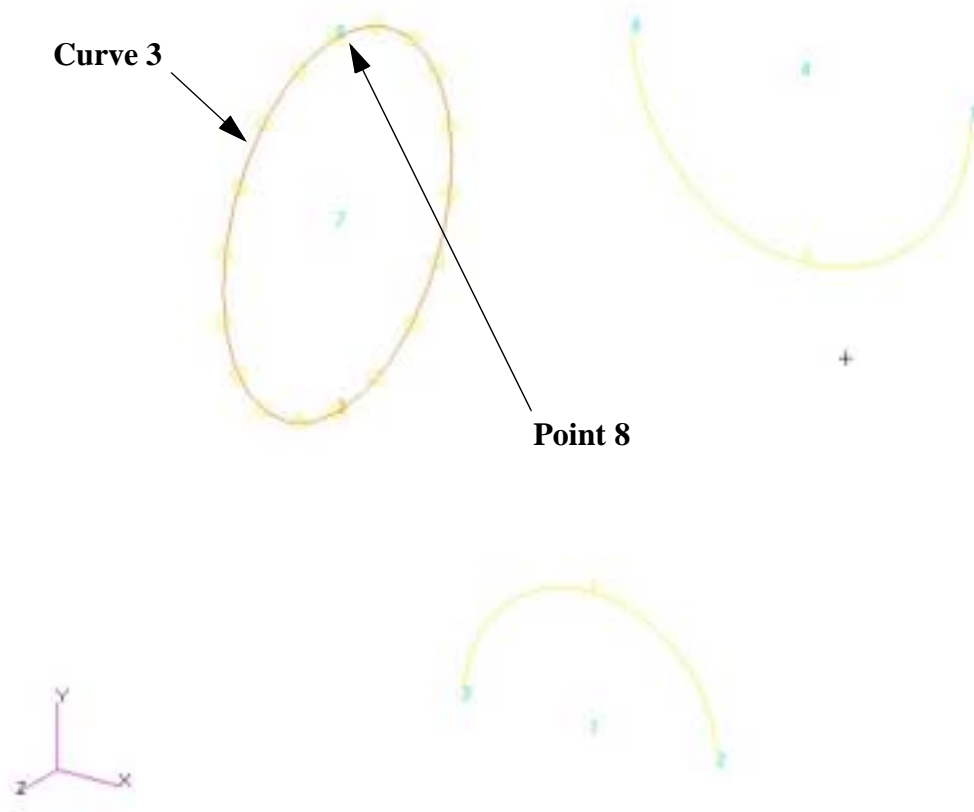
<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Curve"/>
<i>Method:</i>	<input type="text" value="Revolve"/>
<i>Axis:</i>	<input type="text" value="{Point 7[1 Y7 Z7]}"/>
<i>Total Angle:</i>	<input type="text" value="360"/>
<i>Point List:</i>	<input type="text" value="Point 8"/>
<input type="button" value="Apply"/>	

8. Create the mesh seed for the circle.

■ Elements

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh Seed"/>
<i>Type:</i>	<input type="text" value="Uniform"/>
<i>Element Edge Length Data:</i>	<input checked="" type="radio" value="Number of Elements"/>
<i>Number:</i>	<input type="text" value="18"/>
<i>Curve List:</i>	<input type="text" value="See Figure 12.3 to choose Curve 3"/>

Figure 12.3 - Point to form curve and curve to place mesh seeds on



- Now you will create the mesh and extrude the elements to represent the pipe.

First, create a group called **fem_pipe**.

Group/Create

New Group Name:

fem_pipe

■ **Make Current**

Group Contents:

Add Entity Selection

Apply

Cancel

Mesh the curve

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh"/>
<i>Type:</i>	<input type="text" value="Curve"/>
<i>Global Edge Length:</i>	<input type="text" value="0.1"/>
<i>Element Topology:</i>	<input type="text" value="Bar 2"/>
<i>Curve List:</i>	<input type="text" value="see Figure 12.3 and choose Curve 3"/>

Now you will extrude the elements

First, be sure to set the view to the default using the following toolbar icon:



Front View

<i>Action:</i>	<input type="text" value="Sweep"/>
<i>Object:</i>	<input type="text" value="Element"/>
<i>Type:</i>	<input type="text" value="Extrude"/>

<i>Mesh Control Data:</i>	<input type="radio"/> Number of Elements
<i>Number:</i>	<input type="text" value="3"/>

<i>Direction Vector:</i>	<input type="text" value="<4.8, 0, 0>"/>
--------------------------	--

Delete Original Elements

<i>Base Entity List:</i>	<input type="text" value="Select the elements you created on the circle (see Figure 12.4)"/>
--------------------------	--

To select these elements you will have to click on the **Elements** icon once you click in the *Base Entity List* databox.



