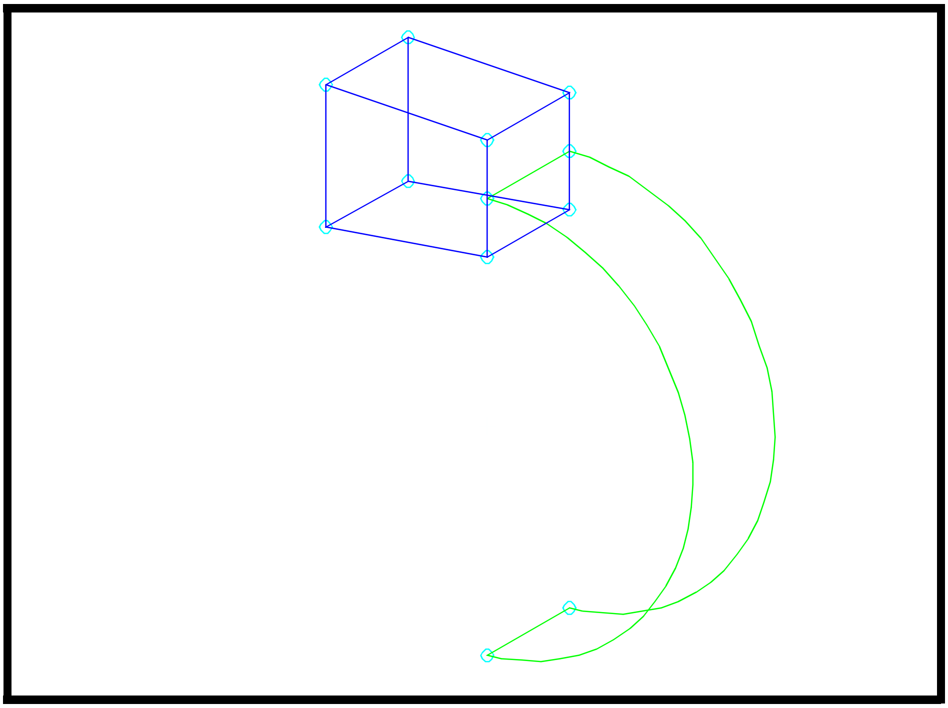
LESSON 9

Modeling a Shell to a Solid Element Transition



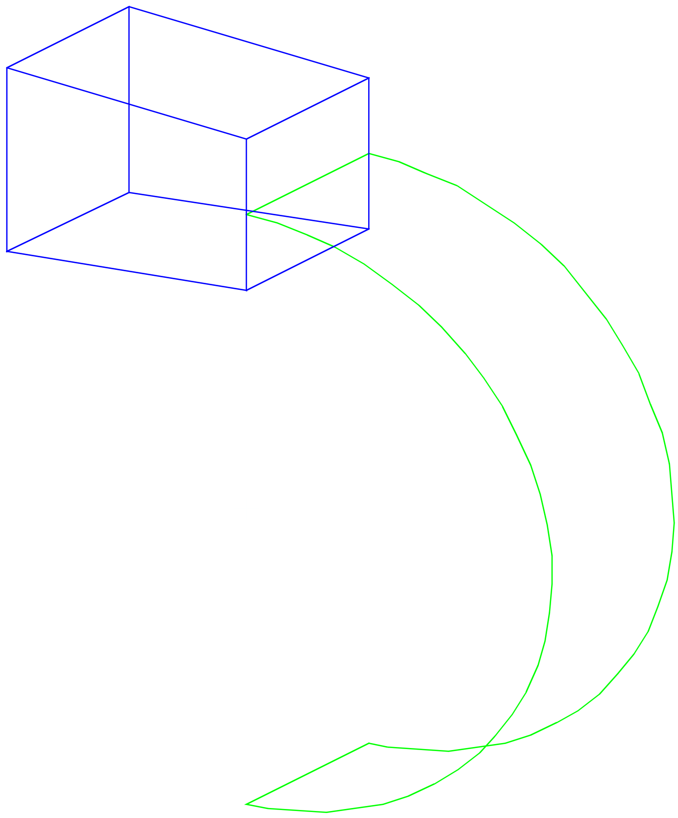
Objectives:

- 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. Through 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.Marc.

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

   Analysis Code: **MSC.Marc**

   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**

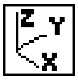
   Method: **XYZ**

   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**

    **Point Size** 

Iso 3 View

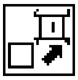
   Action: **Create**

   Object: **Surface**

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

Apply

Click on the **Show Labels** icon.

