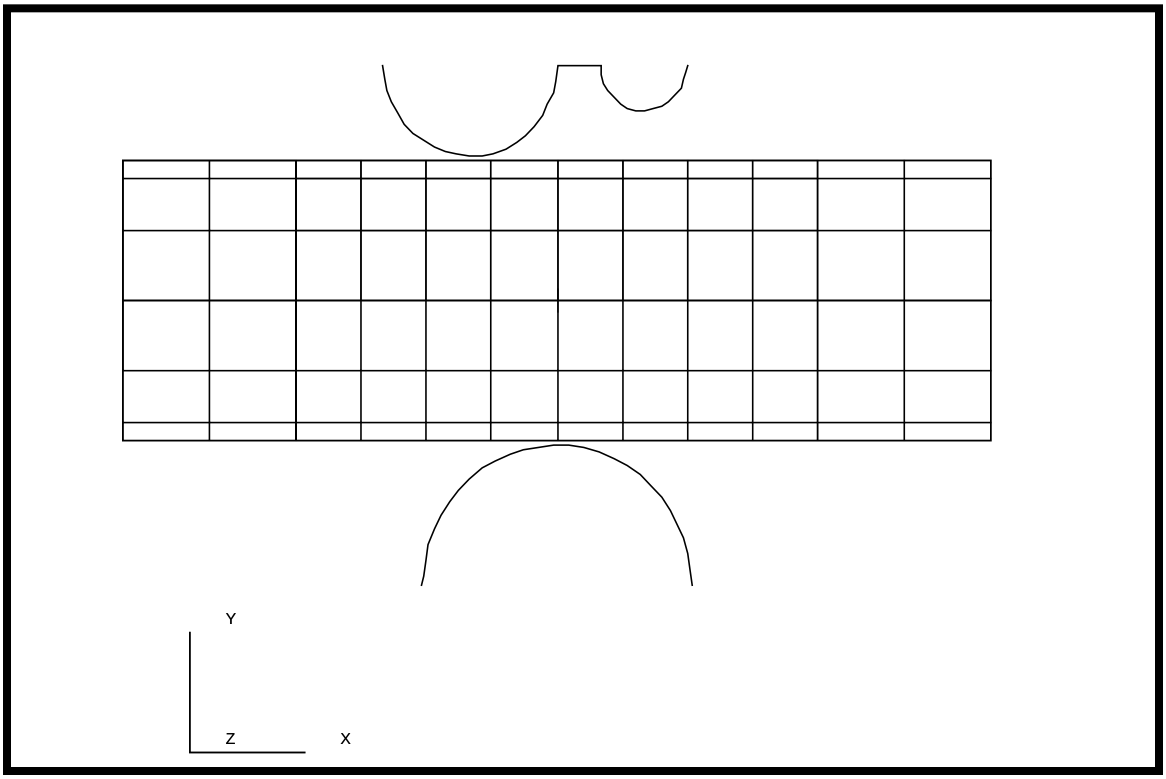
LESSON 7a

Crushed Pipe



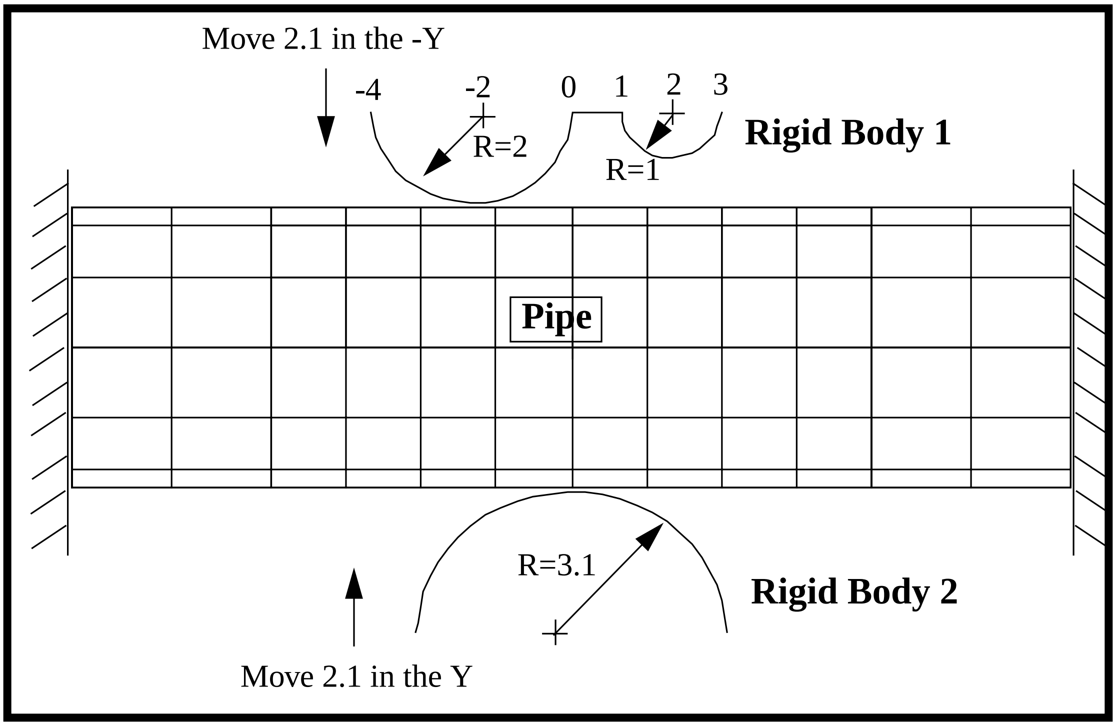
Objectives:

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

Model Description:

In this exercise, a symmetrical 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 the pipe and Rigid Surface (Cylinder) elements to model the crushing rigid bodies. The pipe model will be covered with 2D Interface 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 **pipe_crush**.

File/New ...

Database Name	**pipe_crush.db**

OK

Change the *Analysis Preference* to **MSC/ADVANCED_FEA**.

Analysis Code:	**MSC/ADVANCED_FEA**

OK

2. Create a new group **geo_rigid**.

Group/Create ...

New Group Name	**geo_rigid**

■ **Make Current**

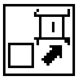
Group Contents:	**Add Entity Selection**

Apply

Cancel

3. Create the model geometry.

To make picking a little easier, use the following toolbar icon:



Show Labels

◆ **Geometry**

Action:	**Create**
Object:	**Point**
Method:	**XYZ**
Points Coordinates List	**[0, -6.3, 0]**

Apply

Create a second point by changing the *Point Coordinates List* to **[3.1, -6.3, 0]**.

Apply

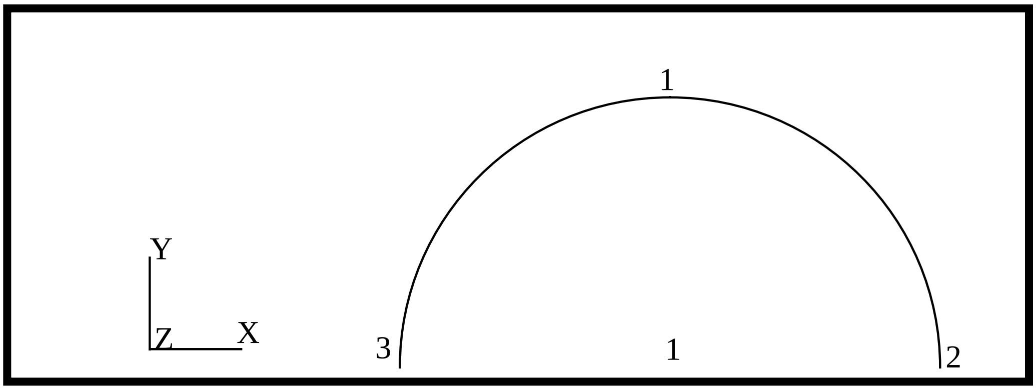
4. Sweep point to make a curve for the lower rigid surface.

Action:	**Create**
Object:	**Curve**
Method:	**Revolve**
Axis	**{Point 1[X1, Y1, 1.0]}**
Total Angle	**180**
Point List	**Point 2**

Apply

The screen should now display the points and curves as shown in Figure 7a.1:

Figure 7a.1 - Lower rigid surface



5. Create the geometric points for the upper rigid surface

◆ **Geometry**

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

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, 0]**
6	**[1.0, 5.2, 0]**
7	**[2.0, 5.2, 0]**

6. Create the curves for the upper rigid surface

Sweep Point 5 into an arc using

Action:	**Create**
Object:	**Curve**
Method:	**Revolve**
Axis	**{Point 5[X5, Y5, 1]}**
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, 1]}**, a *Total Angle* of **180** and the *Point List* set to **Point 6**.

