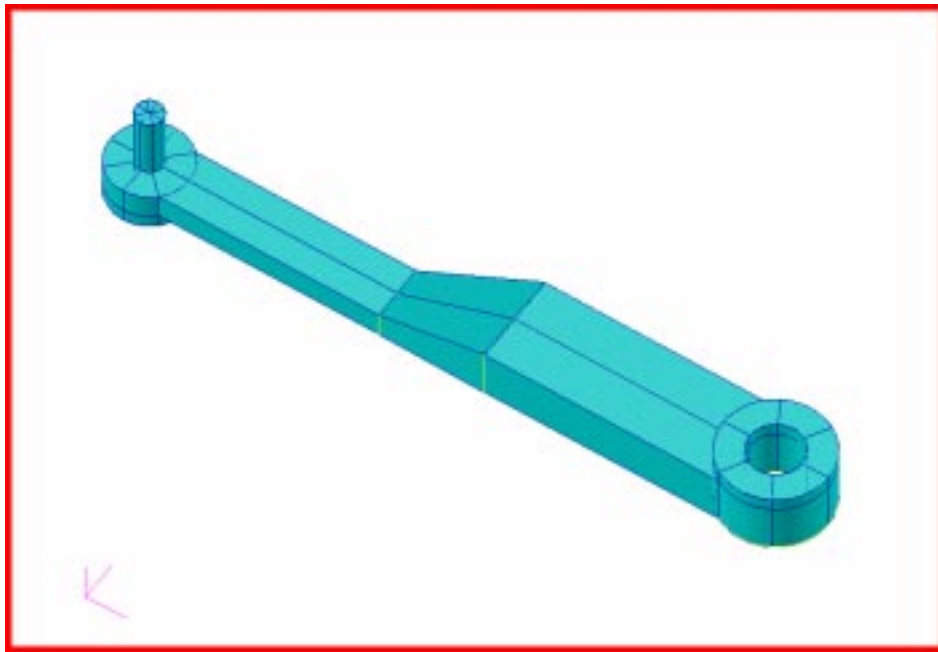


---

## Supplementary Exercise - 2

### *The Pitman Arm*



### **Objectives:**

- Create a pitman arm from scratch.
- Mesh and analyze the pitman arm.



## Model Description:

## Exercise Procedure:

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

### File/New...

*New Database Name*

pitman

OK

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

### New Model Preference

*Tolerance*

◆ Default

*Analysis Code:*

MSC/NASTRAN

OK

3. Open the Geometry form by clicking on the Geometry Radio button in the main form of Patran.

### ◆ Geometry

*Action:*

Create

*Object:*

Curve

*Method:*

Point

*Option:*

2 Point

*Starting Point List*

[0 0 0]

*Ending Point List*

[0.125 0 0]

Apply

4. Turn on all the Geometry Labels.

### Display/Geometry...

Show All Geometry Labels

Create the  
Model using  
the  
Geometry  
Forms

---

**Apply**

**Cancel**

5. Create another curve starting at point 2 with length 0.375.

*Starting Point List*

**Point 2**

*Ending Point List*

**[0.375 0 0]**

**Apply**

6. Create a surface by revolving the two curves that you just created

*Action:*

**Create**

*Object:*

**Surface**

*Method:*

**Revolve**

**■ PATRAN 2 Convention**

*Refer. Coordinate Frame*

**Coord 0**

*Axis*

**Coord 0.3**

*Sweep Parameters*

*Total Angle*

**360.0**

*Offset Angle*

**0.0**

*Surfaces Per Curve*

**8**

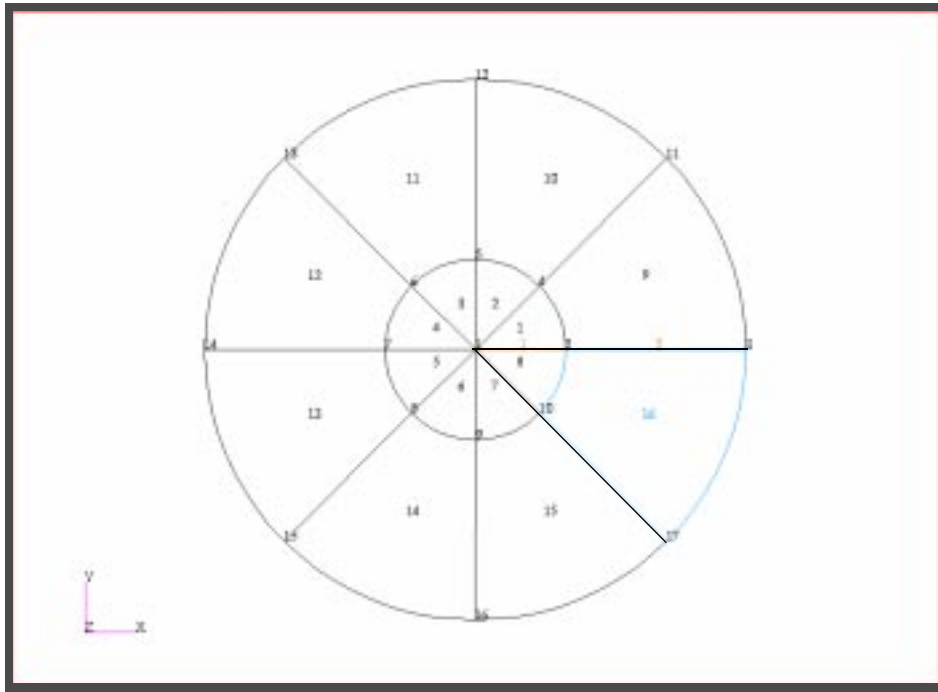
**Auto Execute**

*Curve List*

**Curve 1, 2**

**Apply**

Your model should look like the following:



7. Create a solid using these surfaces.

Action:

Create

Object:

Solid

Method:

Extrude

Translation Vector:

< 0 0 0.125 >

Surface List:

Select all the Surfaces on the screen

Apply

8. Make sure that the picking is set to Enclose Entire Entity and pick only the visible entities.

Preferences/Picking...

◆ Enclose Entire Entity

Now click on the icon Select Visible Entities in the icon form.



9. You are going to extrude the faces of the solids

*Translation Vector:*

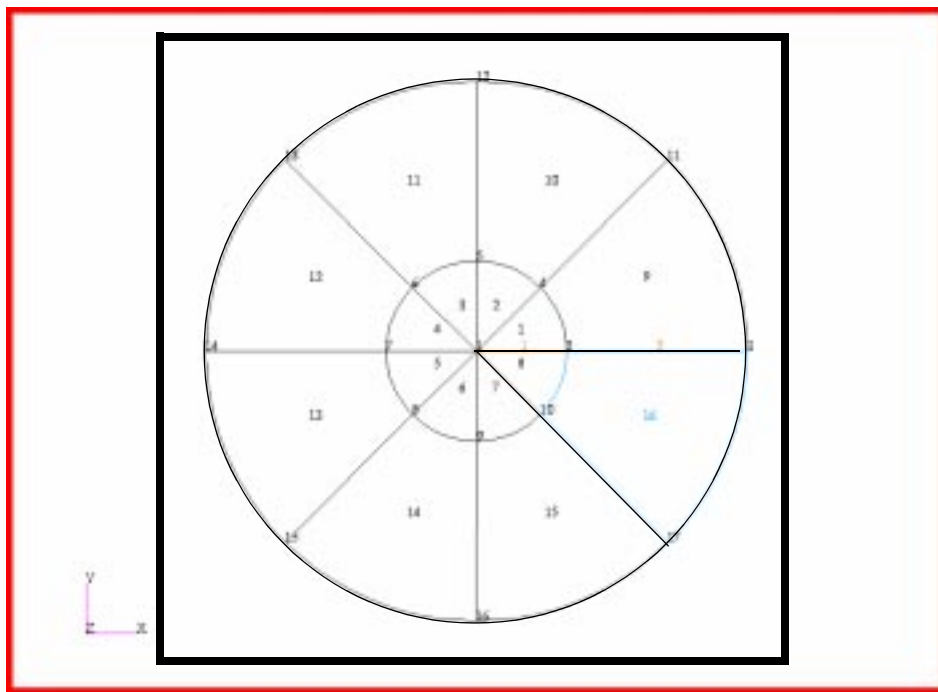
**< 0 0 0.25 >**

*Surface List:*

Select the Faces shown below

**Apply**

Select the following faces.



Rotate the model so that you can be sure that the front of the model was selected.

10. You are now going to extrude the center of the circular model.

*Action:*

**Create**

*Object:*

**Solid**

*Method:*

**Extrude**

*Translation Vector:*

**< 0 0 0.625 >**

**Auto Execute**

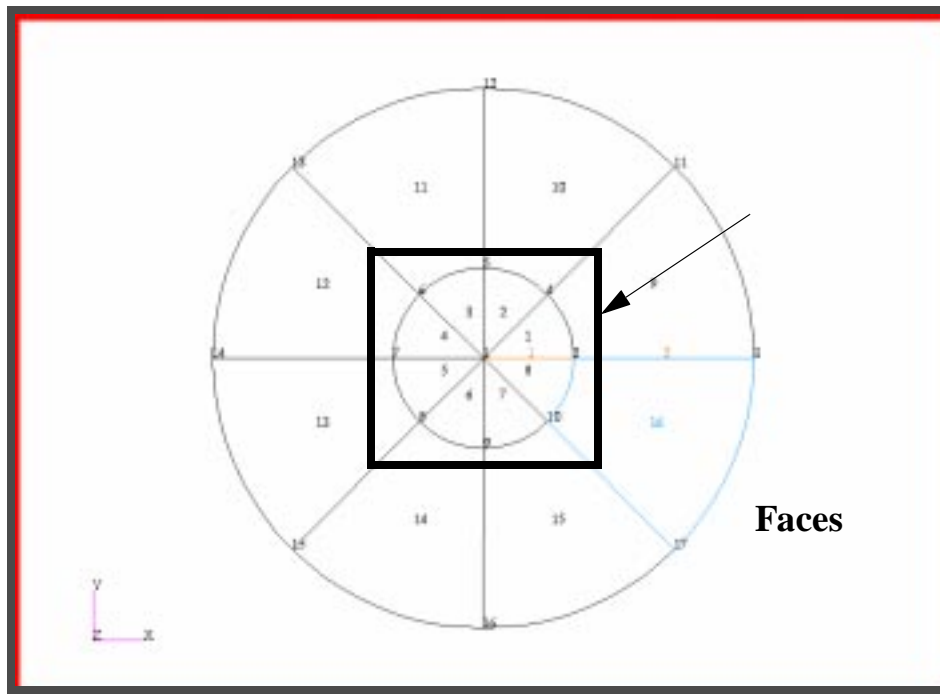
Set the view to the front view by clicking on the front view icon..



*Surface List*

See figure below

Select the solid faces shown below.

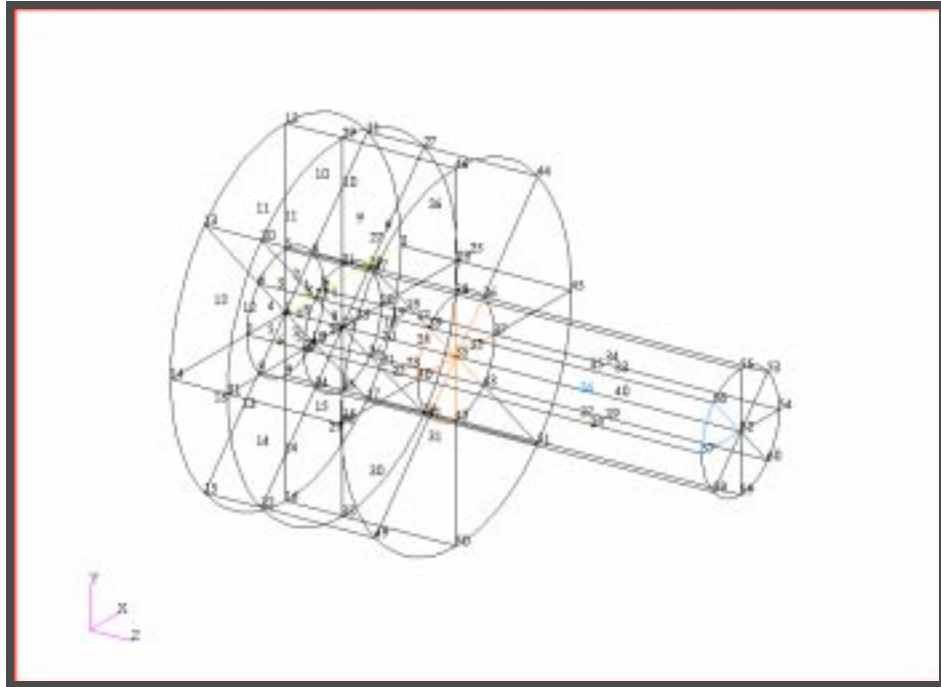


Change the view to Iso 2 View in order to make sure only the middle faces are highlighted.



