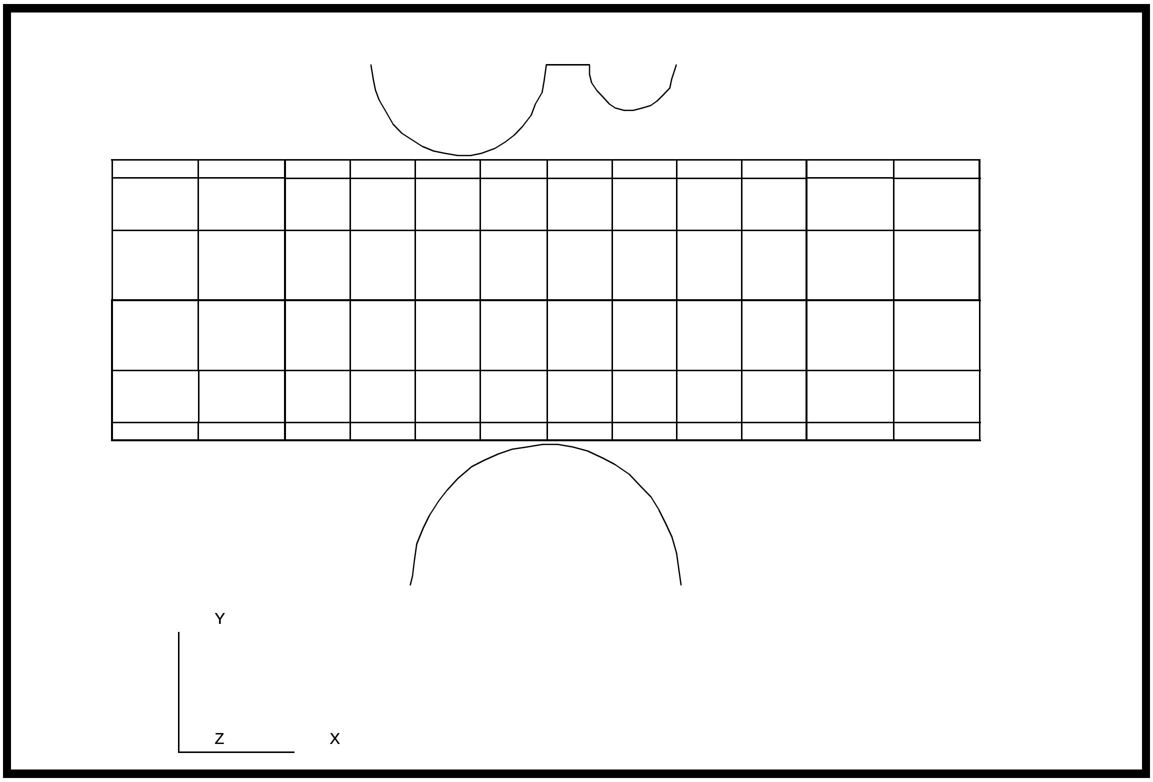
LESSON 10

Crushed Pipe



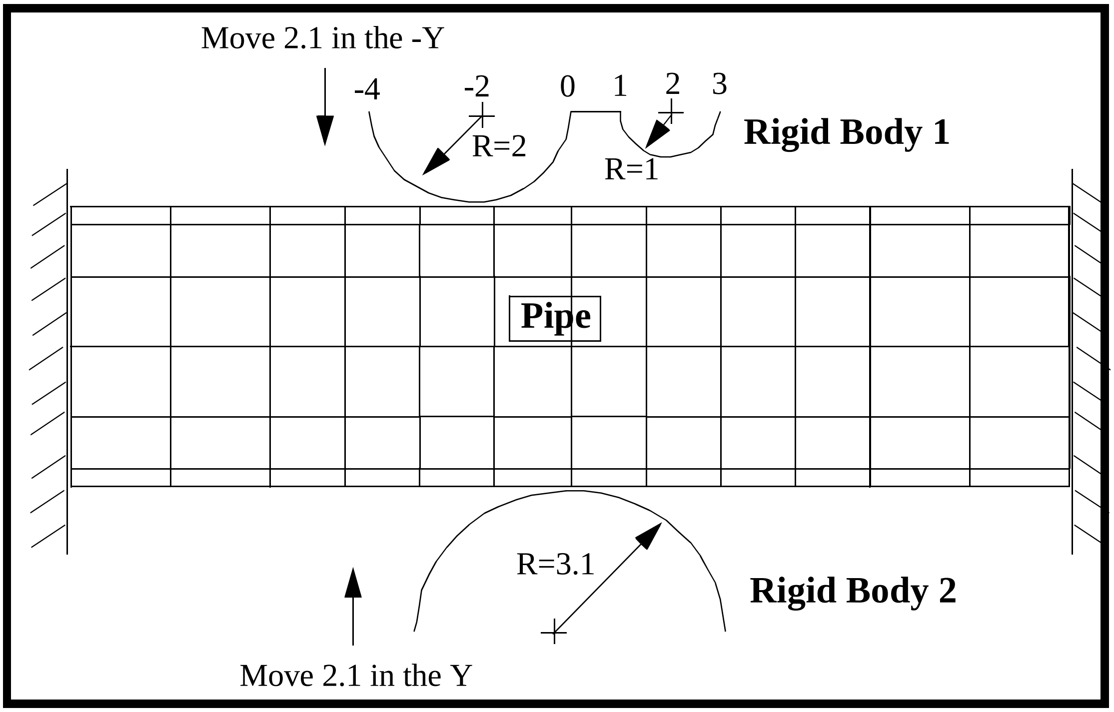
Objectives:

- Large displacement analysis.
- Rigid contact analysis using analytic 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 two rigid bodies.

The model is created using 2D thin shell elements to model both the pipe and the crushing rigid bodies. 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 = 6.2** **Length = 20** **Thickness = 0.4**
Pipe Material:	**Steel, E = 30E6** **nu = 0.3** **Yield Stress = 45,000**
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⁄ADVANCEDFEA**.

Analysis Code: **MSC/ADVANCED_FEA**

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: **[0, -6.3, 4]**

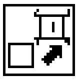
Apply

Create a second point by changing the *Point Coordinates List* to **[3.1, -6.3, 4]**.