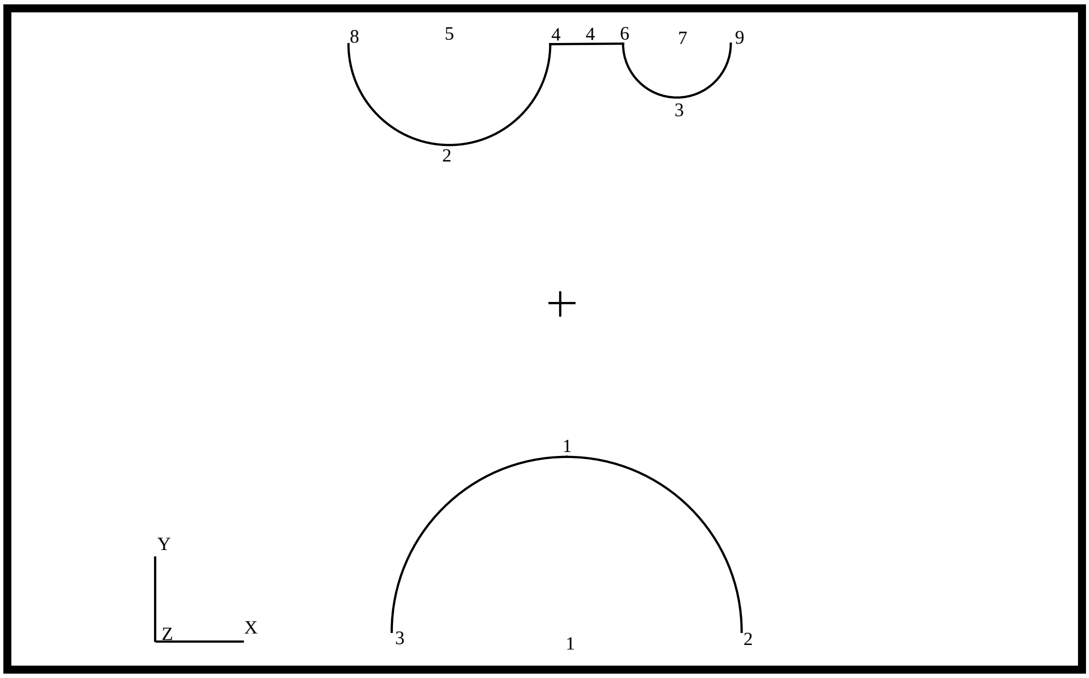
Connect the two half circles with a straight line by using

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

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 7a.2:

Figure 7a.2 - Upper and lower rigid bodies



7. Create a new group **geo_pipe**.

Group/Create ...

New Group Name **geo_pipe**

■ **Make Current**

Group Contents: **Add Entity Selection**

Apply

Cancel

8. Create the geometric points for the Pipe model.

◆ **Geometry**

Action: **Create**

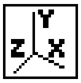
Object:	**Point**
Method:	**XYZ**
Points Coordinates List	**[-10, 0, 0]**

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]**.

9. Create the surfaces for the Pipe model

First, change to the isometric view by clicking on the **Iso 1 View** icon in the toolbar.

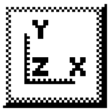


Sweep **Point 11** into two arcs

Action:	**Create**
Object:	**Curve**
Method:	**Revolve**
■ **Patran 2 Convention**	
Axis	**{Point 10[1, Y10, Z10]}**
Total Angle	**180**
Curves per Point	**2**
Point List	**Point 11**

Apply

Change the viewport to **Front View**, using the following toolbar icon:



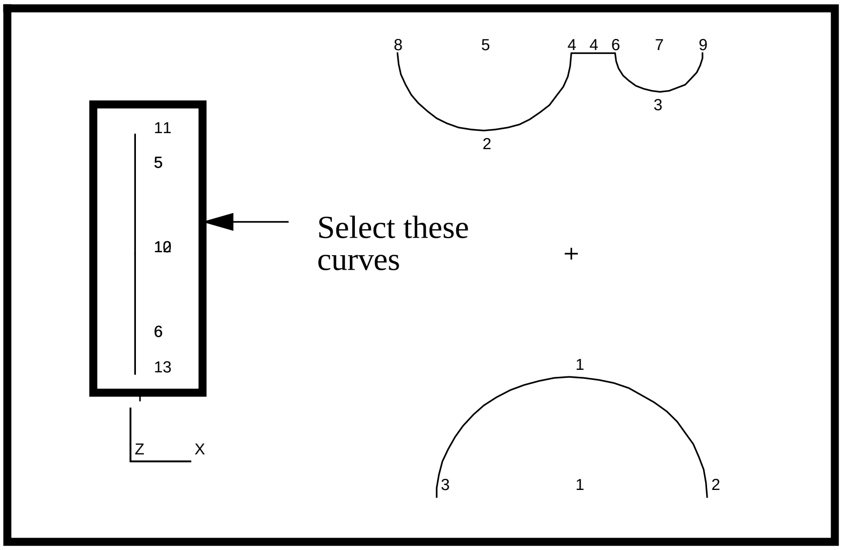
Sweep **Curves 5** and **6** into two surfaces

Action:	**Create**
Object:	**Surface**
Method:	**Extrude**

Translation Vector	**<4, 0, 0>**
Curve List	see Figure7a.3

Apply

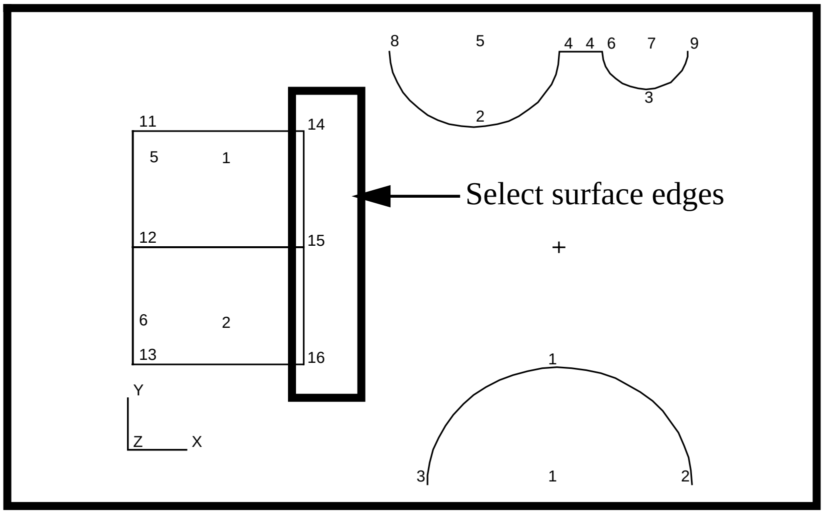
Figure 7a.3 - Curves to sweep into pipe surface



In a similar manner, create two more surfaces from the right edges of Surfaces 1 and 2 using a *Translation Vector* of **<12, 0, 0>**.

Select the surface edges as shown in Figure 7a.4:

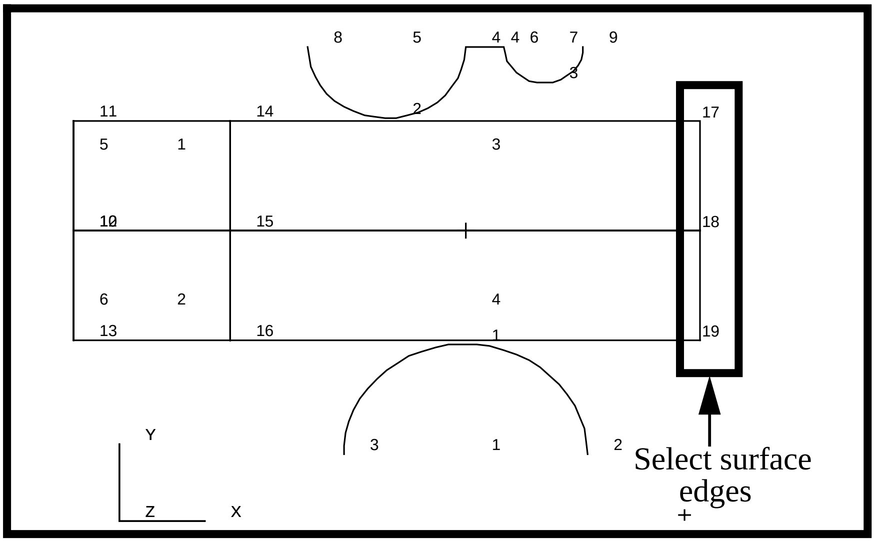
Figure 7a.4 - Surface edges to sweep into pipe surface



The *Curve List* should be **Surface 1.2 2.2**

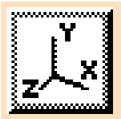
Finally create the last two surfaces using a *Translation Vector* of **<4, 0, 0>**.