**Apply**

Your model should look like the following.



Change back to the left side view.



11. Create a vector with the y and z coordinates of point 48 as the base point

Action:	<input type="text" value="Create"/>
Object:	<input type="text" value="Vector"/>
Method:	<input type="text" value="Magnitude"/>
Vector Direction List:	<input type="text" value="&lt; 1 0 0 &gt;"/>
Base Point List	<input type="text" value="[2.375 y48 z48]"/>
<input type="text" value="Apply"/>	

12. Next, create a plane from the vector that you just created

Action:	<input type="text" value="Create"/>
Object:	<input type="text" value="Plane"/>
Method:	<input type="text" value="Vector Normal"/>

Auto Execute

Vector List:

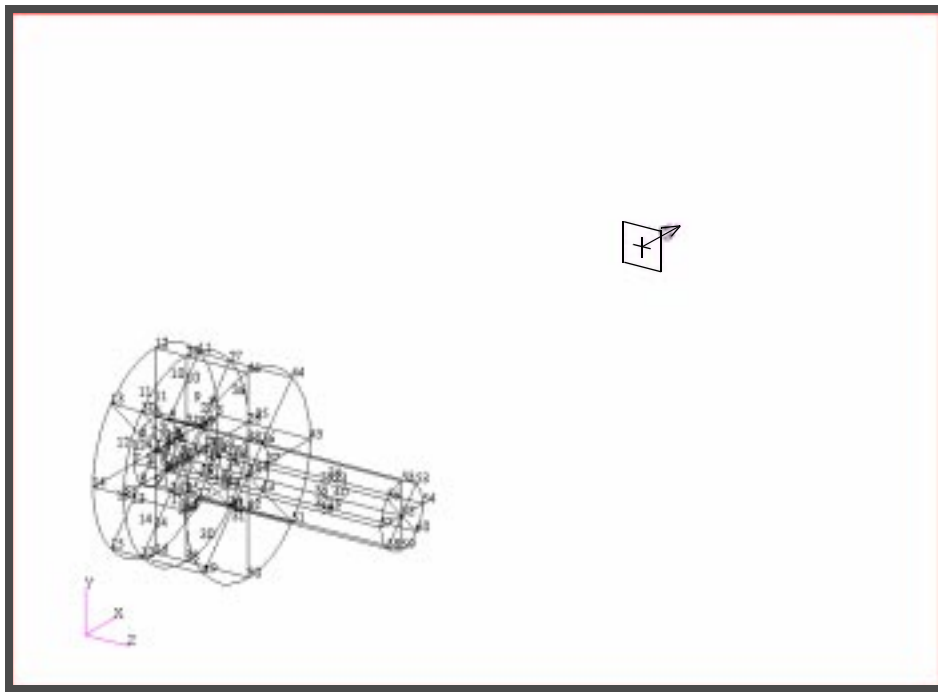
Vector 1

Apply

Change to the Iso 2 View.



Your model should look like the following.



Change to a left side view by using the following icon.



13. Create three curves by projecting three edges from the solid out to the plane that you just created.

Action:

Create

Object:

Curve

Method:

Project

Project Onto:

Plane

Select the edges using the icon below



Auto Execute

Curve List

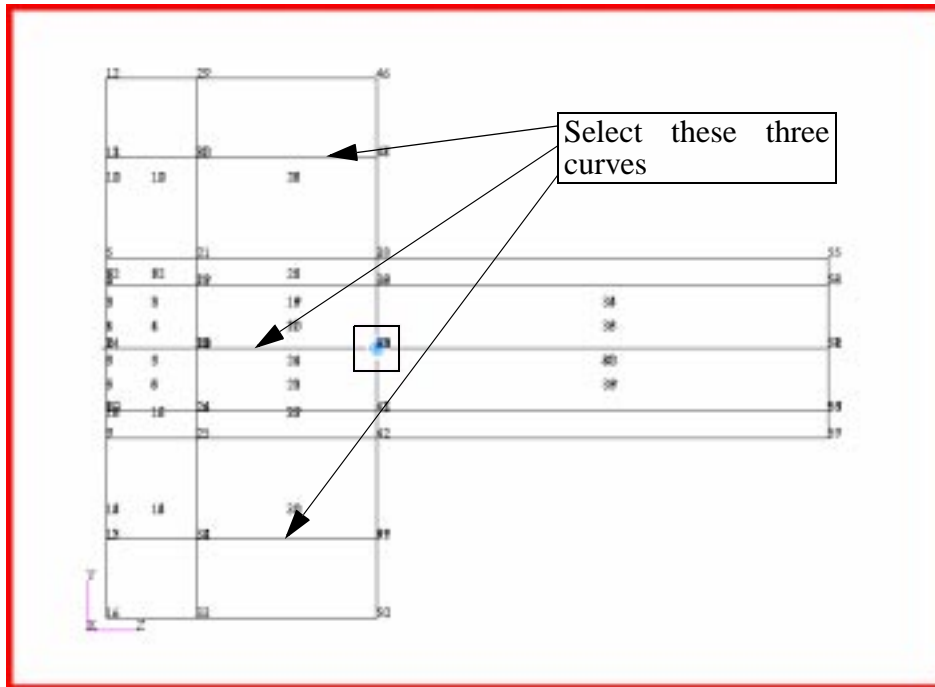
Select the edges shown below

Plane List

Plane 1

Apply

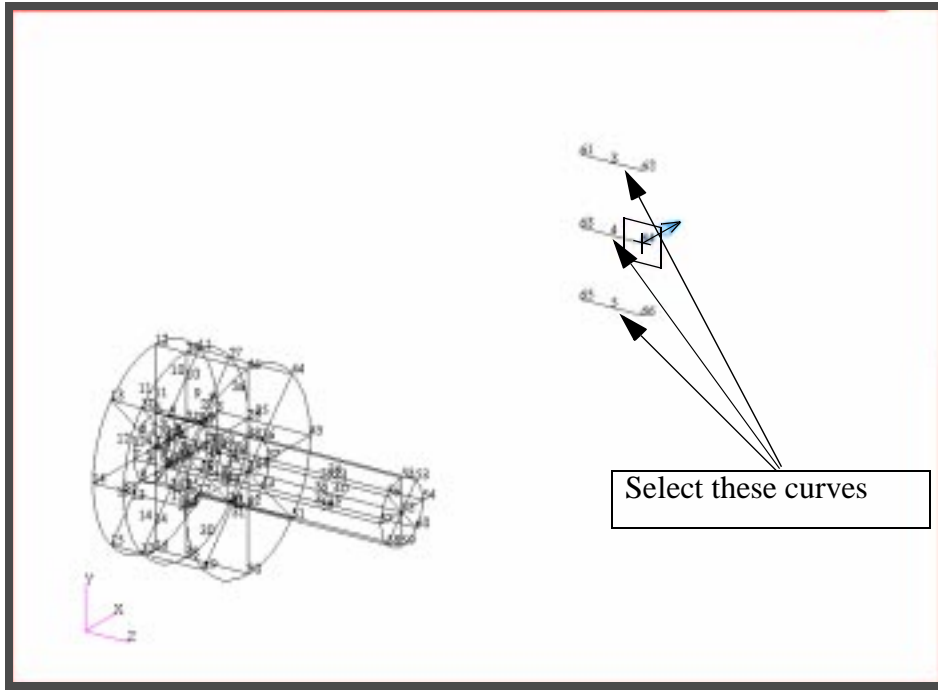
The curves you should select are shown below.



Change back to the Iso 2 View when you are done to make sure that the curves projected out onto the plane.



The model should look like the one shown below.



14. Create the two surfaces from the three curves that you just created.

*Action:*

**Create**

*Object:*

**Surface**

*Method:*

**Curve**

*Option:*

**2 Curve**

**Auto Execute**

Click on the curve icon in the select menu.



*Starting Curve List*

See note below

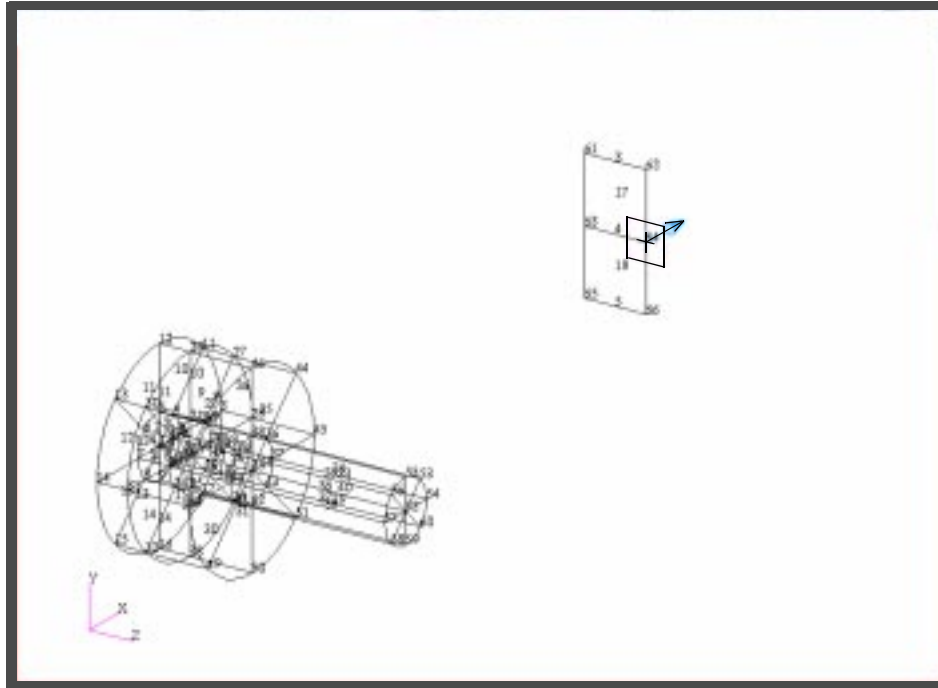
*Ending Curve List*

See note below

**Apply**

**Note:** You are going to select the top curve first as the starting curve and the middle curve as the Ending Curve. Use the same procedure for the bottom curve and the middle curve. See the figure above for the three curves.

Your model should look like the following.



15. Change views to the Iso 1 View



You may want to turn off all the geometry labels in order to see the surfaces of the model easier.

