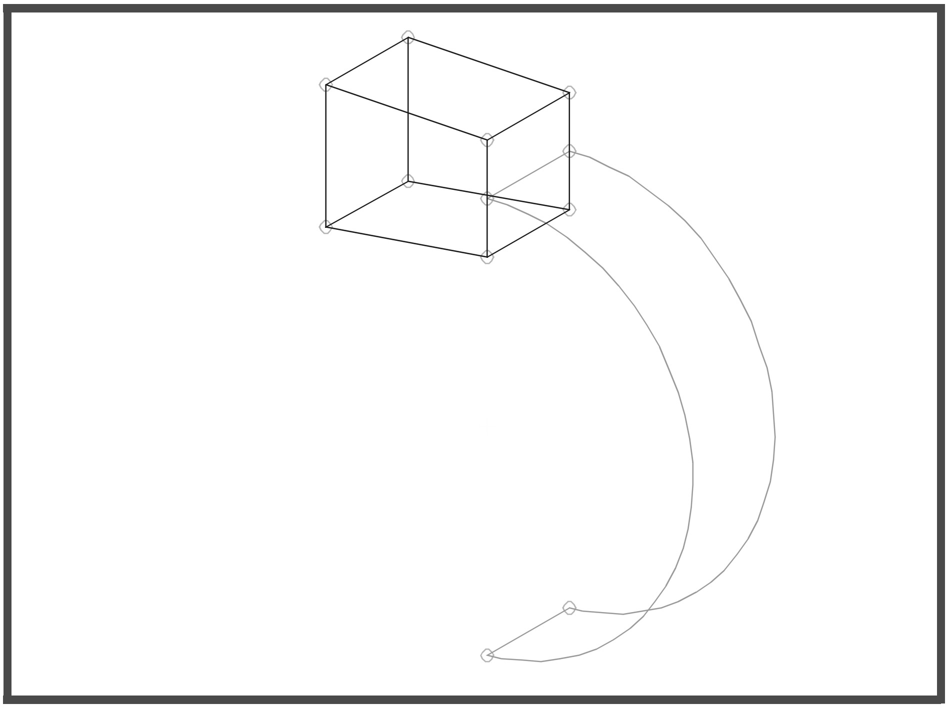
LESSON 6

Modeling a Shell to a Solid Elements Transition



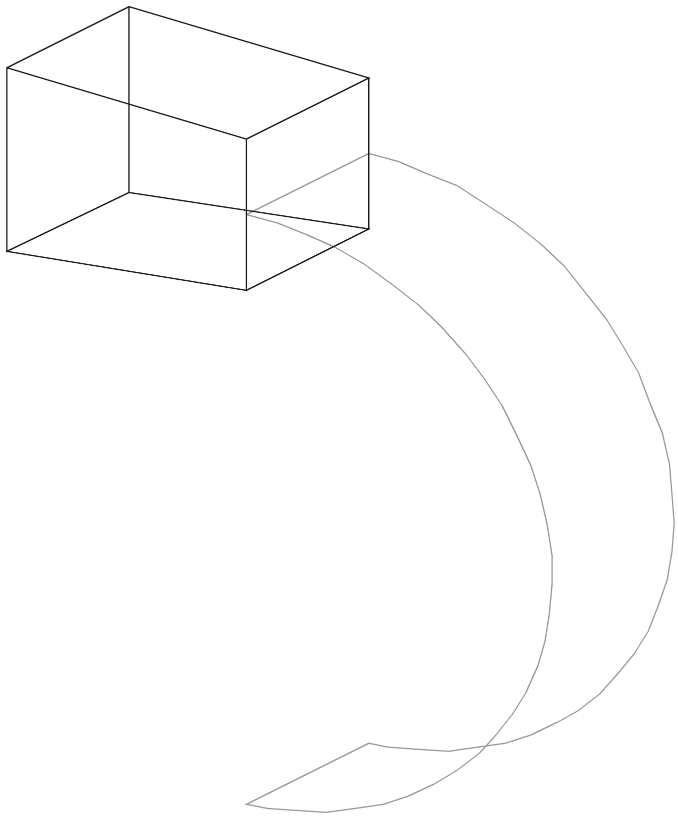
Objectives:

- Use MPCs to replicate a Solid with a Surface
- Compare stress results of the Solid and Surface

Model Description:

In this exercise, you will create a solid/surface transition. Though the use of a thickness field and MPCs the surface will represent a continuation of the solid.

