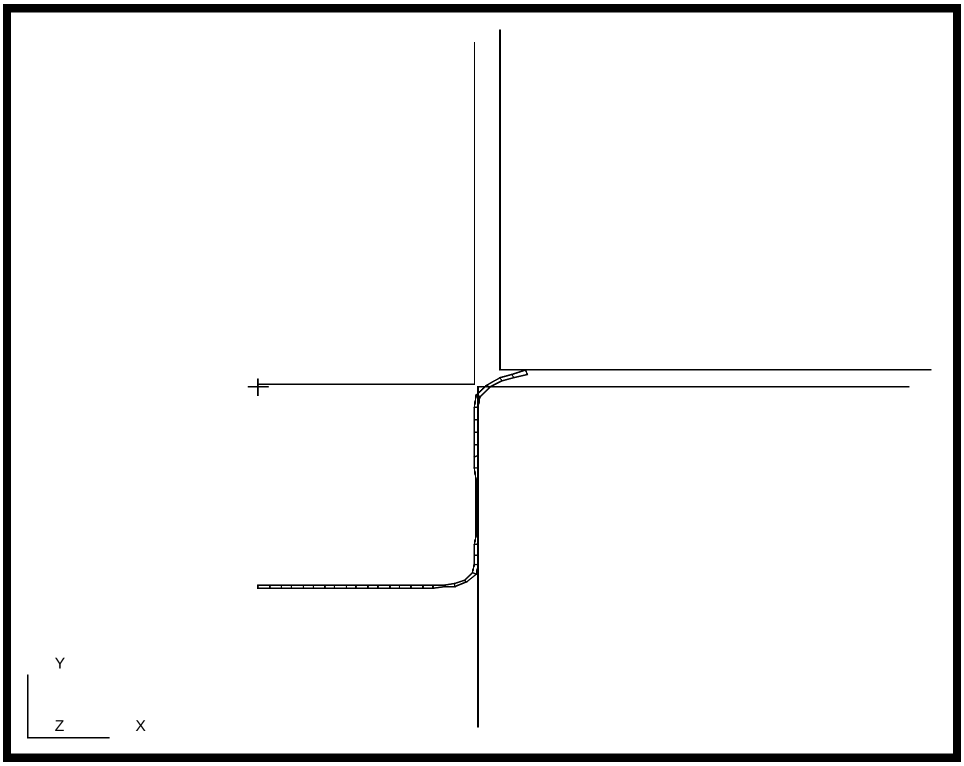
APPENDIX 6

Deep Drawing of a Cylindrical Cup

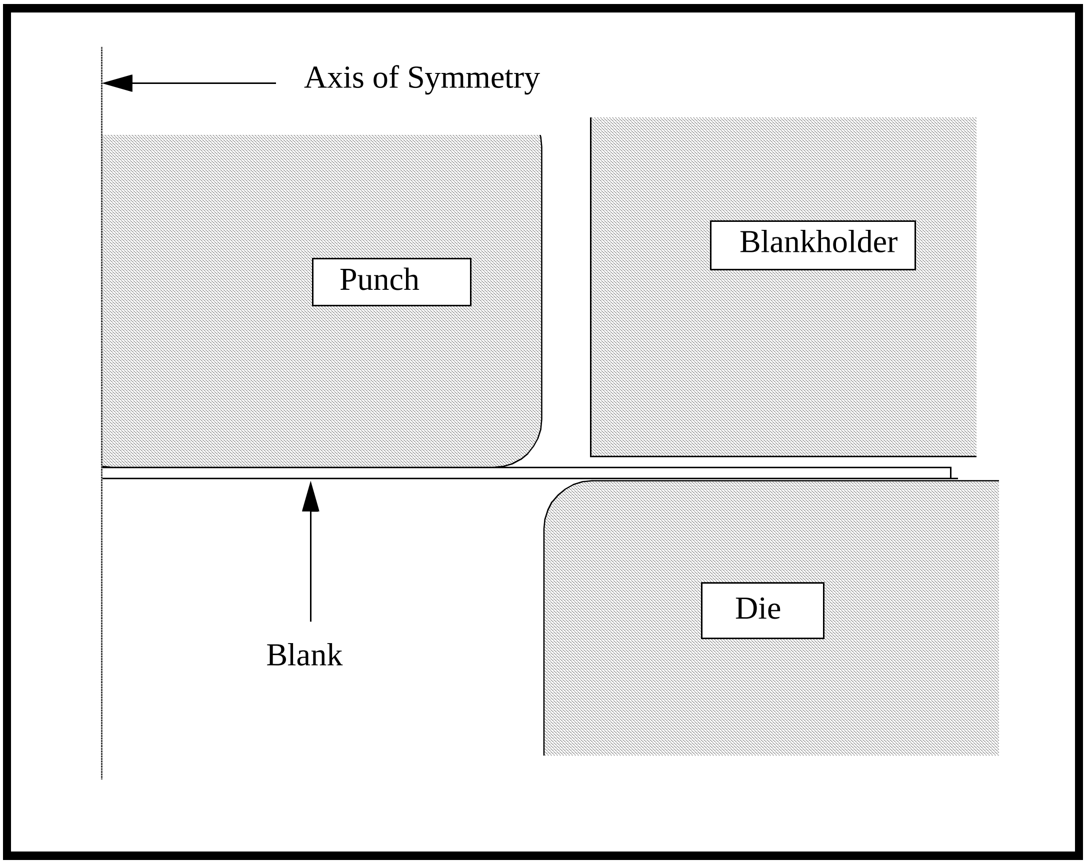


Objectives:

- Large displacement analysis.
- Contact analysis.
- Plasticity theory.

Model Description:

In this Exercise we analyze a manufacturing process which stamps a steel cup. This example problem illustrates the use of multiple steps to model a complicated manufacturing procedure. A flat sheet of steel is loosely secured over a die and plastically deformed with a punch to form a steel cup. The punch is then withdrawn, and the cup released from the die. One might use an analysis such as this to fine tune the manufacturing procedure with respect to parameters such as the clasping force that holds the metal blank over the die, and to examine the resulting residual stress. This problem makes extensive use of rigid surface interfaces. To simplify the analysis this problem takes advantage of the axisymmetric nature of the cup, punch and die. It also includes a strain-hardening plasticity law and large displacements and rotations.



Suggested Exercise Steps:

- Build the geometry
- Mesh the geometry
- Model the Contact Surfaces By Meshing with Bars and creating MPCs
- Define the Materials and Properties
- Define the Load sets
- Group the Load Sets into Load Cases
- Run the analysis
- Evaluate the results

Exercise Procedure:

1. Open a new database called **cup.db**.

File/New ...

Database Name:	**cup.db**

OK

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

Analysis Code:	**MSC/ADVANCED_FEA**

OK

2. Create the geometry for the blank

First, create a *Group* called **blank**.

Group/Create...

New Group Name:	**blank**

■ **Make Current**

Group Contents:	**Add Entity Selection**

Apply

Cancel

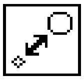
◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinates List:	**<4, .0325, 0>**
Origin:	**[0, 0, 0]**

Apply

3. Increase the point size to make viewing easier.

Increase the *point size* by selecting the following icon.

 **Point Size**

4. Create the Finite Element Mesh for the **blank**.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length:	**0.1**
Element Topology:	**Quad4**
Surface List:	**Surface 1**

Apply

5. Model the die rigid surface elements.

In this step, you are going to coat the bottom surface of the blank with bar elements. Later you are going to assign IRS element properties to these elements. These IRS elements correspond to the die rigid surface to be created later.

Create a new group called **die** and make it current so all the newly created entities will be contained in this group.

Group/Create...

New Group Name:	**die**

■ **Make Current**

Group Contents:	**Add Entity Selection**

Apply

Cancel

6. Create the reference node and rigid surface for the die.

◆ **Geometry**

Action:	**Create**
Object:	**Point**

Method:	**XYZ**
Point Coordinates List:	**[4, -2, 0]**

Apply

This creates Point 5

7. Create the geometry for the die rigid surface.

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinates List:	**<0, -4, 0>**
Origin Coordinates List:	**[2.05, -0.1, 0]**

Apply

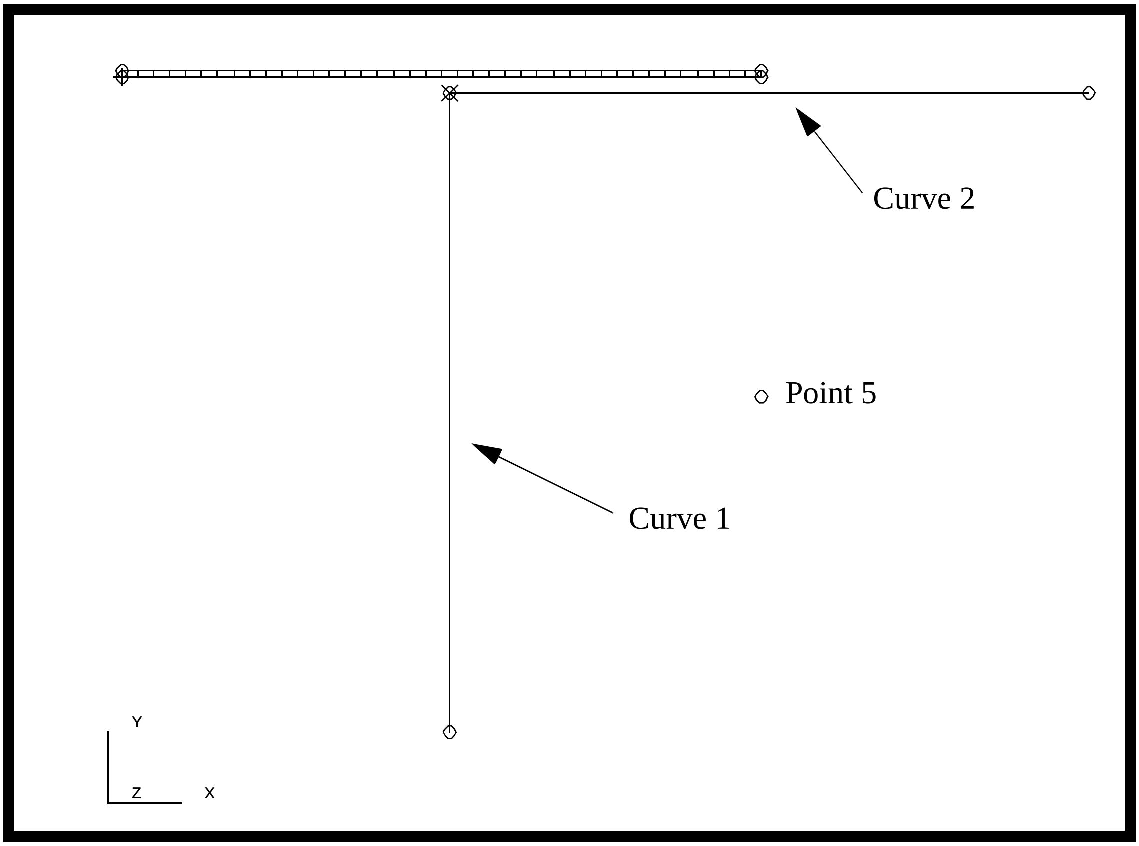
This creates **Curve 1**.