 **Show Labels**

Notice the distance between the planes goes from Δz = 32 at x = -40 to Δz at x = 0. (You may check this by using **Geometry / Point / Distance** on the appropriate points.)

Now create the solid

Action:	**Create**
Object:	**Solid**
Method:	**Surface**
Starting Surface List	**Surface 1** (bottom surface)
Ending Surface List	**Surface 2** (top surface)

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**

Point 1 List **Point 4** (see fig 9.1)

Point 2 List **Point 8**

Apply

This creates point 9. (See figure 9.1)

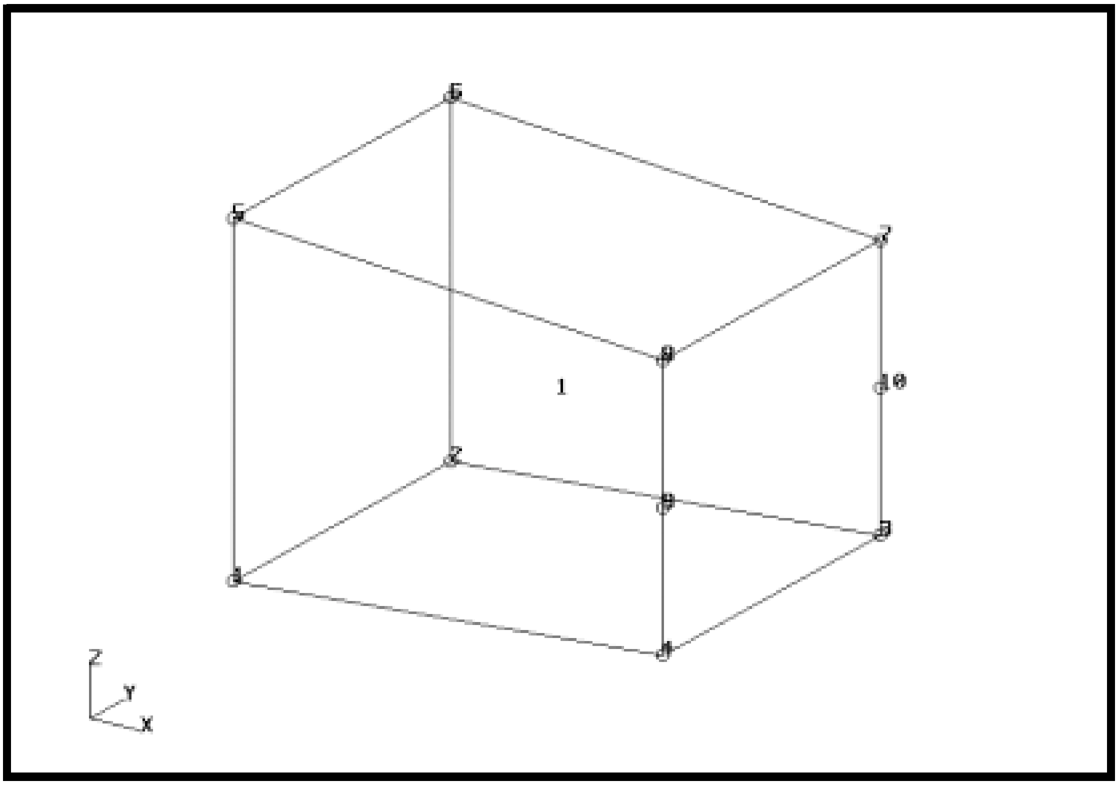
Repeat this procedure with **Points 3** and **7**

Starting Point List **Point 3**

Ending Point List **Point 7**

Apply

Figure 9.1: Make sure to have the Show Labels icon on in order to interpolate between the mentioned points.



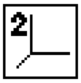
6. Using the two points you just created sweep out the surface

Action: **Create**

Object: **Surface**

Method: **Revolve**

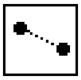
To revolve around the Y-axis, click in the Axis parameter and select this icon then click on the coordinate frame



Coord. Direct. 2

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**.