**Display/Geometry...**

**Hide All Geometry Labels**

**Apply**

**Cancel**

16. Create two solids using the two surfaces that you just created, and two faces from the circular solids..

*Action:*

**Create**

*Object:*

**Solid**

Method:

Surface

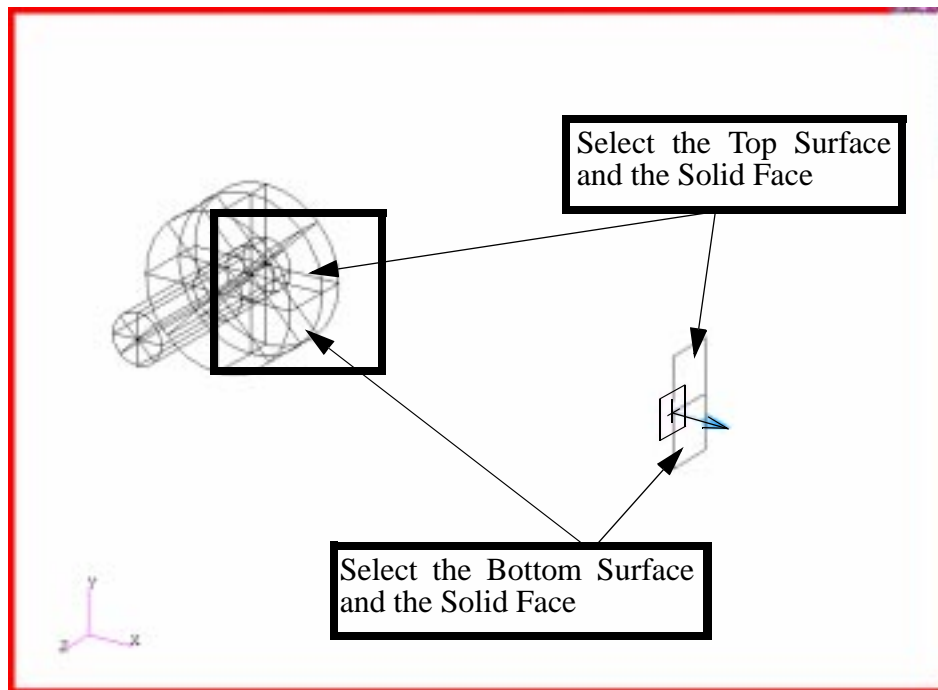
**Auto Execute**

The surface faces you should select are shown below. Select the top surface that you just created as the Starting Surface List.

Next, click in the Ending Surface List box and then click on the Solid Face Icon.

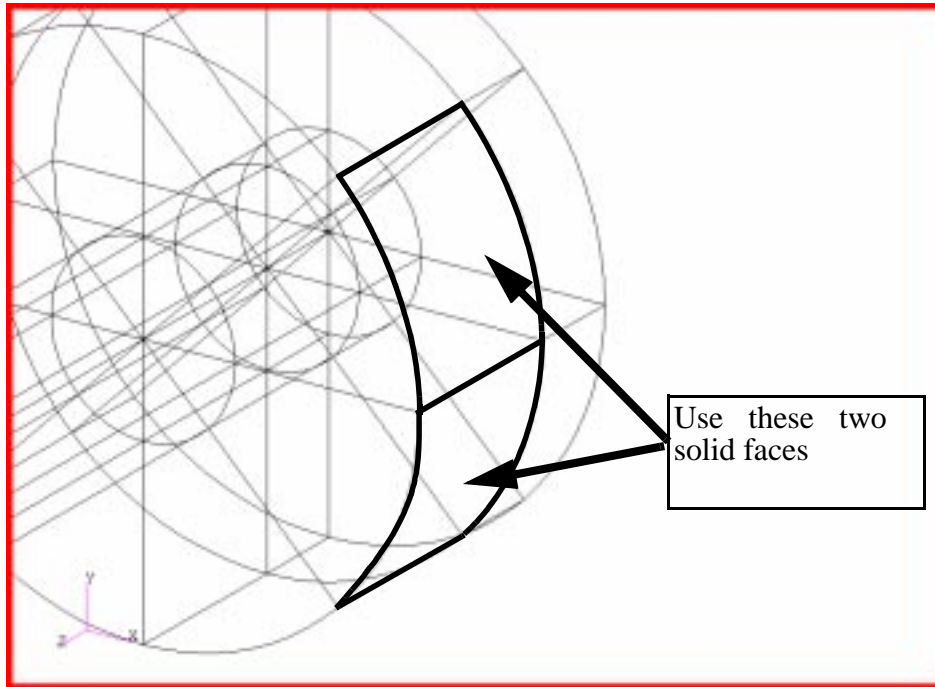


You may want to zoom in on the solid model using the corners tool in order to get a better view of the Solid faces that you are going to be picking. Use the Select Corners Icon shown below to zoom



---

A zoomed in view of the solid.



After selecting the top face shown above click on **Apply** in the Geometry form.

*Starting Surface List*

**Surface 17**

*Ending Surface List*

See figure

**Apply**

The top solid should appear. Now, create the bottom solid. Click in the Starting Surface List and then select the bottom surface, Surface 18.

Click on the fit view icon to fit the entire model to the viewport.



Click in the Ending Surface List box and then click on the solid face icon.



Click on the bottom face that corresponds to the bottom surface.

*Starting Surface List*

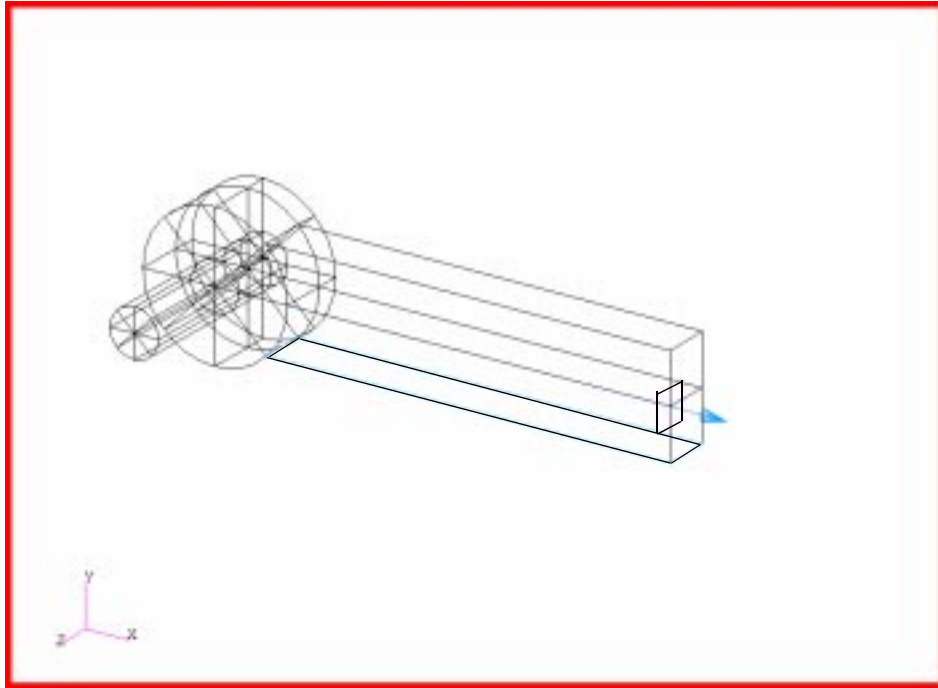
**Surface 18**

*Ending Surface List*

See figure

**Apply**

The model should look like the following when it is completed.



Change to the Front View for the next steps.



17. Next, build the other side of the pitman arm. Create an inner and outer circle by doing the following.

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Curve</b>
<i>Method:</i>	<b>2D ArcAngles</b>
<i>Arc Parameters</i>	
<i>Radius</i>	<b>0.25</b>
<i>Start Angle</i>	<b>120.0</b>
<i>End Angle</i>	<b>-120.0</b>

---

**Project to Plane**

*Construction Plane List*

**Coord 0.3**

*Center Point List*

**[6 0 0.375]**

**Apply**

That will create part of the inner circle. To create the other part simply change the start and end angle.

*Start Angle*

**-120.0**

*End Angle*

**120.0**

**Apply**

18. To create the outer circle change the radius to 0.5.

*Radius*

**0.5**

*Start Angle*

**120.0**

*End Angle*

**-120.0**

*Construction Plane List*

**Coord 0.3**

*Center Point List*

**[6 0 0.375]**

**Apply**

19. Once again change the start and end angle to complete the outer circle.

*Start Angle*

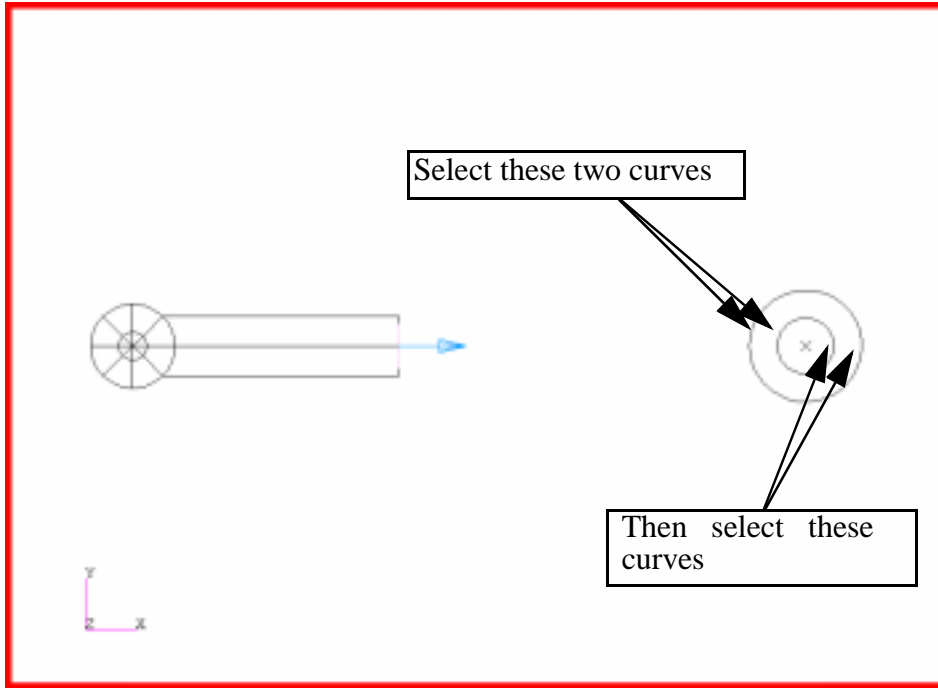
**-120.0**

*End Angle*

**120.0**

**Apply**

Your model should look like the following



20. Create two surfaces between these inner and outer circles. Use the figure above to select the curves which will be used for the surfaces.

*Action:*

**Create**

*Object:*

**Surface**

*Method:*

**Curve**

*Starting Surface List*

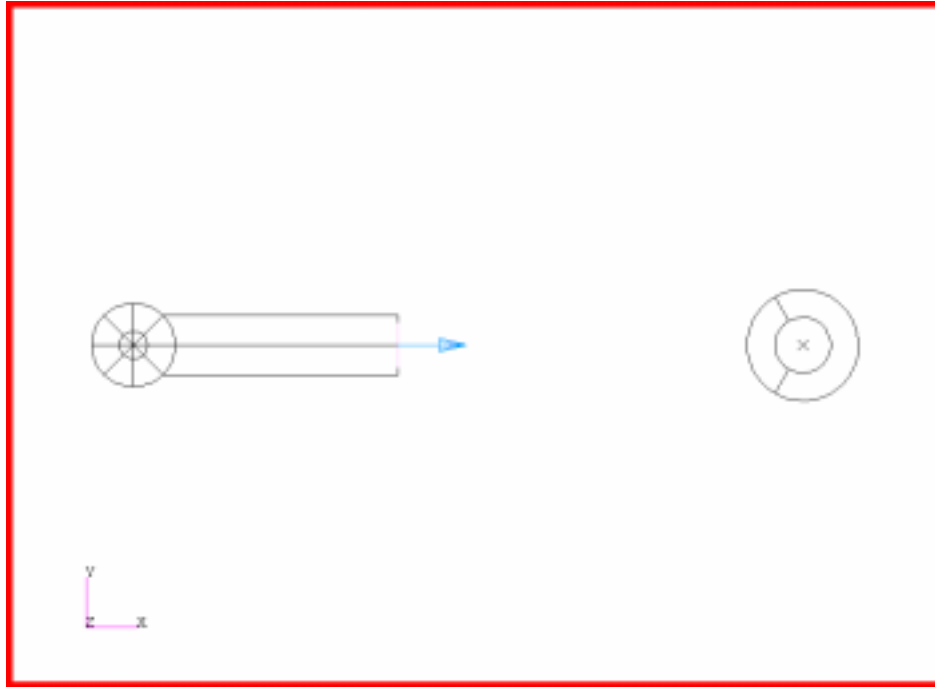
See figure above

*Ending Surface List*

See figure above

**Apply**

You figure should look like the following



21. Break these two surfaces into six surfaces

Action:

**Edit**

Object:

**Surface**

Method:

**Break**

Option:

**Parametric**

**Constant v Direction**

Option:

**u = 0.5**

**Delete Original Surfaces**

**Auto Execute**

Click on the surface icon in the select menu.