Apply

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

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



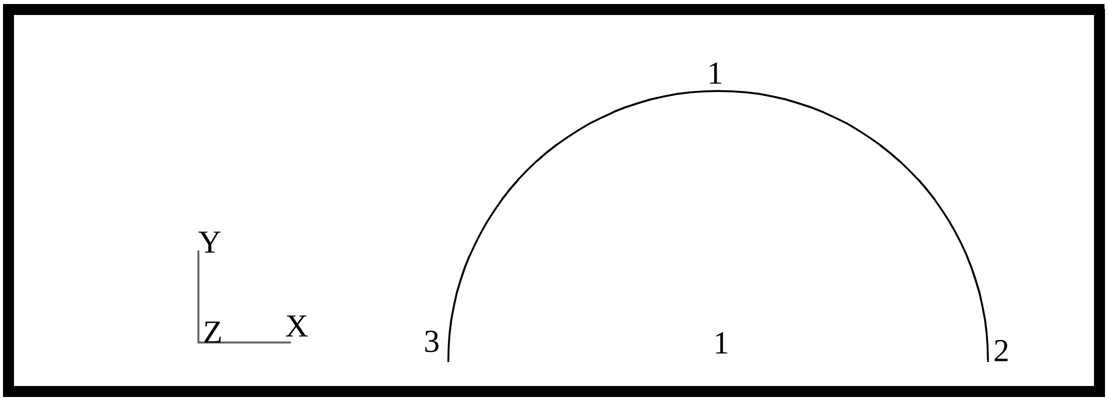
Show Labels

Action:	**Create**
Object:	**Curve**
Method:	**Revolve**
Axis:	**{Point 1[X1 Y1 5.0]}**
Total Angle:	**180**
Point List:	**Point 2**

Apply

The screen should now display the points and curves as shown in Figure 8.1:

Figure 8.1 - Curve for lower rigid surface



5. Create the geometric points for the upper rigid surface

Action:	**Create**
Object:	**Point**
Method:	**XYZ**
Points Coordinates List:	**[0, 5.2, 4]**

Apply

This will create **Point 4**

In a similar manner, create points 5, 6 and 7 using the **Create, Point, XYZ** option. The data for the transformations is given below.

Table 1: List of Points to Create

New Point	Point Coordinate List
5	**[-2.0, 5.2, 4]**
6	**[1.0, 5.2, 4]**
7	**[2.0, 5.2, 4]**

6. Create the curves for the upper rigid surface

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

Action:	**Create**
Object:	**Curve**
Method:	**Revolve**
Axis:	**{Point 5[X5 Y5 5]}**
Total Angle:	**-180**
Point List:	**Point 4**

Apply

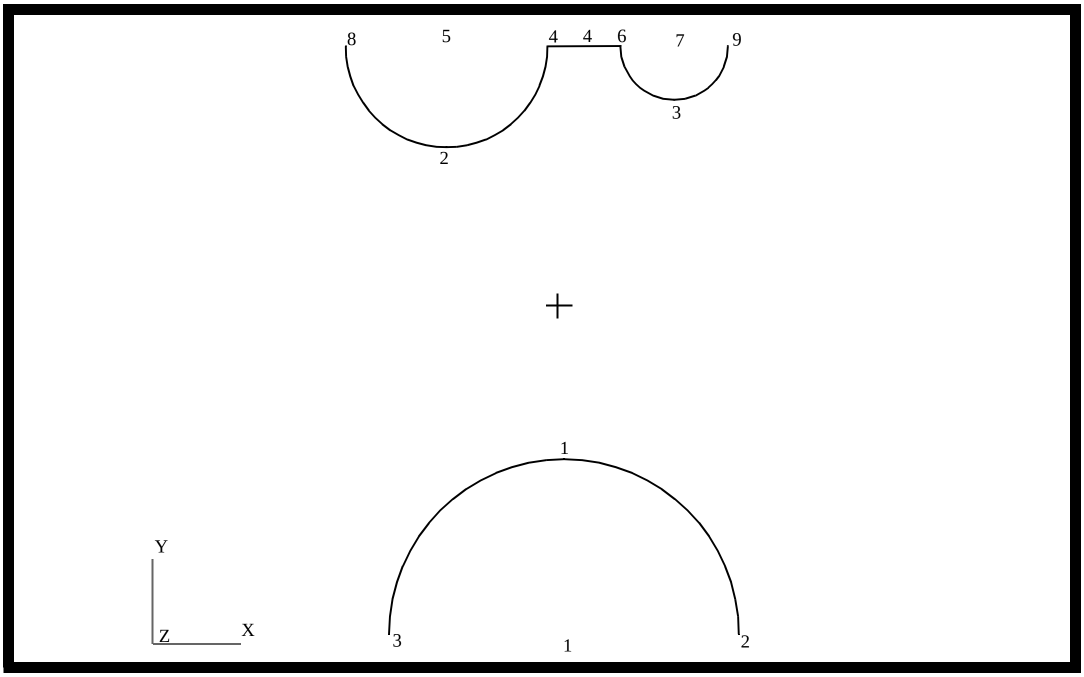
In a similar manner create a curve from **Point 6** using an *Axis* definition of **{point 7[X7 Y7 5]}**, a *Total Angle* of **180** and the *Point List* set to **Point 6**.

Connect the two half circles with a straight line.

Action:	**Create**
Object:	**Curve**
Method:	**Point**
Options:	**2 Point**
Starting Point List:	**Point 4**
Ending Point List:	**Point 6**

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 8.2:

Figure 8.2 - Curves to form rigid surfaces

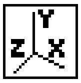


7. Create a new group **pipe** and the geometry for the pipe.

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

Action:	**Create**

Object:	**Point**
Method:	**XYZ**
Points Coordinates List:	**[-10 0 0]**

Apply

In a similar manner, using the **Create, Point, XYZ** method to create **Point 11** by changing the *Point Coordinates List* to **[-10 3.1 0]**.