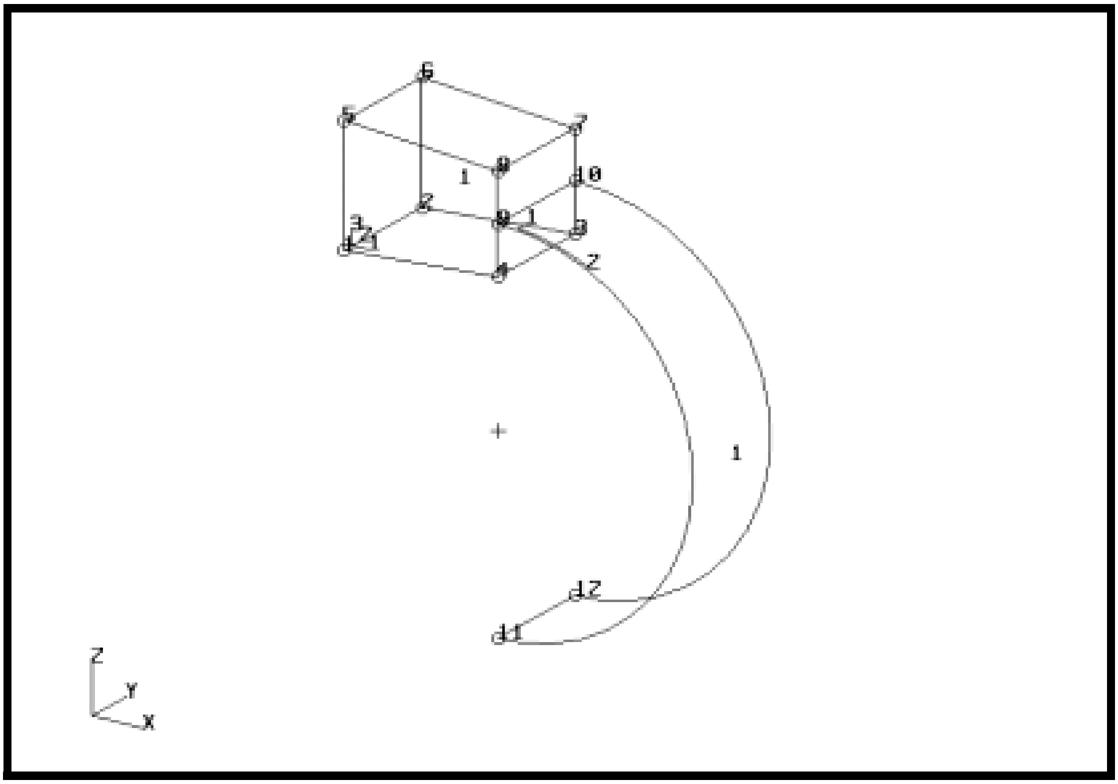


Two Points

Apply

Your model should now appear as in Figure 9.2 below.

Figure 9.2: **The complete geometry should appear as this.**



7. Display the parametric direction

Display/Geometry

■ **Show Parametric Direction**

Apply

Cancel

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

Notice the thickness of the solid at x = 0 is equal to 26, and C2 has a value of 0 at the edge connecting points 9 and 10 and a value of 2 at the face edge connecting points 11 and 12 (See Fig. 9.2). Thus the thickness of the shell varies from 26 where it joins the solid to 2 at the free edge.

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 Ratio	**0.30**

Density	**2.7E-9**
Thermal Expansion Coeff	**2.32E-5**
Reference Temperature	**30**

OK

Apply

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

Cancel

11. Now Mesh the solid

■ **Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Solid**
Elem Shape:	**Hex**
Global Edge Length	**8**
Solid List	**Solid 1**

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

Cancel

13. Now Mesh the surface

■ **Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length	**8**
Surface List	**Surface 1**

Apply

14. Create duplicate nodes on the curve connecting the solid and the shell elements. To view easier, post the surface FEM body.

Group/Post

Select Groups to Post **fem_sur**

Apply

Cancel

Action:	**Transform**
Object:	**Node**
Method:	**Translate**
Direction Vector:	**<0 0 0>**
Node List:	*click on nodes connecting shell and solid elements,* **Node 66:90:6**

Apply

15. Verify there are no additional free edges and the normals to the surface point outwards. First post all the groups.

Group/Post

Select All

Apply

Cancel

Action:	**Verify**
Object:	**Element**
Test:	**Boundaries**

● **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:	**Element**
Type:	**Normals**

● **Draw Normal Vectors**

Apply

If the elements point inward then perform this step. (You might want to just post the necessary **fem_sur** group for a better view of the elements.)