Elements

Then click on the **Beam Element** icon.



Beam Element

Figure 12.4 - Bar elements to sweep into surface elements



Apply

Repeat this process with

Action:

Sweep

Object:

Element

Type:

Extrude

Mesh Control...

Mesh Control Data:

● Number of Elements

Number:

12

OK

Direction Vector:

<14.4, 0, 0>

Base Entity List:

see Figure 12.5

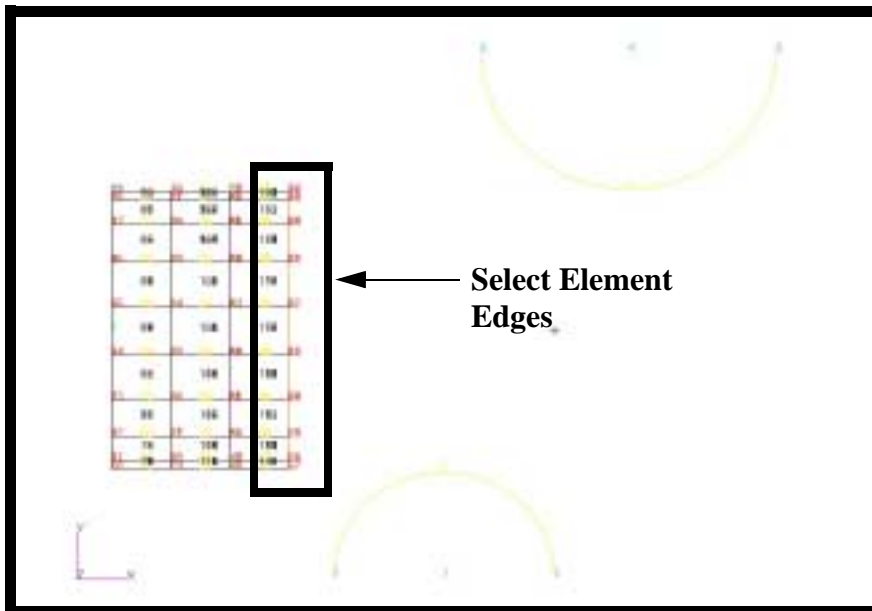
First, click on the **Element's Edge** icon



Element's Edge

Select the edge of the element as shown in Figure 12.5:

Figure 12.5 - Element edges to sweep into surface elements



Apply

Repeat the process once again with the original settings

Action:

Sweep

Object:

Element

Type:

Extrude

Mesh Control...

Mesh Control Data:

Number of Elements

Number:

3

OK

Direction Vector:

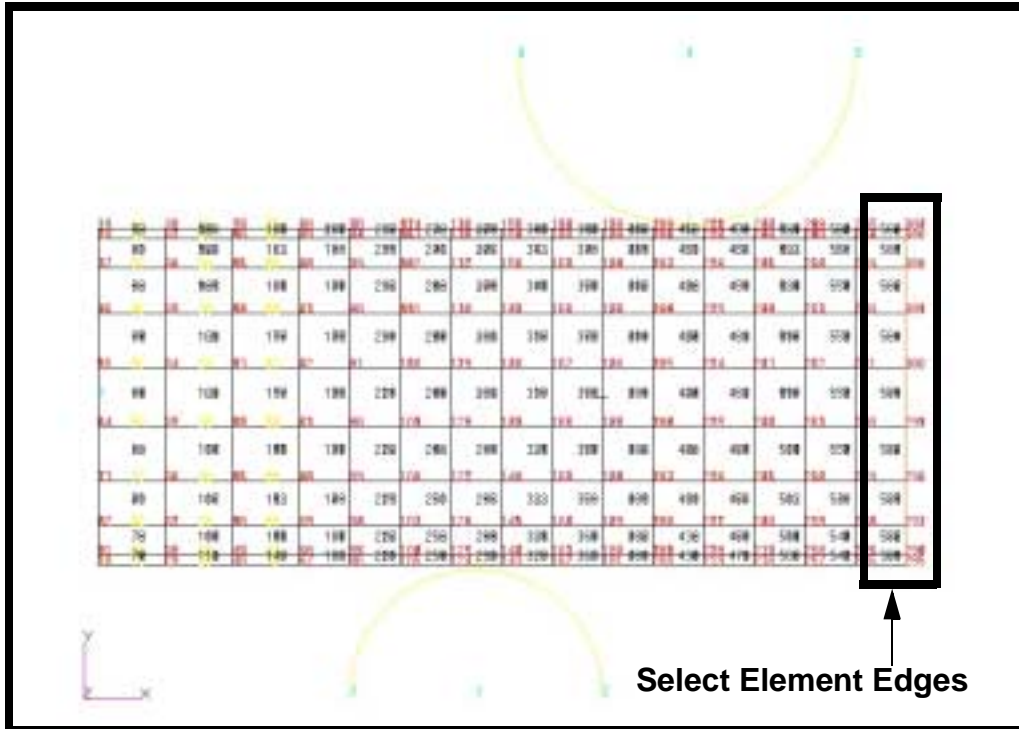
<4.8, 0, 0>

Base Entity List:

see Figure 12.6

Apply

Figure 12.6 - Element edges to sweep into surface elements



10. Create the upper rigid surface elements.

First, post only the group **rigid**

Group/Post...

Select Groups to Post:

rigid

Apply

Cancel

Create a beam mesh on the upper rigid bodies.

Action:

Create

Object:

Mesh

Type:

Curve

Global Edge Length:

1

Element Topology:

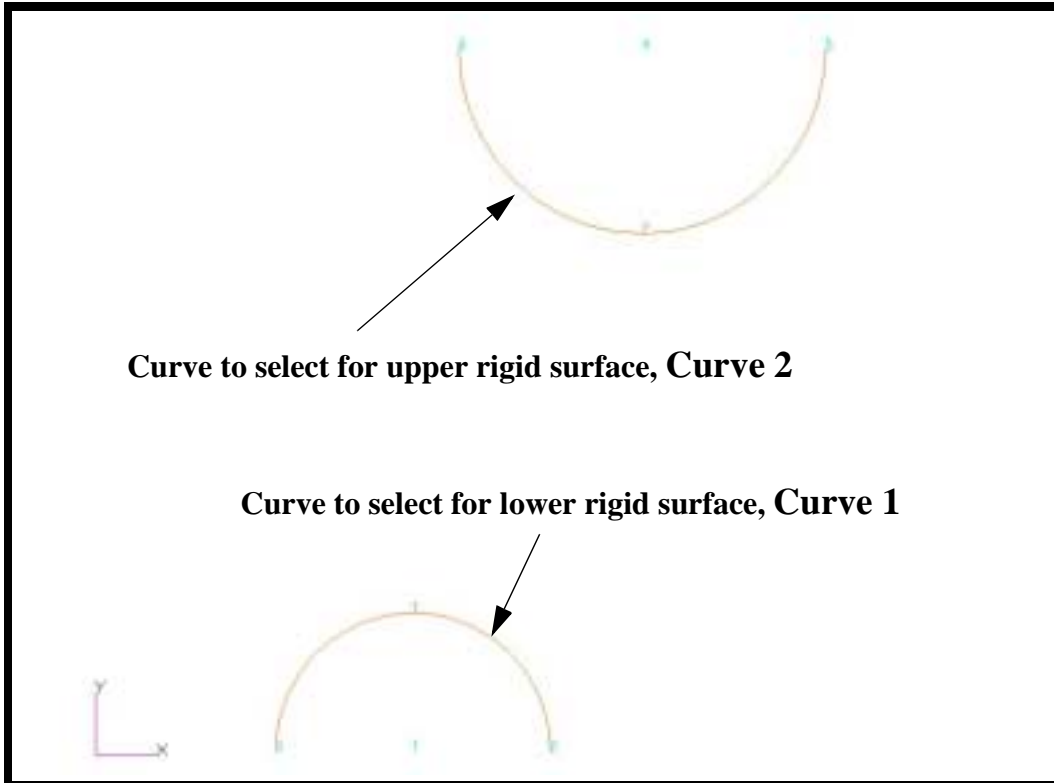
Bar2

Curve List:

select the upper curve as in
Figure 12.7, **Curve 2**

Apply

Figure 12.7 - Curves to sweep into rigid surface elements



11. Create the elements for the lower rigid surface.

In a similar manner, create a beam mesh on the lower rigid body. The *Global Edge Length* is = **1** and pick the curve shown at the bottom of Figure 12.8. The *Curve List* should be **Curve 1**. See Figure 12.7.