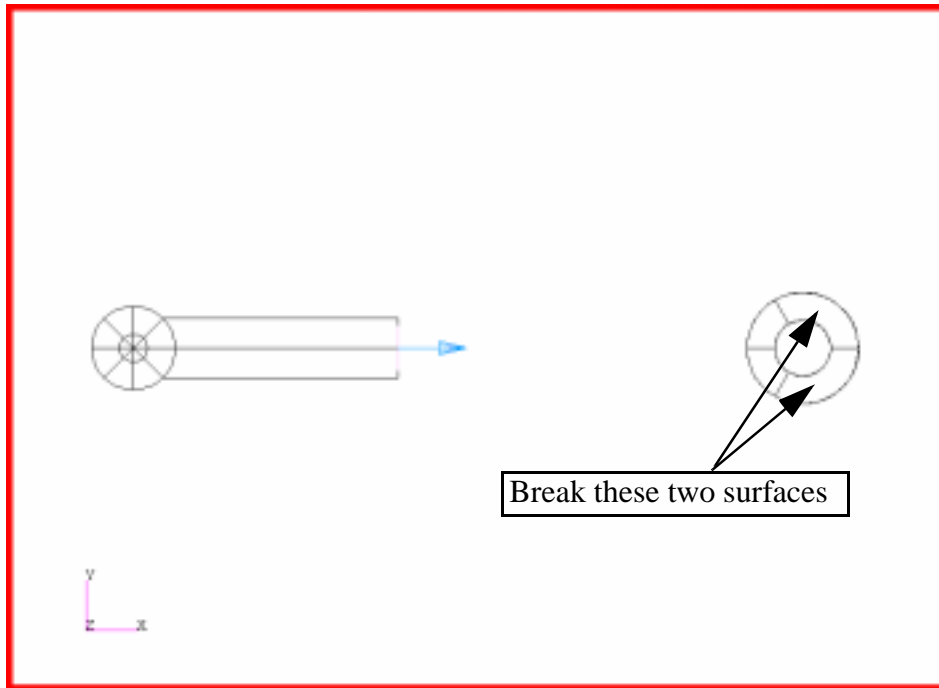
Surface List

Select all the surfaces

**Apply**

When asked if you want to delete the original surfaces click **yes**.

The model should now look like this.



22. Now break two of the surfaces that were just created in half. Break the two surfaces shown above.

*Surface List*

See Figure Above

**Apply**

Click on **YES** when you are asked if you want to delete the original surfaces.

23. Extrude these surfaces in order to make solids. The procedure is the same as the first side of the pitman arm.

*Action:*

**Create**

*Object:*

**Solid**

*Method:*

**Extrude**

*Translation Vector*

< **0 0 0.5** >

*Surface List*

Select all surfaces just created on the right side

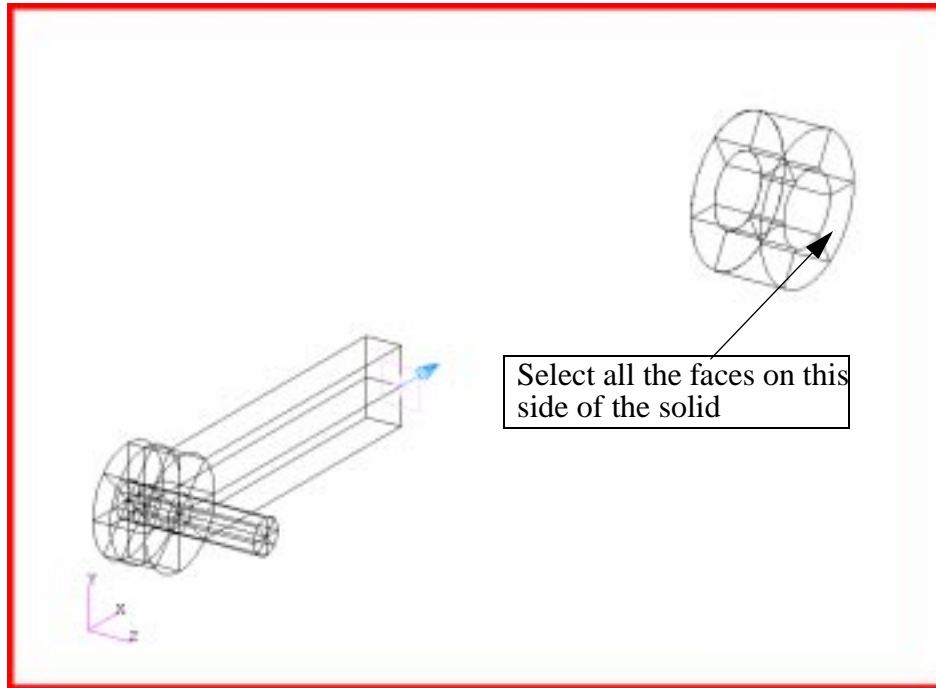
**Apply**

---

Change to the Iso 2 View to make sure that the surfaces extruded correctly.



The Model should look like the following.



24. Select all the faces of the solid that are pointing in the positive Z-direction. Shift click the faces of the solid.

*Translation Vector*

**< 0 0 0.125 >**

Use the Solid Face Icon shown below to pick the faces.

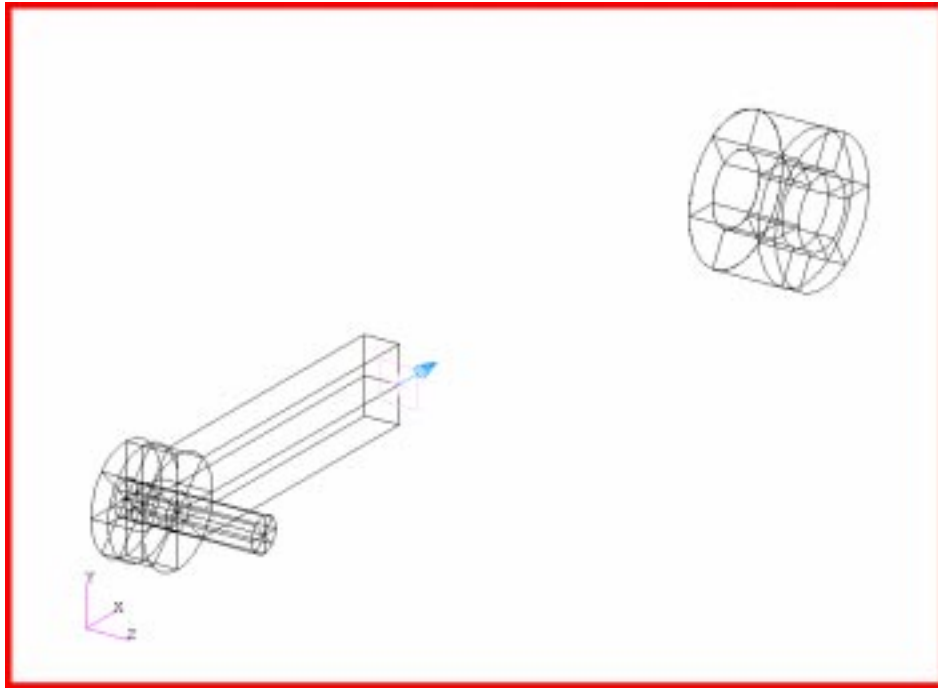


*Surface List*

See figure above

**Apply**

The model should look like the one shown below.



25. Turn the Point labels back on by doing the following.

**Display/Geometry...**

*Colors and Labels*

*Point:*

**Label**

**Apply**

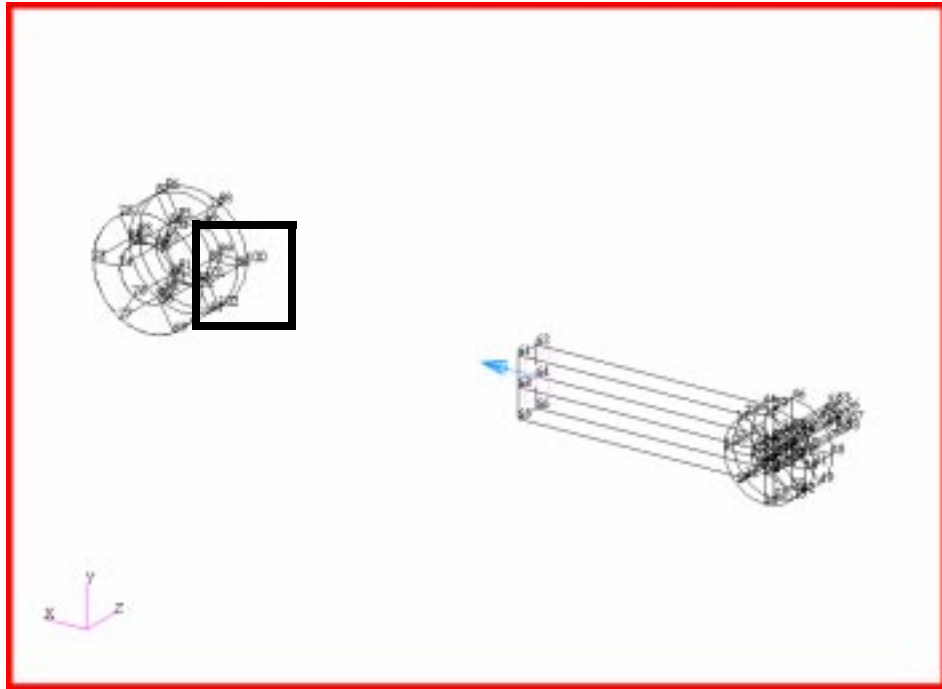
**Cancel**

Change to the Iso 4 View.

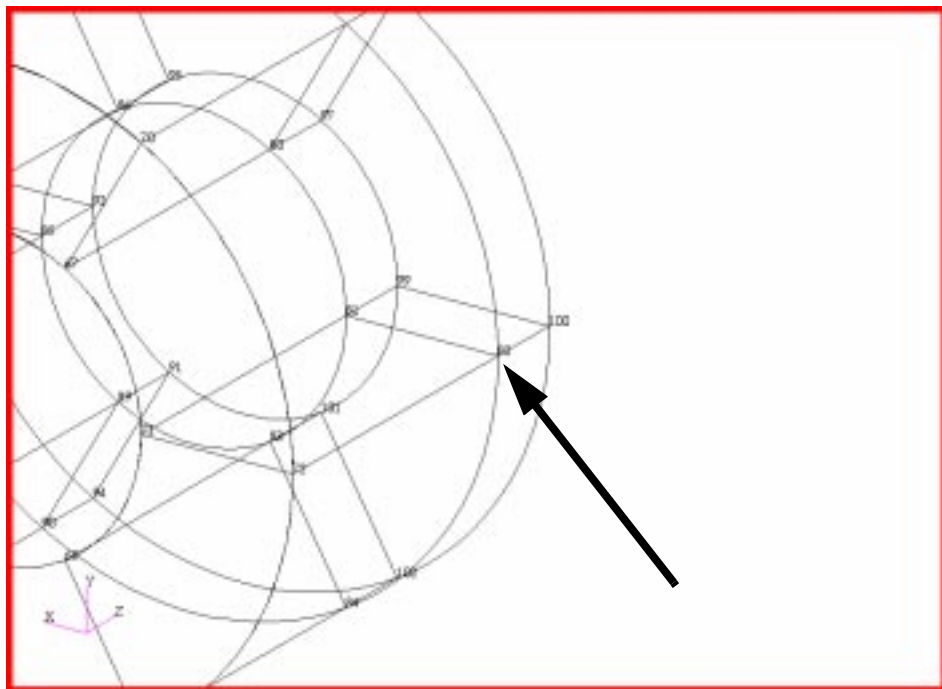


---

Use the View Corners icon to zoom in on the location shown below.



Use the Y and Z location of the point shown below to create a vector.



26. Create a vector like that of the other side of the arm. The point that will be used is 82. The point on your model might be different.