Repeat the procedure with these changes

Vector Coordinates List:	**<4, 0, 0>**
Origin Coordinates List:	**[2.05, -0.1, 0]**

Apply

Figure 5.1 - Created geometry for die rigid surface



8. Define the finite elements for the die slave surface and die reference node.

◆ **Finite Elements**

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node Location List:	**Point 5**

Apply

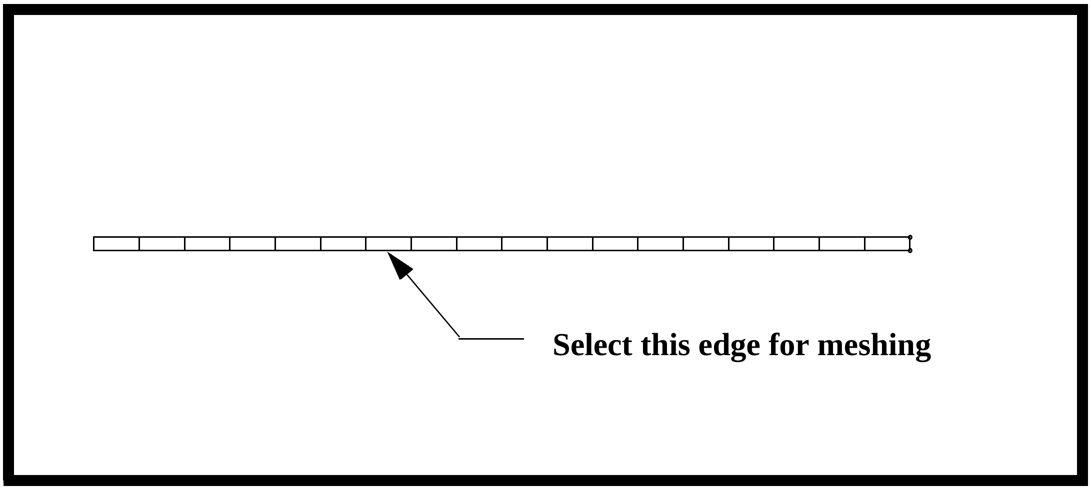
This is the control node for the die rigid surface. Now create the slave surface elements for the die**.** These are located along the bottom of **Surface 1** as shown in Figure 5.2.

Action:	**Create**
Object:	**Mesh**

Type:	**Curve**
Global Edge Length:	**0.1**

Select the lower edge of the blank, **Surface 1.4**

Figure 5.2 - Blank edge to contact die surface.



Curve List:	**Surface 1.4**

Apply

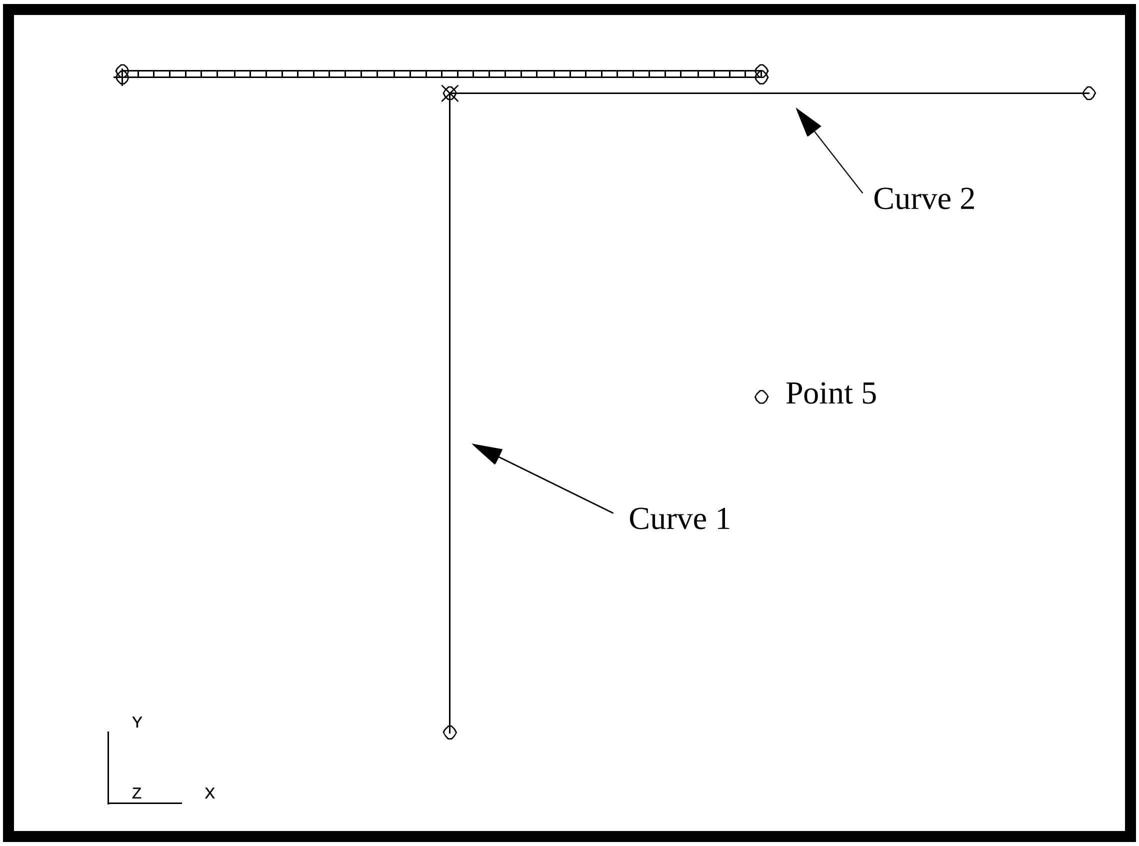
Duplicate nodes are created when the Bar2 elements are created on the edge of the surface. You will use equivalencing in a later step to remove all duplicate nodes.

9. Create the finite elements for the die rigid surface.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**4**
Curve List:	**Curve 1, 2**

Figure 5.3 - Geometry to create die surface elements upon



Apply

10. Create the punch rigid surface elements.

In this step, you are going to coat the top surface of the blank with bar elements again. Later you are going to assign IRS element properties to these elements. These IRS elements will correspond to the punch rigid surface to be created later.

Group/Create...

New Group Name: **punch**

■ **Make Current**

Group Contents: **Add Entity Selection**

Apply

Cancel

11. Create the geometry for the punch reference node and rigid body.

◆ **Geometry**

Action:	**Create**
Object:	**Point**
Method:	**XYZ**
Point Coordinates List:	**[0, 2, 0]**

Apply

This results in **Point 9**.

12. Create the geometry for the punch rigid surface.

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinates List:	**<0, 4, 0>**
Origin Coordinates List:	**[2.0, 0.1325, 0]**

Apply

This creates **Curve 3**

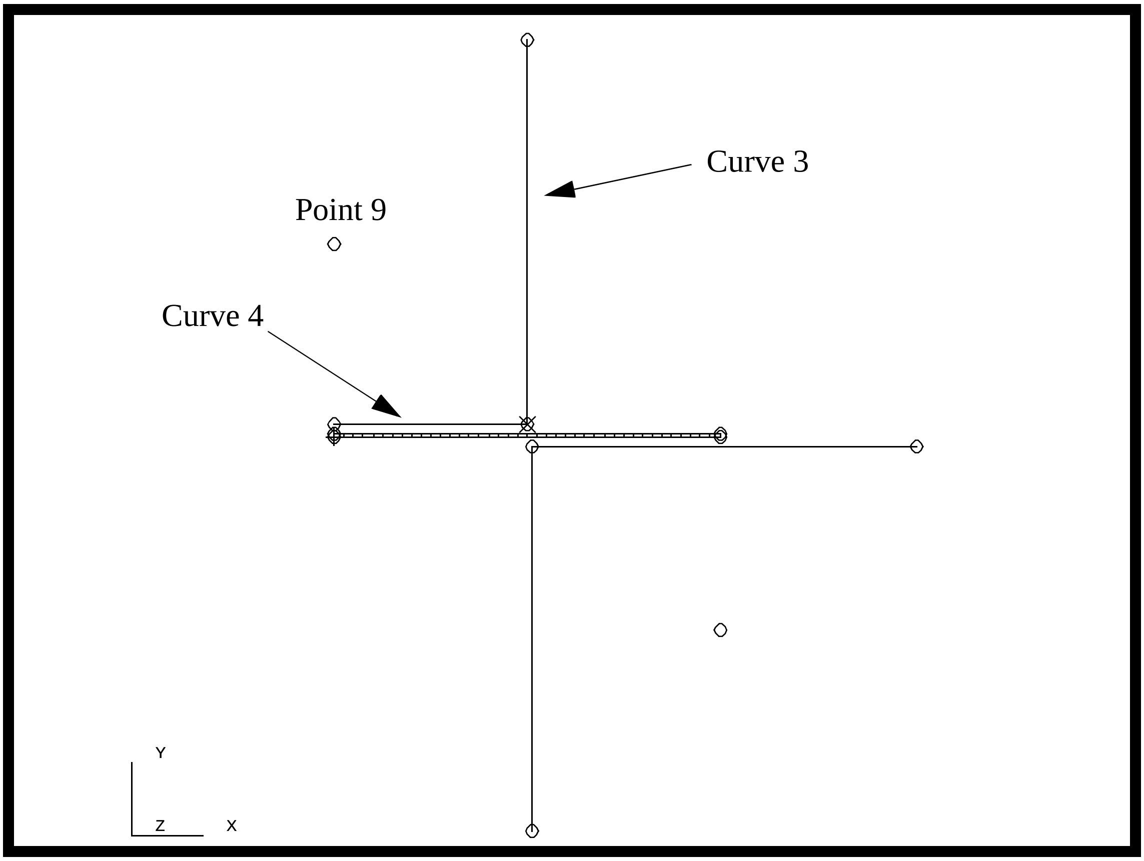
Repeat the procedure with these changes

Vector Coordinates List:	**<-2, 0, 0>**
Origin Coordinates List:	**[2, 0.1325, 0]**

Apply

This creates **Curve 4.**

Figure 5.4 - Created geometry for punch rigid surface



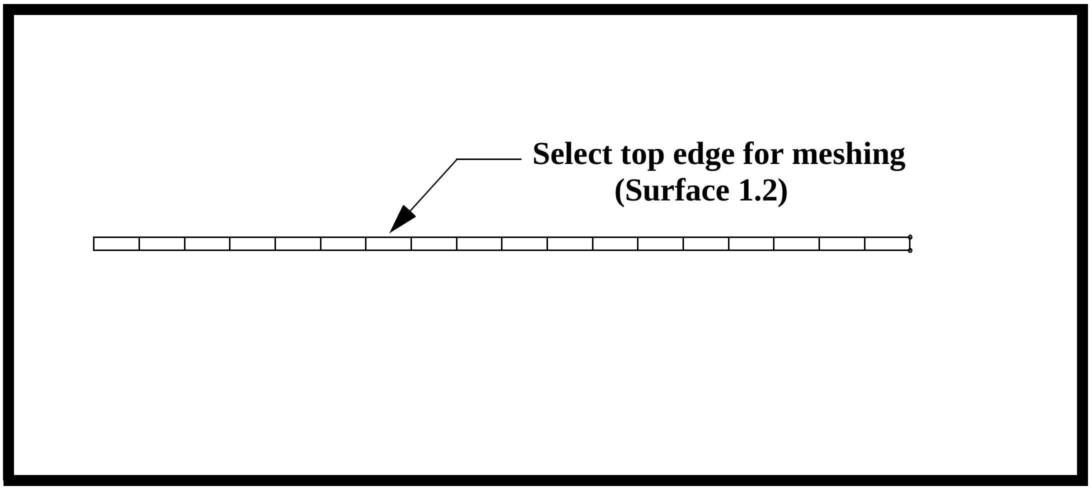
13. Mesh the top edge of surface 1 with bar2 elements to define the slave contact surface for the punch.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**0.1**
Curve List:	**Curve 7, 8**

Apply

Figure 5.5 - Blank edge to contact punch surface



14. Create the finite elements for the punch rigid surface and reference node.

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**4**
Element Topology:	**Bar2**
Curve List:	**Curve 3 4**

Apply

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node Location List:	**Point 9**

Apply

This is the control node for the punch rigid surface.

15. Create the holder interface rigid surface (IRS) elements

In the next step, you are going to coat the half of the top surface of the blank with bar elements again. Later you are going to assign IRS element properties to these elements. These IRS elements will correspond to the holder rigid surface to be created later. Across this section you will have coincident bar2 elements, one set corresponding to the punch IRS elements and the other corresponding to the holder IRS elements. We will use groups to keep track of these elements.