12. Give the rigid surfaces depth by using

Action:

Sweep

Object:

Element

Type:

Extrude

Mesh Control...

Mesh Control Data:

● **Number of Elements**

Number:

Direction Vector:

■ **Delete Original Elements**

Base Entity List:

Again you will have to click on these two icons to select the bar elements



Elements



Beam Element

13. Equivalence the finite element nodes (eliminate any extra, overlapping nodes created by the mesher).

Action:

Object:

Type:

14. Verify that the Rigid Body element normals face the pipe.

In a later step, these elements will be used to define a rigid boundary. The rigid body outward direction is defined by the element's normal direction. Therefore, we need to be sure that the normals for each of the rigid boundaries are pointing towards the pipe.

Action:

Object:

Test:

● **Draw Normal Vectors**

All of the element normals for the rigid surfaces should point towards the pipe. If any of them do not, reverse their direction by following this procedure:

<i>Action:</i>	<input type="text" value="Modify"/>
<i>Object:</i>	<input type="text" value="Element"/>
<i>Method:</i>	<input type="text" value="Reverse"/>
<i>Element List:</i>	<input type="text" value="select the elements which had normals facing the wrong direction"/>
<input type="text" value="Apply"/>	

Remove the entity labels using the following toolbar icon:



Hide Labels

15. Create a new group called **fem_all** which will be used for post processing later.

Group/Create...

<i>New Group Name:</i>	<input type="text" value="fem_all"/>
■ Make Current	
<i>Group Contents:</i>	<input type="text" value="Add All FEM"/>
<input type="text" value="Apply"/>	
<input type="text" value="Cancel"/>	

16. Create the material properties for the pipe.

■ **Materials**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Isotropic"/>
<i>Method:</i>	<input type="text" value="Manual Input"/>
<i>Material Name:</i>	<input type="text" value="steel"/>
<input type="text" value="Input Properties..."/>	
<i>Constitutive Model:</i>	<input type="text" value="Elastic"/>

Elastic Modulus:

Poisson's Ratio:

The model will also experience yielding during the analysis, so a Plastic Constitutive Model needs to be defined.

Input Properties...

Constitutive Model:

Type:

Yield Stress:

17. Post the group **fem_pipe** only.

Group/Post...

Select Groups to Post:

18. Now create the Element Properties for your model.

■ **Properties**

Action:

Dimension:

Type:

Property Set Name:

Option(s):

Input Properties...

Material Name:

steel

Shell Thickness:

0.4

OK

Select Members:

select the 2D Element icon in the Select menu and then select *all* elements displayed



Shell Element

Add

Apply

19. Now apply the loads and boundary conditions to the end of the pipe.

The ends of the pipe are to be fixed in all translations. The upper rigid surface will then be forced down into the pipe and lower rigid surface forced up into the pipe.

■ **Loads/BCs**

Action:

Create

Object:

Displacement

Method:

Nodal

New Set Name:

end_disp

Input Data...

Translations <T1,T2,T3>:

< 0, 0, 0 >

Rotations <R1,R2,R3>:

< 0, 0, 0 >

OK

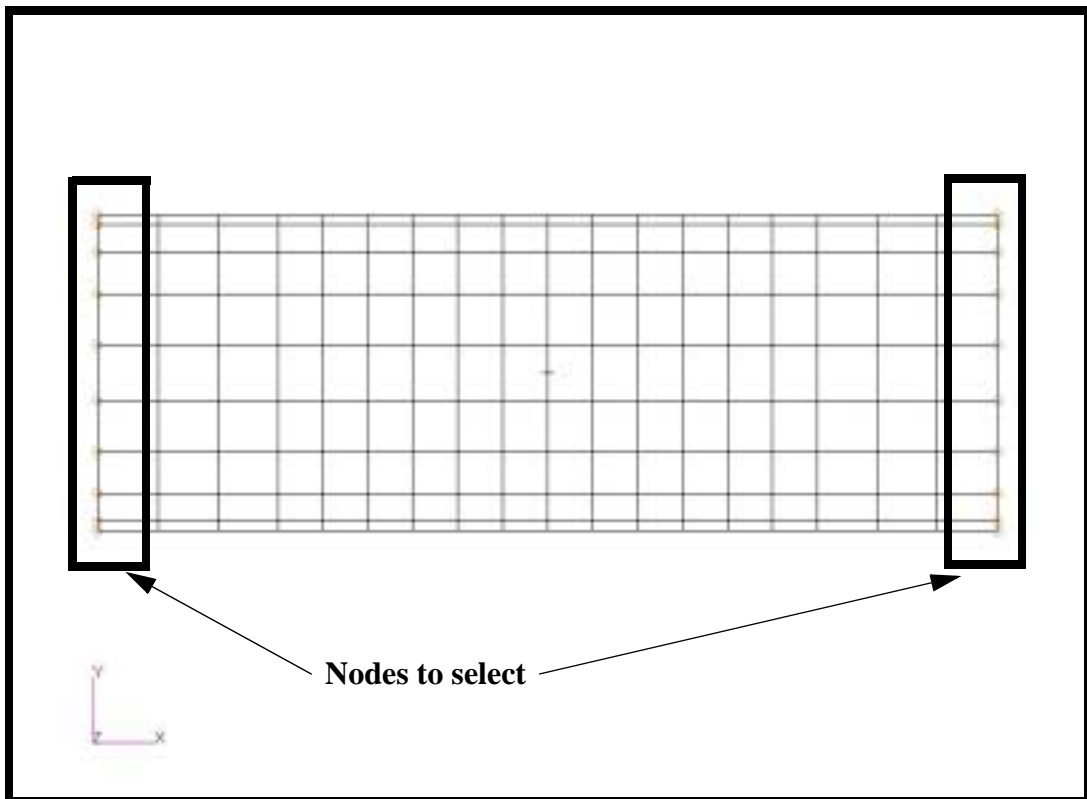
Select Application Region...

Geometry Filter:

● FEM

Click in the *Select Nodes* databox, and screen select the nodes (while holding down shift) at both ends of the pipe as shown in Figure 12.8:

Figure 12.8 - Nodes at fixed ends of the pipe



20. Now create the **Load and Boundary Conditions** for the contact. First, begin with the pipe as the deformable body.

Post the group **fem_all**.

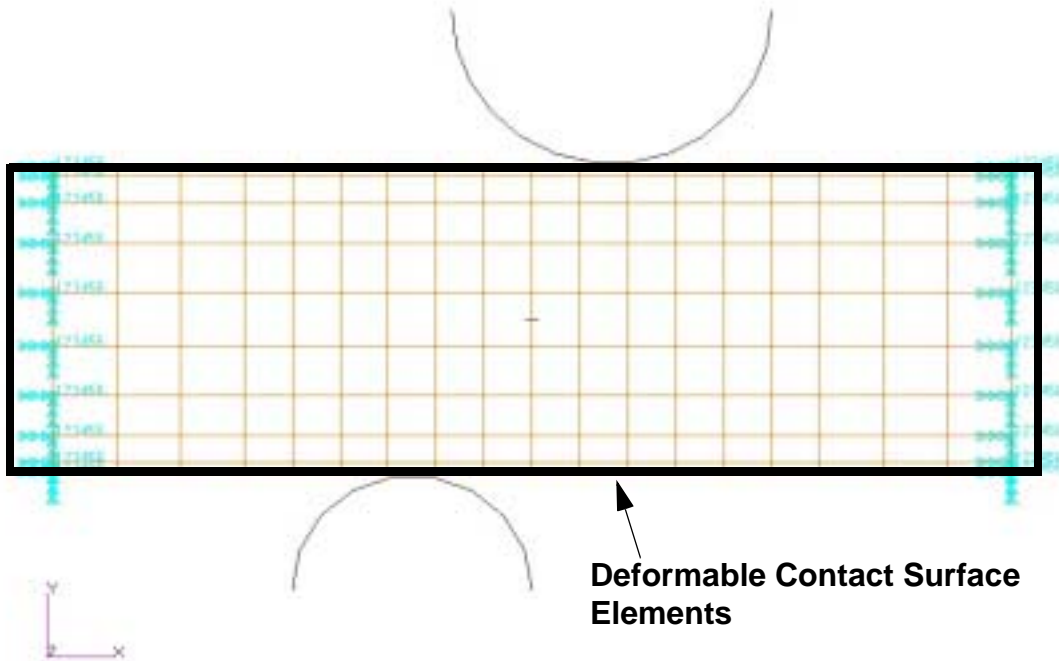
Group/Post...