Action:	**Modify**
Object:	**Element**
Method:	**Reverse**
Element List	Select all Surface Elements, **Elm 81:160**

Apply

16. Create a group named **fem_all**

Group/Create

New Group Name	**fem_all**

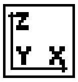
■ **Make Current**

Groups Contents | **Add All FEM**

Apply

Cancel

17. Adjust the viewport. It may be more convenient for screen selecting to switch to the **bottom view**

 **Bottom View**

Viewing/Transformations...

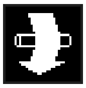
Options...

Rotation increment (deg): | **10**

● **Model Relative**

OK

Click once right X rotation.

 **Right X Rotation**

Click once left Z rotation.:

 **Left Z Rotation**

OK

Using View Corners, the viewport should look like Fig 9.3.:

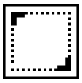
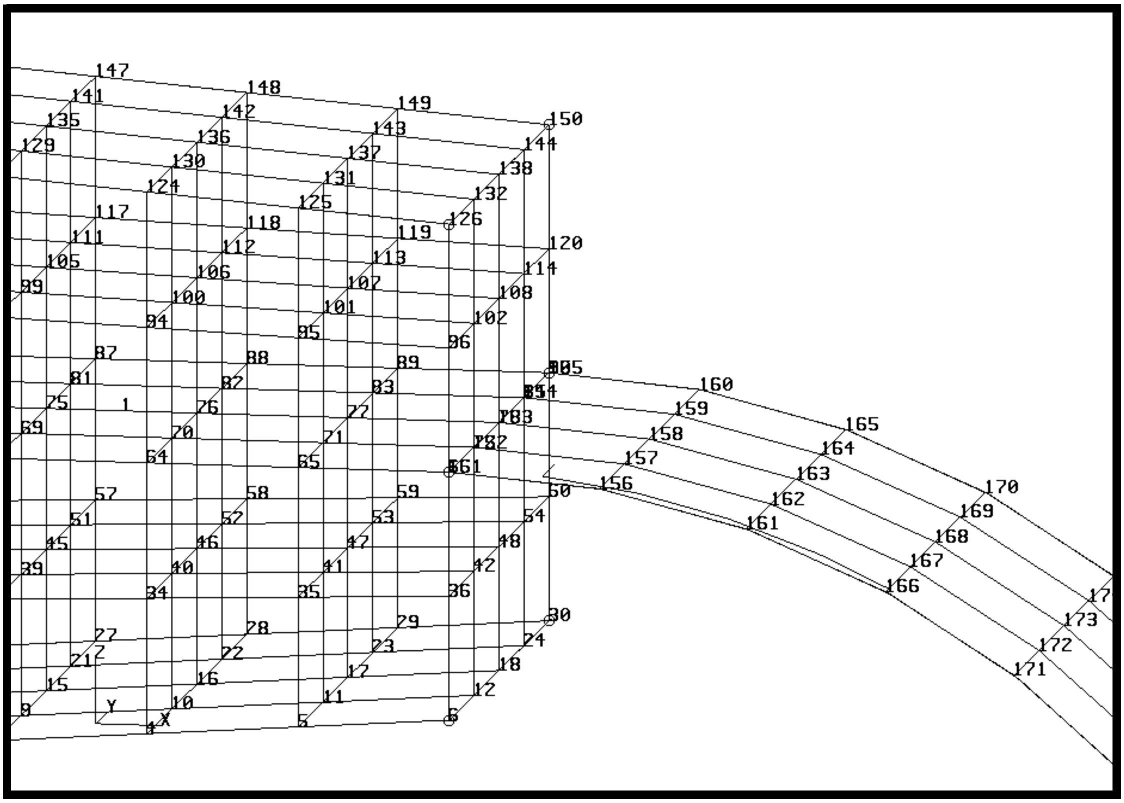
 **View Corners**

Figure 9.3: Zoom in on the specified area.



Preferences/Graphics..

Automatic View Settings:	❐ **Auto Extend (Off)**

Apply

Cancel

Group/Post...