Sweep **Point 11** into a circle

Action:	**Create**
Object:	**Curve**
Method:	**Revolve**
Axis:	**{Point 10[1 Y10 Z10]}**
Total Angle:	**360**
Point List:	see Figure 8.3

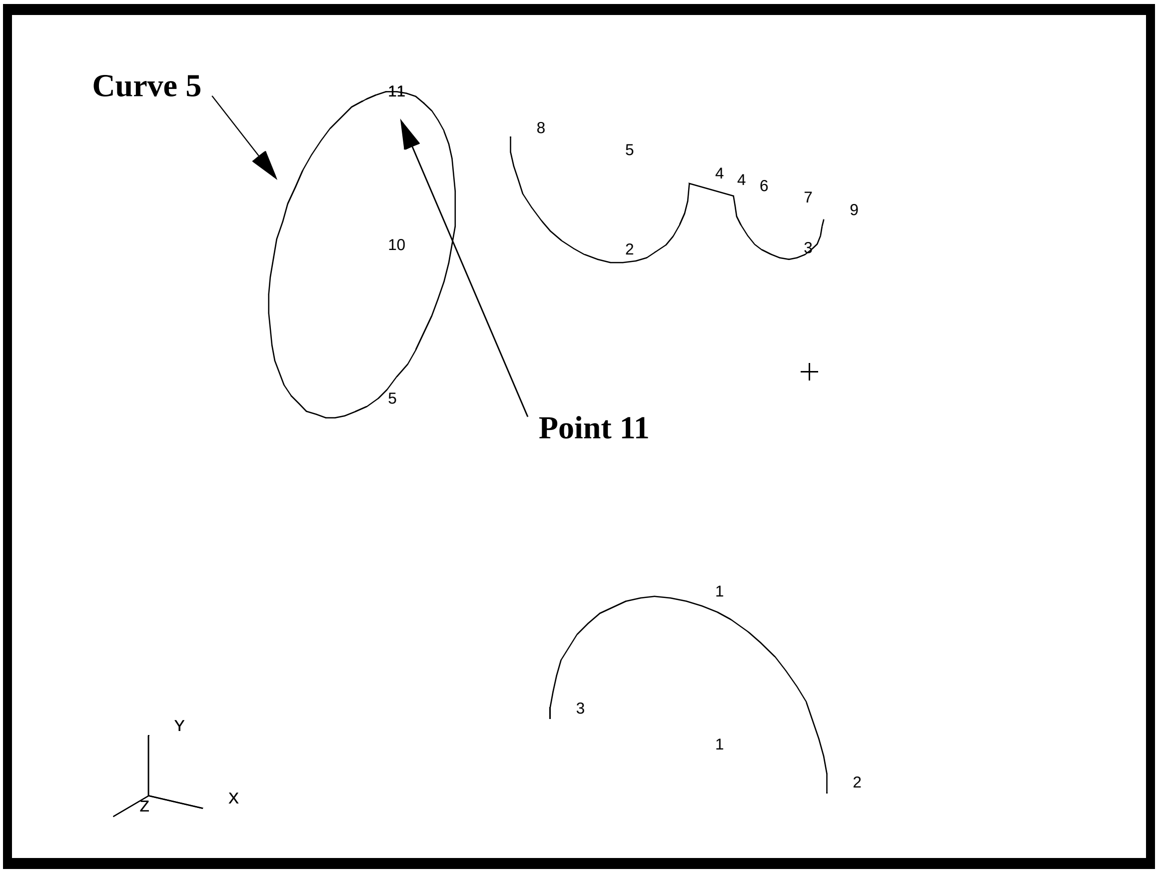
Apply

8. Create the mesh seed for the circle.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**
Element Edge Length Data:	◆ **Number of Elements**
Number:	**12**
Curve List:	See Figure 8.3

Figure 8.3 - Point to form curve and curve to place mesh seeds on



9. 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

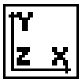
Mesh the curve

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**0.1**
Element Topology:	**Bar 2**
Curve List:	see Figure 8.3

Apply

Now you will extrude the elements

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



Front View

Action:	**Sweep**
Object:	**Element**
Type:	**Extrude**

Mesh Control...

Mesh Control Data:	◆ **Number of Elements**
Number:	**2**

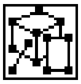
OK

Direction Vector:	**<4, 0, 0>**

■ **Delete Original Elements**

Base Entity List:	Select the elements you created on the circle (see Figure 8.4)

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.

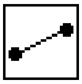
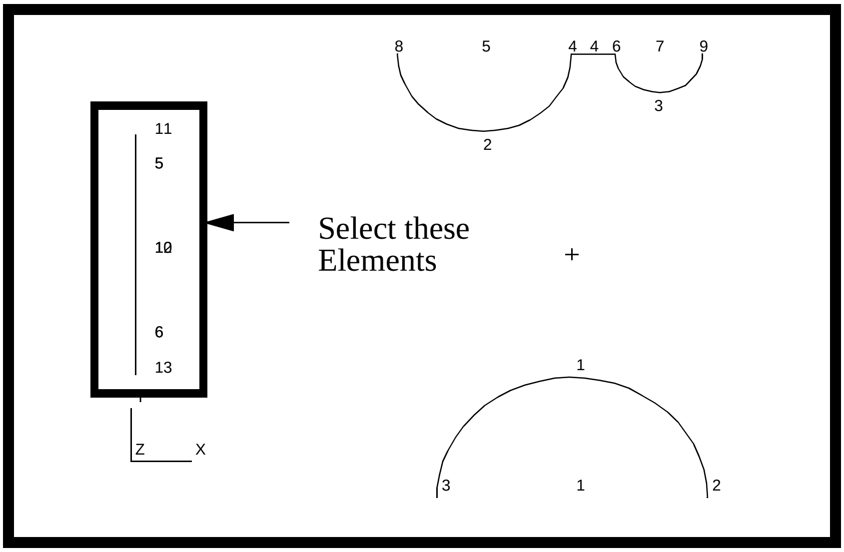

Beam Element

Figure 8.4 - 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**

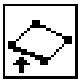
Number: **8**

OK

Direction Vector: **<12, 0, 0>**

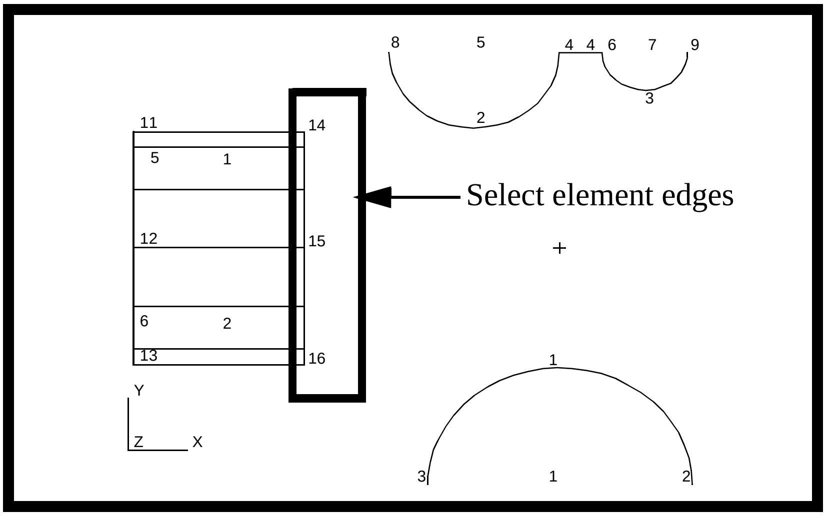
Base Entity List: see Figure 8.5

First, click on the **Edge of Element** icon


Edge of Element

Select the element’s edges as shown in Figure 8.5:

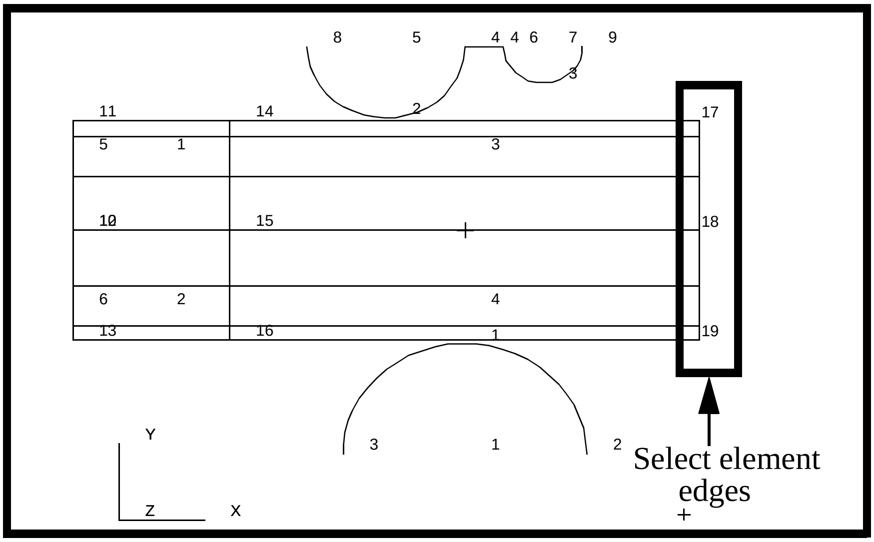
Figure 8.5 - Element edges to sweep into surface elements



Repeat the process once again with the original settings

Action:	**Sweep**
Object:	**Element**
Type:	**Extrude**
Mesh Control...	
Mesh Control Data:	◆ **Number of Elements**
Number:	**2**
OK	
Direction Vector:	**<4, 0, 0>**
Base Entity List:	see Figure 8.6

Figure 8.6 - Element edges to sweep into surface elements



10. Define the reference nodes for the upper rigid surface.

Add a “reference” node to the upper and lower rigid bodies. This node will be used to control the motion of the rigid bodies.

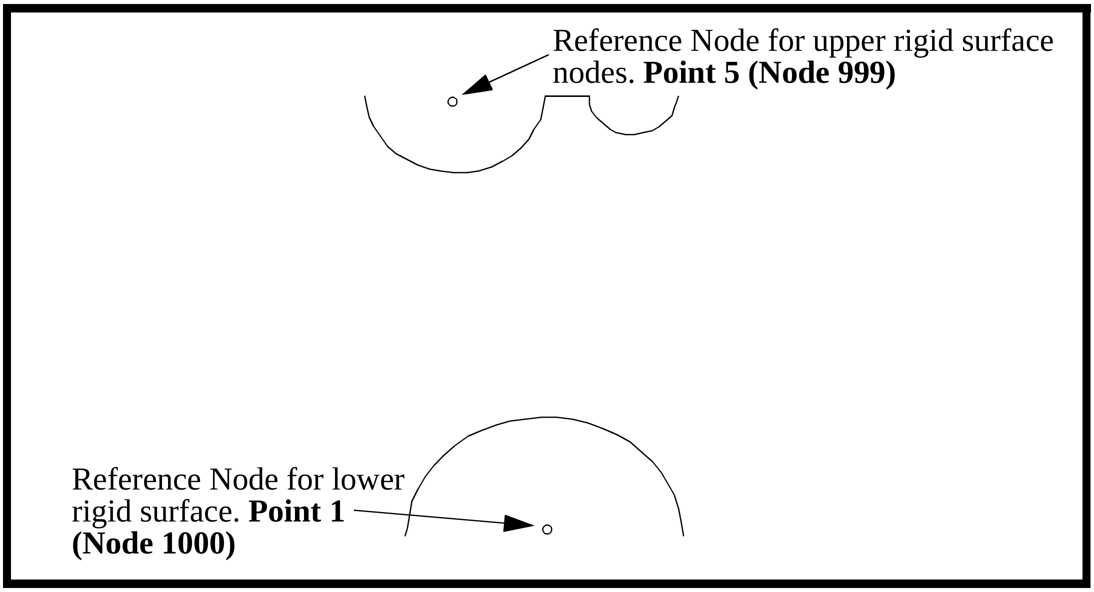
First, post only the group **rigid**

Group/Post...

Select Groups to Post:	**rigid**
Apply	
Cancel	

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node ID List:	**999**
Node Location List:	see Figure 8.7
Apply	

Figure 8.7 - Locations of reference nodes



11. Define the reference nodes for the lower rigid surface.

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node ID List:	**1000**
Node Location List:	see Figure 8.7

Apply

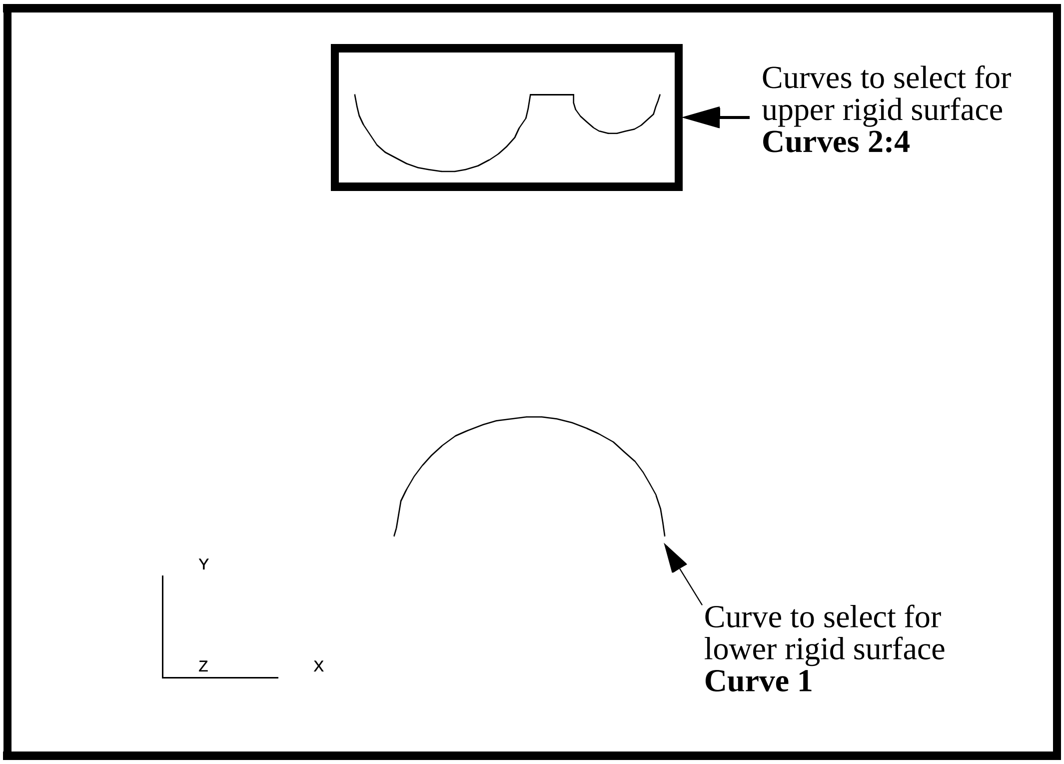
12. Create the upper rigid surface elements.