The Model appears below



Exercise Procedure:

1. Open a new database. Name it **shell_solid**

File/New ...

New Database Name: **shell_solid**

OK

The viewport (PATRAN's graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC/PATRAN.

2. In the *New Model Preference* form pick the following options

Analysis Code: **MSC/ADVANCED_FEA**

Analysis Type: **Structural**

OK

3. To create necessary geometry for the solid.

Start with the two surfaces that will define the solid

◆ **Geometry**

Action: **Create**

Object: **Surface**

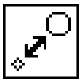
Method: **XYZ**

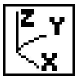
Vector Coordinate List **<40, 30, 3>**

Origin Coordinate List **[-40, 0, 34.93]**

Apply

Increase the **Point Size** in order to see the points more clearly and change the view to **Isoview 3**





Action: **Create**

Object: **Surface**

Method:	**XYZ**
Vector Coordinate List	**<40, 30, -3>**
Origin Coordinate List	**[-40, 0, 66.93]**

Apply

Now create the solid

Action:	**Create**
Object:	**Solid**
Method:	**Surface**
Starting Surface List	**Surface 1**
Ending Surface List	**Surface 2**

Apply

4. Delete the original surfaces

Action:	**Delete**
Object:	**Surface**
Surface List:	**Surface 1 2**

Apply

5. Create two points that will define the surface later

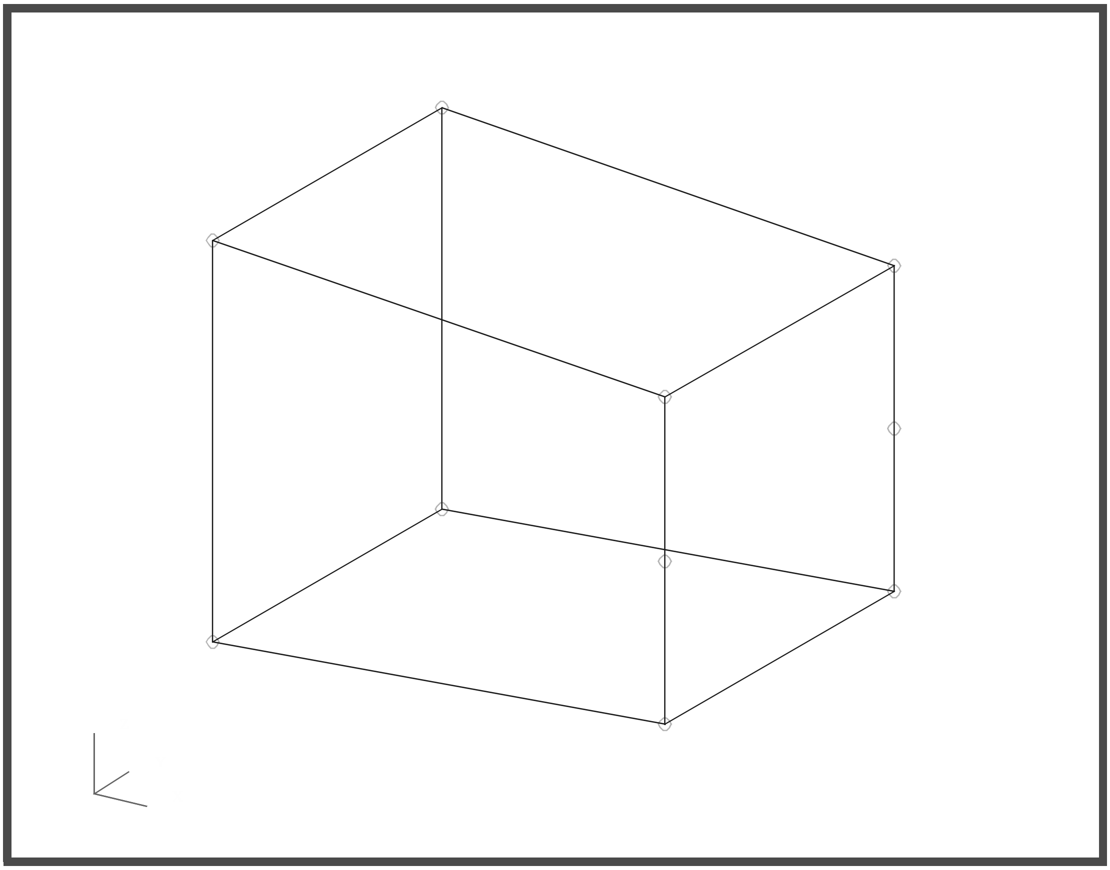
Action:	**Create**
Object:	**Point**
Method:	**Interpolate**
Starting Point List	**Point 4** (see fig -1)
Ending Point List	**Point 8**