Action:

Create

Object:

Vector

Method:

Magnitude

Vector Direction List:

< 1 0 0 >

Base Point List

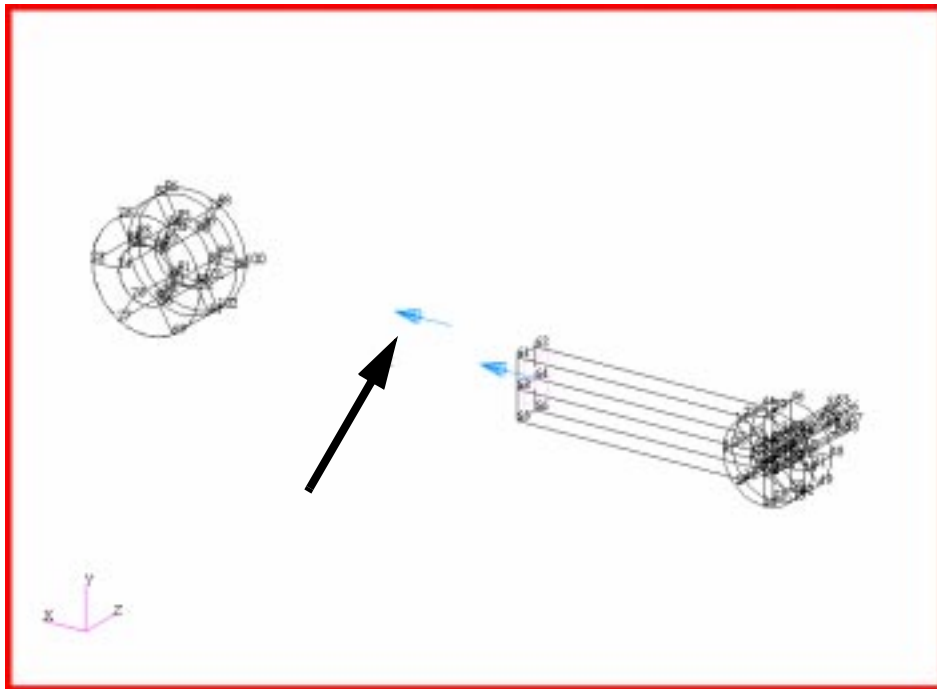
[3.5 y82 z82]

Apply

Click on the Fit View Icon to verify that the Vector was created.



The vector should now be in between the two solids. The model should look like the one shown below.



27. Create a plane from the vector that you just created.

Action:

Create

*Object:*

**Plane**

*Method:*

**Vector Normal**

*Vector List:*

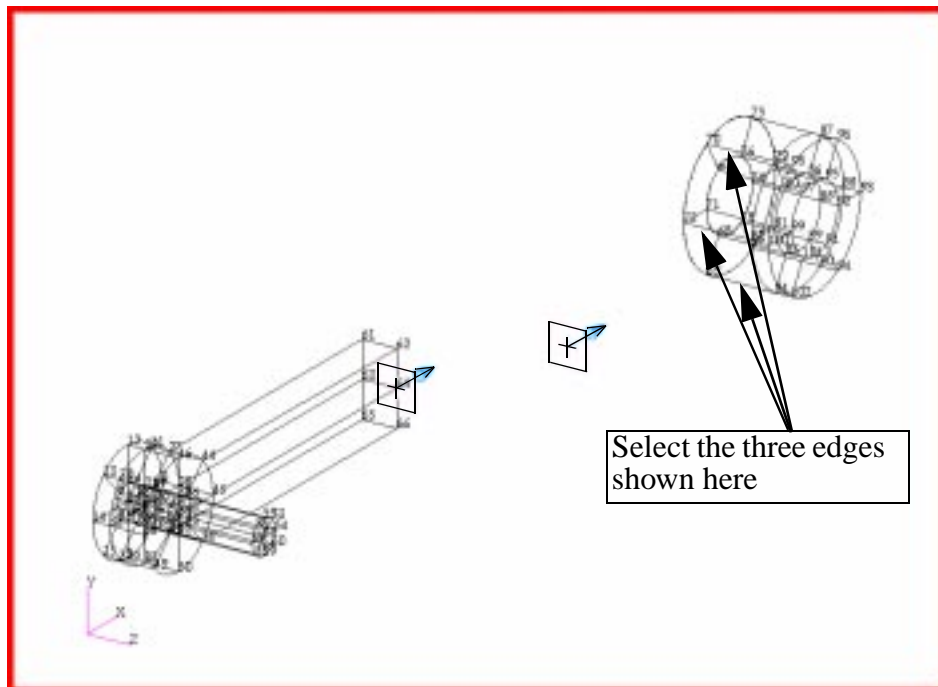
**Vector 2**

**Apply**

Change views to the Iso 2 View.



Use the figure below to select the three edges of the solids that are going to be projected onto the plane.



28. Create three curves by projecting edges of the solid to the plane. This was also done on the first part of the pitman arm.

*Action:*

**Create**

*Object:*

**Curve**

*Method:*

**Project**

*Project Onto:*

**Plane**

Use the Edge icon to select the edges of the solid.



You can use the Shift Click Option to select all the surface edges.

*Curve List*

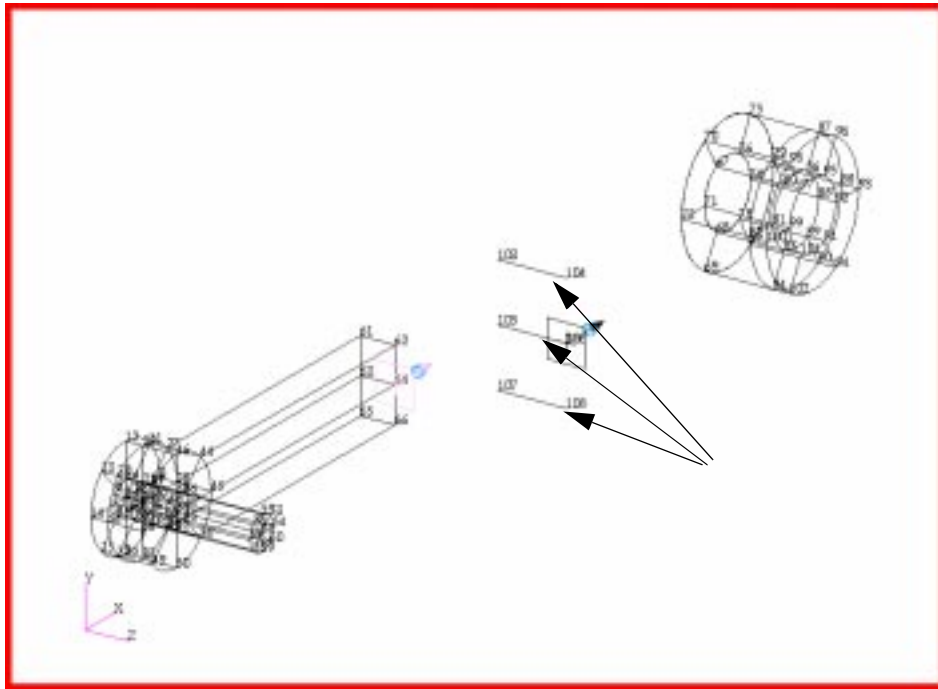
Select the curves shown  
above

*Plane List*

**Plane 2**

**Apply**

The curves should appear as follows.



29. Now create the two surfaces from the three curves as before. The three curves are shown above. Once again use the top curve as the starting curve and the middle as the ending curve for the top surface. Reverse this step for the bottom surface.

---

Use the Curve Icon to select the curves.



*Action:*

**Create**

*Object:*

**Surface**

*Method:*

**Curve**

*Starting Curve List*

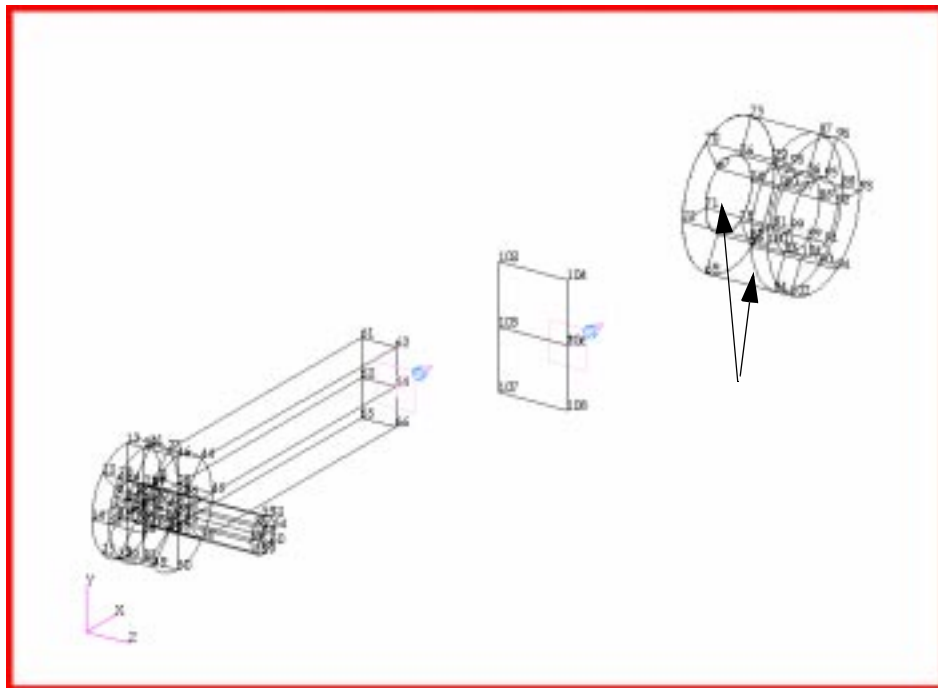
See figure

*Ending Curve List*

See figure

**Apply**

Use the figure below to select the correct surfaces for creating the two solids in the next step.



30. Now create the two solids by using the two surfaces that you just created along with two faces from the solids.

*Action:*

**Create**

*Object:*

**Solid**

*Method:*

**Surface**

Select the top Surface on the plane.

Use the Solid Face icon when selecting the faces that are pointed out in the figure.



*Starting Surface List*

See figure

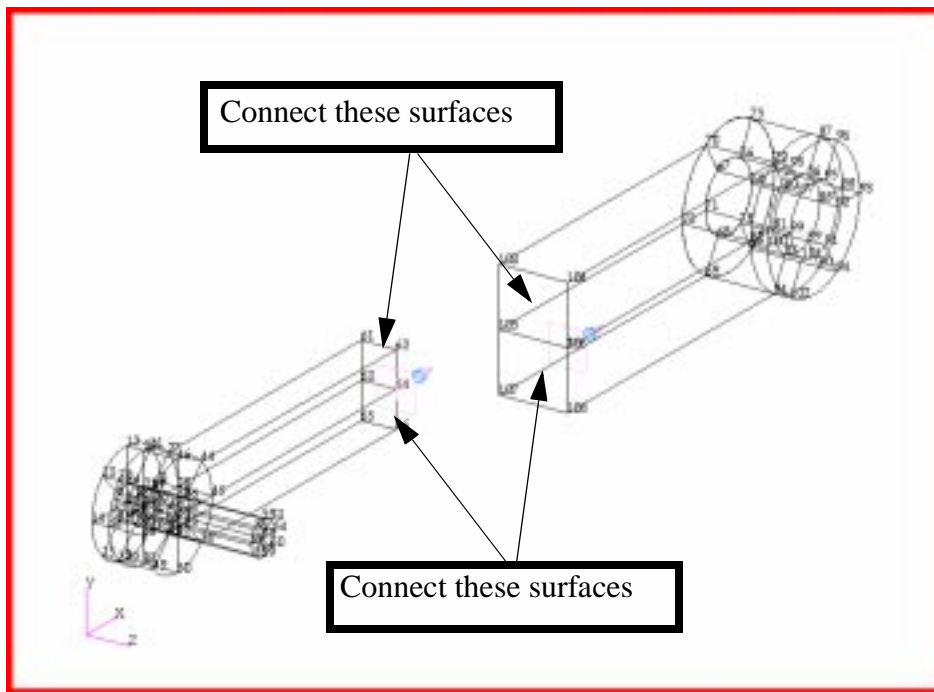
*Ending Surface List*

See figure

**Apply**

Repeat for the bottom surface on the plane.

Your model should look like the one shown below.



31. Connect the four surfaces that you created on the two planes by creating two solids between them. Use the surface icon to select the two surfaces.