Create a beam mesh on the upper rigid bodies.

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**1**
Element Topology:	**Bar2**
Curve List:	see Figure 8.8

Apply

Figure 8.8 - Curves to sweep into rigid surface elements



13. 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 8.8. The *Curve List* should be **Curve 1**. See Figure 8.8.

14. Give the rigid surfaces depth by using

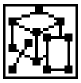
Action:	**Sweep**
Object:	**Element**
Type:	**Extrude**
Mesh Control...	
Mesh Control Data:	◆ **Number of Elements**
Number:	**1**
OK	
Direction Vector:	**<0, 0, -8>**

■ **Delete Original Elements**

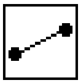
Base Entity List: Select all posted bars

Apply

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



Elements



Beam lement

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

Action: **Equivalence**

Object: **All**

Type: **Tolerance Cube**

Apply

16. 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: **Verify**

Object: **Element**

Test: **Normals**

◆ **Draw Normal Vectors**

Apply

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:

Action: **Modify**

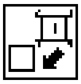
Object: **Element**

Method: **Reverse**

Element List: select the elements which had normals facing the wrong direction

Apply

Remove the entity labels using the following toolbar icon:

 **Hide Labels**

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

Group/Create...

New Group Name: **fem_all**

■ **Make Current**

Group Contents: **Add All FEM**

Apply

18. Create the material properties for the pipe.

◆ **Materials**

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name: **steel**

Input Properties...

Constitutive Model: **Elastic**

Elastic Modulus: **30E6**

Poisson's Ratio: **0.30**

Apply

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

Constitutive Model:	**Plastic**
Hardening Rule:	**Perfect Plasticity**
Yield Stress:	**45000**

Apply

Cancel

19. Post the group **fem_pipe** only.

Group/Post...

Select Groups to Post:	**fem_pipe**

Apply

Cancel

20. Now create the Element Properties for your model.

◆ **Properties**

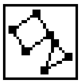
Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name:	**pipe**
Options:	**Thin**
	Homogeneous

Input Properties...

Material Name:	**steel**
Shell Thickness:	**0.4**

OK

Click in the *Select Members* databox. Select the 2D Element icon in the Select menu and then select ***all*** elements displayed.



2D Element

Add

Apply

21. 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 >**

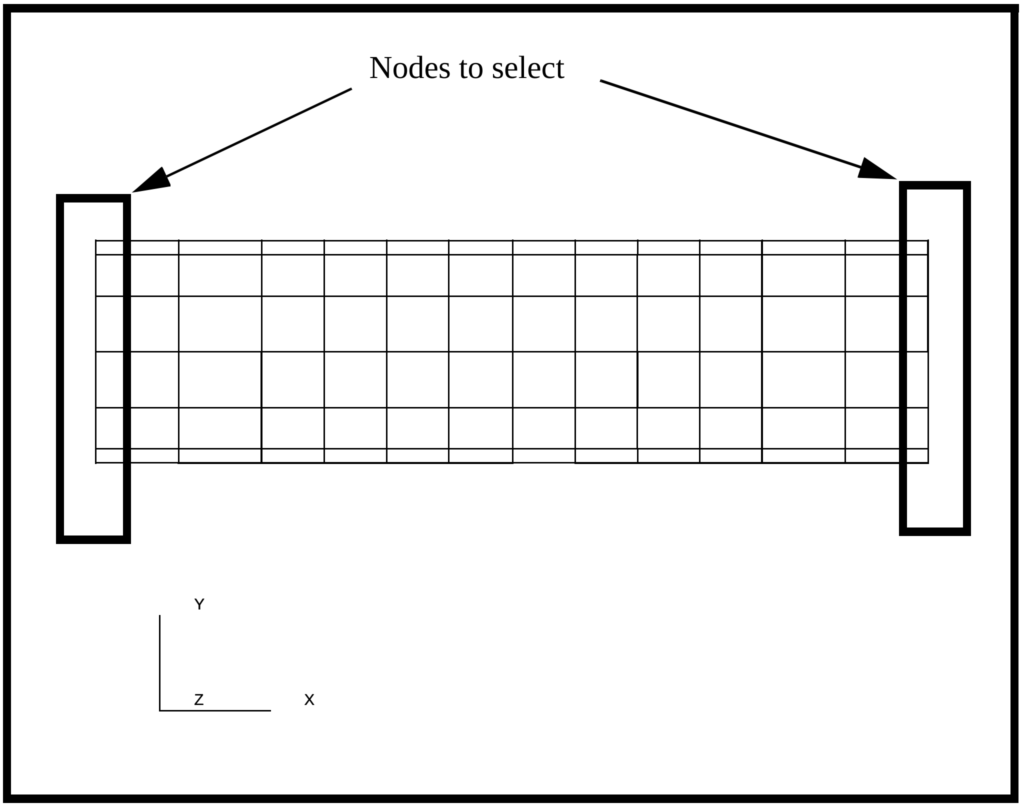
OK

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 8.9:

Figure 8.9 - Nodes at fixed ends of the pipe



22. Next, you will create the enforced displacement on the upper rigid body.

Post the group **fem_all**.

Group/Post...