Select Groups to Post:	**fem_sol**

Apply

Cancel

18. Now create the **MPCs**

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

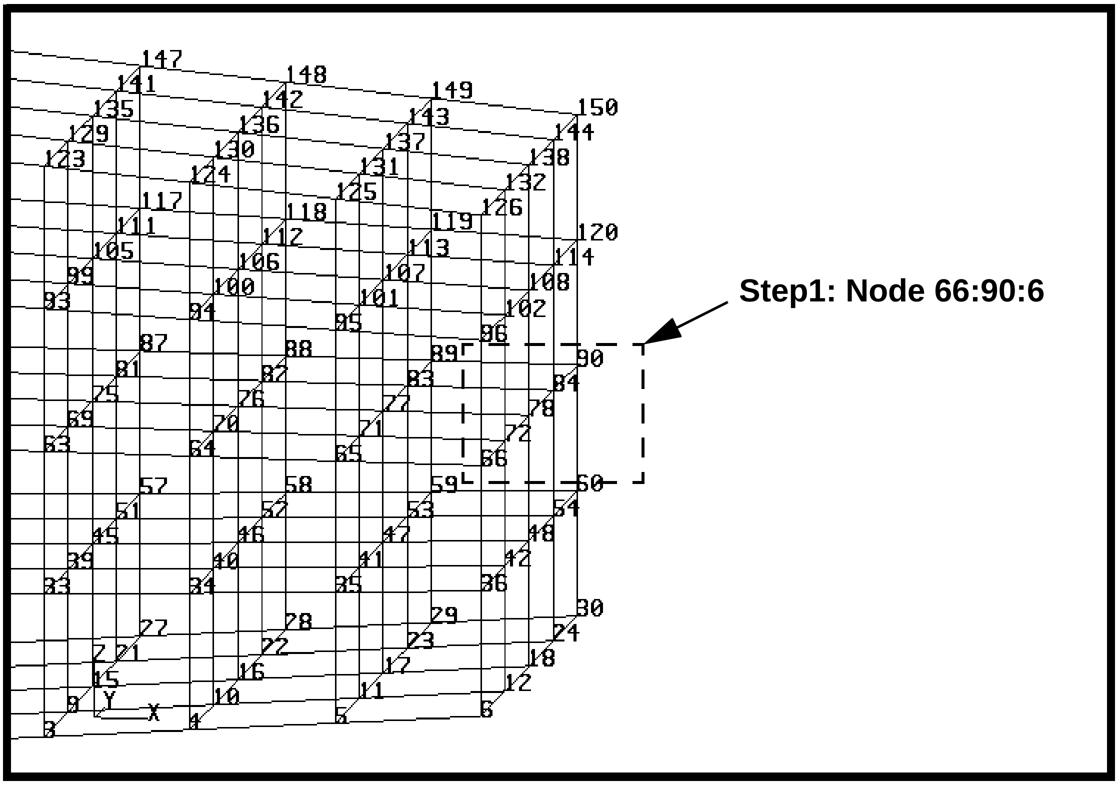
Type: **Rigid Link**

Define Terms ...

● **Create Dependent**

Using the mouse, select the 5 nodes along the center row of the right face by creating a rectangle around the nodes as shown in the figure below.

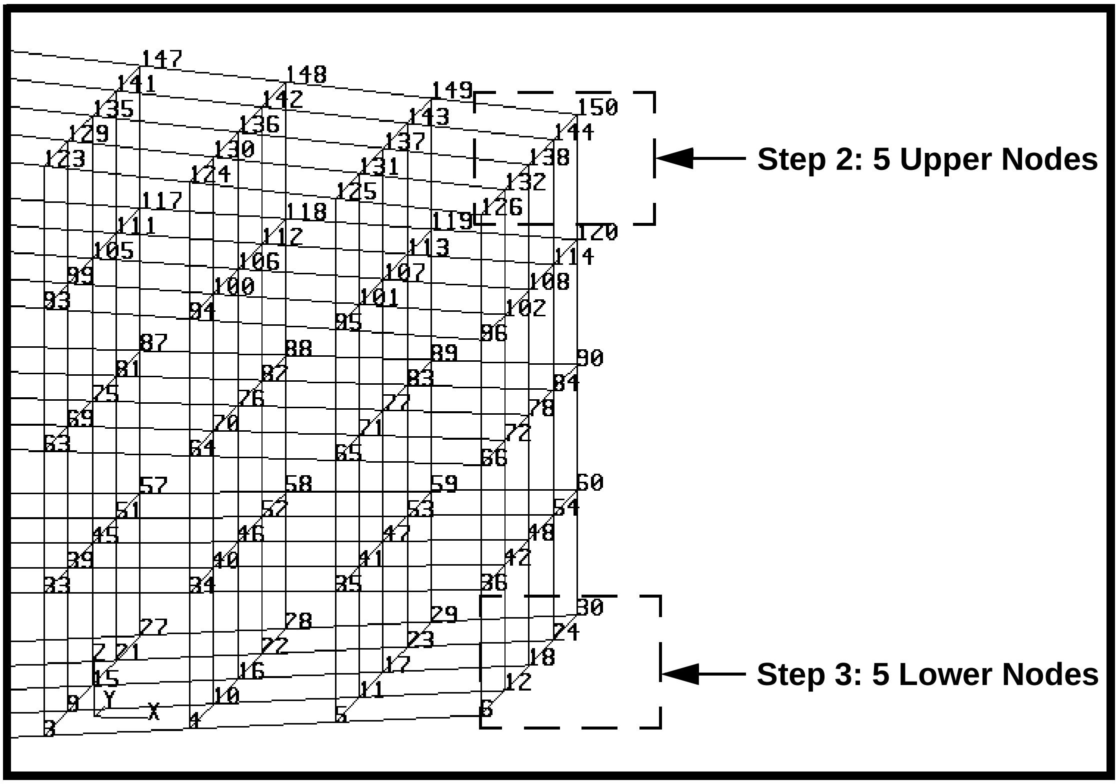
Figure 9.4: Select the given row to be dependent nodes on this MPC.