Apply

Repeat this procedure with **Points 3** and **7**

Starting Point List	**Point 3**
Ending Point List	**Point 7**
Apply	

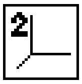
Figure 6-1



6. Using the two points you just created sweep out the surface

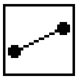
Action:	**Create**
Object:	**Surface**
Method:	**Revolve**

To revolve around the Y-axis select this icon then click on the coordinate frame



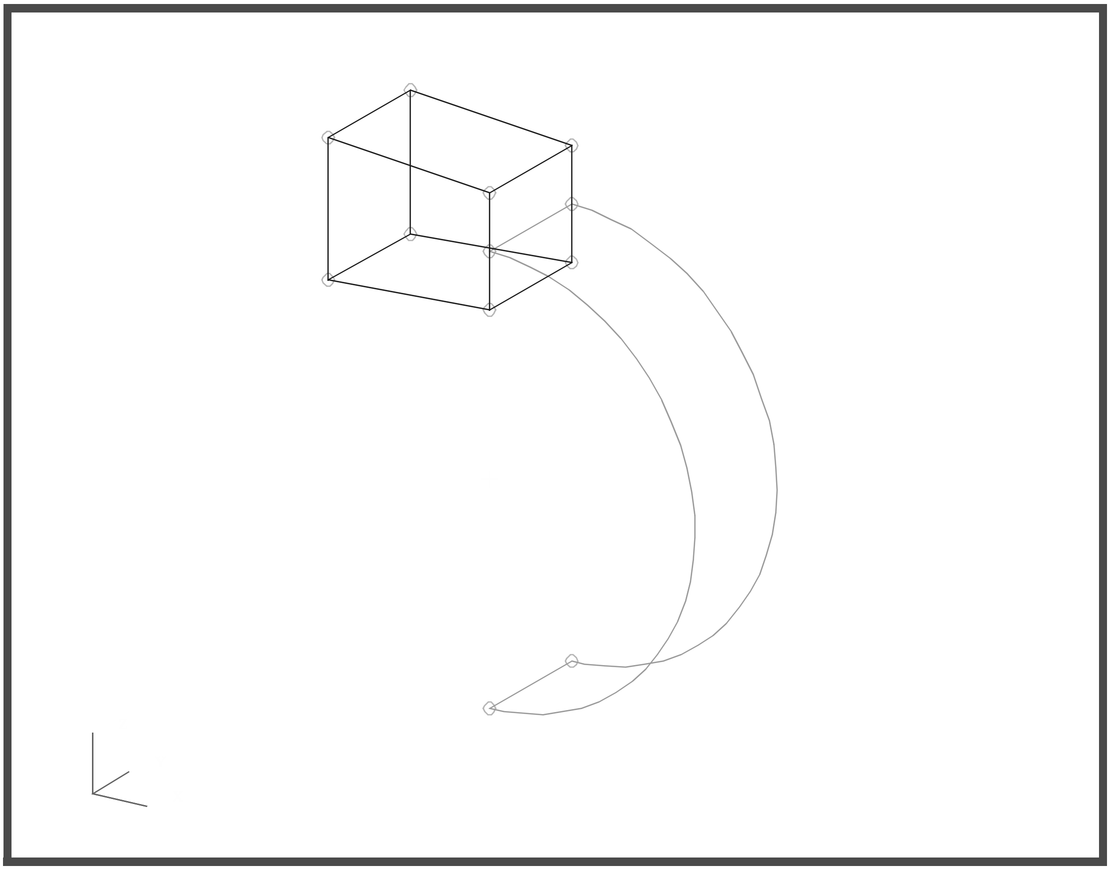
Axis	**Coord 0.2**
Total Angle	**180**
Curve List:	

Click in the Curve List databox and select the Two Point icon. The curve will be defined by selecting **Points 9** and **10**.