Select Groups to Post:	**fem_all**
Apply	
Cancel	
New Set Name:	**top_rigid_down**
Input Data...	
Translations <T1,T2,T3>:	**< 0, -2.2, 0>**
Rotations <R1,R2,R3>:	**< 0, 0, 0 >**
OK	
Select Application Region...	

Geometry Filter: ◆ **FEM**

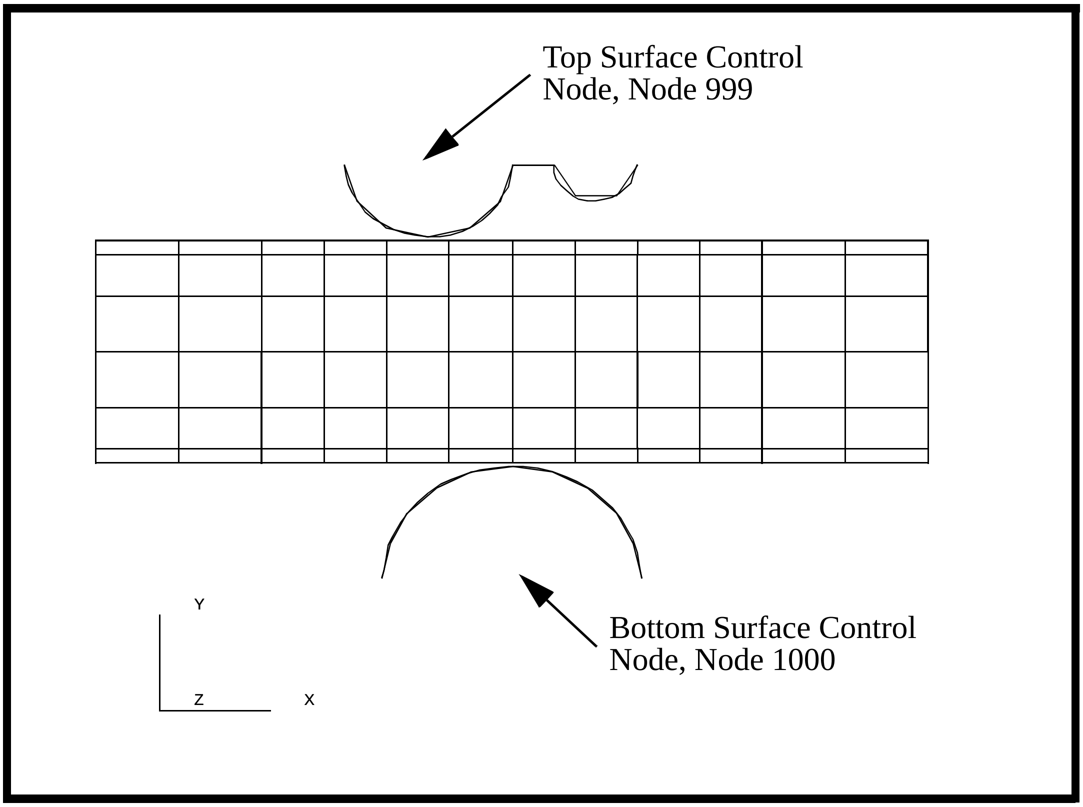
Select Nodes: see Figure 8.10

Add

OK

Apply

Figure 8.10 - Reference nodes for rigid surface displacements



23. The last boundary condition is the enforced displacement on the lower rigid body.

New Set Name: **bottom_rigid_up**

Input Data...

Translations <T1,T2,T3>: **< 0, 2.2, 0>**

Rotations <R1,R2,R3>: **< 0, 0, 0 >**

OK

Select Application Region...

Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 1000**

Add

OK

Apply

24. Now create the **Load and Boundary Conditions** for the contact.

Action:	**Create**
Object:	**Contact**
Method:	**Element Uniform**
Option:	**Rigid-Deform**
New Set Name:	**contact_top**

Input Data...

Reference Node:	Select the top reference node. See Figure 8.10

OK

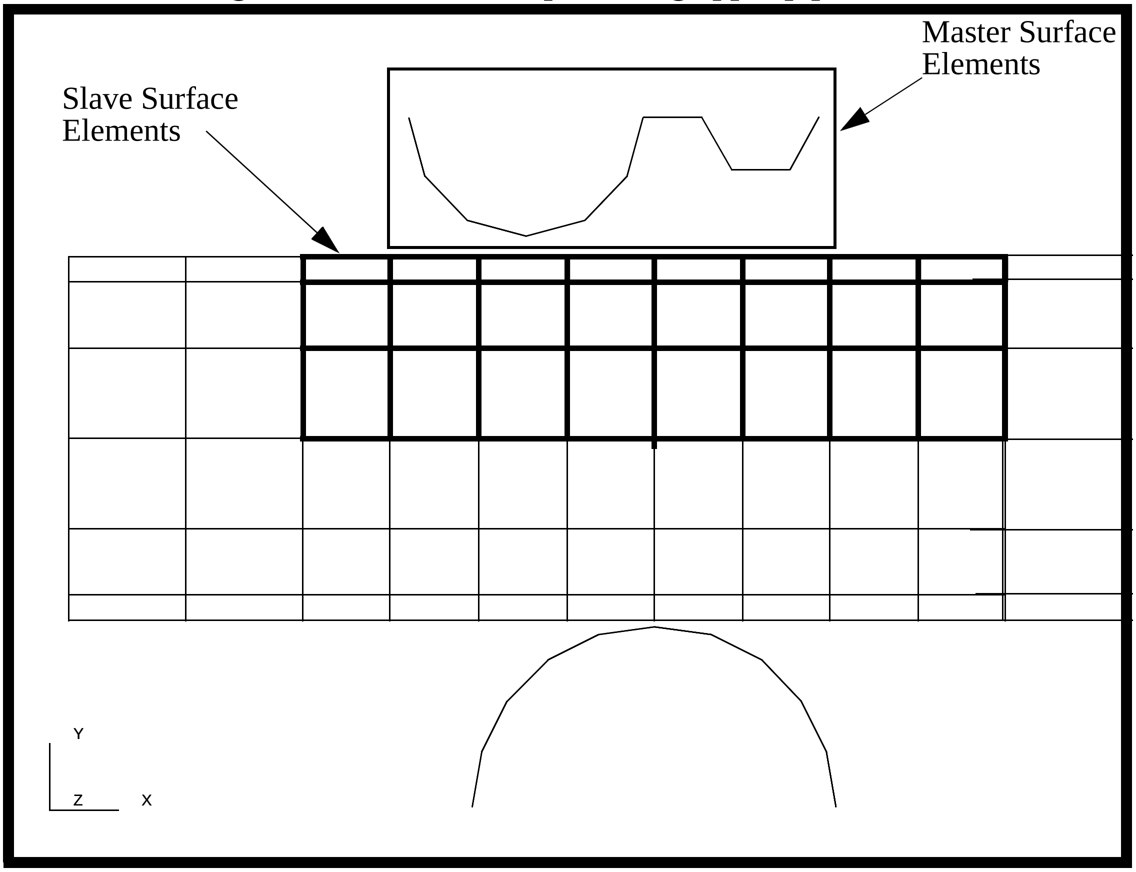
Select Application Region...