Figure 7a.5 - Surface edges to sweep into pipe surface



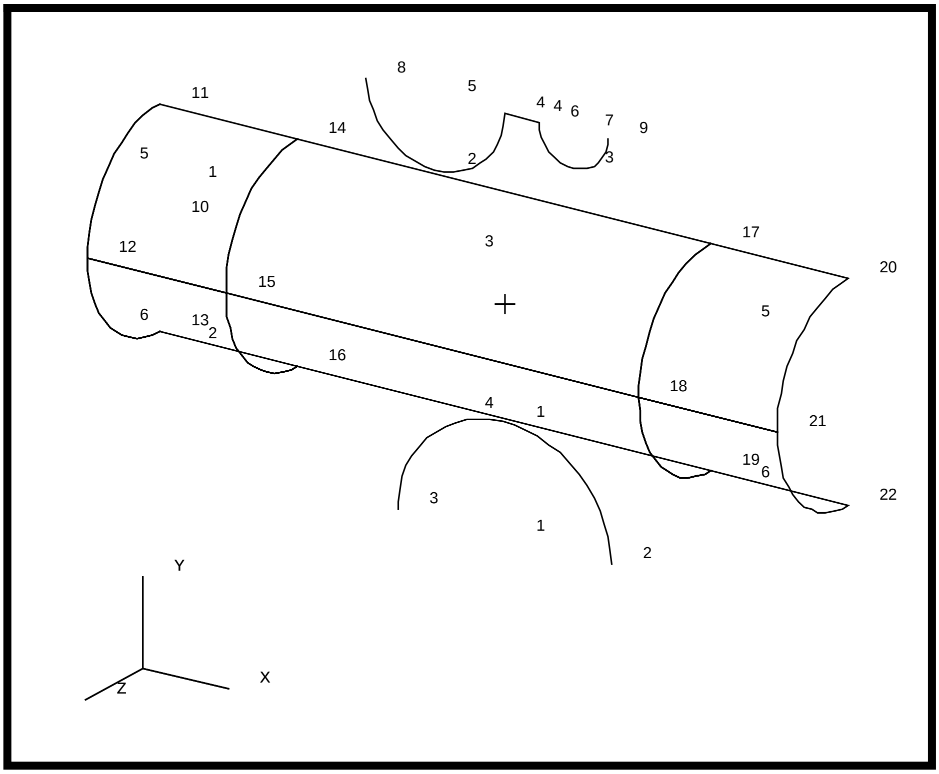
The *Curve List* should be **Surface 3.2 4.2**.

Change the display of the model.Click on the following icon to change to isometric view:



Your model should look like the one shown in Figure 7a.6:

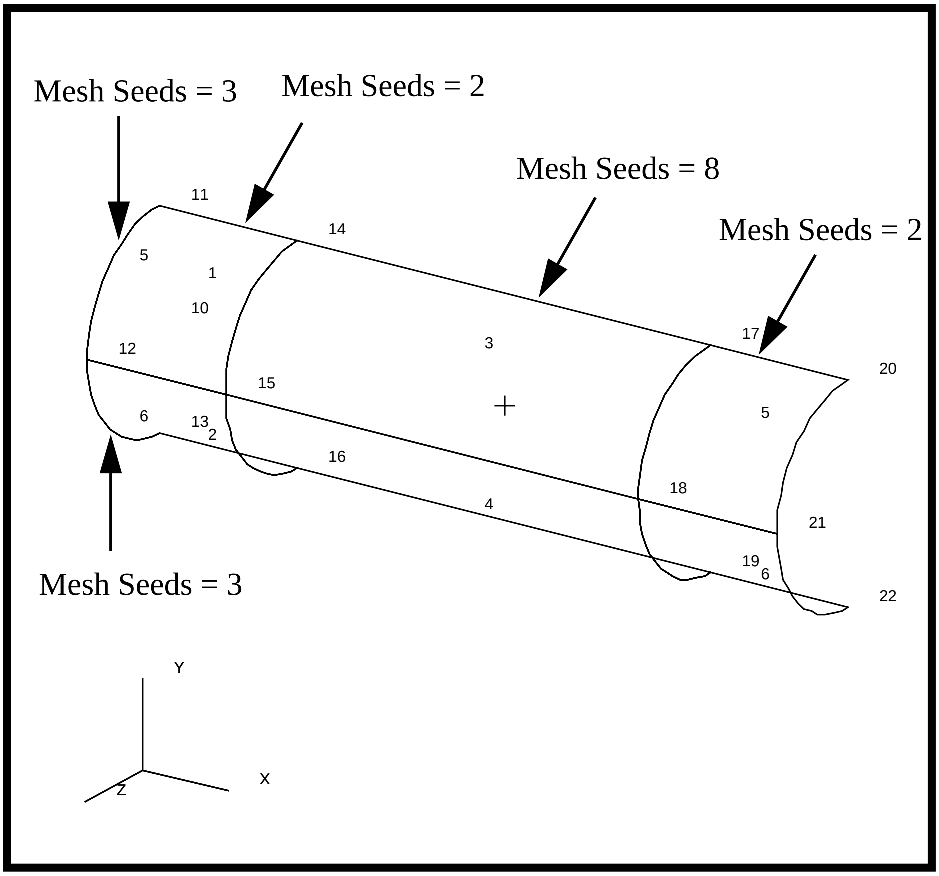
Figure 7a.6 - Isometric view of pipe surface and rigid bodies



10. Create the mesh seeds for the finite element mesh.

First, we will define the mesh seed for the body of the pipe. You will define the mesh seeds as shown in Figure 7a.7:

Figure 7a.7 - Mesh seed locations



◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Method:	**Uniform**

◆ **Number of Elements**

Number	**2**
Curve List	**Surface 1.1 5.1**

NOTE: To select multiple entries, click in the *Curve List* databox and screen select the top edge of Surface 1 (**Surface 1.1**) and the top edge (while holding down the shift key) of Surface 5 (**Surface 5.1**).

Apply

In a similar manner set *Number* = **3** and a *Curve List* of **Curve 5 6** (the left edge of Surface 1 and Surface 2) and click on **Apply**.

Again, set *Number* = **8** and a *Curve List* of **Surface 3.1** (the top of Surface3) and click on **Apply**.

11. Create a new group **fem_pipe**.

Group/Create ...

New Group Name **fem_pipe**

■ **Make Current**

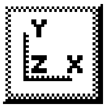
Group Contents: **Add Entity Selection**

Apply

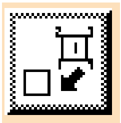
Cancel

12. Create the mesh for the pipe body.

Change the view to a front view by using the following toolbar icon:



Click the **Hide Labels** icon in the toolbar menu to turn off all the entity labels.



Action: **Create**

Object: **Mesh**

Type: **Surface**

Global Edge Length **1**

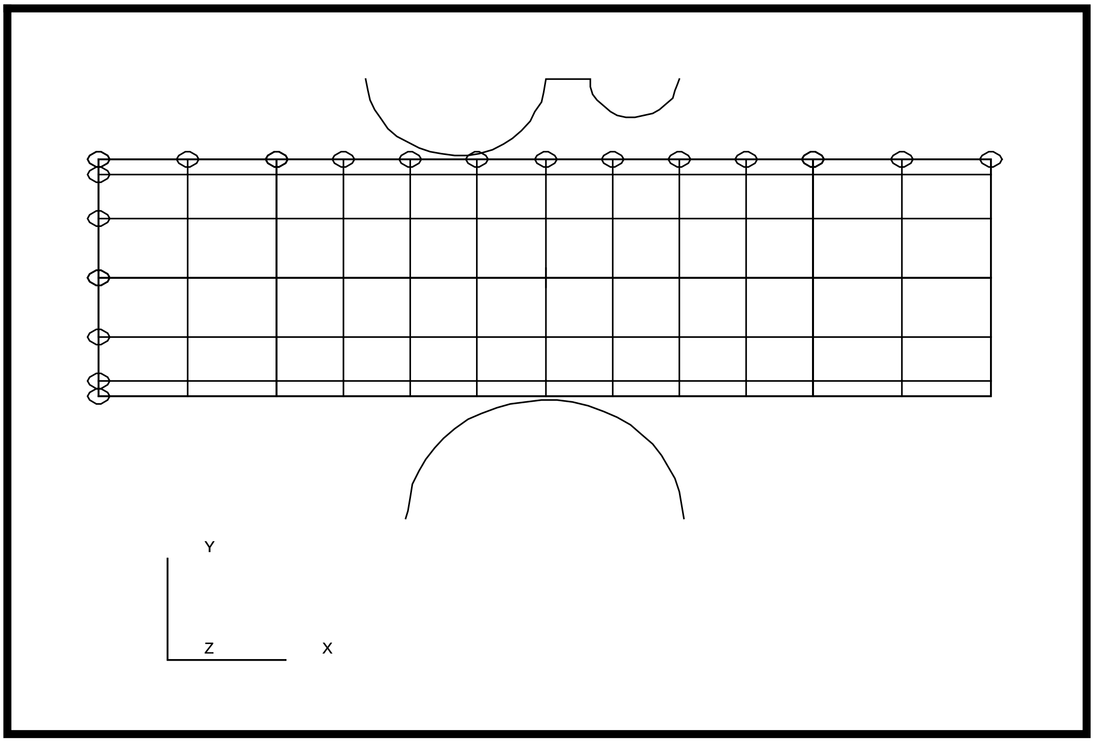
Element Topology **Quad4**

Surface List select all surfaces in model

Apply

Your model should look like the one shown in Figure 7a.8:

Figure 7a.8 - Meshed pipe surface



13. Create a new group **fem_irs_top**.

Group/Create ...

New Group Name **fem_irs_top**

■ **Make Current**

Group Contents: **Add Entity Selection**

Apply

Cancel

14. Create IRS elements for the top surface.

Next, we need to create a series of duplicate **Quad4** elements lying on top of the elements that make up the upper, central portion of the pipe model. These elements will be used to define our contact surfaces.