*Action:*

**Create**

*Object:*

**Solid**

*Method:*

**Surface**

*Starting Surface List*

See figure

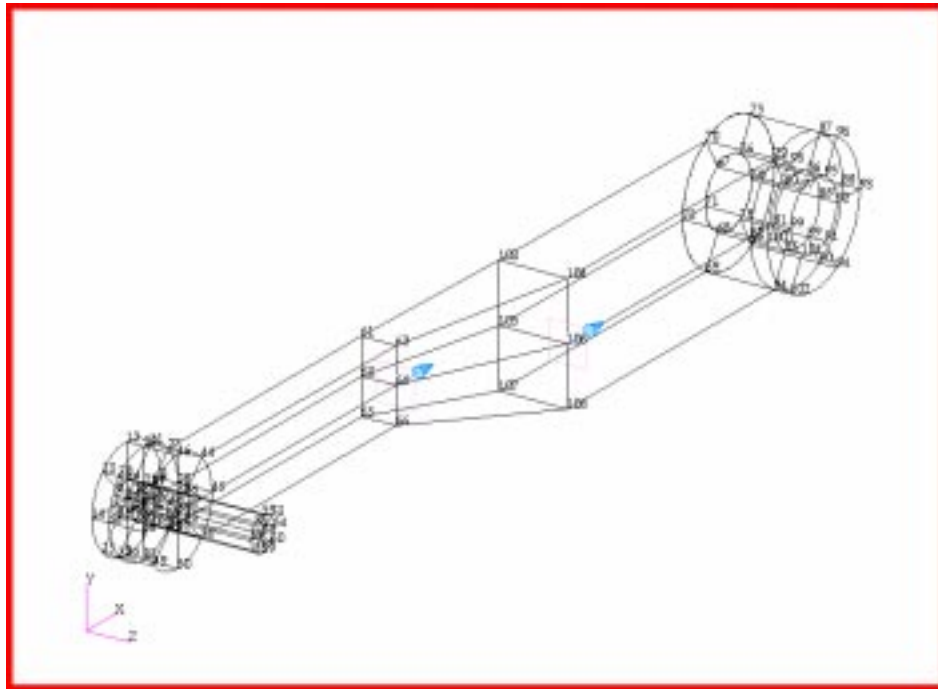
---

*Ending Surface List*

See figure

**Apply**

Your model should now look like the one shown below.



Create a group for the geometry of the model called geom\_parametric.

**Group/Create...**

*Action:*

**Create**

*New Group Name*

**geom\_parametric**

**Make Current**

*Group Contents:*

**Add All Geometry**

**Apply**

Rotate the model to get an idea of what the model looks like.

The next part of the exercise deals with meshing the pitman arm.

32. Look at the figure below to place the mesh seeds on the proper solid edges.

◆ **Finite Elements**

Action:

Create

Object:

Mesh Seed

Type:

Uniform

◆ **Number of Elements**

Number =

See figures below

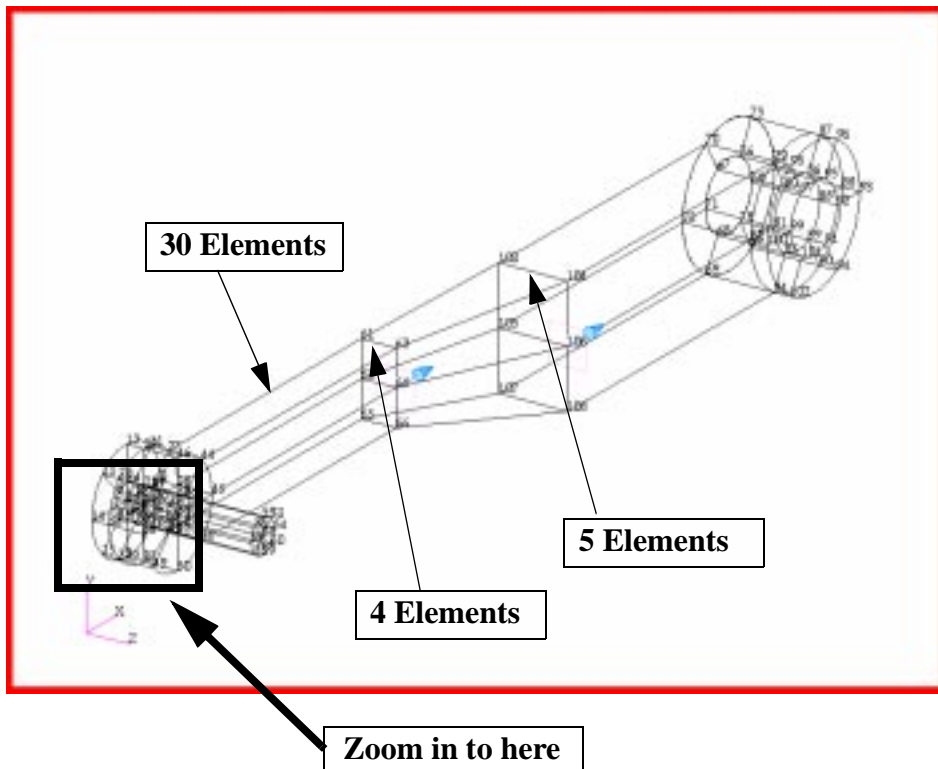
**Auto Execute**

Curve List

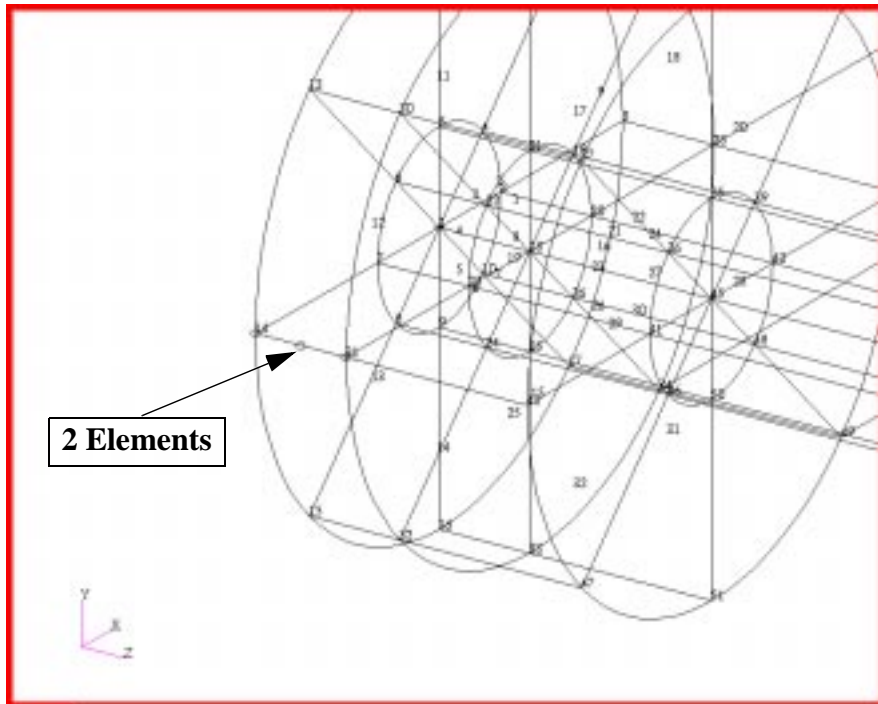
See figures below

**Apply**

Zoom in on the area shown below



You should have two elements located on the following part of the pitman arm.



33. Create a Finite Element Group.

**Group/Create...**

*New Group Name*

fem\_isomesh

**Make Current**

*Group Contents*

**Add Entity Selection**

*Entity Selection:*

(Leave Blank)

**Apply**

**Cancel**

34. Mesh the entire model using the Isomesh utility.

*Action:*

**Create**

*Object:*

**Mesh**

*Type:*

**Solid**

*Global Edge Length*

0.1

**IsoMesh**

<i>Element Topology</i>	<input type="text" value="Hex8"/>
<i>Solid List</i>	<input type="text" value="Solid 1:58"/>
<input type="button" value="Apply"/>	

Click OK on the message warning about the wedge elements that were created while meshing the model. The wedge elements should show up in red on the model.

35. Turn off all the FEM labels if needed.

**Display/Finite Elements...**

	<input type="button" value="Hide All FEM Labels"/>
<input type="button" value="Apply"/>	
<input type="button" value="Cancel"/>	

36. Equivalence all the nodes on the model.

<i>Action:</i>	<input type="text" value="Equivalence"/>
<i>Object:</i>	<input type="text" value="All"/>
<i>Method:</i>	<input type="text" value="Tolerance Cube"/>
<i>Nodes to be excluded</i>	<input type="text"/>
<i>Equivalencing Tolerance</i>	<input type="text" value="0.005"/>
<input type="button" value="Apply"/>	

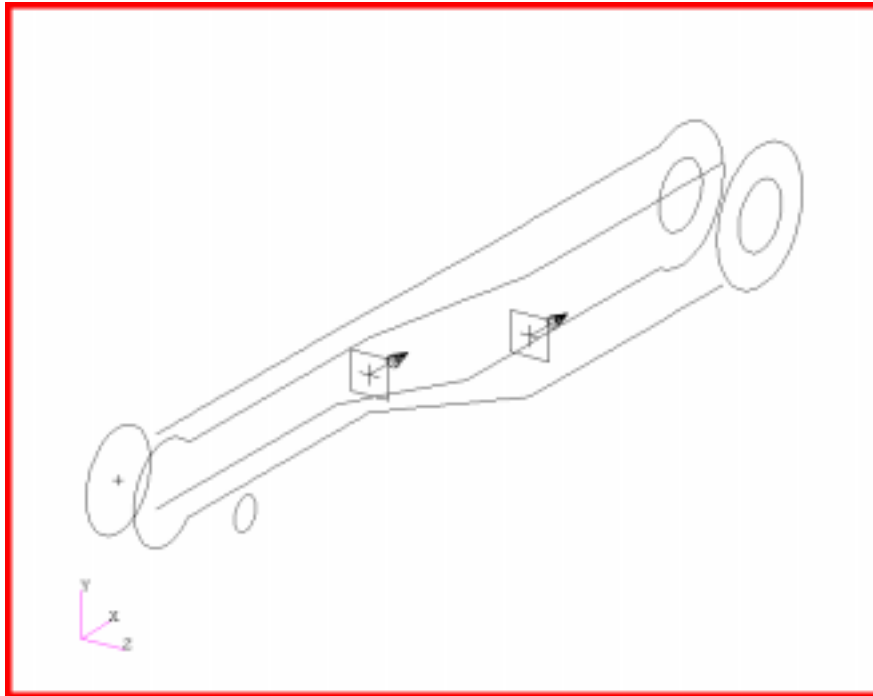
Purple circles should appear where nodes were deleted.

37. Now verify the element boundaries of the model.

<i>Action:</i>	<input type="text" value="Verify"/>
<i>Object:</i>	<input type="text" value="Element"/>
<i>Test:</i>	<input type="text" value="Boundaries"/>
<input type="checkbox"/> <b>Free Edges</b>	
<input type="button" value="Apply"/>	

---

Your model should look like the one shown below.



Change to the Front View for the next steps.



Reset the graphics by clicking on the Reset Graphics Button.

**Reset Graphics**

38. When adding the Loads and Boundary Conditions you may want to erase all the FEM so that the model is clearer. Click on the Plot/Erase icon. Erasing all the Finite Elements will not delete the mesh from the database. Erasing the mesh will just remove it from the screen for the time.



**Erase All FEM**

**OK**

Refresh the graphics in order to draw the geometry once again.