Select Groups to Post:

<i>Action:</i>	Create
<i>Object:</i>	Contact
<i>Method:</i>	Element Uniform
<i>Option:</i>	Deformable Body
<i>New Set Name:</i>	contact_mid
<i>Target Element Type:</i>	2D
Select Application Region...	
<i>Geometry Filter:</i>	● FEM
<i>Select 2D Elements:</i>	select pipe Elements, as in Figure 12.9, Elm 1:324
Add	
OK	
Apply	

Next, develop the contact LBCs for the top rigid body.

Figure 12.9 - Elements representing pipe contact



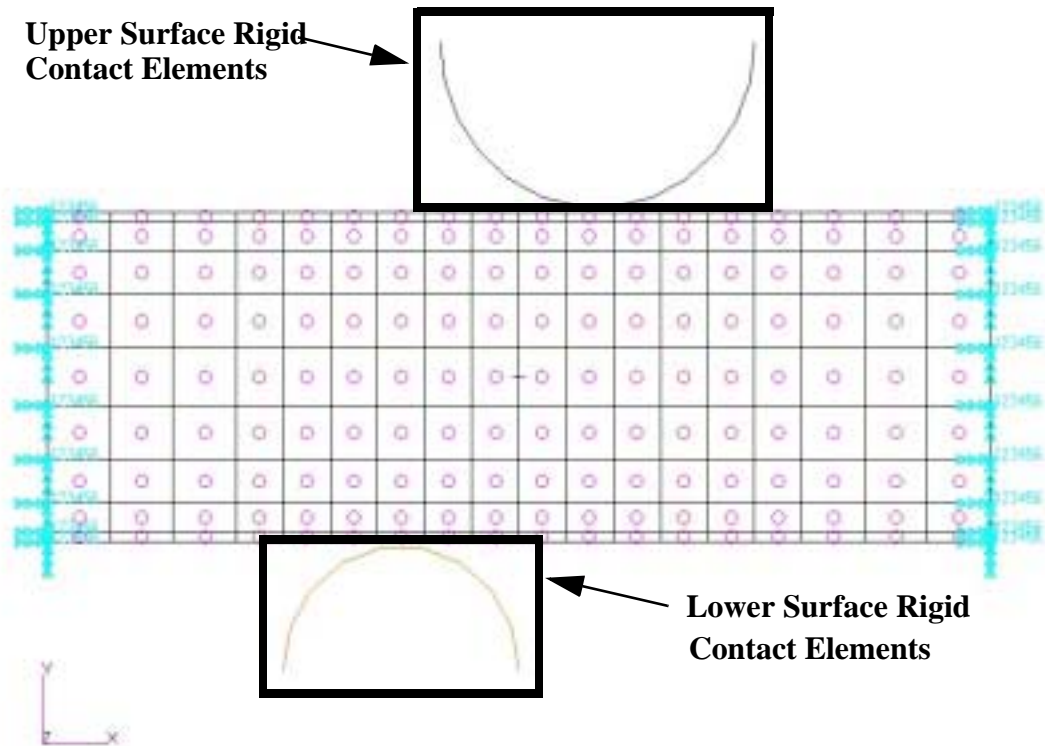
<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Contact"/>
<i>Method:</i>	<input type="text" value="Element Uniform"/>
<i>Option:</i>	<input type="text" value="Rigid Body"/>
<i>New Set Name:</i>	<input type="text" value="contact_top"/>
<i>Target Element Type:</i>	<input type="text" value="2D"/>
<input type="text" value="Input Data..."/>	
<i>Motion Control:</i>	<input type="text" value="Position"/>
<i>Displacement (vector):</i>	<input type="text" value="<0, -2, 0>"/>
<input type="text" value="OK"/>	
<input type="text" value="Select Application Region..."/>	
<i>Select 2D Elements:</i>	<input type="text" value="see Figure 12.10, to select upper surface elements"/>

Add

OK

Apply

Figure 12.10 - Elements representing upper and lower surface contact



Repeat the same procedure for the bottom

<i>Action:</i>	Create
<i>Object:</i>	Contact
<i>Method:</i>	Element Uniform
<i>Option:</i>	Rigid Body
<i>New Set Name:</i>	contact_bottom
<i>Target Element Type:</i>	2D
Input Data...	
<i>Motion Control:</i>	Position

Displacement (vector):

<0, 2, 0>

OK

Select Application Region...