Apply

Your model should now appear as below



7. Display the parametric direction

Display/Geometry

■ **Show Parametric Direction**

Apply

8. Create a field for the thickness of the surface

◆ **Fields**

Action:	**Create**
Object:	**Spatial**
Method:	**PCL Function**
Field Name:	**thick**
Coordinate System Type	◆ **Parametric**

Geometric Entity	**Surface 1**
Scalar Function	**26-24*'C2**

Apply

9. Now create the relevant material properties for aluminum_iso_SI_mm.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**aluminum_iso_SI_mm**

Input Properties ...

Elastic Modulus	**7000**
Poisson's Ratio	**0.30**
Density	**2.7E-6**
Thermal Expansion Coeff	**2.32E-5**

Apply

Cancel

10. Create a group for the mesh on the solid named **fem_sol**

Group/Create

New Group Name	**fem_sol**

■ **Make Current**

Groups Contents	**Add Entity Selection**

Apply

11. Now Mesh the solid

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Solid**
Global Edge Length	**8**

Apply

12. Create a group for the mesh on the surface named **fem_sur**

Group/Create

New Group Name	**fem_sur**

■ **Make Current**

Groups Contents	**Add Entity Selection**

Apply

13. Now Mesh the surface

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length	**8**

Apply

14. Verify there are no free edges and the normals to the surface point outwards

◆ **Finite Elements**

Action:	**Verify**
Object:	**Elements**
Type:	**Boundary**

◆ **Free Edges**

Apply

The display should be yellow indicating the only free edges are on the outside of the model. Now check the element normals.

Action:	**Verify**
Object:	**Elements**
Type:	**Normals**

◆ **Draw Normal Vectors**

Apply

If the elements point inward then perform this step

Action:	**Modify**
Object:	**Elements**
Type:	**Reverse**
Element List	Select all Surface Elements

Apply

15. Create a group named **fem_all**

Group/Create

New Group Name	**fem_all**

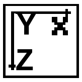
■ **Make Current**

Groups Contents	**Add All FEM**

Apply

16. Now create the **MPCs**

It may be more convenient for screen selecting to switch to the **top view**



Action:	**Create**
Object:	**MPC**

Type: **SS Linear (13)**

Define Terms ...

■ **Create Dependent**

Node List **Node 151:155**

Nodes 151:155 exist as part of the surface mesh and are coincident with nodes 66:90:6 on the solid. Be sure to select the proper nodes.