◆ **Finite Elements**

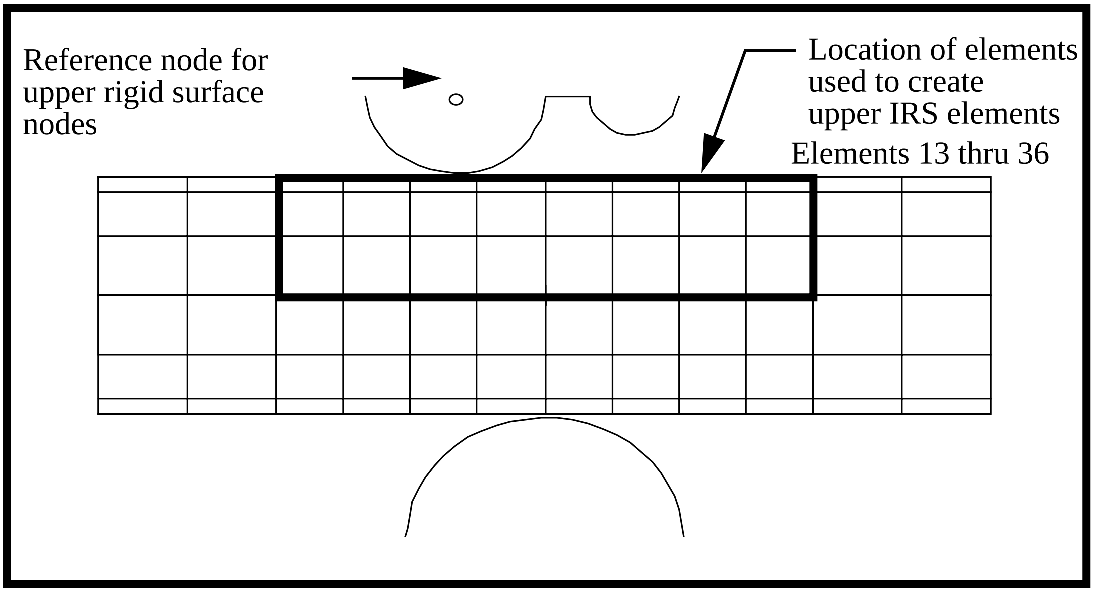
Action: **Transform**

Object: **Element**

Method:	**Translate**
Translation Vector	**<0, 0, 0>**
Element List	see Figure 7a.9

Apply

Figure 7a.9 - Elements used for upper contact



15. 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.

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node ID List	**999**
Node Location List	see Figure 7a.9

Apply

16. Create a new group **fem_irs_bottom**.

Group/Create...

New Group Name	**fem_irs_bottom**
■ **Make Current**	
Group Contents:	**Add Entity Selection**

Apply

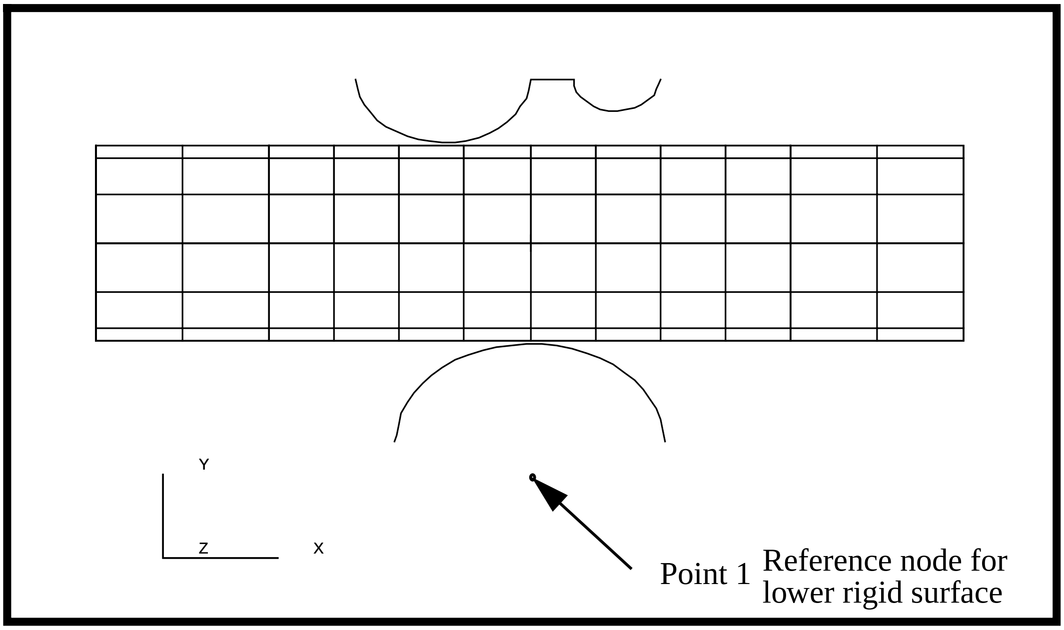
Cancel

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

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node ID List	**1000**
Node Location List	see Figure 7a.10

Apply

Figure 7a.10 - Location of lower rigid surface reference node



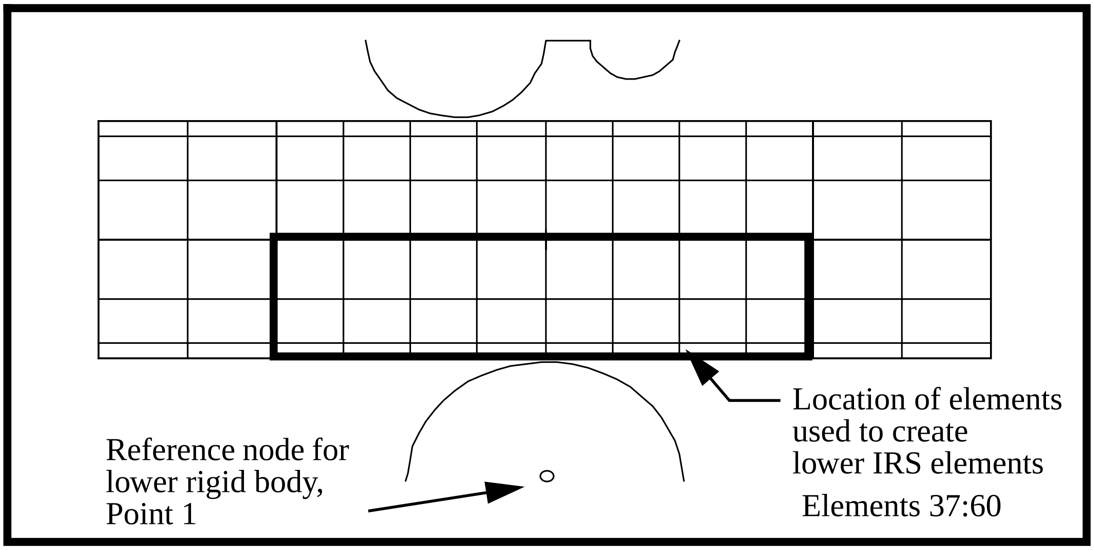
18. Create the lower irs elements.

In a similar manner, define a series of duplicate elements for the lower, central portion of the pipe model.

Action:	**Transform**
Object:	**Element**
Method:	**Translate**
Translation Vector	**<0, 0, 0>**
Element List	select the elements as shown in Figure 7a.11

Apply

Figure 7a.11 - Elements used for lower contact



19. Create a new group **fem_rigid_top**.

Group/Create

New Group Name	**fem_rigid_top**
■ **Make Current**	
Group Contents:	**Add Entity Selection**

Apply

Cancel

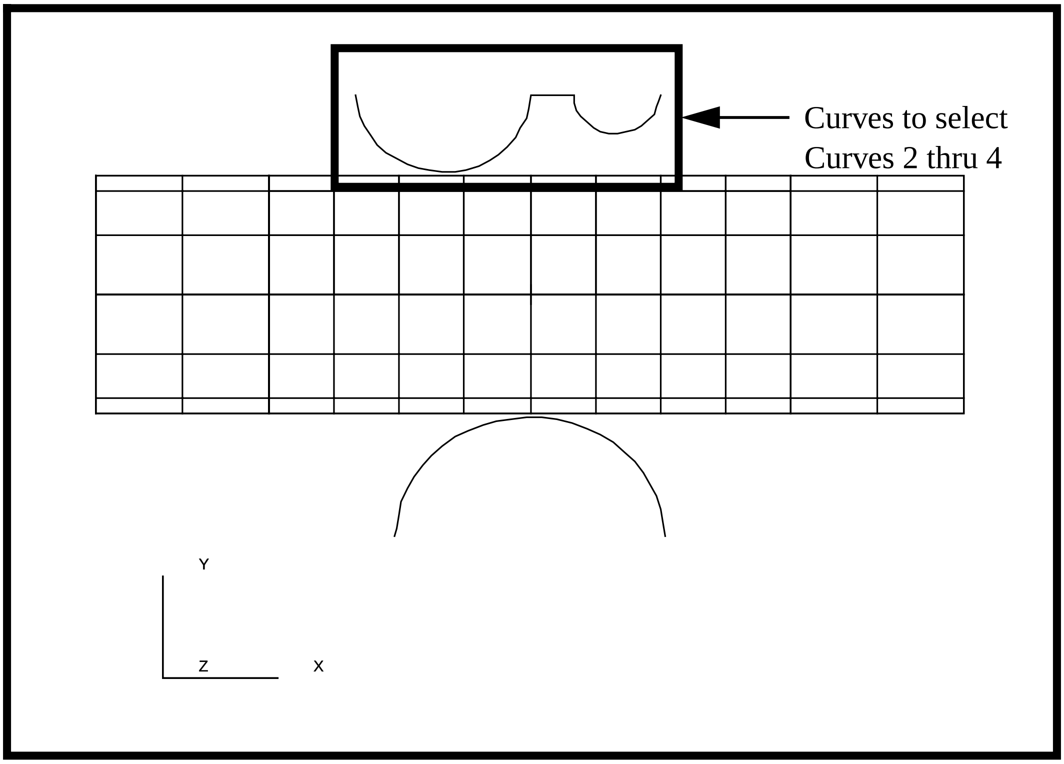
20. Create the upper rigid surface elements.