Notice how the nodes are added to the **Dependent Terms (1)** panel automatically because the **Auto Execute** is ON. Also, the **Create Independent** switch is turned ON automatically without clicking it again in the panel.

Make a rectangle selection of the upper row of nodes, then select the lower row of nodes by creating a rectangle around them as well.

Figure 9.5: Node picking for steps 2 and 3 are shown.



Notice the upper 5 nodes are listed in the first row in the **Independent Terms (2)** panel and the lower 5 nodes are listed on a secound row in the same panel. There is a vertical correspondence between the nodes in the three lists.

Cancel	(Click in the Define Terms form)
Apply	(Click in the Finite Element form)

This action creates the multiple (5 in 1) MPC.

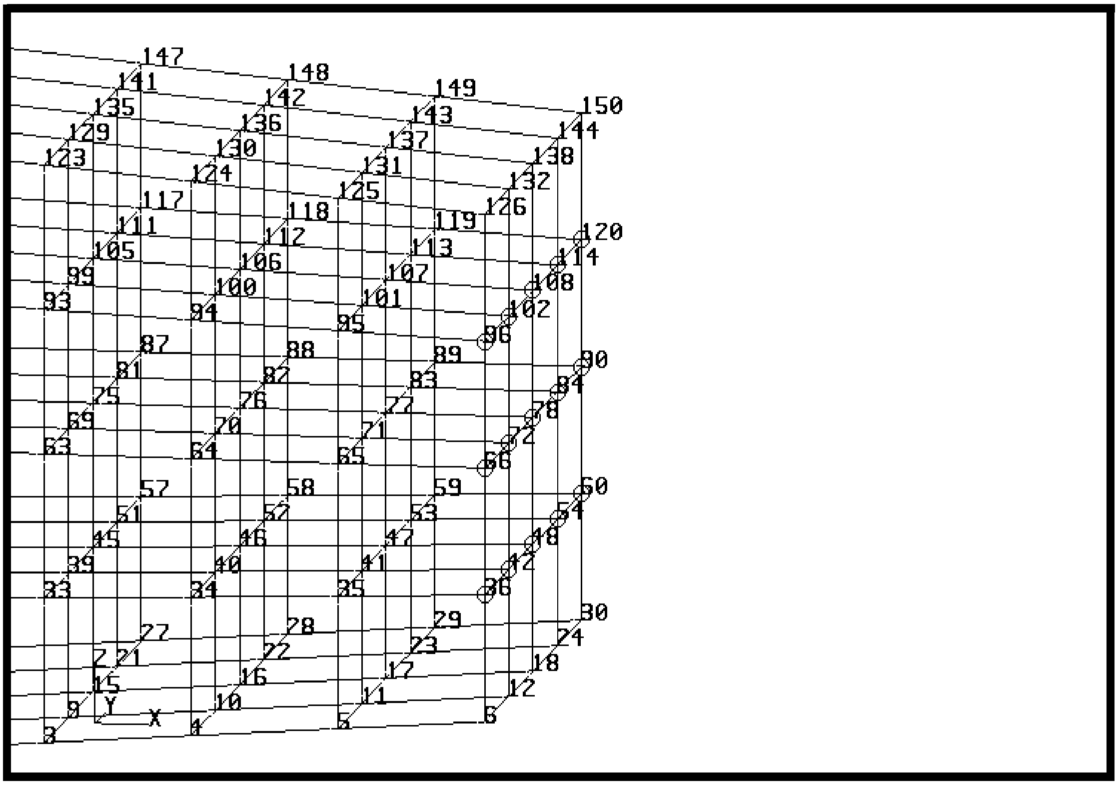
Inspect the markers in the viewport. The dependent nodes are marked with a circle. What this does is to assume the dependent nodes will stay on the straight line correcting the corresponding upper and lower, independent nodes, even as the later were around. That is the reason why this MPC is called a Rigid Link.