Group/Create ...

■ **Make current**

New Group Name:	**holder**
	Add Entity Selection

Apply

Cancel

◆ **Geometry**

Action:	**Create**
Object:	**Point**
Method:	**XYZ**
Point Coordinates List:	**[4, 2, 0]**

Apply

This creates **Point 13.**

The holder only comes in contact with half the length of the blank. Using Geometry editing, we break the curve represented by the top surface edge at its halfway point and create two curves.(Hint: you will need to use the Curve Edge Select Filter). This creates two curves, **Curve 5, 6**.

Action:	**Edit**
Object:	**Curve**
Method:	**Break**
Option:	**Parametric**
Break Point:	**0.5**

Select Curve:	Select the top edge of Surface 1 (Surface 1.2)

Apply

16. Create the geometry for the holder rigid surface.

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinates List:	**<0, 4, 0>**
Origin Coordinates List:	**[2.25, 0.1325, 0]**

Apply

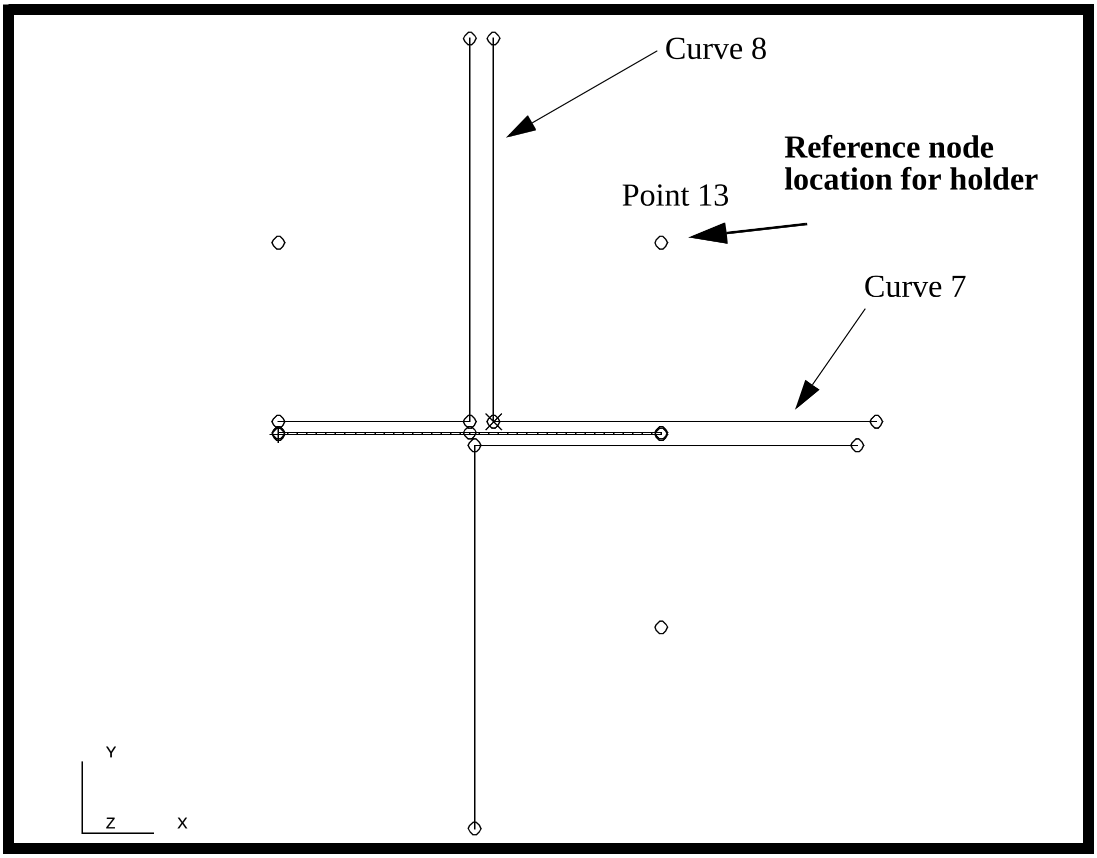
This creates **Curve 7**.

Vector Coordinates List:	**<4, 0, 0>**
Origin Coordinates List:	**[2.25, 0.1325, 0]**

Apply

This creates **Curve 8.**

Figure 5.6 - Created geometry for holder rigid surface.



17. Now create the holder reference node and mesh the slave surface.

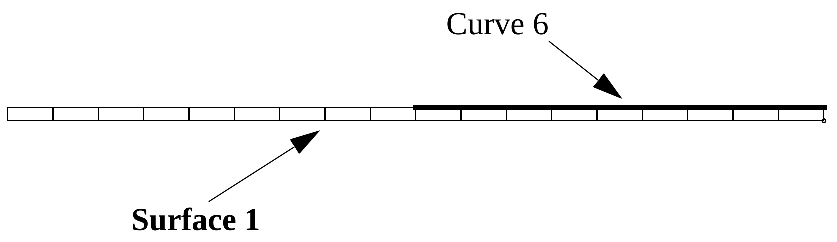
◆ **Finite Elements**

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node Location List:	see Figure 5.6

Apply

This is the control node for the holder. Now mesh where the holder will contact the **Surface 1**.

Figure 5.7 - Blank edge to contact holder surface



Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**0.1**
Element Topology:	**Bar2**
Curve List:	**Curve 6**

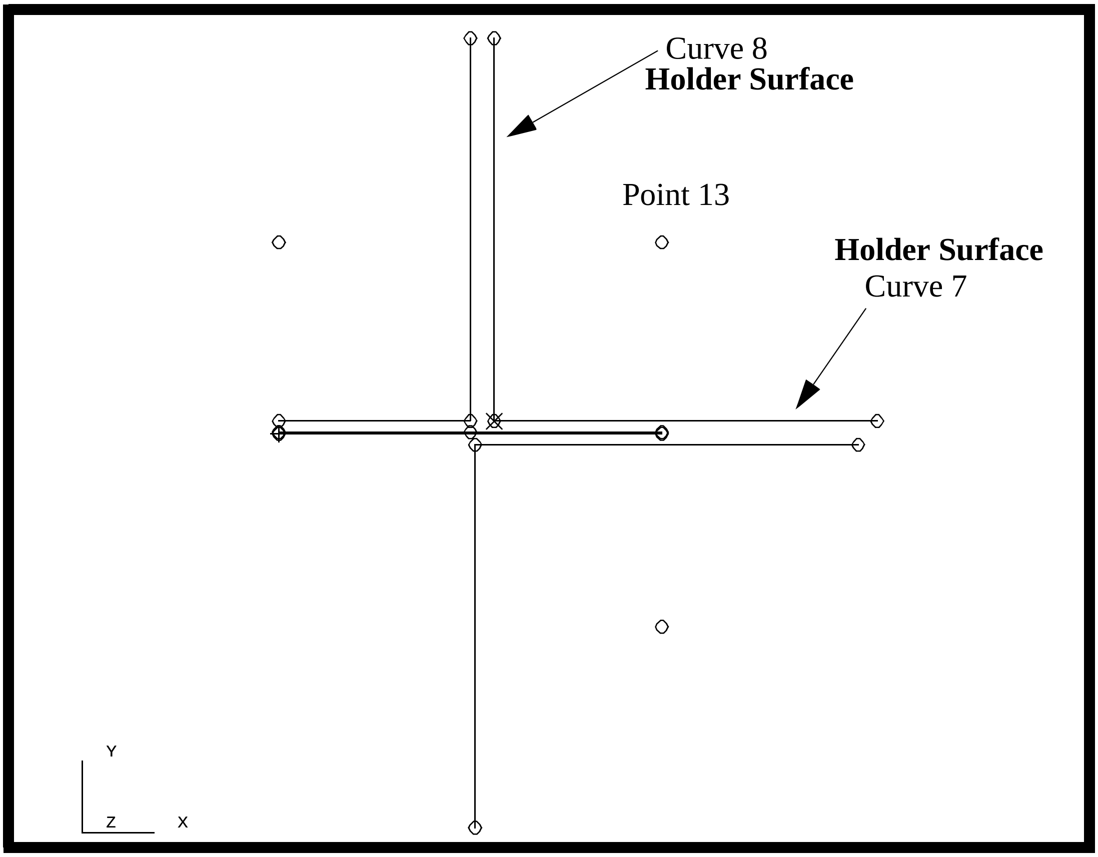
Apply

20 elements should be created as a result of this meshing operation.

18. Create the finite elements for the holder rigid surface.

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**4**
Element Topology:	**Bar2**
Curve List:	see Figure 5.8

Figure 5.8 - Geometry to create holder surface elements upon



Apply

19. Equivalence all duplicate nodes created during the meshing operation.

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

Apply

106 duplicate node will be removed.

20. Create the Hardening Curve needed for Material Properties.

In this step you will create a tabular field which represents strain hardening of the blank. This will be done by importing an existing hardening curve using the session file feature of MSC/PATRAN

File/Session/Play ...

Session Files List: **s_vs_e_us.ses**

Apply

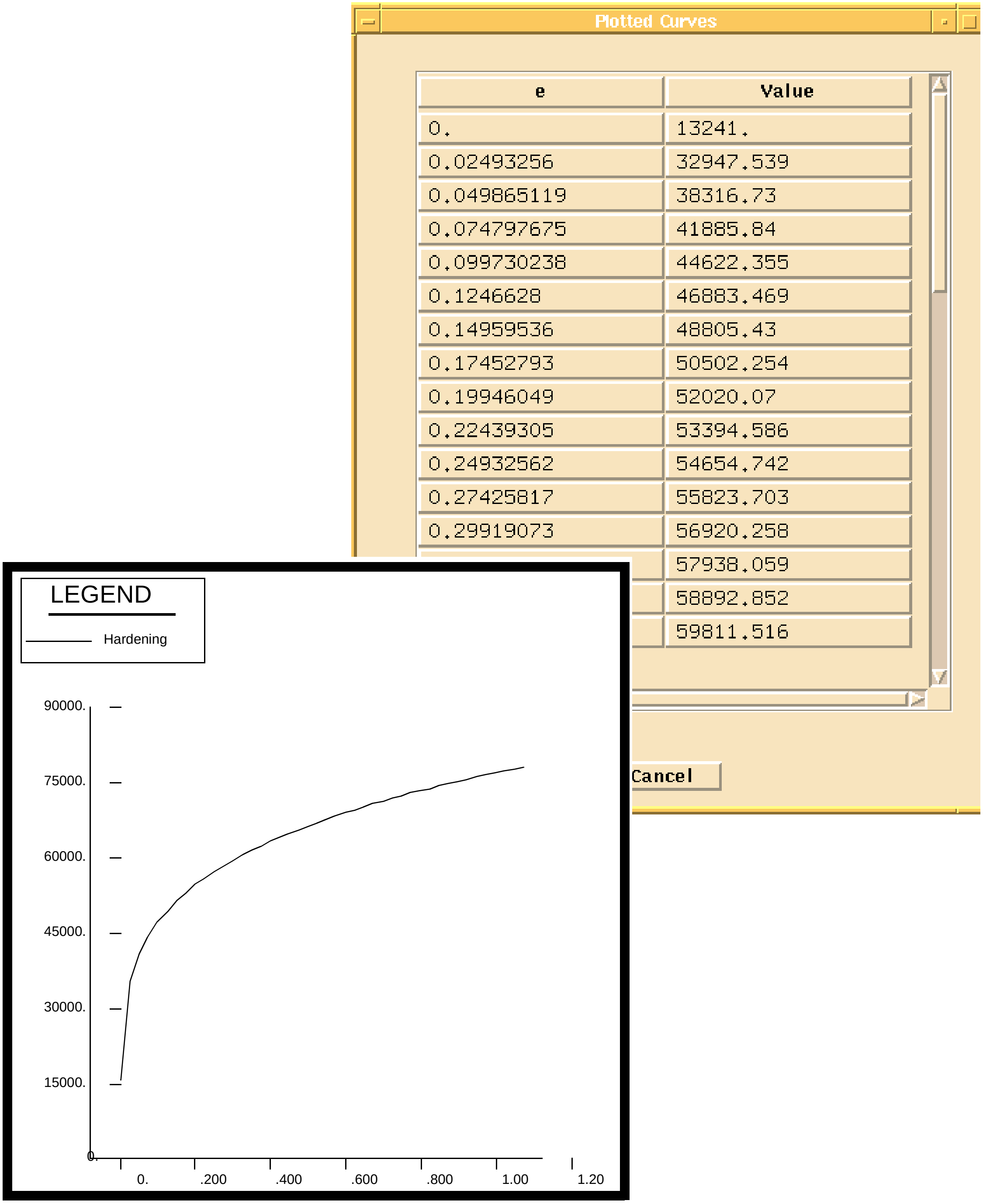
Now, you will look at the strain hardening field that was created by playing the hardening session file.