Next, 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 7a.12

Apply

Figure 7a.12 - Curves to select for upper rigid surface



21. Create a new group **fem_rigid_bottom**.

Group/Create ...

New Group Name	**fem_rigid_bottom**

■ **Make Current**

Group Contents:	**Add Entity Selection**

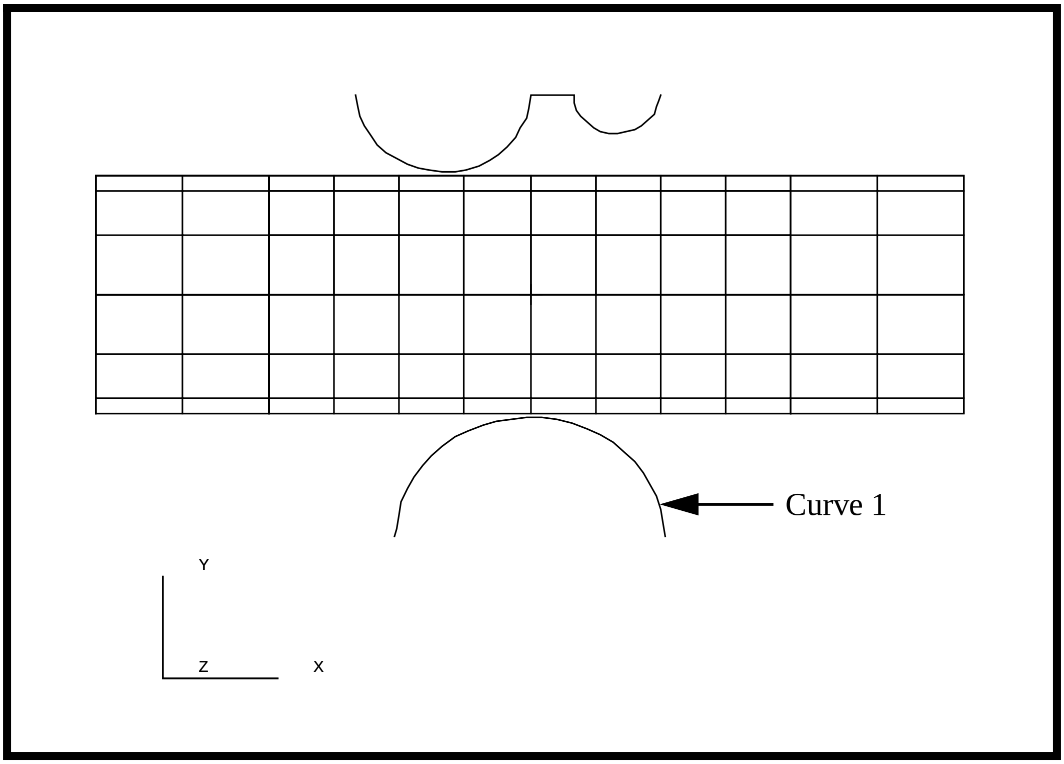
Apply

Cancel

22. 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 in Figure 7a.13:

Figure 7a.13 - Curve to select for lower rigid surface



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

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

Apply

24. 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

25. Create a new group **all**, which will be used to apply LBCs onto.

New Group Name **all**

■ **Make Current**

Group Contents: **Add All Entities**

Apply

Cancel

26. 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

27. Post the group **fem_pipe** only.

Group/Post ...

Select Groups to Post **fem_pipe**

Apply

Cancel

You may have to hit the *Refresh* button in the *Main window* in order to see your model.

28. Now create the Element Properties for your model.

◆ **Properties**

Action: **Create**

Dimension: **2D**

Type **Shell**

Property Set Name **pipe**

Options: **Thin**

Homogeneous

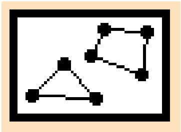
Input Properties...

Material Name **m:steel**

Shell Thickness **0.4**

OK

Click in the *Select Members* databox. Select the surface element icon in the Select menu and then select ***all*** elements displayed.



Add

Apply

29. Post the group **fem_irs_top** only.

Group/Post ...

Select Groups to Post **fem_irs_top**

Apply

Cancel

30. Create the element property for the top IRS elements.

Use the following toolbar icon to increase the node size to make picking easier:

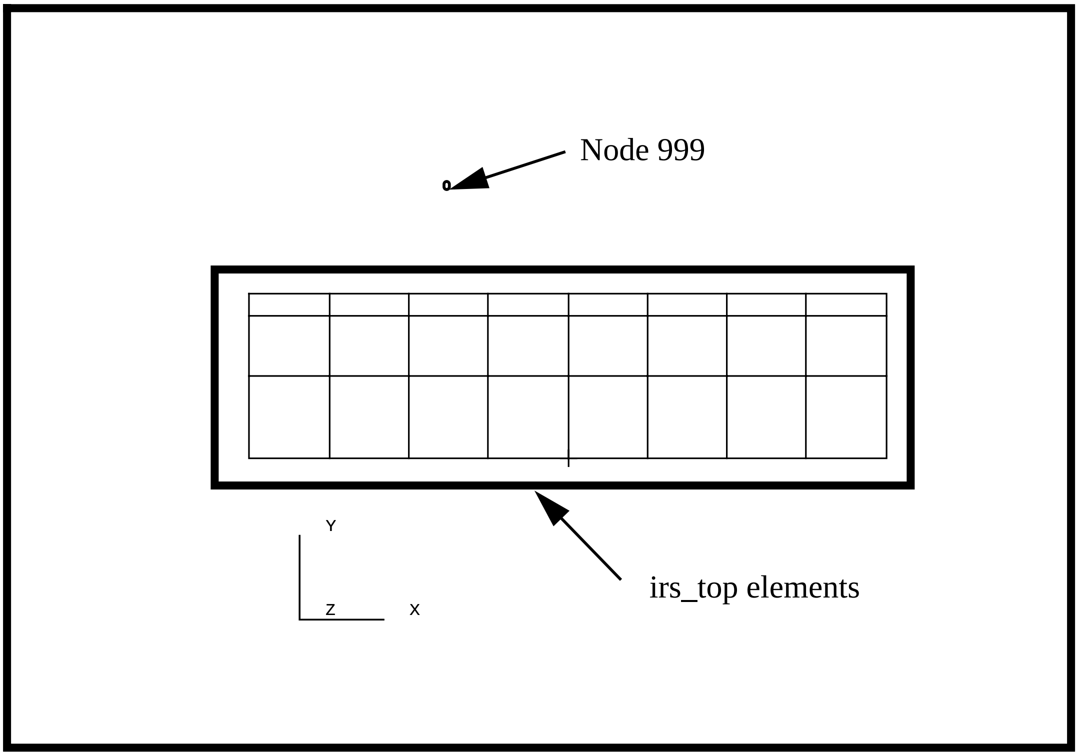
◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type	**IRS (shell/solid)**
Property Set Name	**irs_top**
Options:	**Elastic Slip Soft Contact**

Input Properties...

ELSET Name	**top**
Reference Node	see Figure 7a.14

Figure 7a.14 - Pipe elements/reference node for upper contact



OK

Click in the *Select Members* databox and screen select ***all*** members on the screen.

Add

Apply

31. Post the group **fem_irs_bottom** only.

Group/Post ...