Apply

■ **Create Independent**

Node List **Node 126:150:6**

Apply

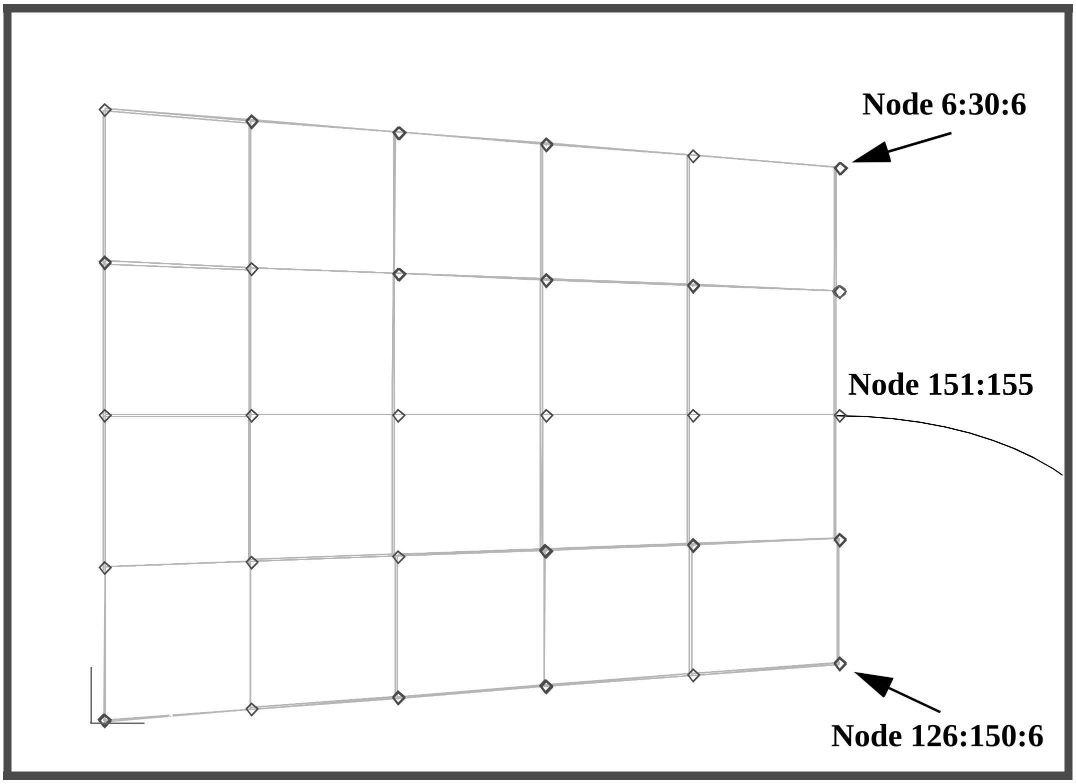
Node List **Node 6:30:6**

Apply

Cancel

Apply

Figure 6-2

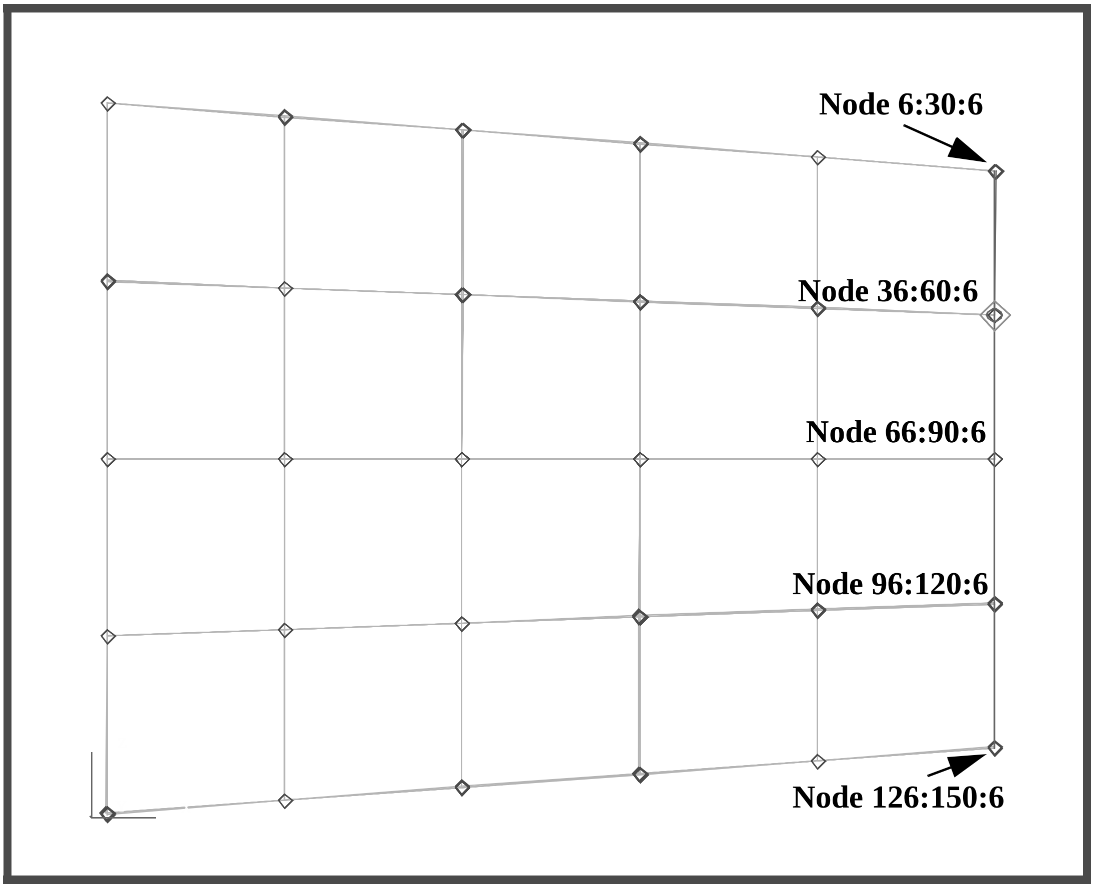


17. Post the **fem_sol** group and create a **Slider MPC**

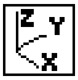
Action:	**Create**
Object:	**MPC**
Type:	**SS Slider (12)**
Define Terms ...	
■ **Create Dependent**	
Node List	**Node 96:120:6** *(see figure 6-3)*
Apply	
■ **Create Independent**	
Node List	**Node 126:150:6**
Apply	
Node List	**Node 6:30:6**
Apply	
Cancel	
Apply	