◆ **Fields**

Action: **Show**

Select Field to Show: **Hardening**

Apply


Plotted Curves

e	Value
0.	13241.
0.02493256	32947.539
0.049865119	38316.73
0.074797675	41885.84
0.099730238	44622.355
0.1246628	46883.469
0.14959536	48805.43
0.17452793	50502.254
0.19946049	52020.07
0.22439305	53394.586
0.24932562	54654.742
0.27425817	55823.703
0.29919073	56920.258
	57938.059
	58892.852
	59811.516

This will display the Field as an XY Plot and also in tabular format. Press **Cancel** in the Plotted Curves form and press **Unpost Current XYWindow** in the *Fields* Application form.

Cancel

Unpost Current XY Window

21. Create the material **steel** for the blank.

In this step you will create one material property with two constitutive material properties, linear elastic and plastic. The plastic constitutive material property will be created from the imported hardening curve. First, create the linear elastic constitutive model.

◆ **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

Define the plastic portion of the curve.

Constitutive Model:	**Plastic**
Yield Criteria:	**Mises/Hill**
Hardening Rule:	**Isotropic**
Stress vs. Plastic Strain:	**Hardening**

Apply

Cancel

22. Create the Element Properties for the blank.

In this step you will create a Element Property for the blank. The blank will be modeled as an axisymmetric 2D Solid and use the previously created steel material.

First post only the **blank** group

Group/Post...

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

◆ **Properties**

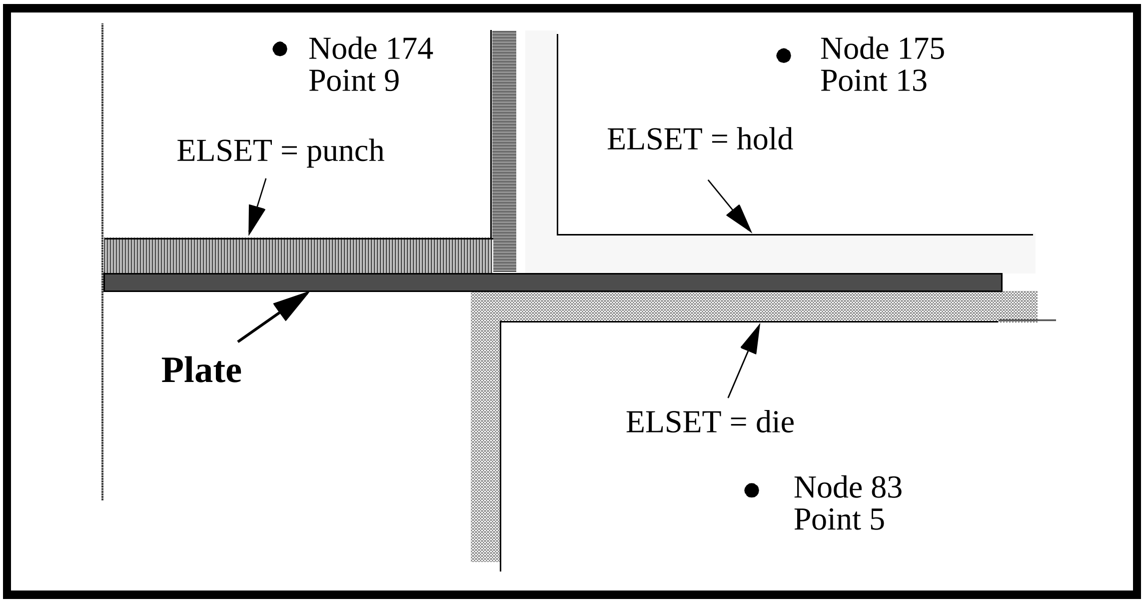
Action:	**Create**
Dimension:	**2D**
Type:	**2D Solid**
Property Set Name:	**blank**
Options:	**Axisymmetric**
	Reduced Integration
Input Properties...	
Material Name:	**steel**
OK	
Application Region:	**Surface 1**
Add	
Apply	

23. Create the Contact Surface Properties for the die

In this step, you will create element properties for the die rigid surface/ IRS pair. You will first create the element properties for the slave contact surface, the die IRS elements. Then you will create the element properties for the master contact surface, the die rigid surface. These two property sets will be associated to each other by a ELSET named die.

To aid in applying the elements properties to the appropriate elements, you will make use of the previously created groups. You will be performing similar steps for the punch and holder too. The ELSET Names for all rigid surface/IRS pairs are listed in Figure 5.9:

Figure 5.9 - Diagram of all surfaces (and reference nodes) and corresponding ELSET pairs



Post the group **die**.

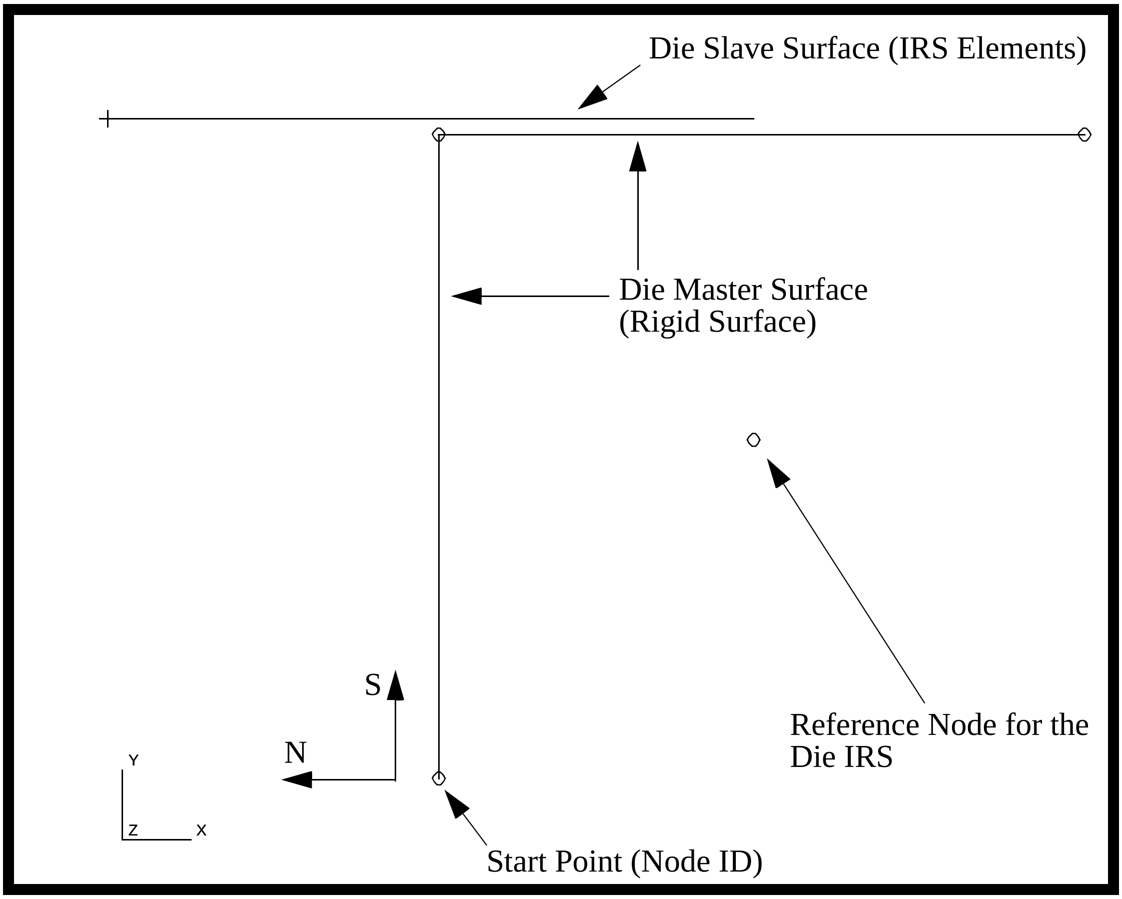
Group/Post...

Select Groups to Post: **die**

Apply

Cancel

Figure 5.10 - Die surface definitions



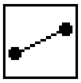
Now create the *Element Properties* for the **Slave Surface**

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type:	**IRS (planar/axisym)**
Property Set Name:	**die_irs**
Options:	**Axisymmetric**
	Elastic Slip Hard Contact
Input Properties...	
ELSET Name:	**die**
Reference Node:	see Figure 5.10
[Friction in Dir_1]:	**0.125**

OK

You will have to use the following element select icon.

 **Beam Element**

Application Region: select the slave surface

Add

Apply

Now create the *Element Properties* for the master surface.

Action: **Create**

Dimension: **1D**

Type: **RigidSurf(Seg)**

Property Set Name: **die_rigid_surface**

Input Properties...

ELSET Name: **die**

Start Point (Node_id): see Figure 5.10

Smooth Param Value: **0.2**

OK

Application Region: select the master surfaces

Add

Apply

24. Create the Contact Surface Properties for the Punch.

Post only the **punch** group using **Group/Post...**

◆ **Properties**

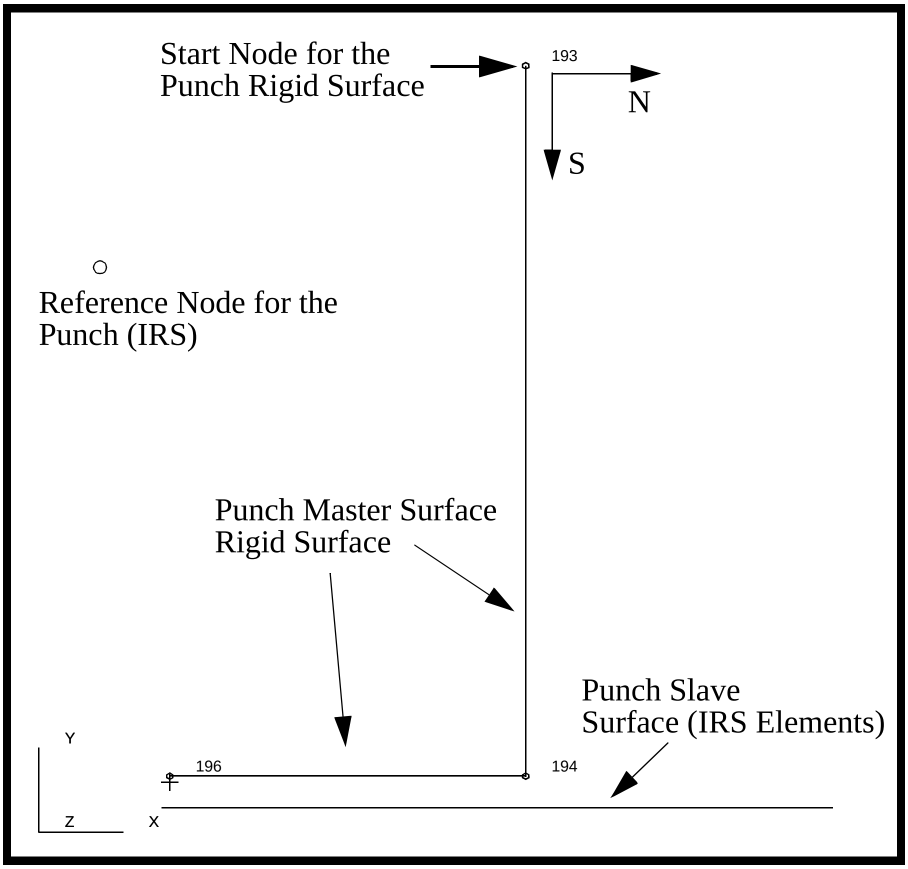
Action: **Create**

Dimension: **1D**

Type:	**IRS (planar/axisym)**
Property Set Name:	**punch_irs**
Options:	**Axisymmetric**
	Elastic Slip Hard Contact
Input Properties...	
ELSET Name:	**punch**
Reference Node:	see Figure 5.11
[Friction in Dir_1]:	**0.125**
OK	
Application Region:	select the slave surface
Add	
Apply	