Geometry Filter:	◆ **FEM**
Master Surface:	**Rigid Surface**
Slave Surface:	**Shell Surface**
Active Region:	**Master**
Select Shell Elements:	see Figure 8.11

Add

Active Region:	**Slave**
Select Shell Elements:	see Figure 8.11

Figure 8.11 - Elements representing upper pipe contact



Add

OK

Apply

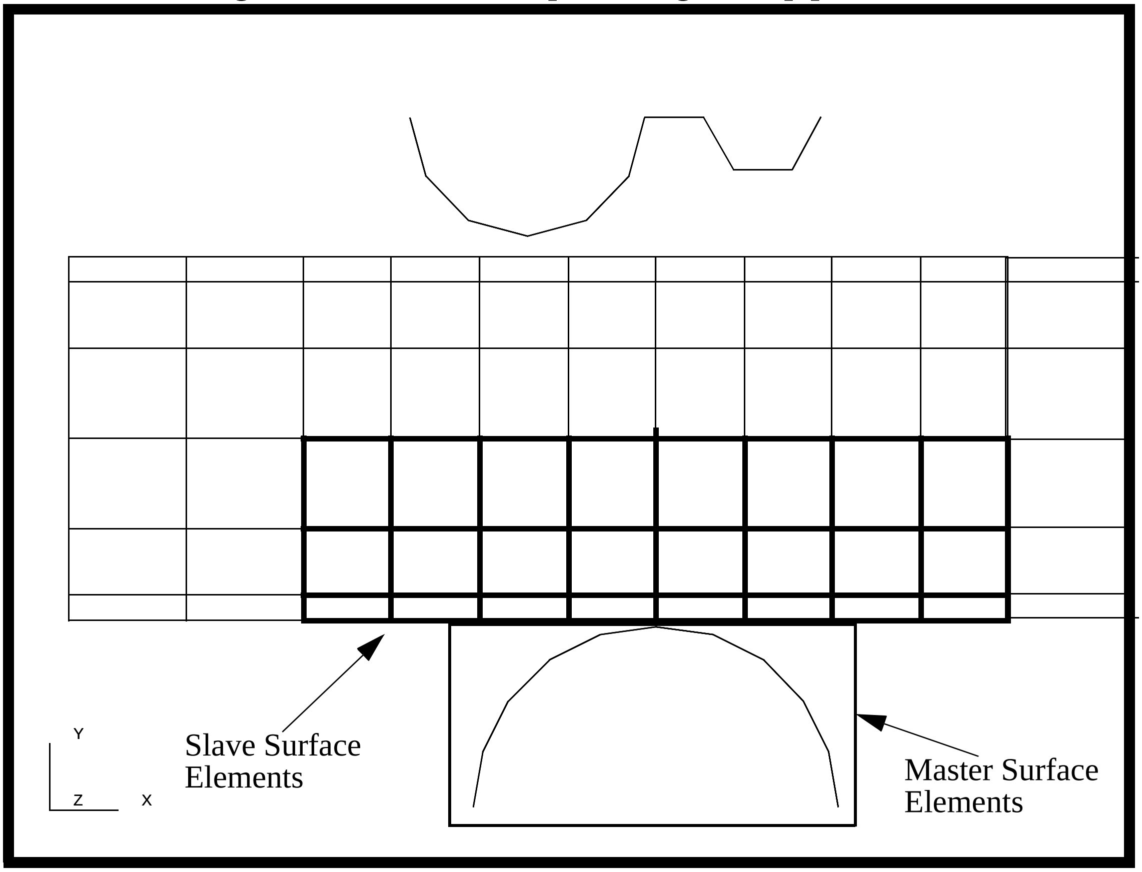
Repeat the same procedure for the bottom

Action:	**Create**
Object:	**Contact**
Method:	**Element Uniform**
Option:	**Rigid-Deform**
New Set Name:	**contact_bottom**

Input Data...

Reference Node:	Select the bottom reference node. See Figure 8.10
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**
Master Surface:	**Rigid Surface**
Slave Surface:	**Shell Surface**
Active Region:	**Master**
Select Shell Elements:	see Figure 8.12
Add	
Active Region:	**Slave**
Select Shell Elements:	see Figure 8.12