Select Groups to Post **fem_irs_bottom**

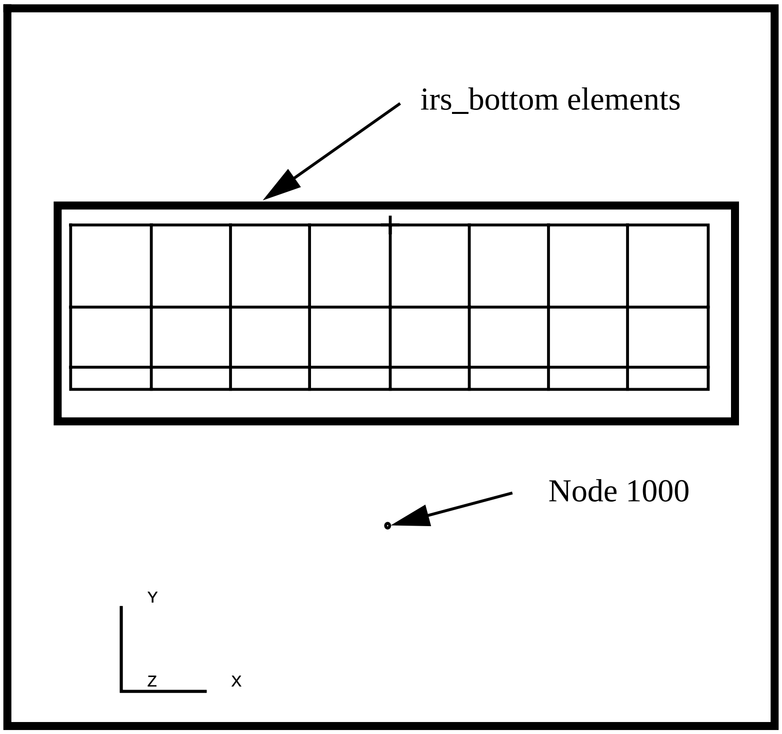
Apply

Cancel

32. In a similar manner, enter the information for the bottom IRS elements on the underside of the pipe.

Property Set Name	**irs_bottom**
Options:	**Elastic Slip Soft Contact**
Input Properties...	
ELSET Name	**bottom**
Reference Node	see Figure 7a.15

Figure 7a.15 - Pipe elements/reference node for lower contact



OK

Click in the *Select Members* databox and screen select ***all*** members on the screen.

Add

Apply

33. Post the group **fem_rigid_top** and **fem_rigid_bottom**.

Group/Post ...

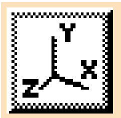
Select Groups to Post — **fem_rigid_top** **fem_rigid_bottom**

Apply

Cancel

A caution form will appear asking you to select a current group. Select **fem_rigid_top.** Press **OK,** then **Cancel**.

Click on the following icon to change to isometric view:



34. Finally, create the element properties for the upper and lower rigid surfaces.

To do this, you will have to define the normal which should point away from the rigid body and to the contact IRS elements. To do this, we will need to define a surface generation vector.

The *Surface Generator Direction* and the *Start Node* are used to determine the "inside" and the "outside" of the rigid body. We want the "outside" of the rigid body to contact the pipe model. A more formal definition of these terms is:

The generator direction follows the right hand rule and is specified as:

$$\hat{n} = \hat{s} \times \hat{z}$$

where

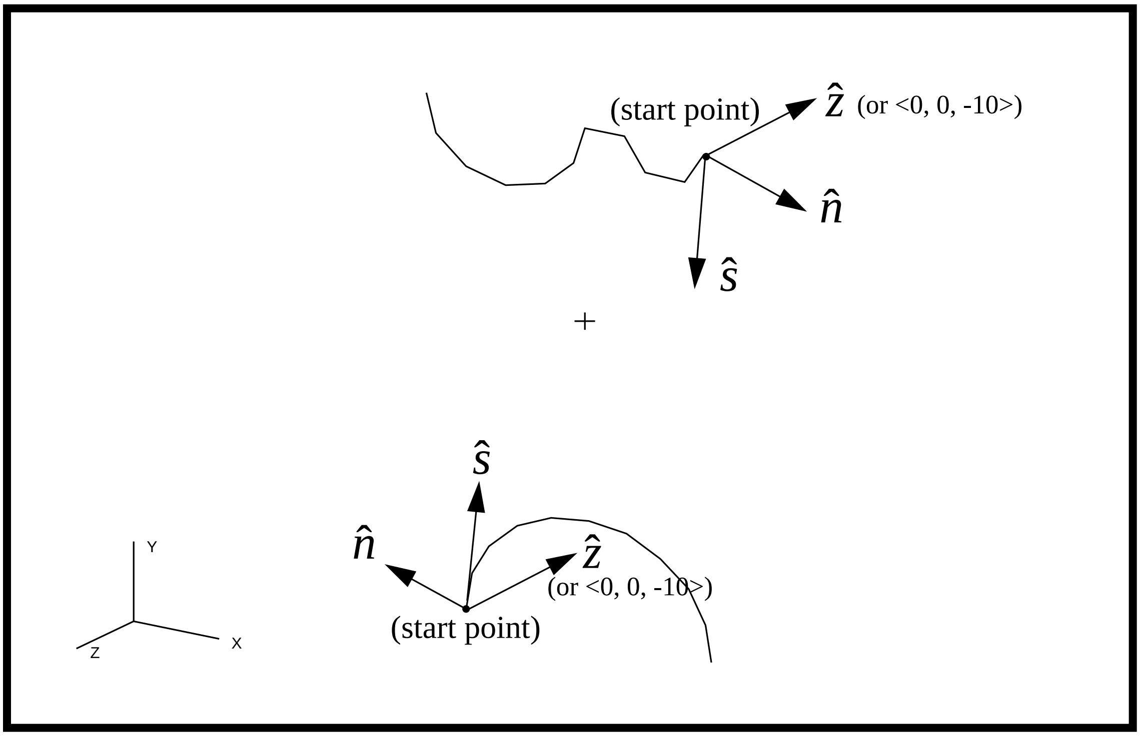
$\hat{n}$ is the unit normal direction.

$\hat{s}$ is the cord length direction.

$\hat{z}$ is the generator direction.

These vectors are shown for each of the rigid bodies in the sketch shown in Figure 7a.16:

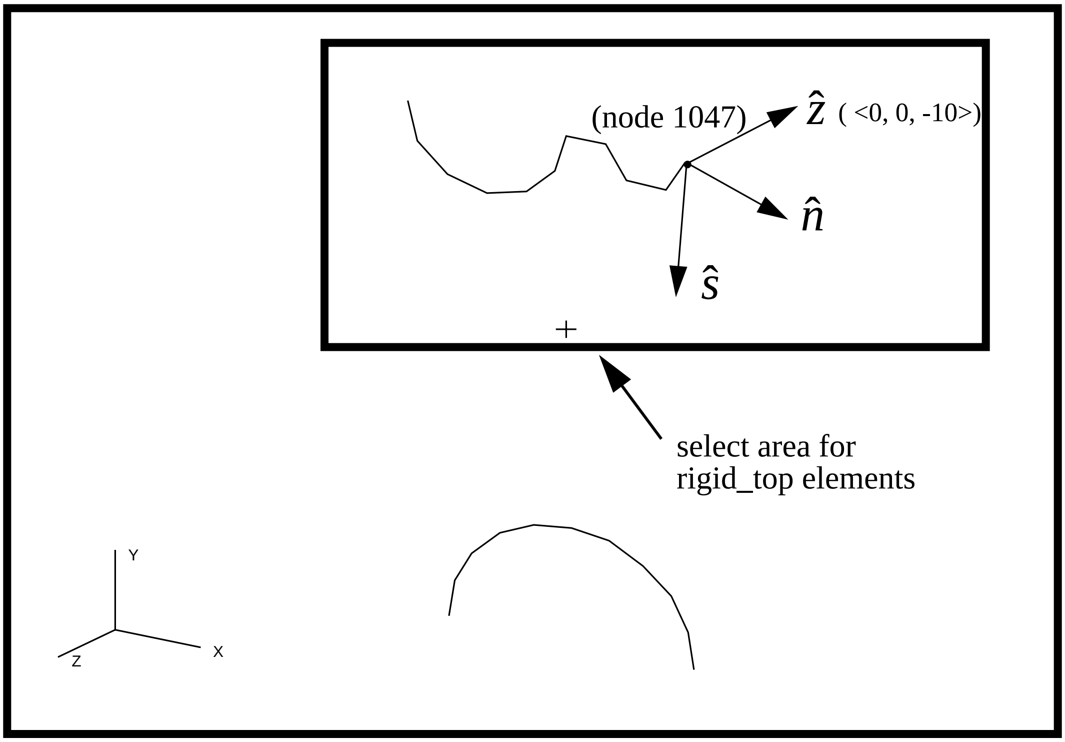
Figure 7a.16 - Sketch of normal directions for rigid bodies



◆ **Properties**

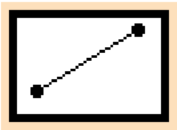
Action:	**Create**
Dimension:	**1D**
Type	**RigidSurf(Cyl)**
Property Set Name	**rigid_top**
Input Properties...	
ELSET Name	**top**
Surface Gen. Direction	**< 0, 0, -10>**
Start Point (Node_id)	see Figure 7a.17

Figure 7a.17 - Normal definitions for upper rigid body



OK

Click on *Select Members* databox, then select the Beam Element icon in the Select Menu.