39. Fix one end of the Pitman Arm

◆ **Loads/BCs**

*Action:*

**Create**

*Object:*

**Displacement**

*Type:*

**Nodal**

*New Set Name*

**fix\_pivot\_point**

**Input Data ...**

*Translations <T1 T2 T3>*

**<0 0 0>**

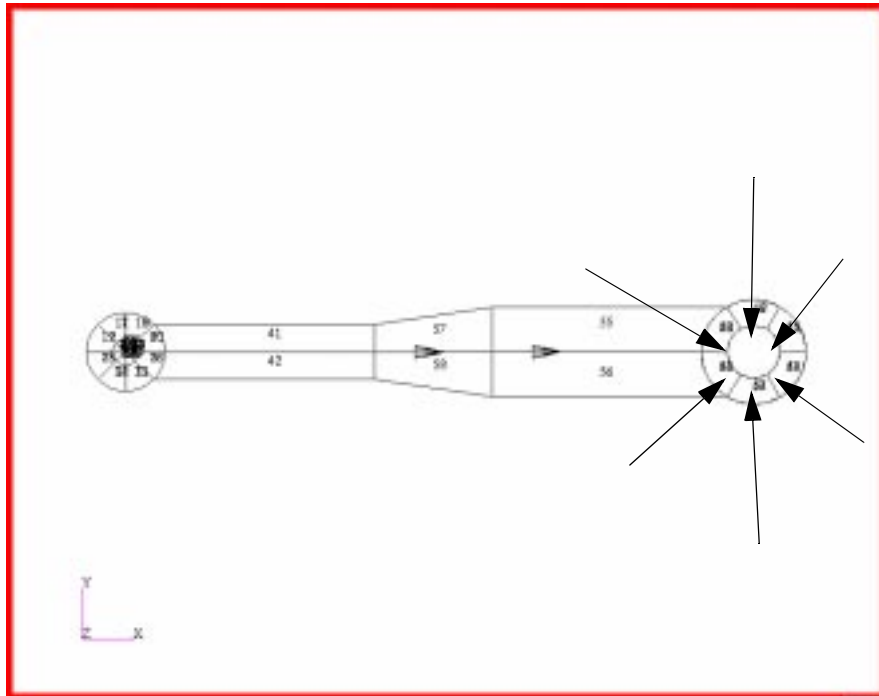
**OK**

**Select Application Region**

◆ **Geometry**

Select the Surface or Face Icon in the select menu.

Select each of the twelve faces on the inside of the hole individually, by shift clicking the solid faces.



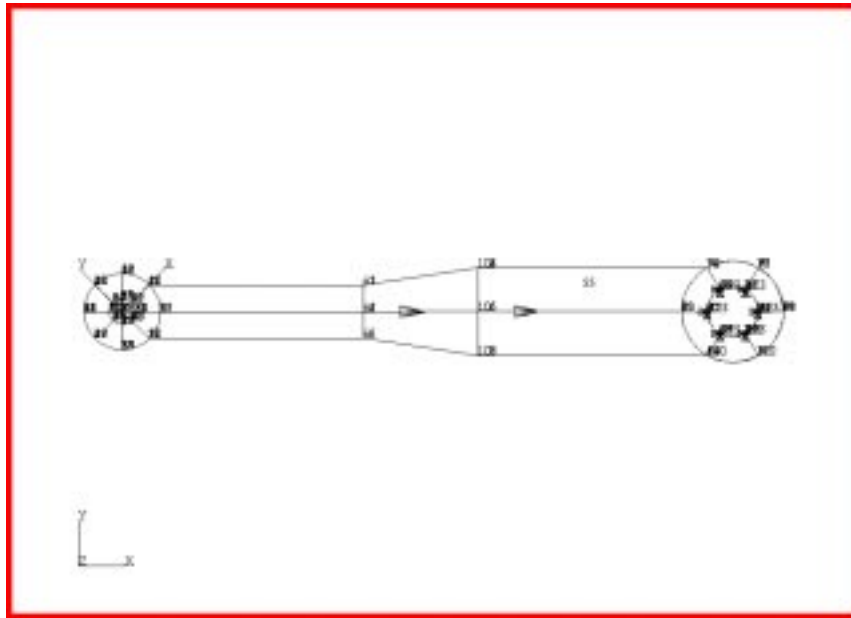
Rotate the figure around so that you can be sure that only the faces of the solid were selected.

**Add**

**OK**

**Apply**

Your figure should now look like the one shown below.



40. Create a coordinate axis

◆ **Geometry**

*Action:*

**Create**

*Object:*

**Coord**

*Method:*

**3Point**

*Type:*

**Rectangular**

*Origin:*

**[0 0 0]**

*Point On Axis 3:*

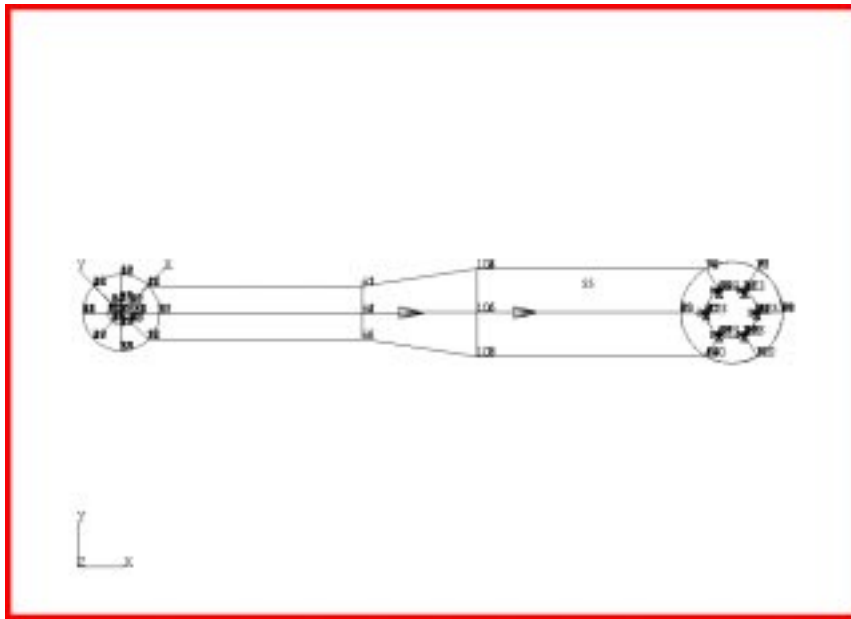
**[0 0 1]**

*Point On Plane 1-3:*

**[1 1 0]**

**Apply**

The axis should appear on the left side of the model.



41. Create a field called force\_zero.

◆ **Fields**

Action:

**Create**

Object:

**Spatial**

Method:

**PCL Function**

Field Name

**force\_0deg**

Field Type

◆ **Vector**

Coordinate System Type

◆ **Real**

Coordinate System

**Coord 0**

Second Component

$6400*(0.125**2 - 'X**2)$

**Apply**

42. Create another field called force\_45deg.

Action:

**Create**

Object:

**Spatial**

Method:

**PCL Function**

Field Name

**force\_45deg**

*Field Type*◆ **Vector***Coordinate System Type*◆ **Real***Coordinate System***Coord 1***Second Component* $6400*(0.125**2 - 'X**2)$ **Apply**

Now with your model in the front view you are going to rotate the model to look at the end of the arm.

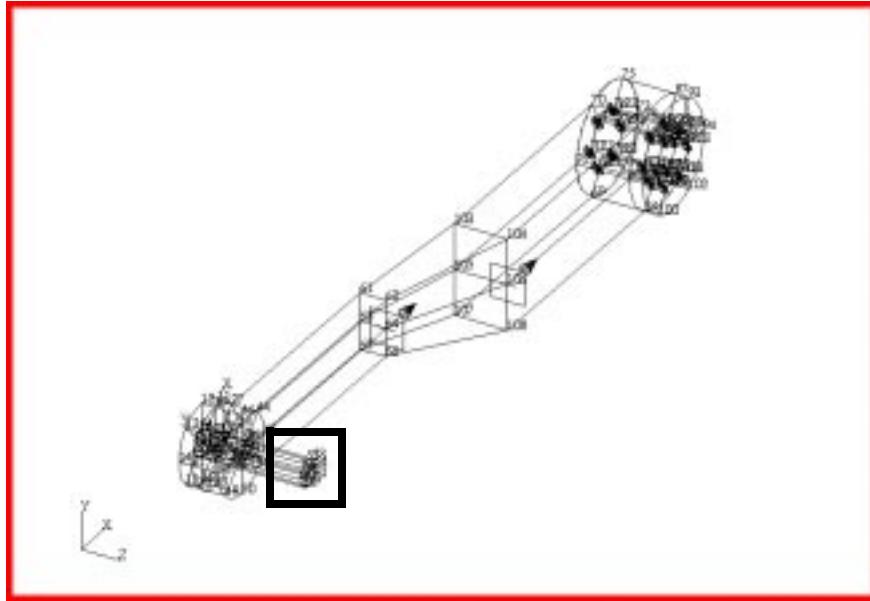
**Viewing/Transformations...**

Rotate the model to the right twice and then down once.



---

Your model should now look like the following.



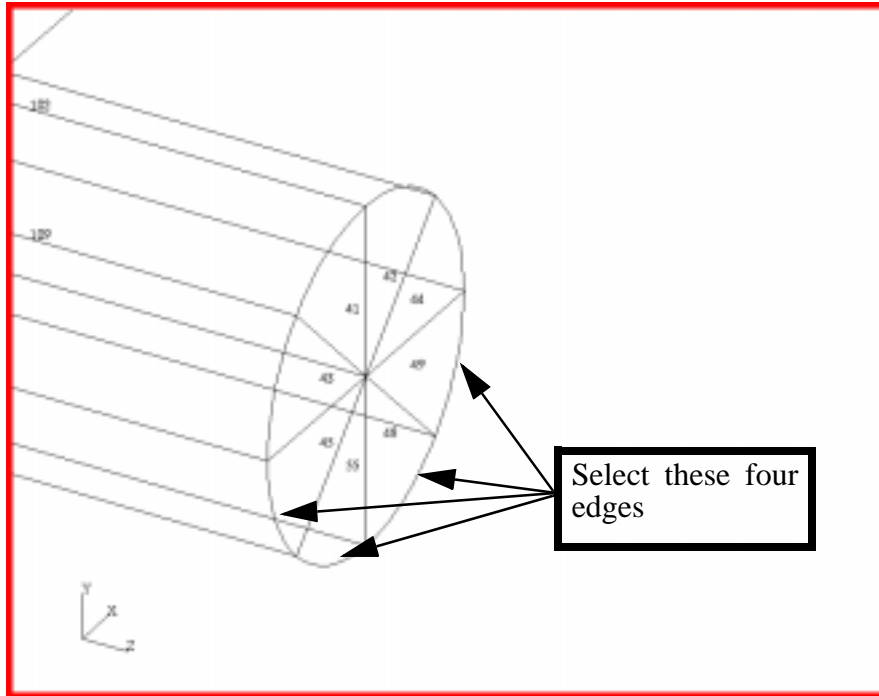
Click OK to close the form.

**OK**

Zoom in on the area shown above by using the View Corners Icon in the Main Form of Patran.



You should now clearly be able to see the end of the arm.



43. Create a force at one end of the arm. Select the four bottom surfaces in the figure shown above.

◆ **Loads/BCs**

Action:

Create

Object:

Force

Type:

Nodal

New Set Name

force\_0deg

Input Data ...

Spatial Fields

force\_0deg

Force <F1 F2 F3>

f:force\_0deg

OK

Select Application Region

◆ **Geometry**

Click on the Select Curve or Edge Icon in the select menu.



Now Shift Click the edges shown in the previous figure.

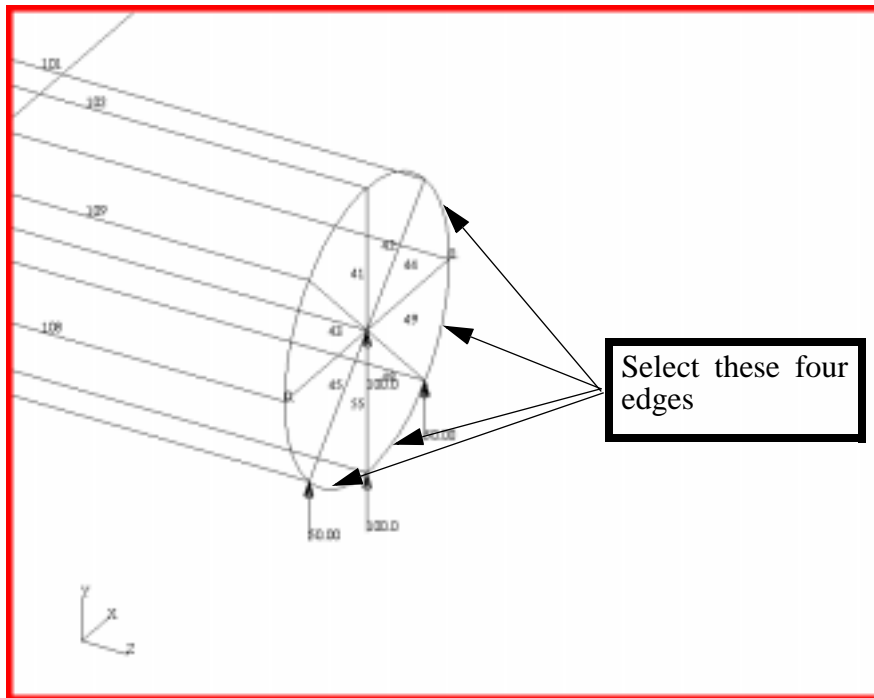
Select Geometry Entities

See figure above

OK

Apply

The forces should appear as shown below.