Figure 8.12 - Elements representing lower pipe contact



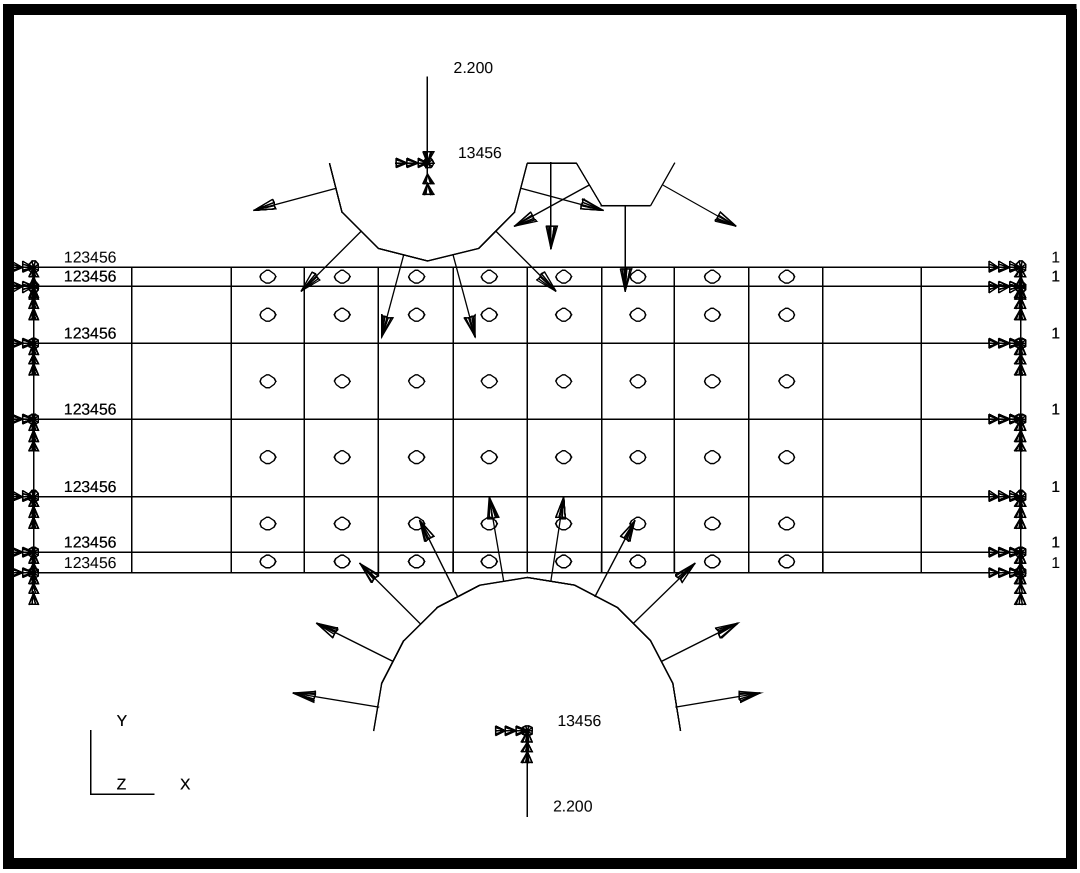
Add

OK

Apply

Your model should now look like the picture shown in Figure 8.13:

Figure 8.13 - Correct contact normal definitions



Note: If the arrows on your model are pointing the opposite direction then you must perform this step. If not skip to the next step.

◆ **Load/BCs**

Action: **Modify**

Object: **Contact**

Method: **Element Uniform**

Option: **Rigid-Deform**

Select Set to Modify: **contact_bottom or top**

Select Application Region...

■ **Reverse Contact Direction**

OK

Apply

25. Your model is now ready for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**crush**

Step Creation...

Job Step Name:	**pipe_crush**
Solution Type:	**Nonlinear Static**

Solution Parameters...

Large Deflections/Strains:	**ON**
Max No. of Increments Allowed:	**100**
Riks Method:	**OFF**
Automatic Load Increments:	**ON**

OK

Apply

Cancel

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

Step Selection...

Selected Job Steps:	**pipe_crush**

Apply

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) 5 to 10 minutes to run. When the job is done there will be a results file titled **crush.fil** in the same directory you started MSC/PATRAN in. The **crush.023** file will disappear.

Again, you can monitor the progression of the job by looking at **crush.msg** and **crush.sta** as well as using the UNIX command ***ps -a*** and ***tail -lf crush.msg***.

26. Read in the results.

◆ **Analysis**

Action: **Read Results**

Select Results File...

Selected Results File: **crush.fil**

OK

Apply

27. Change the Display Properties for results.

Display/Results...

Scale Factor: **1.0**

◆ **Direct Multiplication**

❒ **Show Undeformed Entities**

Apply

Cancel

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

To display the results, click on the **Results** switch in the *Main Window.*

◆ **Results**

Action: **Create**

Object: **Quick Plot**

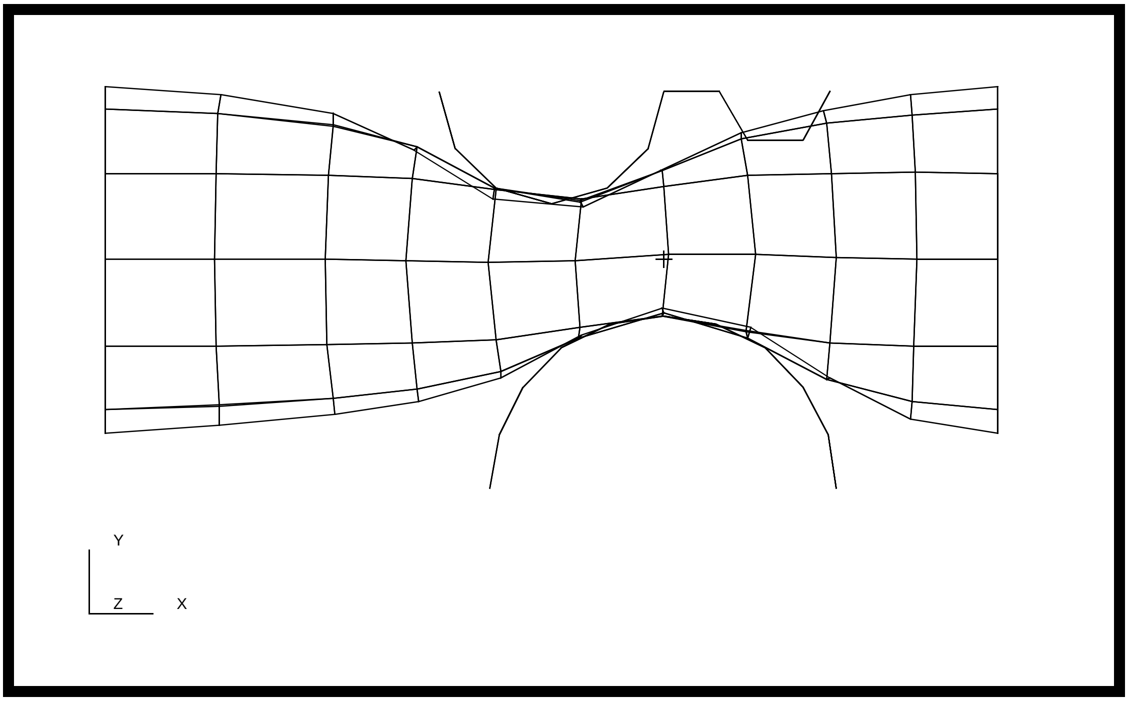
Select Results Cases: Select the last increment

Select Deformation Result: **Deformation, Displacement**

Apply

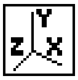
Your model should appear as shown in Figure 8.14:

Figure 8.14 - Deformation of pipe



29. 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:

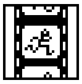


Iso 1 View

Action:	**Create**
Object:	**Quick Plot**
Select Results Cases:	Select the last increment
Select Fringe Results:	**Stress, Components**
Results Position:	**At SECTION_POINT_1**
Result Quantity:	**Von Mises**
Select Deformation Result:	**Deformation, Displacement**

■ **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 exericse.