Action:	**Create**
Dimension:	**1D**
Type:	**RigidSurf(Seg)**
Property Set Name:	**punch_rigid_surface**
Input Properties...	
ELSET Name:	**punch**
Start Point (Node_id):	see Figure 5.11
Smooth Param Value:	**0.013**
OK	
Application Region:	select the master surfaces
Add	
Apply	

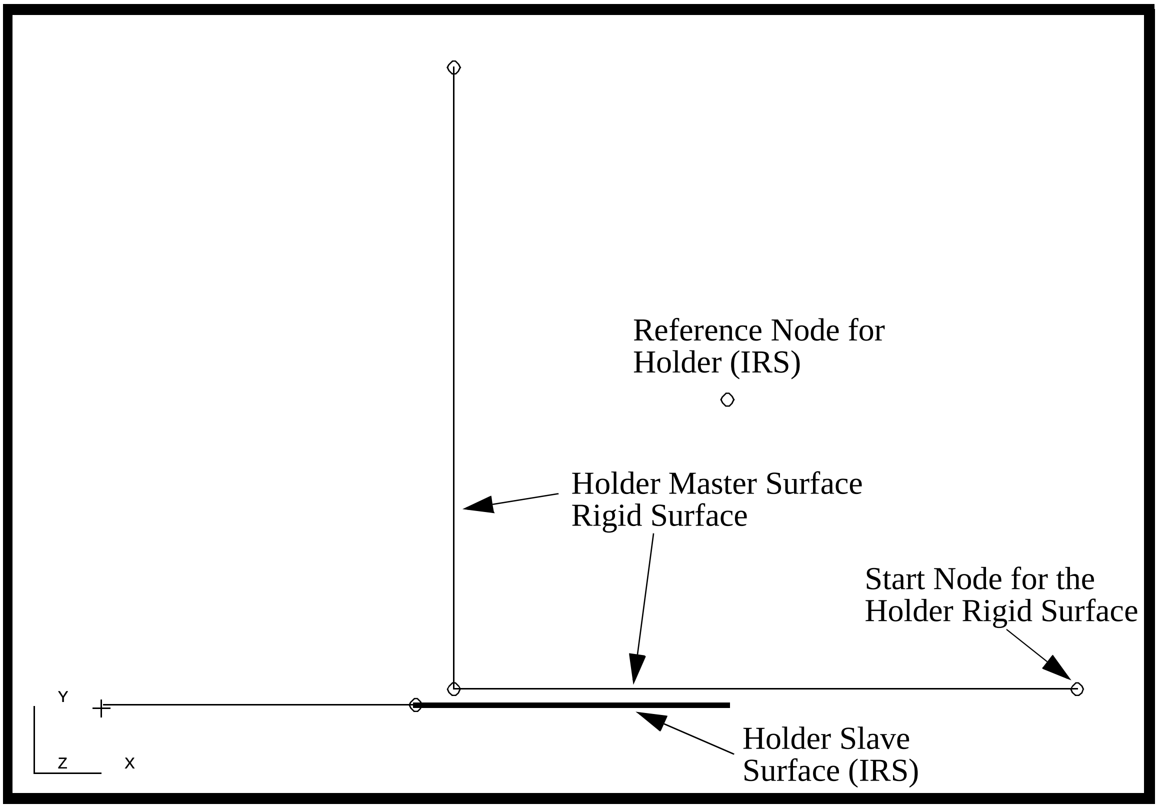
Figure 5.11 - Punch surface definitions



25. Create the Contact Surface Properties for the Holder

Post only the **holder** group using **Group/Post...**.

Figure 5.12 - Holder surface definitions



Action:	**Create**
Dimension:	**1D**
Type:	**IRS (planar/axisym)**
Property Set Name:	**holder_irs**
Options:	**Axisymmetric**
	Elastic Slip Hard Contact
Input Properties...	
ELSET Name:	**holder**
Reference Node:	see Figure 5.12
[Friction in Dir_1]:	**0.125**
OK	
Application Region:	select the slave surface
Add	

Apply

Action:	**Create**
Dimension:	**1D**
Type:	**RigidSurf(Seg)**
Property Set Name:	**holder_rigid_surface**

Input Properties...

ELSET Name:	**holder**
Start Point (Node_id):	see Figure 5.12
Smooth Param Value:	leave blank

OK

Application Region:	select the master surfaces

Add

Apply

26. Create the individual Loads and Boundary Conditions needed.

First, you will create a group that contains all geometry and finite element entities.

Group/Create...

New Group Name:	**all**

■ **Make Current**

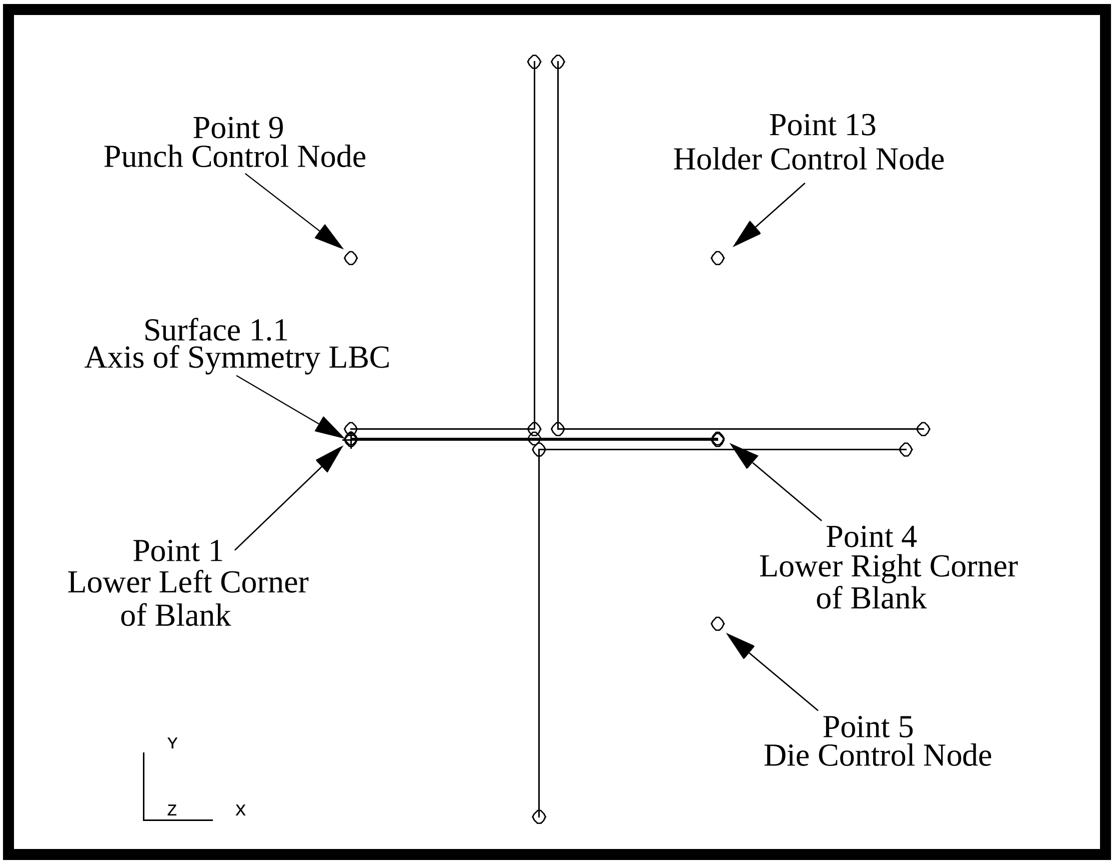
■ **Unpost All Other Groups**

Group Contents:	**Add All Entities**

Apply

You will use Figure 5.13 in the next steps as an aid to create Loads and Boundary Conditions.

Figure 5.13 - Reference nodes and application points for Loads/BCs



◆ **Load/BCs**

Action:	**Create**
Object:	**Displacement**
Method:	**Nodal**
New Set Name:	**axis_of_symmetry**

Input Data...

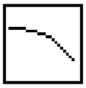
Translations <T1,T2,T3>:	**<0, , >**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

To select the left edge of the blank you will have to change the select menu option to curve by selecting this icon.

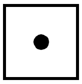
 **Curve**

Select Geometry Entities:	see Figure 5.13
Add	
OK	
Apply	

Create a new LBC named **die_fix**

New Set Name:	**die_fix**
Input Data...	
Translations <T1,T2,T3>:	**<0, 0.1, >**
Rotations <R1,R2,R3>:	**< , , 0 >**
OK	
Select Application Region...	
Geometry Filter:	◆ **Geometry**

Change the select icon for a point or vertex.

 **Point**

Select Geometry Entities:	**Point 5**
Add	
OK	
Apply	

Create an LBC which holds the punch initially fixed.

New Set Name:	**punch_fix**
Input Data...	

Translations <T1,T2,T3>: **<0, -.093, >**

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

OK

Select Application Region...

◆ **Geometry**

Select Geometry Entities: **Point 9**

Add

OK

Apply

Create a Load and Boundary Condition for the initial displacement of the holder.

New Set Name: **holder_disp**

Input Data...

Translations <T1,T2,T3>: **<0, -.1001, >**

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

OK

Select Application Region...

◆ **Geometry**

Select Geometry Entities: **Point 13**

Add

OK

Apply

Create an LBC guide the path of the holder.

New Set Name: **holder_y_guide**

Input Data...

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

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

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Select Geometry Entities: **Point 13**

Add

OK

Apply

Create an LBC to represent the movement of the punch.

New Set Name: **punch_move**

Input Data...

Translations <T1,T2,T3>: **< 0, -2.45, >**

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

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Select Geometry Entities: **Point 9**

Add

OK

Apply

Create an LBC to represent lifting up the holder at the end.

New Set Name: **holder_raise**

Input Data...

Translations <T1,T2,T3>: **< 0, 0.2, >**

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

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Select Geometry Entities: **Point 13**

Add

OK

Apply

Create an LBC which lifts the newly formed cup from the die.

New Set Name: **cup_lift**

Input Data...

Translations <T1,T2,T3>: **< 0, 0.2, >**

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

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Select Geometry Entities: **Point 4**

Add

OK

Apply

Create an LBC which slightly moves the die out of the way.

New Set Name: **die_release**

Input Data...

Translations <T1,T2,T3>: **< 0.4, -0.4, >**

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

OK

Select Application Region...

Select Geometry Entities: **Point 5**

Add

OK

Apply

Create an LBC to fix the cup from movement.

New Set Name:	**cup_fix**
Input Data...	
Translations <T1,T2,T3>:	**< , 0, >**
Rotations <R1,R2,R3>:	**< >**
OK	
Select Application Region...	
Select Geometry Entities:	**Point 1**
Add	
OK	
Apply	

Create the force LBC which keeps the blank held stationary.