Repeat this procedure twice more changing only the *Dependent Node List* to **Node 66:90:6** and **36:60:6**

Figure 6-3



Change the view back to **Iso_view 3**



18. Apply the properties to the model.

First the solid

◆ **Properties**

Action:	**Create**
Dimension:	**3D**
Type:	**Solid**
Property Set Name:	**sol**
Input Properties ...	
Material Name:	**m:aluminium_iso_SI_mm**

OK	
Select Members:	**Solid 1**
Add	
Apply	

Now the Surface

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name:	**sur**
Input Properties ...	
Material Name	**m:aluminium_iso_SI_mm**
Shell Thickness	**f:thick**
OK	
Select Members:	**Surface 1**
Add	
Apply	

If you would like to view the surface thickness

Action:	**Show**
Existing Properties	**Shell Thickness**
Display Method	**Scalar Plot**
Group Filter	**fem_all**
Apply	

19. Create the LBCs. One to fix the solid and one to pull on the surface

◆**Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **fix**

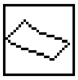
Input Data...

Translation **<0, 0, 0>**

OK

Select Application Region...

In order to select the appropriate solid faces, use the following entity select icon:



Surface or Face

Select Geometric Entities: **Solid 1.1** (opposite the surface)

Add

OK

Apply

Next, create displacement to pull on the end of the surface.

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **pull**

Input Data...

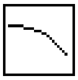
Translation **<-20, 0, 0>**

Rotation **<0, 0, >**

OK

Select Application Region...

In order to select the appropriate edge, use the following entity select icon:



Curve or Edge

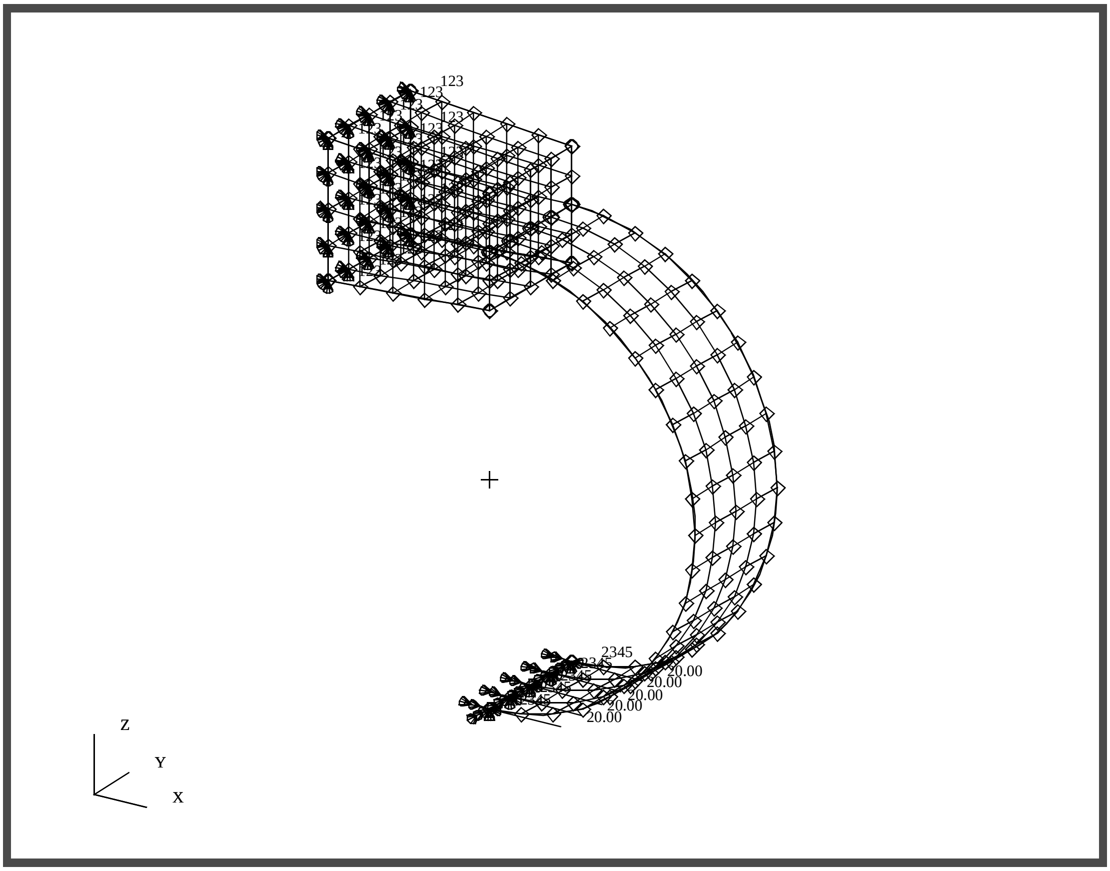
Select Geometric Entities: **Surface 1.2**