Now screen select the members shown in the figure above. In a similar

Add

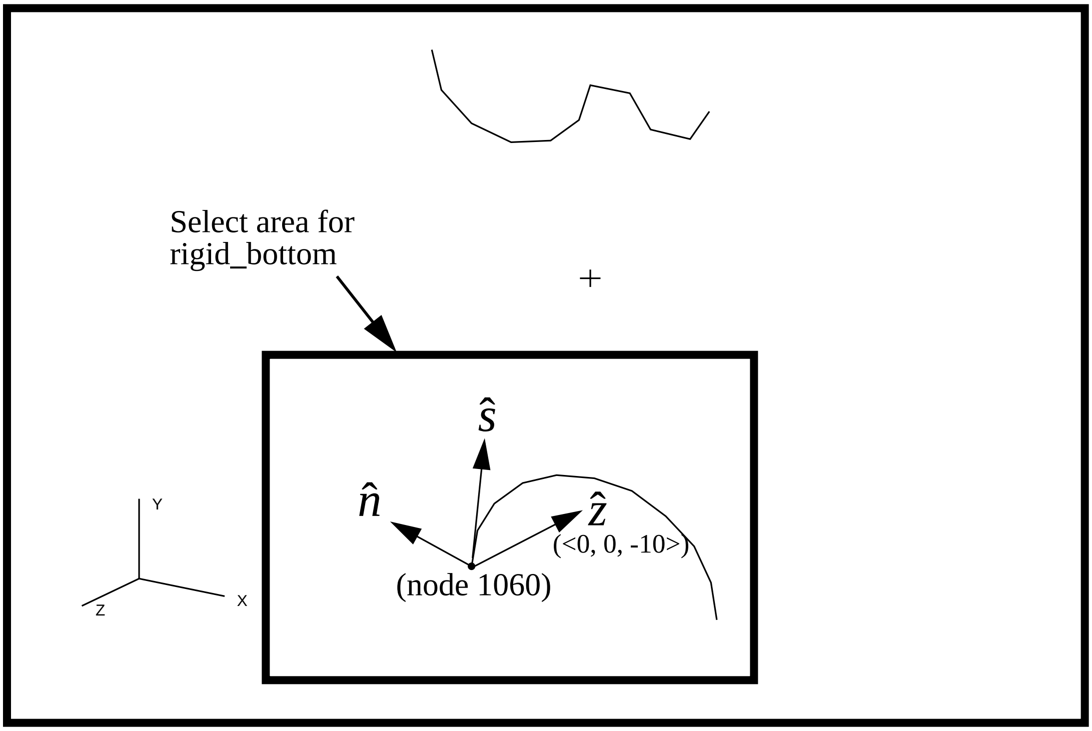
Apply

manner, enter the information for the Rigid Surface elements for the lower rigid surface.

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type	**RigidSurf (Cyl)**
Property Set Name	**rigid_bottom**
Input Properties...	
ELSET Name	**bottom**
Surface Gen. Direction	**< 0, 0, -10>**
Start Point (Node_id)	select the start node shown in Figure 7a.18
OK	
Select Members	shown in Figure 7a.18
Add	
Apply	

Figure 7a.18 - Normal definitions for lower rigid body

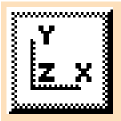


35. Post the group “**all**” only. We will define the loads and boundary conditions.

Group/Post ...

Select Groups to Post	**all**
Apply	
Cancel	

Select the following toolbar front view icon to change the display.



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

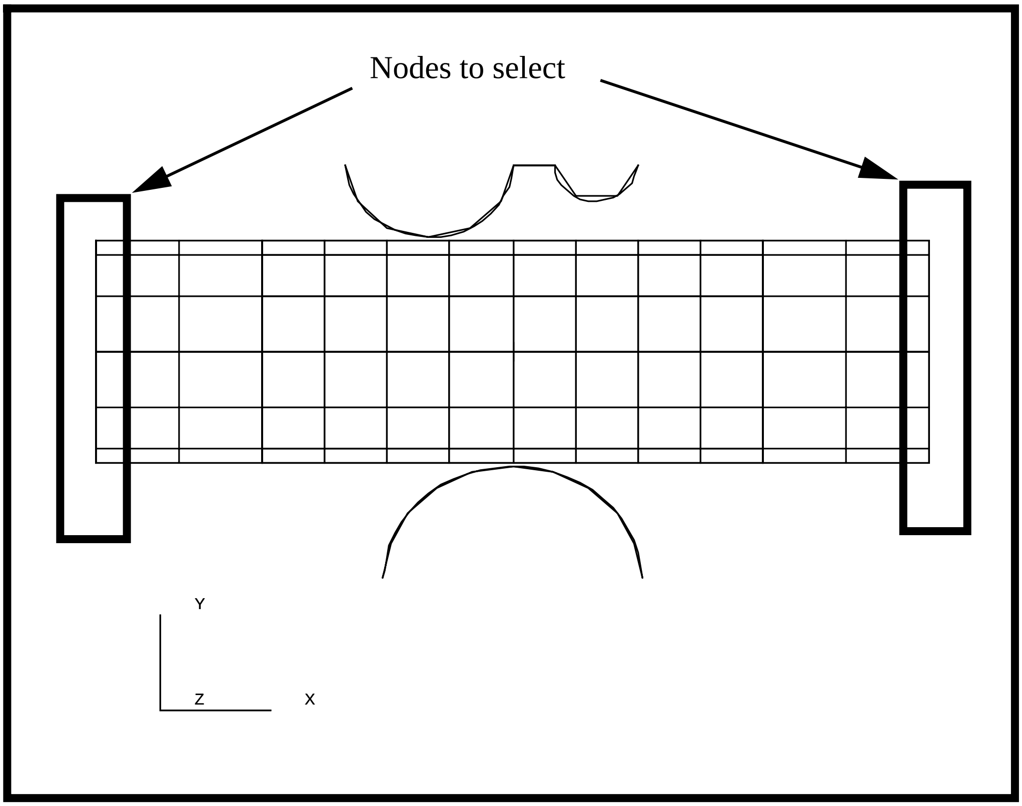
The ends of the pipe are to be fixed in all translations. The upper rigid surface will then be forced down into the pipe and lower rigid surface forced up into the pipe.

◆ **Load/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 at both ends of the pipe as shown in Figure 7a.19:

Figure 7a.19 - Fixed ends of pipe



Add

OK

Apply

37. Next create the symmetrical edge constraints for the top and bottom edges of the pipe model.

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name	**symmetry**

Input Data...

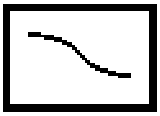
Translations <T1,T2,T3>	**< , , 0>**
Rotations <R1,R2,R3>	**< 0, 0, >**

OK

Select Application Region...

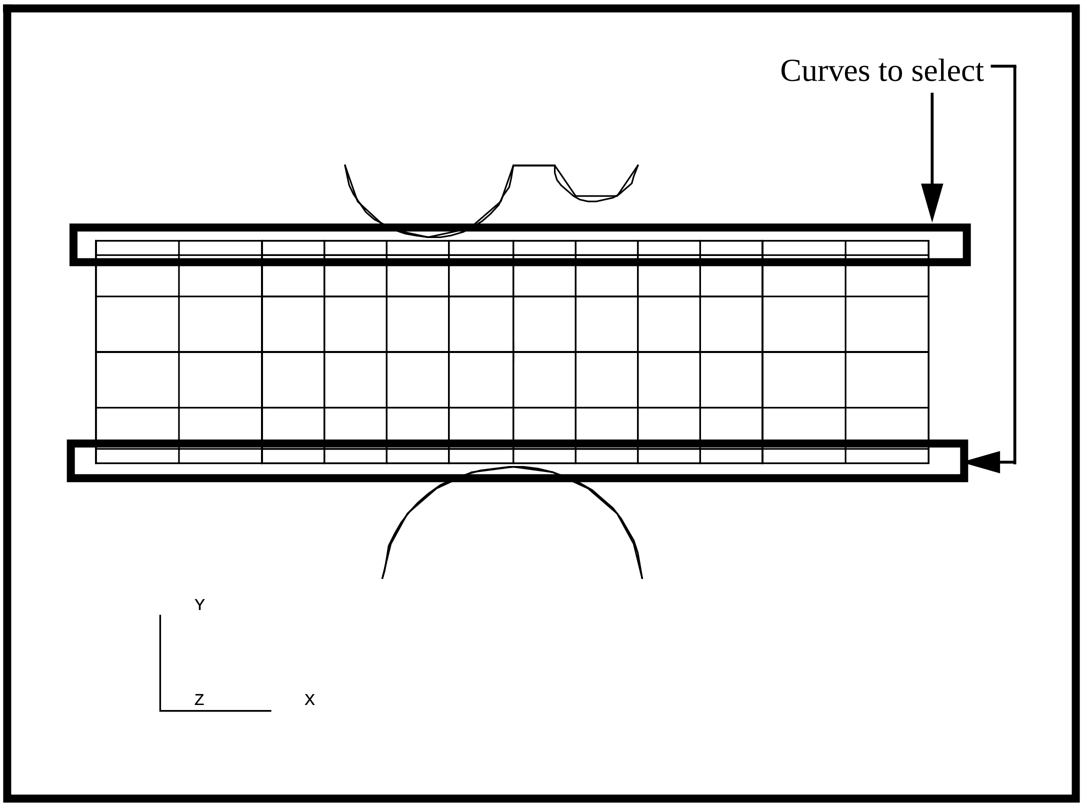
Geometry Filter ◆ **Geometry**

Click in the *Select Geometry Entities* databox, change the select menu icon at the bottom to *Select a Curve*,



Screen select the surface edges at both ends of the pipe as shown in Figure 7a.20:

Figure 7a.20 - Locations for symmetry BCs



Add

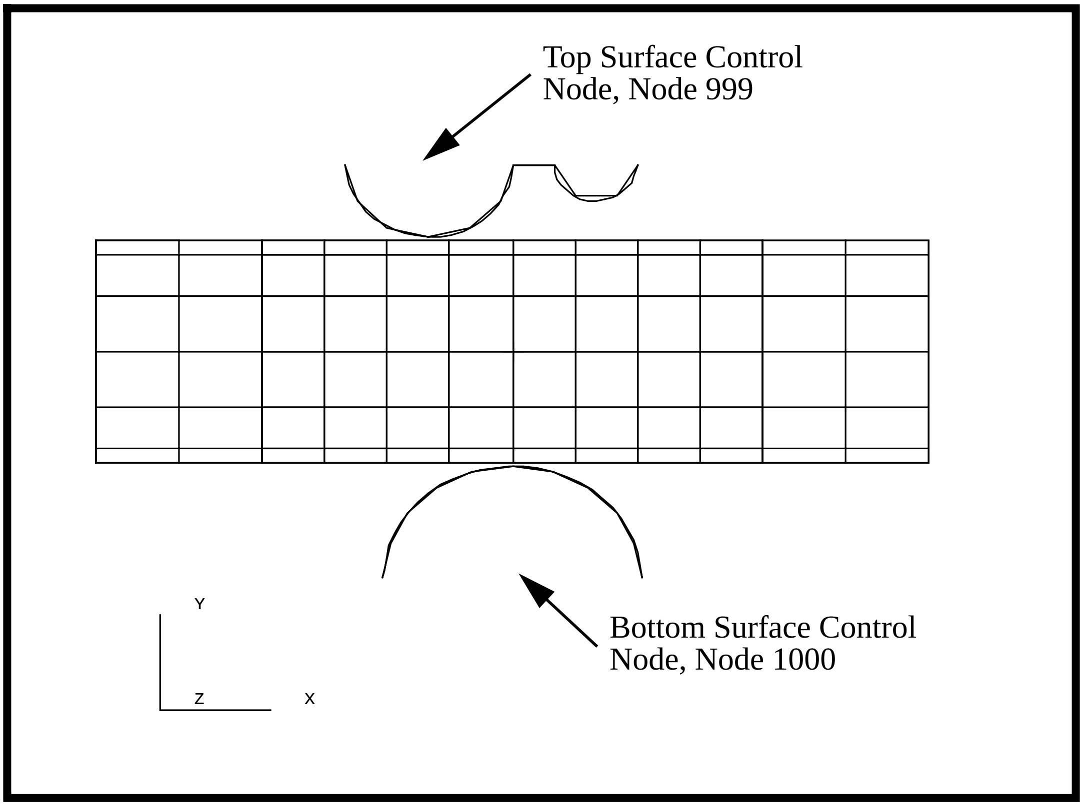
OK

Apply

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

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	**Node 999**
Add	
OK	
Apply	

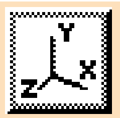
Figure 7a.21 - Reference nodes for rigid bodies



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

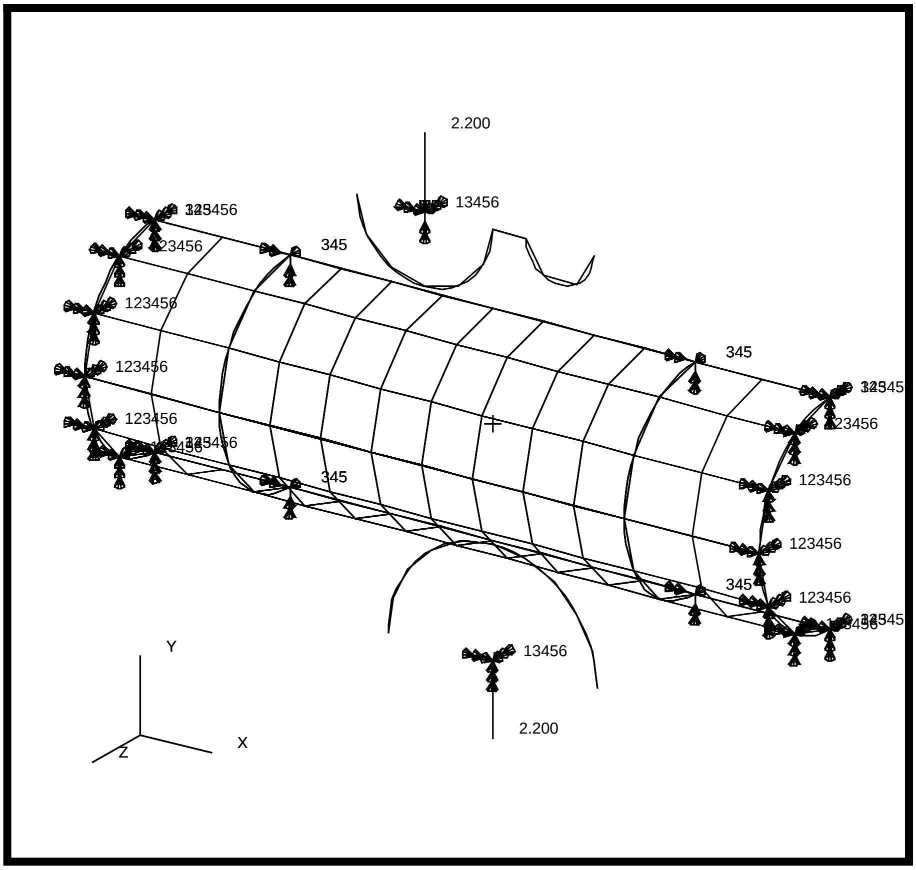
New Set Name	**bot_rigid_up**
Input Data...	
Translations <T1,T2,T3>	**< 0, 2.2, 0>**
Rotations <R1,R2,R3>	**< 0, 0, 0 >**
OK	
Select Application Region...	
Geometry Filter	◆ **FEM**
Select Nodes	**Node 1000**
Add	
OK	
Apply	

Change the display of the model using the isometric view icon.



Your model should now look like the picture shown in Figure 7a.22:

Figure 7a.22 - Model with all Loads/BCs applied



40. Your model is now ready for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name	**pipe_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 “pipe_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 **pipe_crush.fil** in the same directory you started MSC/PATRAN in and the **pipe_crush.023** file will disappear.

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

41. Read in the results.

◆ **Analysis**

Action: **Read Results**

Select Results File...

Selected Results File **pipe_crush.fil**

OK

Apply

42. Post the group “fem_all” only for post processing.

Group/Post ...

Select Groups to Post	**fem_all**

Apply

Cancel

43. Change the Display Properties for results.

Display/Results ...

Scale Factor	**1.0**

◆ **Direct Multiplication**

❐ **Show Undeformed Entities**

Apply

Cancel

Hit the *Refresh* button in the *Main window* in order to redisplay your model.

44. Use results to post process your results of the analysis.

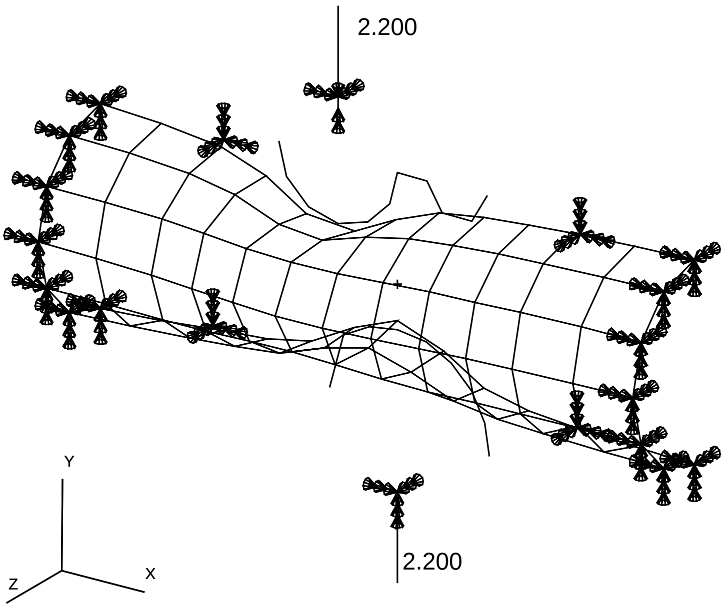
◆ **Results**

Form Type:	**Basic**
Select Results Cases	select the last increment
Select Deformation Result	**Deformation, Displacement**

Apply

Your model should appear as shown in Figure 7a.23:

Figure 7a.23 - Deformed pipe model



45. Create an animation of the deformation and Von Mises stresses using:

Form Type:	**Basic**
Select Result Type	select the last increment
Select Fringe Result	**Stress, Components**
Result Position	**Section Point 1**
Result Type	**Von Mises**
Select Deformation	**Deformation, Displacement**

■ **Animate Results**

Animation Options ...

■ **Animate Fringe**

■ **Animate Deformation**

Animation Method	◆ **Ramped**
Animation Graphics	◆ **2D**
Number of Frames	**15**

OK

Apply