Action:	**Create**
Object:	**Force**
Method:	**Nodal**
New Set Name:	**holder_force**
Input Data...	
Force <F1,F2,F3>:	**< , -2250, >**
OK	
Select Application Region...	
Geometry Filter:	◆ **Geometry**
Select Geometry Entities:	**Point 13**
Add	
OK	

Apply

Type **holder_reduce** as the *New Set Name* then press *Input Data....*

New Set Name:	**holder_reduce**
Input Data...	
Force <F1,F2,F3>:	**< , -10, >**
OK	
Select Application Region...	
Geometry Filter:	◆ **Geometry**
Select Geometry Entities:	**Point 13**
Add	
OK	
Apply	

The table below is provided to aid in reviewing/verifying the loads and boundary condition created in this step

Table 1: Summary of Loads and Boundary Conditions

Name	LBC	Application Region	Translation	Rotations
axis_of_symmetry	displacement	Surface 1.1	<0, , >	
die_fix	displacement	Point 5	<0, 0.1, >	< , , 0>
punch_fix	displacement	Point 9	<0, -0.93, >	< , , 0>
holder_disp	displacement	Point 13	<0, -0.1001, >	< , , 0>
holder_y_guide	displacement	Point 13	<0, , >	< , , 0>
punch_move	displacement	Point 9	<0, -2.45, >	< , , 0>
holder_raise	displacement	Point 13	<0, 0.2, >	< , , 0>
cup_lift	displacement	Point 4	<0, 0.2, >	< , , 0>
die_release	displacement	point 5	<.4, -0.4, >	< , , 0>
cup_fix	displacement	Point 1	< , 0, >	
holder_force	force	Point 13	< ,-2250, >	
holder_reduce	force	Point 13	< ,-10, >	

27. Create the Load Cases for the analysis.

In this step you are going to “group” the Loads and Boundary Conditions in an order that simulates the various manufacturing processes required to create the deep drawn cup. In all, there will be 6 Load Case. These load cases were are based on the steps in the physical process of creating the drawn cup.

◆ **Load Cases**

Action: **Create**

Load Case Name: **step_1_close_blankholder**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet:

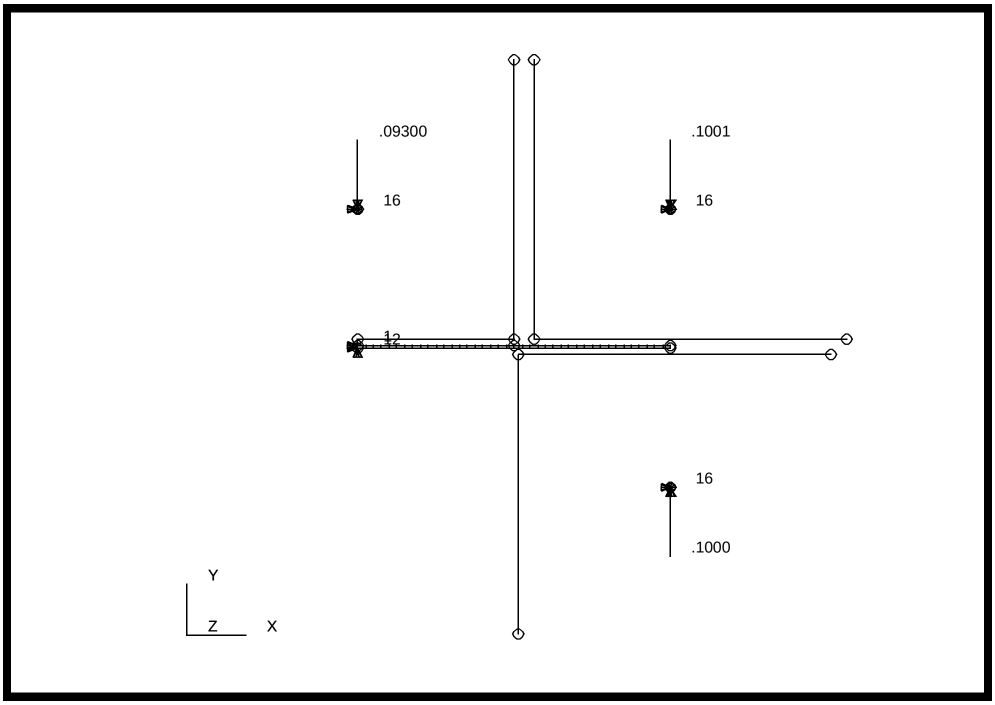
Displ_axis_of_symmetry
Displ_cup_fix
Displ_die_fix
Displ_holder_disp
Displ_punch_fix

OK

Apply

Your viewport should look like Figure 5.14:

Figure 5.14 - Resulting Loads/BCs for step_1



Load Case Name: **step_2_pressurize_blankholder**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet:

Displ_holder_y_guide
Force_holder_force

(select rows containing)

Displ_cup_fix
Displ_holder_disp

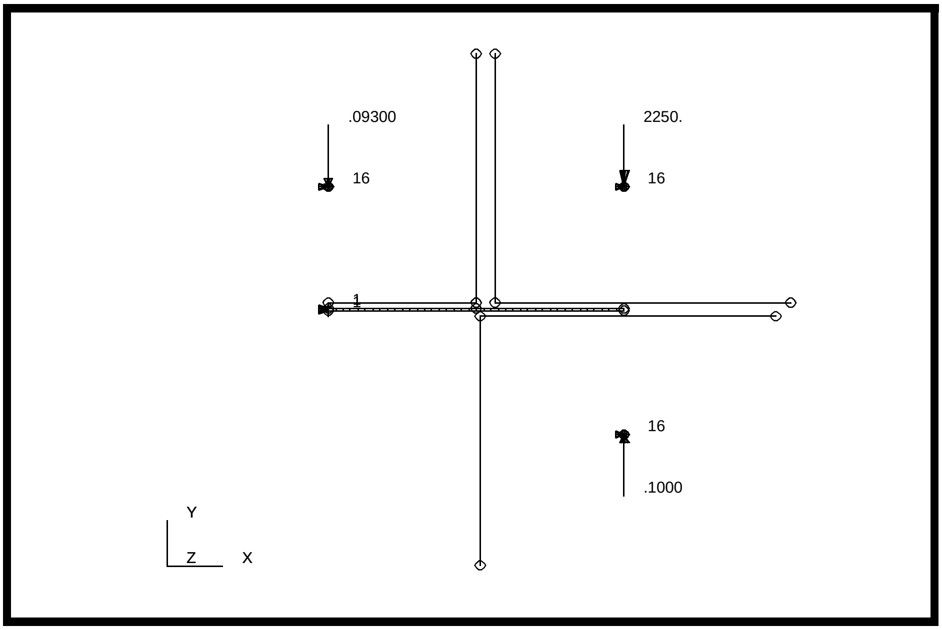
Remove Selected Rows

OK

Apply

Your viewport should look like Figure 5.15:

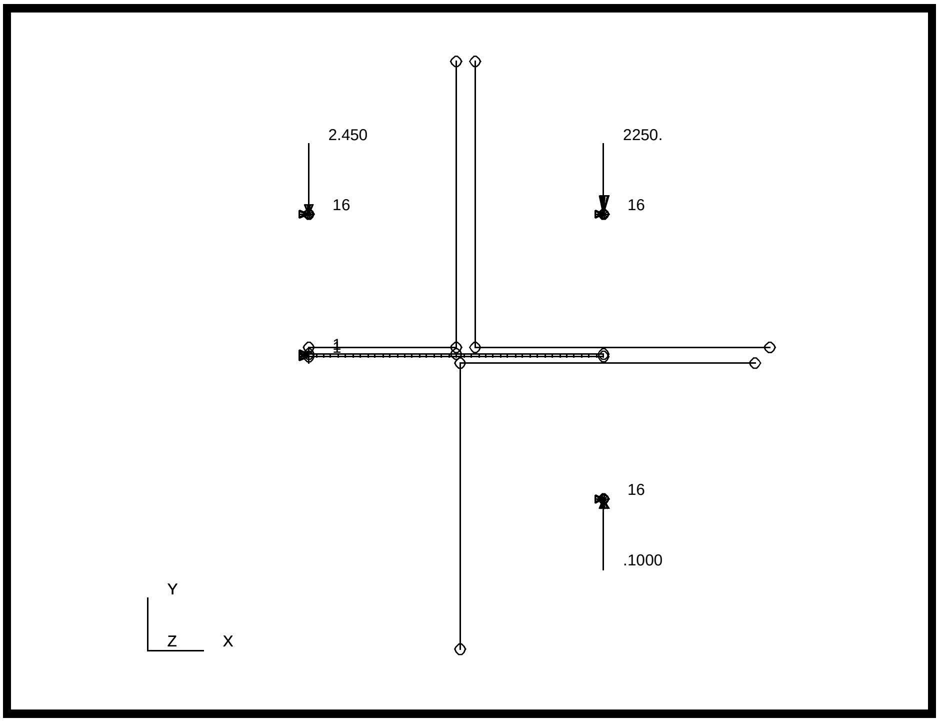
Figure 5.15 - Resulting Loads/BCs for step_2



Load Case Name:	**step_3_move_punch**
Assign/Prioritize Loads/BCs	
Select LBCs to Add to Spreadsheet:	**Displ_punch_move**
(select rows containing)	**Displ_punch_fix**
Remove Selected Rows	
OK	
Apply	

Your viewport should look like Figure 5.16:

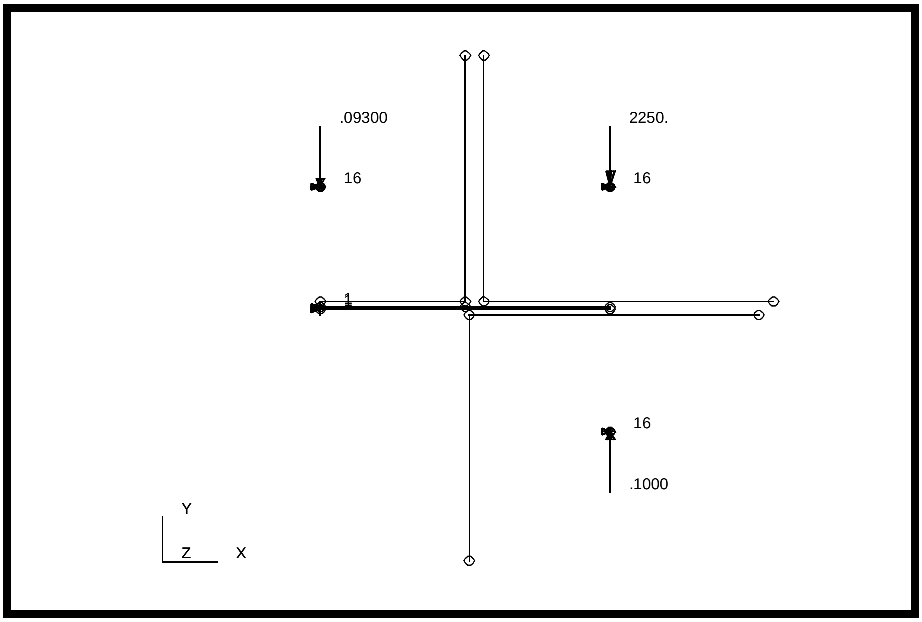
Figure 5.16 - Resulting Loads/BCs for step_3



Load Case Name:	**step_4_release_punch**
Assign/Prioritize Loads/BCs	
Select LBCs to Add to Spreadsheet:	**Displ_punch_fix**
(select rows containing)	**Displ_punch_move**
Remove Selected Rows	
OK	
Apply	

Your viewport should look like Figure 5.17:

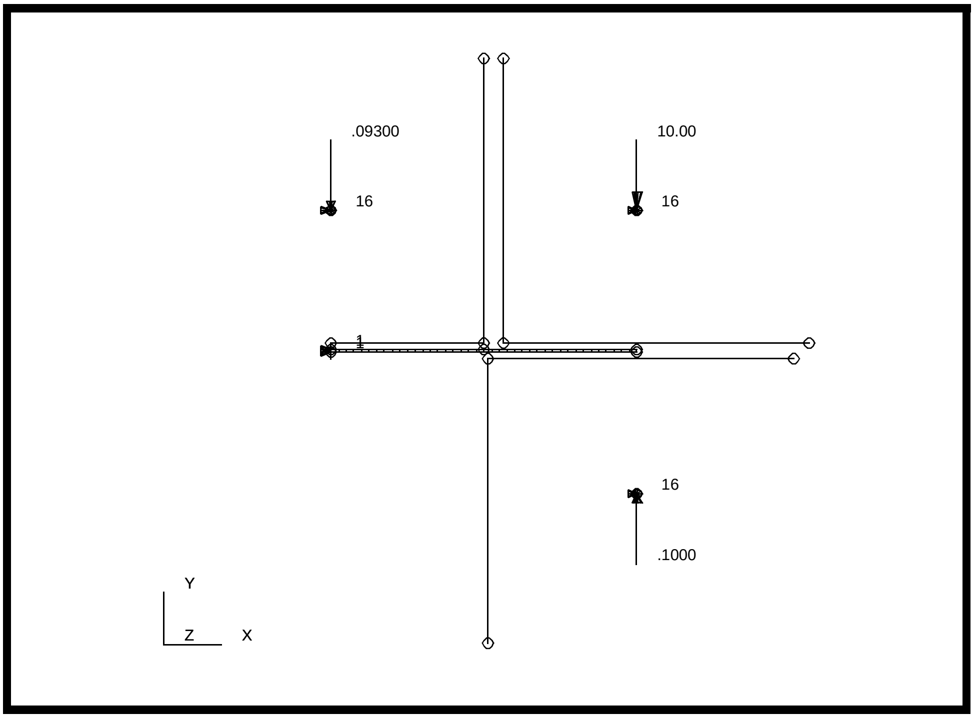
Figure 5.17 - Resulting Loads/BCs for step_4



Load Case Name:	**step_5_release_holder**
Assign/Prioritize Loads/BCs	
Select LBCs to Add to Spreadsheet:	**Force_holder_reduce**
(select rows containing)	**Force_holder_force**
Remove Selected Rows	
OK	
Apply	

Your viewport should look like Figure 5.18:

Figure 5.18 - Resulting Loads/BCs for step_5



Load Case Name: **step_6_release_die**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet:

Displ_cup_lift
Displ_die_release
Displ_holder_raise

(select rows containing)

Displ_die_fix
Displ_holder_y_guide
Force_holder_reduce

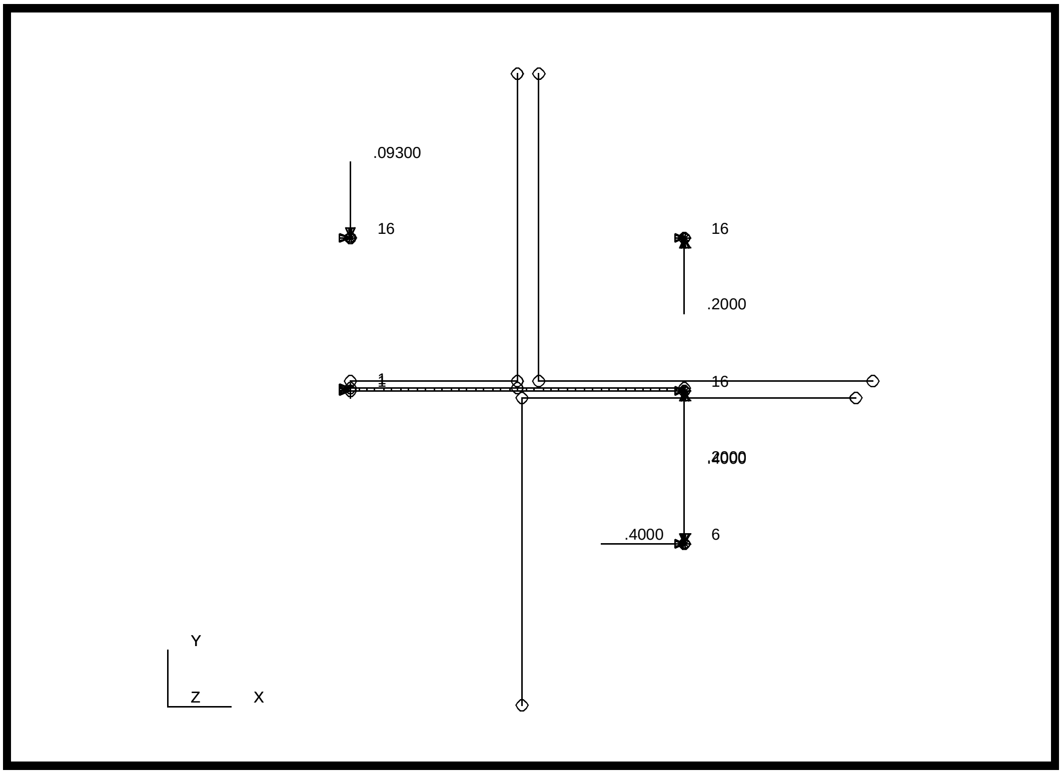
Remove Selected Rows

OK

Apply

Your viewport should look like Figure 5.19:

Figure 5.19 - Resulting Loads/BCs from step_6



28. Create the Analysis Steps and submit the Analysis.

In this section, you are going to create each of the consecutive steps for the analysis. For each step, you will define the step name, solution type, solution parameters, corresponding load case, and output requests.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**cup**
Optional Controls...	
Results File Format:	**ASCII**
Apply	
Step Creation...	
Job Step Name:	**step_1**
Solution Type:	**Nonlinear Static**
Solution Parameters...	
Max No. of Increments Allowed:	**100**
Minimum Delta-T:	**1.0E-5**
OK	
Select Load Cases...	
Available Load Cases:	**step_1_close_blankholder**
OK	
Output Requests...	
Stress Components:	**Integ Point**
Strain Components:	**None**
Plastic Strains:	**Integ Point**
Displacement:	**ON**
OK	
Apply	

Repeat the Step Creation Procedure for remaining 5 Load Cases with the parameters and names given below. Use the same Output Requests for all steps

Table 2: STEP 2

Step Creation	
Job Step Name:	**step_2**
Solution Type:	**Nonlinear Static**
Solution Parameters	
Max. No. Increments Allowed :	**100**
Minimum Delta-T:	**1.0E-5**
Load Case	
Load Case:	**step_2_pressurize_blankholder**
Output Requests	
Stress Components:	**Integration Point**
Plastic Strains:	**Integration Point**
Displacements:	**ON**

Table 3: STEP 3

Step Creation	
Job Step Name:	**step_3**
Solution Type:	**Nonlinear Static**
Solution Parameters	
Max. No. Increments Allowed :	**500**
Minimum Delta-T:	**1.0E-8**
Load Case	
Load Case:	**step_3_move_punch**

Table 3: STEP 3

Output Requests	
Stress Components:	**Integration Point**
Plastic Strains:	**Integration Point**
Displacements:	**ON**

Table 4: STEP 4

Step Creation	
Job Step Name:	**step_4**
Solution Type:	**Nonlinear Static**
Solution Parameters	
Max. No. Increments Allowed:	**200**
Minimum Delta-T:	**1.0E-8**
Load Case	
Load Case:	**step_4_release_punch**
Output Requests	
Stress Components:	**Integration Point**
Plastic Strains:	**Integration Point**
Displacements:	**ON**

Table 5: STEP 5

Step Creation	
Job Step Name:	**step_5**
Solution Type:	**Nonlinear Static**
Solution Parameters	
Max. No. Increments Allowed :	**100**
Minimum Delta-T:	**1.0E-5**
Load Case	
Load Case:	**step_5_release_holder**
Output Requests	

Table 5: STEP 5

Stress Components:	**Integration Point**
Plastic Strains:	**Integration Point**
Displacements:	**ON**

Table 6: STEP 6

Step Creation	
Job Step Name:	**step_6**
Solution Type:	**Nonlinear Static**
Solution Parameters	
Max. No. Increments Allowed :	**120**
Minimum Delta-T:	**1.0E-5**
Load Case	
Load Case:	**step_6_release_die**
Output Requests	
Stress Components:	**Integration Point**
Plastic Strains:	**Integration Point**
Displacements:	**ON**

When you are finished creating each of the steps, press **Cancel** on the *Step Create* form to close it.

Cancel

Step Selection...

Selected Job Steps: **Step_1, Step_2, Step_3, Step_4, Step_5, Step_6**

(be sure to select in order)

Apply

Apply

After the analysis is submitted, close the database using **File/Close**.

File/Close

At this point the analysis is running. You can monitor the analysis from a UNIX windows by monitoring the ***cup.sta*** and ***cup.msg*** files. You can use the UNIX commands **more**, **cat** or **tail**. The usage of tail is given below. The usage may very slightly depending upon platform (you can use **man tail** to get UNIX manual help on tail)

tail -lf cup.msg

tail -lf

29. Reading Results

Once the analysis has finished you will read in the results into a database. However, often times multi-step nonlinear jobs are rather large in size and may require several refinements on the analysis to obtain the correct results and convergence. You can read in the results to the same database if you wish. However, often times its more efficient to read in the results into a NEW database. Therefore, if the results are not exactly as you wish, you have not increased the size of the original model database significantly.

File/New ...

Database Name:	**cup_results.db**
OK	
Analysis Code:	**MSC/ADVANCED_FEA**
Approximate Maximum Model Dimension:	**10**
OK	

Switch the Application to *Analysis*

◆ **Analysis**

Action:	**Read Results**
Object:	**Both**
Method:	**Translate**
Select Results File...	

Selected Results File: **cup.fil**

OK

Apply

Note: when both is selected, only the node and element connectivity is read in, none of the Material data, LBC's or Element Property data is available in the database.

When the model re-appears, the results translation is complete.

Select **Display** from the Main Menu.

Display/Finite Elements...

Show Only Free: ◆ **Edges**

Apply

Cancel

Display/Load/BC/Elem.Props...

Loads/BCs: **Hide All**

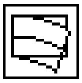
Apply

Cancel

Select **Results** from the *Display* menu.

◆ **Results**

Select the **Deformation Attributes** icon



Scale Interpretation: ◆ **True Scale**

Scale Factor: **1.0**

■ **Show Undeformed Entities**

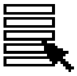
Switch the Application to *Results*

Use the Quick Plot option to look at the results at the end of each step.