Add

OK

Apply

Your Model should appear as below



20. Submit the model for analysis.

◆**Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Full Run**

Job Name: **pull**

Step Creation...

Job Step Name: **pull1**

Solution Type **Nonlinear Static**

Apply

Cancel

Step Selection...

Selected Job Steps: **pull1**

Apply

Apply

You can monitor the progression of the job by looking at **pull1.msg** and **pull1.sta** files using the UNIX command ***tail -lf [filename]***. You can also monitor the analysis in the background using the UNIX command ***ps -a***.

21. Once the analysis is complete read the results back into the database

◆Analysis

Action: **Read Results**

Object: **Results Entities**

Method: **Translate**

Select Results File ...

Available Files: **pull1.fil**

OK

Apply

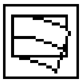
22. Create a fringe plot of the stresses for the model

◆Results

Action: **Create**

Object: **Quick Plot**

Before viewing the stresses change the **Deformation Attributes**



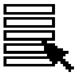
Render Style **Free Edge**

Scale Interpretation ◆ **True Scale**

Scale Factor **1.0**

❒ **Show Undeformed**

Now switch back to the **Select Results** form



Select Results Case: **Step1, Total Time=1**

Select Fringe Results: **Stress, Components**

Position...((NON-LAYERED))

Quantity: **Von Mises**

Selected Deformation Results: **Deformation, Displacement**

Apply

Notice that no stresses appear on the surface. This is because the results for shell elements have layers and cannot be displayed on a single surface. Create a new viewport so stresses on the solid and the surface can be displayed at the same time.

Viewport/Create

New Viewport Name **another**

Apply

Cancel

To see them side by side select:

Viewport/Tile

To make the new viewport active click on the edge until the border appears. Now post the **default** and **fem_all** groups in that viewport.

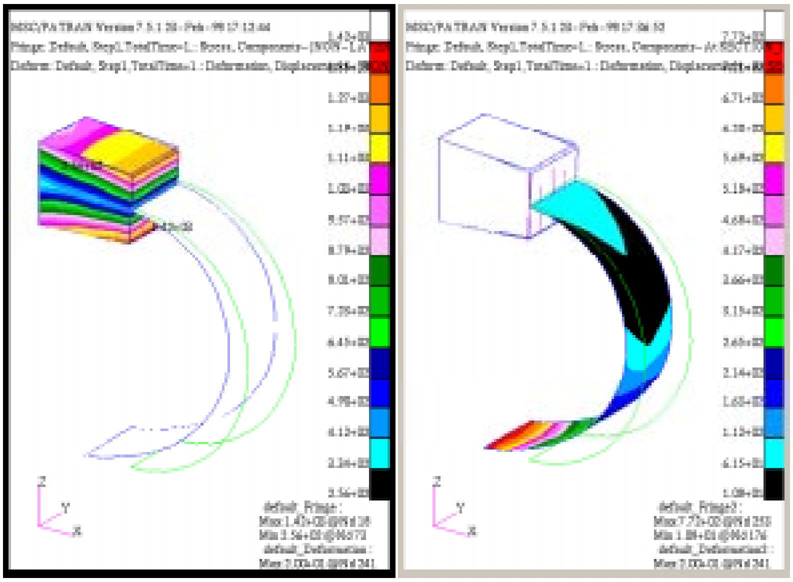
Group/Post

Select Groups to Post	**default_group** **fem_all**
Apply	
Cancel	

Now create a fringe plot of the surface in **another** viewport. Remember the surface has 5 different layers numbered 1 to 5, the bottom being the lowest number. Lets start with the middle layer.