Do the same with the second row of nodes from the top (nodes 96:120:6) as dependent nodes, using the same independent nodes as before. (Click **Define Terms**, rectangle pick the second row, rectangle pick the upper row, and then the lower row, click **Apply** on the **Finite Element** form.)

Repeat the process with the second row from the bottom (nodes 36:60:6).

The veiwport should look like the following.

Figure 9.6: Currently, your model should appear as such.



With these you have ensured that the entire cross section (right-front face) moves so that every point stays on a straight line connecting the upper and lower edge of the face.

The intent is to provide a consistent connection with the shell that hangs from this section, which has the same thickness as the height of the face of the solid. The solid and shell are physically a single piece. We chose to represent one position with 3D elements and the other partial with 2D/ Shell elements to save computational efforts. (In other cases we my need this to account for thin geometries.)

19. Now connect the shell to the solid.

Action:	**Create**
Object:	**MPC**
Method:	**Axi Shell-Solid**

Define Terms...

To save yourself from unnecessary clicking and to practice a second way of doing things, turn the **Auto Execute** off in the **Define Terms (MPC)** form.

- **Create Independent**

Rectangle pick the center nodes as before (node 66:90:6). Notice this time the nodes were not sent to the **Independent Terms** panel above.

Apply	(Click in the Define Terms form)

The dependent nodes are Shell nodes, so perform the following:

Group/Post...

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

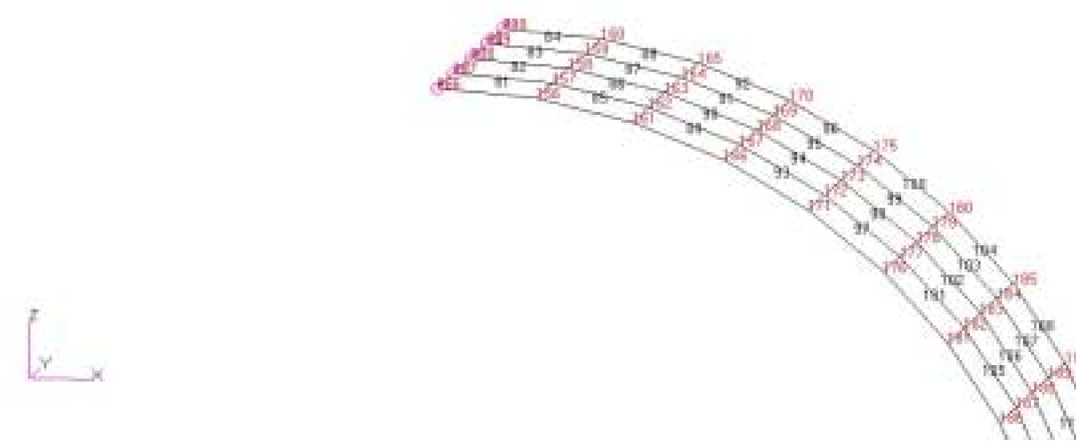
The **Create Dependent** option will automatically be selected.

Holding the SHIFT key, select the 5 Shell nodes at the end of the surface, **Node 151:155**.

Apply	(Click in the Define Terms form)
Cancel	(Click in the Define Terms form)
Apply	(Click in the Finite Element form)

The viewport should look like the following figure.

Figure 9.7: Axi Shell-Solid MPC on surface



The Axi Shell-Solid MPC as seen added together with the nodes on the solid that they are connected to. This MPC ensures that the shell nodes stay in the center of the straight line connecting the upper and lower nodes of the solid on each column. Also, the rotation of the Shell node is the same as the rotation of the straight line connecting the Independent (solid) nodes. Therefore the cross section of the shell stays with the face of the solid. Notice each column may rotate differently in consistancy with Shell Theory.

20. Apply the properties to the model.

Post the related group, then apply the properties to the solid.

Group/Post...

Select Groups to Post: **default_group**

Apply

Cancel

■ **Properties**

Action: **Create**

Object: **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**

Object: **2D**

Type: **Thick Shell**

Property Set Name: **sur**

Input Properties ...