◆ **Results**

Action:	**Create**
Object:	**Quick Plot**

Click on the **Select Results** icon



Select Results Cases:	select the last increment of each step.
Select Deformation Result:	**Deformation, Displacement**

Apply

The figures of the deformed plots at the point of full punch penetration are shown below:

Figure 5.20 - Resulting deformation from step_1

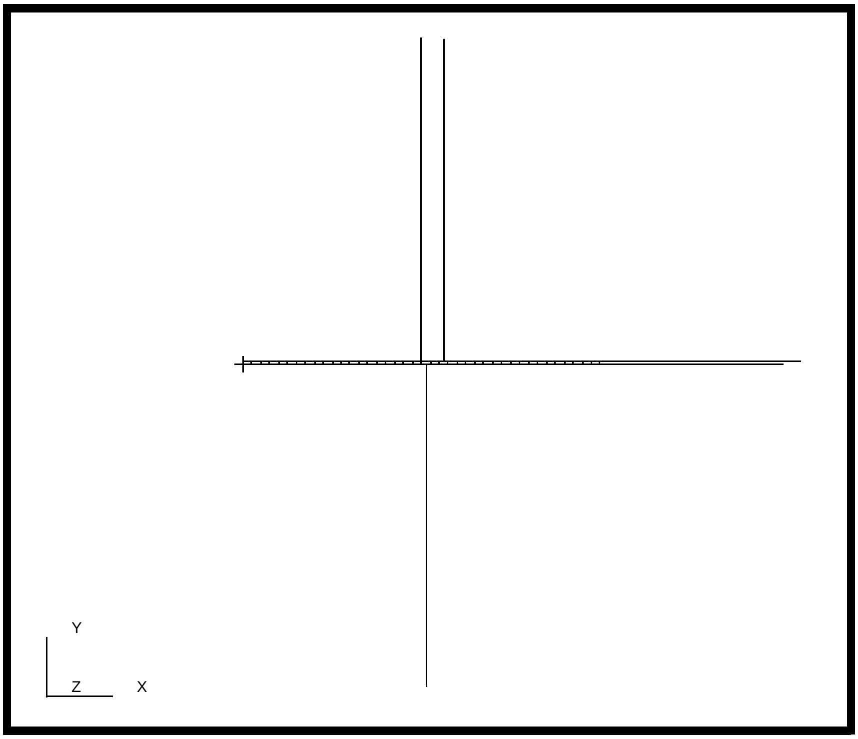


Figure 5.21 - Resulting deformation from step_2

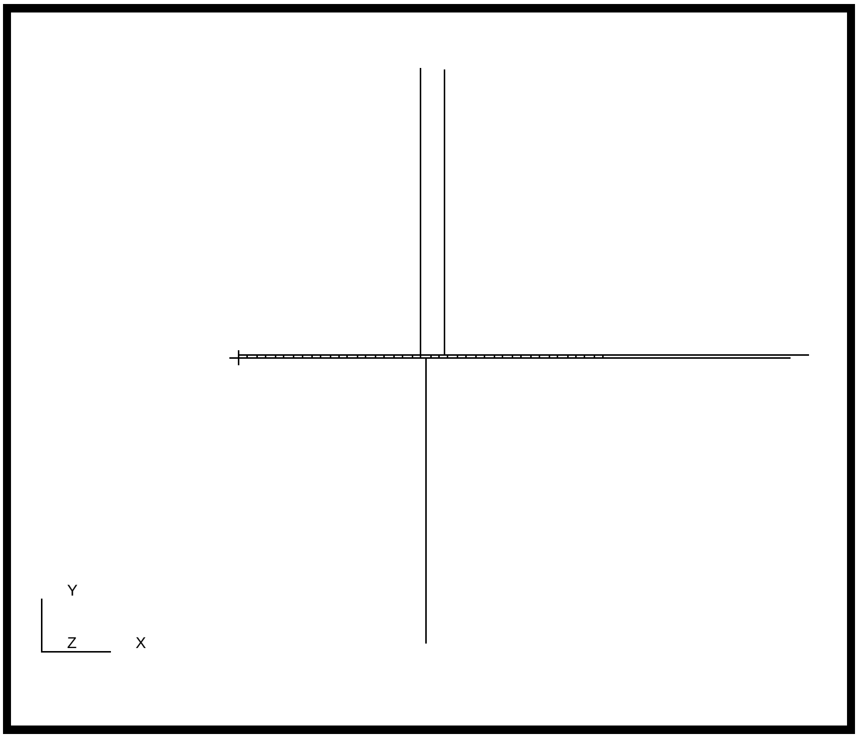


Figure 5.22 - Resulting deformation from step_3

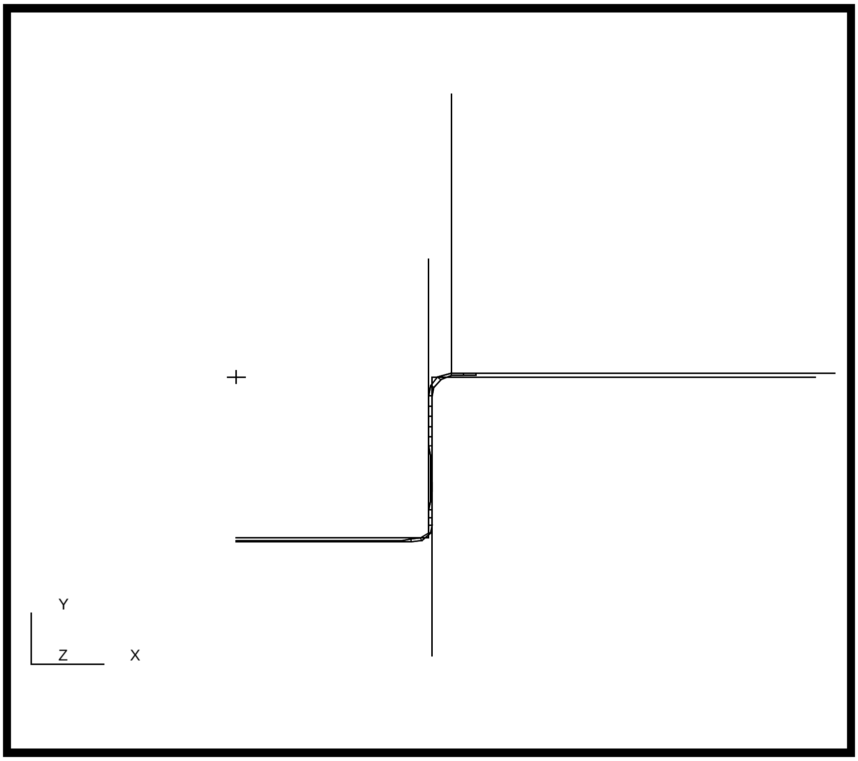


Figure 5.23 - Resulting deformation from step_4

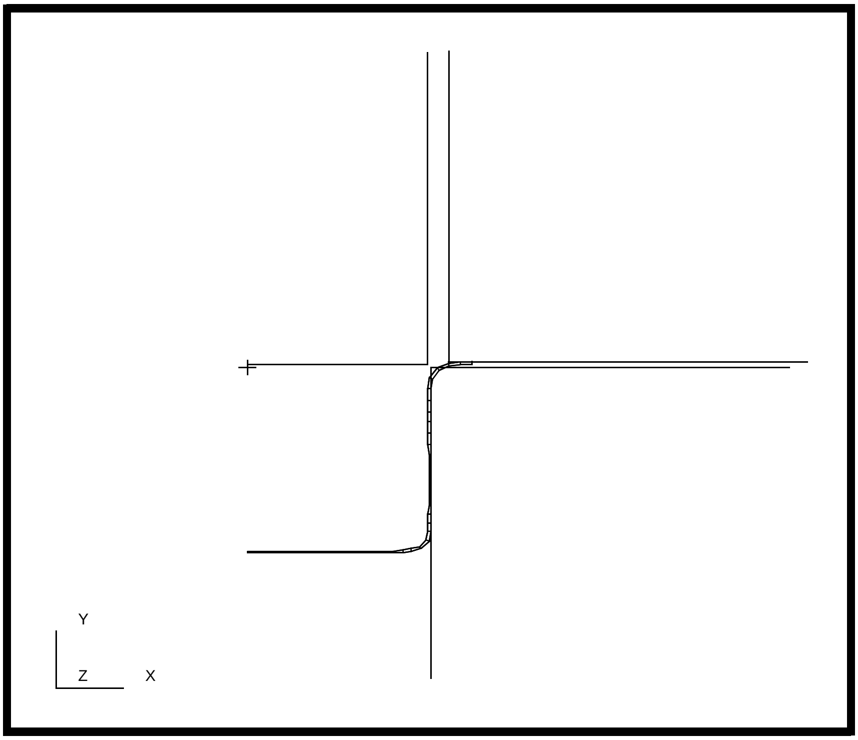


Figure 5.24 - Resulting deformation from step_5

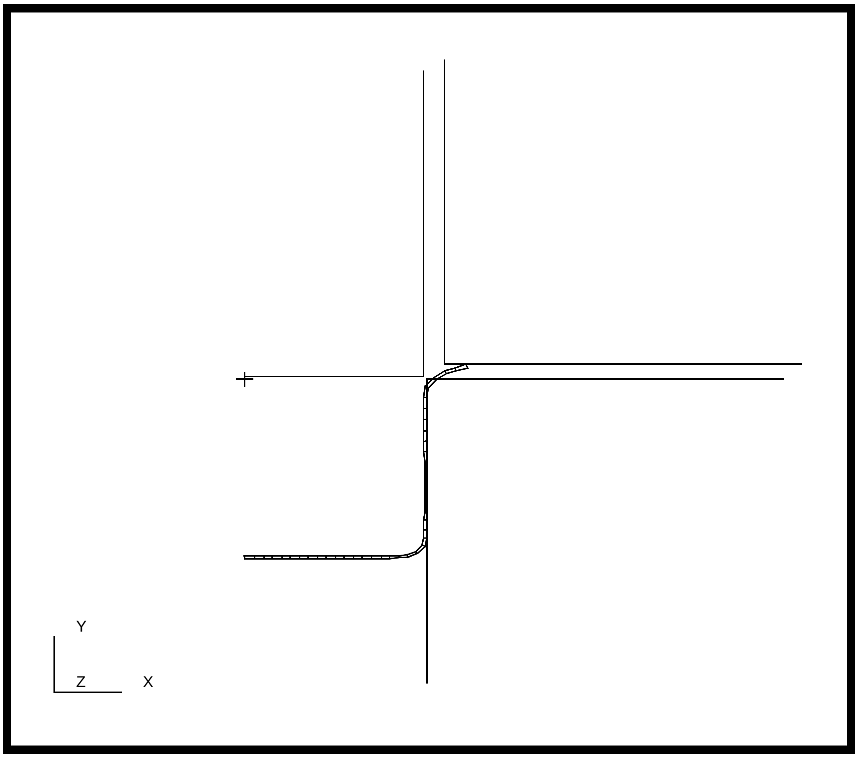
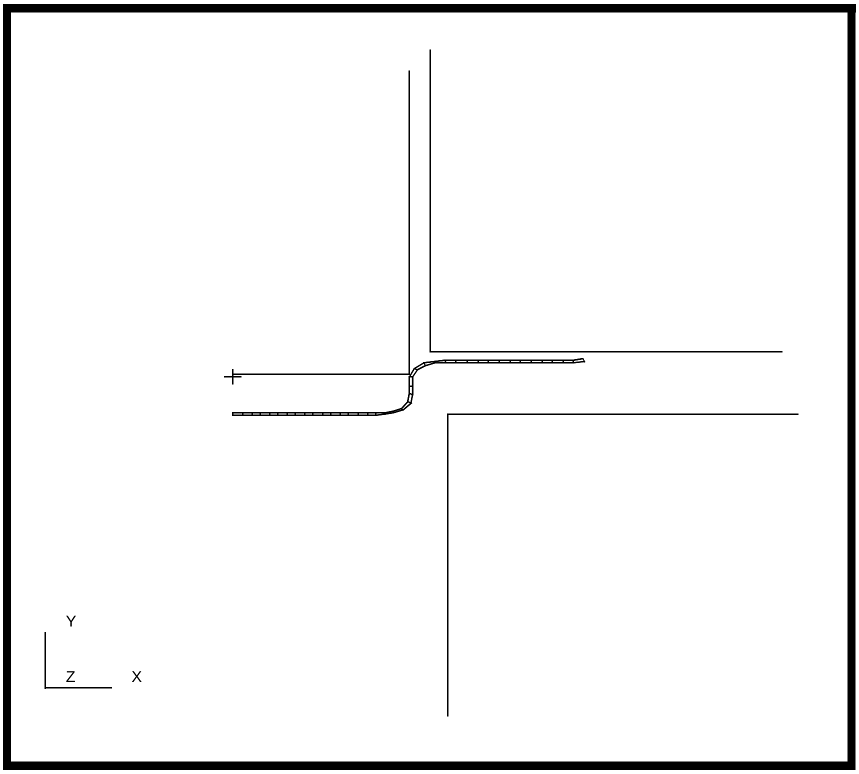


Figure 5.25 - Resulting deformation from step_6



When done viewing, close the database and quit PATRAN.

This concludes the exercise.