Select Results Case:	**Step1, Total Time=1**
Select Fringe Results:	**Stress, Components**
Position...((At SECTION_POINT_3))	
Option:	**Average**
Close	
Quantity:	**Von Mises**
Selected Deformation Results:	**Deformation, Displacement**
Apply	

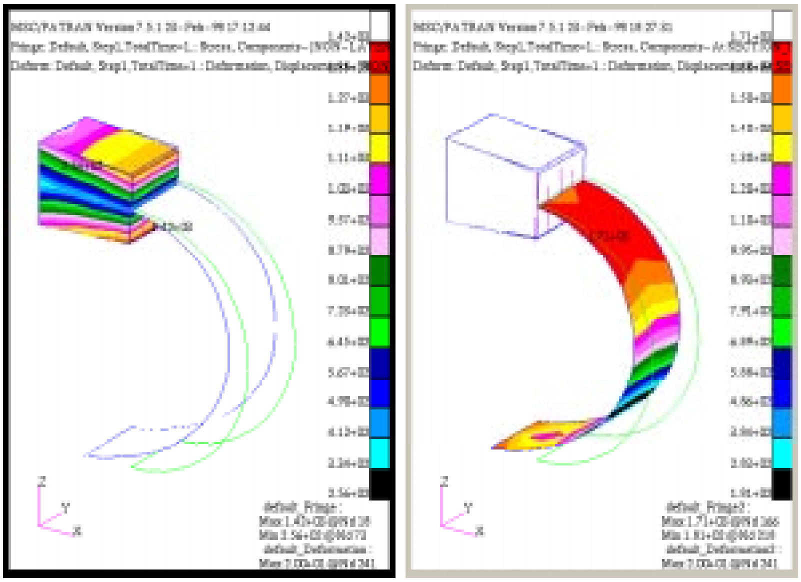
Notice the stress in the middle of the solid matches the stress on the surface.



Now lets look at the stress at the top of the model.

Select Results Case:	**Step1, Total Time=1**
Select Fringe Results:	**Stress, Components**
Position...((At SECTION_POINT_5))	
Option:	**Average**
Close	
Quantity:	**Von Mises**
Selected Deformation Results:	**Deformation, Displacement**
Apply	

You’ll notice now there is a discrepancy in the stress values for the top of the model.



Shell theory in this case over estimated the stress values. If this is within a given tolerance level the problem can be ignored. If not then there are two things that can be done. Patran has the option to create 2D Solids which is a very similar to produce and is designed for thicker surfaces. Another option is to extend the solid part of the model further.

23. Now we will animate the Translational Displacement along with a fringe plot of the Z-Component.

First delete the extra viewport

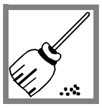
Viewport/Delete

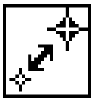
New Viewport Name **another**

Apply

Cancel

Next, reset the graphics and make the node size small again





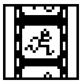
Action: **Create**

Object: **Deformation**

Select Results Case	select all
Select Deformation Result	**Defomation, Displacement**

■ **Animate**

Before hitting Apply select the Animation Options icon



Animation Method	**Global Variable**
Select Global Variable	**Time**
Animation Graphics	**3D**
Number of Frames	**12**

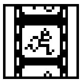
Apply

Once the animation is set up pause it and create a second fringe animation.

Action:	**Create**
Object:	**Fringe**
Select Results Case	select all
Select Deformation Result	**Stress, Component**
Quantity:	**Z Component**

■ **Animate**

Again select the Animation Options icon

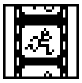


Animation Method	**Global Variable**
Select Global Variable	**Time**
Animation Graphics	**3D**
Number of Frames	**12**

Apply

The two animations will appear together

You can play with different aspects of the animation, for example, pause the animation and select the Display Attributes icon



Style **continuous**

Apply

Another feature on the Animation Control form is Cycle and Bounce. Change it to Bounce.

This ends the exercise, you may quit Patran.