Select 2D Elements:

see Figure 12.10 for lower surface elements

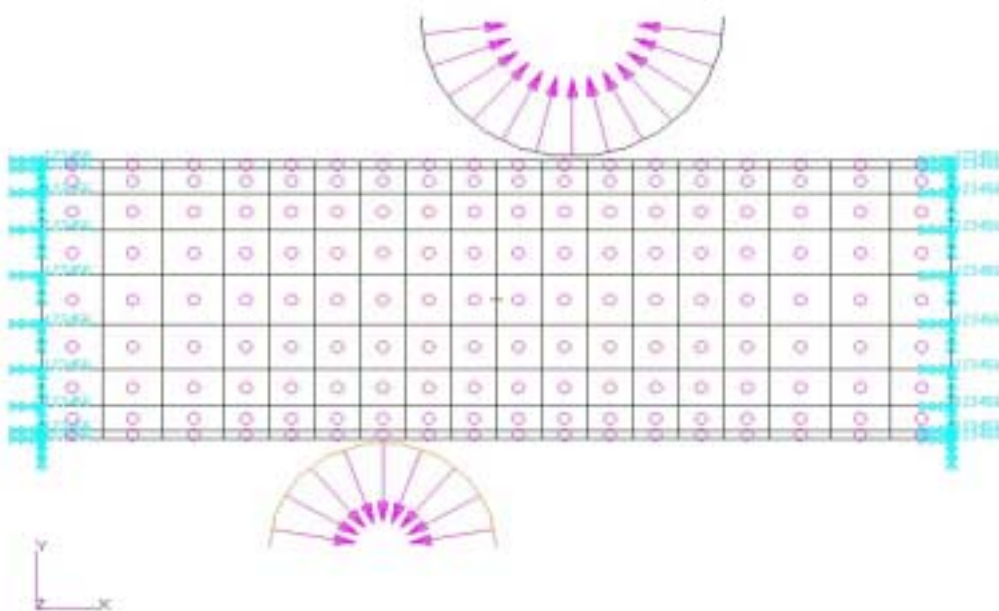
Add

OK

Apply

Your model should now look like the picture shown in Figure 12.11:

Figure 12.11 - Correct contact normal definitions



Note: If the arrows on your model are pointing outward, towards the pipe, then you must perform this step. If not, skip to the next step.

■ Load/BCs

Action:

Modify

<i>Object:</i>	<input type="text" value="Contact"/>
<i>Method:</i>	<input type="text" value="Element Uniform"/>
<i>Option:</i>	<input type="text" value="Rigid Body"/>
<i>Select Set to Modify:</i>	<input type="text" value="contact_bottom or top"/>

■ Flip Contact Side

21. Your model is now ready for analysis.

■ Analysis

<i>Action:</i>	<input type="text" value="Analyze"/>
<i>Object:</i>	<input type="text" value="Entire Model"/>
<i>Method:</i>	<input type="text" value="Full Run"/>
<i>Job Name:</i>	<input type="text" value="crush"/>

<i>Job Step Name:</i>	<input type="text" value="pipe_crush"/>
<i>Solution Type:</i>	<input type="text" value="Static"/>

<i>Linearity:</i>	<input type="text" value="◆ NonLinear"/>
<i>Nonlinear Geometric Effects:</i>	<input type="text" value="Large Displacement/
Large Strains"/>

<i>Arclength Method:</i>	<input type="text" value="Modified Riks/Ramm"/>
<i>Max # of Increments:</i>	<input type="text" value="100"/>

<i>Max # of Increments per Increment:</i>	<input type="text" value="75"/>
---	---------------------------------

Contact Table...

*Parameters Defining Contact
Between Bodies:*

Glue All**OK****OK****Apply****Cancel**

Note: The default load case contains all the loads required to run this step and is selected by default.

Load Step Selection...

Selected Job Steps:

pipe_crush**OK****Apply**

The non-linear analysis job **crush** will then be submitted for analysis to the workstation designated in the Submit Script (usually your local workstation).

The analysis job will take (on average) 1 to 2 minutes to run. When the job is done there will be a results file titled **crush.t16** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at **crush.log** and **crush.sts** and viewing the **crush.out** file, or using the **Monitor** application.

Action:

Monitor

Object:

Job**View Status File...**

A job finishing successfully will have end with: Job ends with exit number: 3004.

22. Read in the results.

■ Analysis

Action:

Read Results

Select Results File...