44. Now create another force at a 45 degree angle to the other forces.

New Set Name

force\_45deg

Input Data ...

Spatial Fields

force\_45deg

Force <F1 F2 F3>

f:force\_45deg

Analysis Coordinate Frame

Coord 1

OK

**Select Application Region**

◆ **Geometry**

*Select Geometry Entities*

See figure above

**Add**

**OK**

**Apply**

45. Show the vectors for the 45 degree force that you just created.

*Action:*

**Plot Markers**

*Assigned Load/BC Sets*

**Force\_force\_45deg**

*Select Groups*

**geom\_parametric**

**Apply**

46. Create the material properties for the model.

◆ **Materials**

*Action:*

**Create**

*Object:*

**Isotropic**

*Method:*

**Manual Input**

*Material Name*

**material**

**Input Properties...**

*Constitutive Model*

**Linear Elastic**

*Elastic Modulus*

**30E6**

*Poisson Ratio*

**0.3**

**Apply**

**Cancel**

Click on the Fit View icon in the main menu.



---

47. Create the properties for the model.

◆ **Properties**

<i>Action:</i>	<b>Create</b>
<i>Dimension:</i>	<b>3D</b>
<i>Type:</i>	<b>Solid</b>
<i>Property Set Name</i>	<b>solid_prop</b>
<b>Input Properties...</b>	
<i>Material Property Sets</i>	<b>material</b>
<i>Material Name</i>	<b>m:material</b>
<b>OK</b>	
<i>Application Region</i>	
<i>Select Members</i>	<b>Select All the Solids</b>
<b>Add</b>	
<b>Apply</b>	

48. Create the Load Cases for the analysis.

◆ **Load Cases**

<i>Action:</i>	<b>Create</b>
<i>Load Case Name</i>	<b>force_0deg</b>
<i>Description</i>	<b>Add description here</b>
<b>Assign/Prioritize Loads/BCs</b>	
<i>Select Loads/BCs to Add to Spreadsheet</i>	<b>Force_force_0deg Displ_fix_pivot_point</b>
<b>OK</b>	
<b>Apply</b>	

49. Create the load case for the 45 degree force.

<i>Action:</i>	<b>Create</b>
<i>Load Case Name</i>	<b>force_45deg</b>
<i>Description</i>	<b>Add description here</b>

**Assign/Prioritize Loads/BCs****Remove All Rows***Select Loads/BCs to Add to Spreadsheet***Force\_force\_45deg  
Displ\_fix\_pivot\_point****OK****Apply**

50. Once again post all the FEM in the model.

**Plot All Posted FEM****OK**

The Elements and the Mesh should once again appear on the model.

51. When you created the force\_45deg load case that one was made current. Change that so the force\_zero load case is current.

*Action:***Modify***Select Load Case to Modify***force\_0deg****■ Make Current****Apply**

Click on Cancel in the Assign/Prioritize Load/BCs in order to close the form that popped up when you clicked on force\_zero.

**Cancel**

52. Set up the subcases for running the analysis.

**◆ Analysis***Action:***Analyze***Object:***Entire Model***Method:***Full Run**

---

<i>Job Name</i>	<b>pitman_arm</b>
<b>Subcase Create...</b>	
<i>Available Subcases</i>	
<i>Subcase Name</i>	<b>force_0deg</b>
<i>Subcase Description</i>	subcase for force_0deg loading
<i>Available Load Cases</i>	<b>force_0deg</b>
<b>Apply</b>	

53. Do the same thing for the force\_45deg subcase.

<i>Available Subcases</i>	
<i>Subcase Name</i>	<b>force_45deg</b>
<i>Subcase Description</i>	subcase for force_45deg loading
<i>Available Load Cases</i>	<b>force_45deg</b>
<b>Apply</b>	
<b>Cancel</b>	

54. Next select the subcases that you want Natran to analyze.

<b>Subcase Select...</b>	
<i>Subcases for Solution Sequence</i>	<b>force_0deg force_45deg</b>
<i>Subcases Selected</i>	<b>force_0deg force_45deg</b>
<b>OK</b>	
<b>Apply</b>	

Note in the Subcases Selected list clicking on the Default Subcase will remove it from the selected subcases, this is what you want.

55. Close this database and create a new one, then read in the .op2 file to the new database.

<b>File/Close</b>
-------------------

**File/New...**

*New Database Name*

**pitman\_res**

Read in the results file that was just created during the analysis.

◆ **Analysis**

*Action:*

**Read Output2**

*Object:*

**Both**

*Method:*

**Translate**

**Select Results File...**

*Available Files*

**pitman\_arm.op2**

**OK**

**Apply**

56. Create a deformed shape plot of the Displacement result value for the load force\_0deg.

◆ **Results**

*Action:*

**Create**

*Object:*

**Deformation**

Click on this icon in the results form.



*Select Result Case(s)*

**force\_0deg,...**

*Select Deformation Result*

**Displacements, ...**

*Show As:*

**Component**

■ XX ■ YY ■ ZZ

---

Now click on the Target Entities icon in the results form



*Target Entity*

**Current Viewport**

Next click on the Display Attributes icon in the results form.



*Deformed*

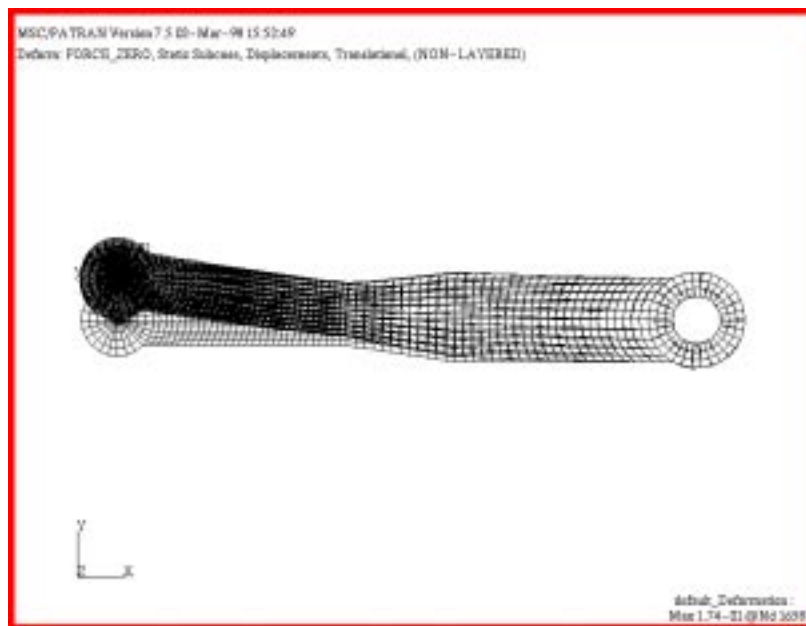
select a color

**Show Undeformed**

select a color

**Apply**

Your model should look like the one shown below.



Clear the deformed plot from the viewport by clicking on the Reset Graphics Icon.



57. Create a Fringe Plot of von Mises stress for the two loading sets. To do this create a second group and viewport for displaying the results for loading force\_45deg.

**Group/Create...***Action:***Create***New Group Name***grp\_force\_45deg***Group Contents***Add All Entities****Apply****Viewport/Create...***New Viewport Name***vp\_force\_45deg****Apply**

Fit the two vieports on the screen

**Viewport/Tile...**

Post the two stress results for the two loading sets, force\_0deg, and force\_45deg

Make the viewport **default\_viewport** the current viewport by clicking in the border of the viewport.

Make the group default\_group the current group

**Group/Set Current...***Set Current Group***default\_group****Cancel**

Create a fringe plot of Von Mises stress for loading force\_0deg.

**◆ Results***Action:***Create***Object:***Fringe**

Click on the Select Results Icon.



Select Result Case(s)

**force\_0deg**

Select Fringe Result

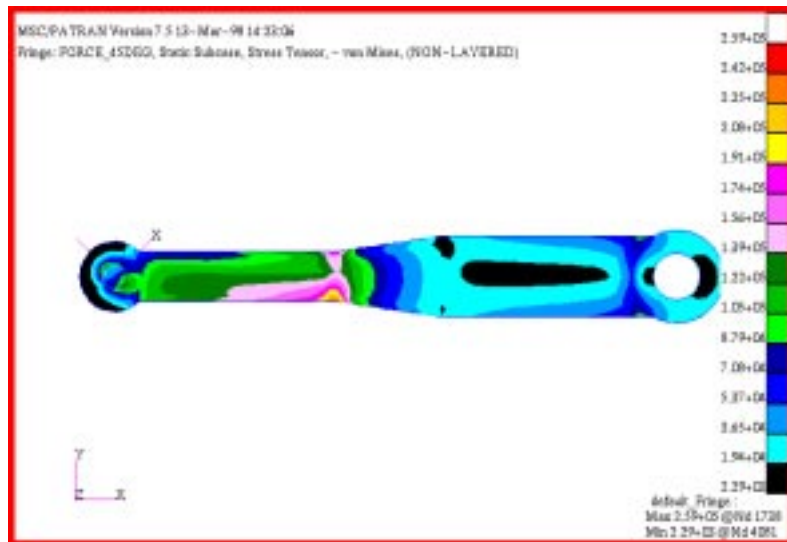
**Stress, Tensor**

Quantity:

**von Mises**

**Apply**

The fringe plot should look like the one shown below.



Now make the viewport vp\_force\_45deg the current viewport by clicking in the boundary of the window.

Make the group grp\_force\_45deg the current group.

**Group/Set Current...**

Set Current Group

**grp\_force\_45deg**

**Cancel**

Create a fringe plot of Von Mises stress for loading force\_45deg.

◆ **Results**

Action:

**Create**

Object:

**Fringe**

Click on the Select Results Icon.



Select Result Case(s)

force\_45deg

Select Fringe Result

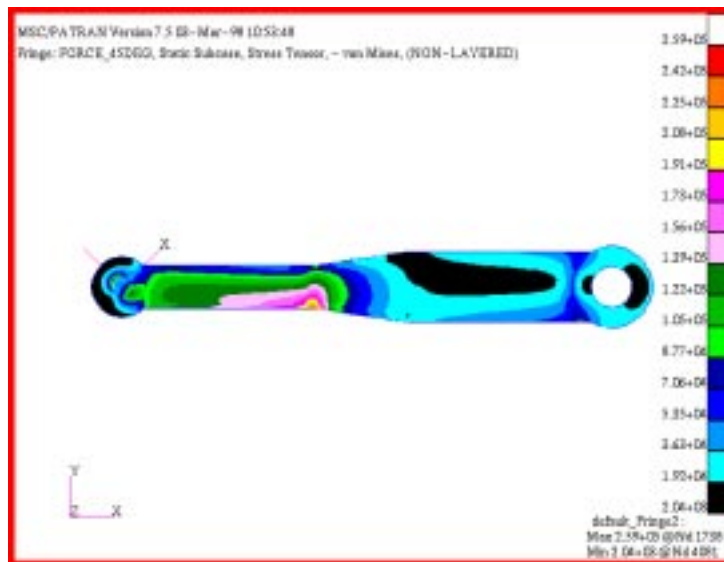
Stress, Tensor

Quantity:

von Mises

Apply

The fringe plot should look like the one shown below.



Both viewports should have fringe plots.

58. This ends the pitman arm lesson. Close the database and quit Patran.

**File/Quit**