Selected Results File:

crush.t16

OK

Apply

23. Use **Results** to post process the results of the analysis.

■ Results

Change the Display Properties for results. Click on the Deform Attributes icon.



Scale Interpretation:

True Scale

Show Undeformed Entities

Choose the Select Results Icon to Choose results data to plot.



Action:

Create

Object:

Quick Plot

Select Results Cases:

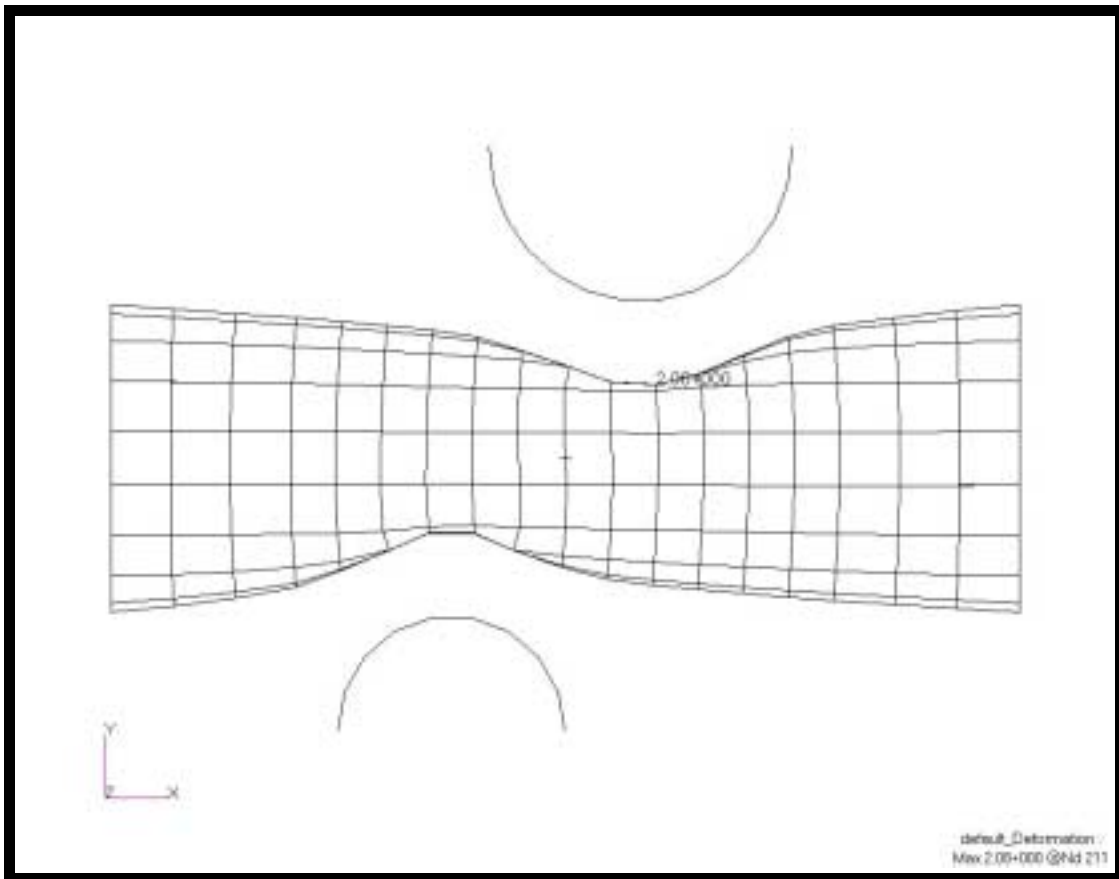
Select the last increment

Select Deformation Result:

Displacement, Translation

Apply

Your model should appear as shown in Figure 12.12:

Figure 12.12 - Deformation of pipe

24. Create an animation of the Deformation Von Mises Stresses.

First, you might wish to view the three-dimensionality of the model by selecting the following toolbat icon:



Iso 1 View

Action:

Create

Object:

Quick Plot

Select Results Cases:

Select the last increment

Select Fringe Results:

Stress, Global System

Results Position:

Position...(At Layer 1)

Quantity:

von Mises

Select Deformation Result:

Displacement, Translation

■ **Animate**

Select the **Animation Options** icon



■ **Animate Fringe**

■ **Animate Deformation**

Animation Method:

● **Ramped**

Animation Graphics:

● **2D**

Number of Frames:

8

Apply

When done viewing the animation, close the database and quit PATRAN.

This concludes the exercise.