Material Name **m:aluminium_iso_SI_mm**

Shell Thickness **f:thick**

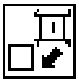
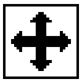
OK

Select Members: **Surface 1**

Add

Apply

Hide the labels and fit the view by using the **Hide Labels** icon and **Fit View** icon.

 **Hide Labels**  **Fit View**

Group/Post...

Select Groups to Post: **fem_all**

Apply

Cancel

View the Surface Thickness.

Action: **Show**

Existing Properties **Thickness**

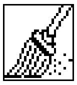
Display Method **Scalar Plot**

● **All Groups**

Group Filter **fem_all**

Apply

Reset the graphics.

 **Reset Graphics**

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

Group/Post...

Select Groups to Post **default_group**

Apply

Cancel

■ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **fix**

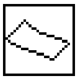
Input Data...

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** (left back face of solid)

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...

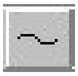
Translation | **<-20, 0, 0>**

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**

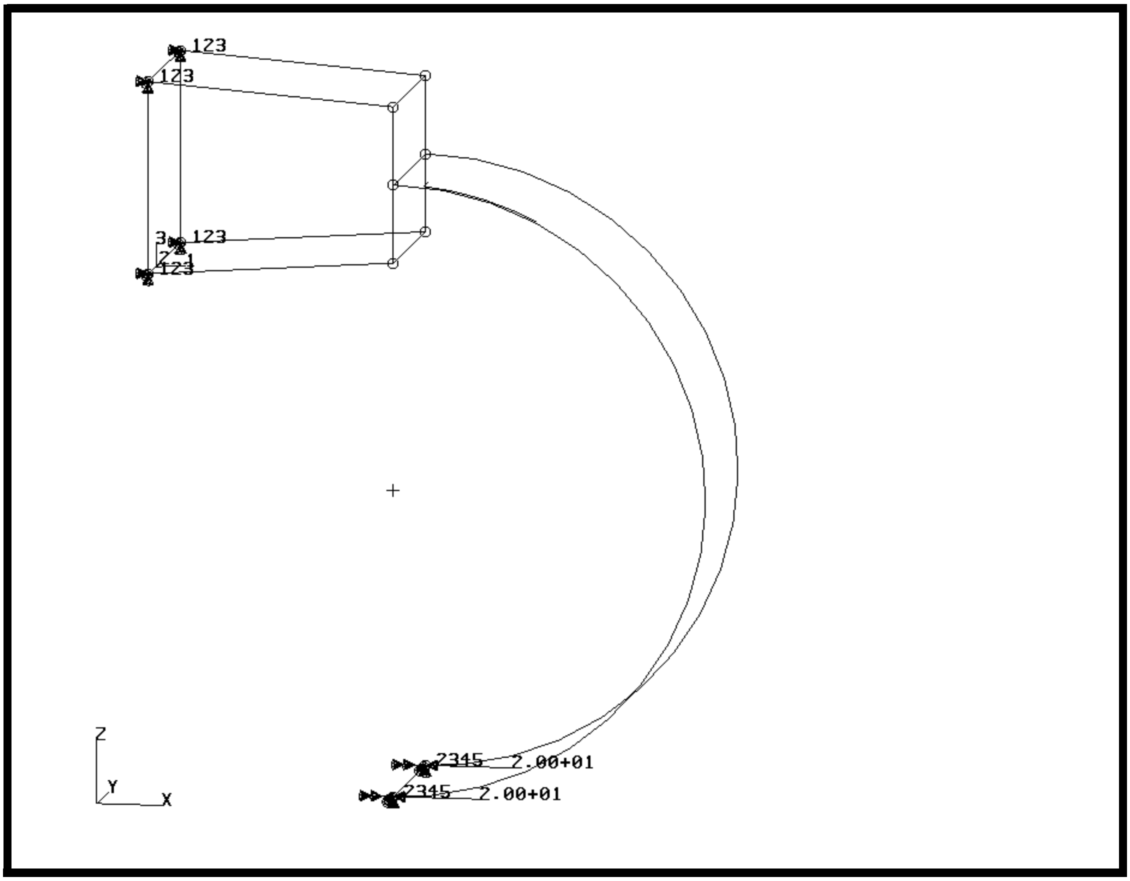
Add

OK

Apply

Your Model should appear as below.

Figure 9.8: The current model with nodal displacement.



22. Submit the model for analysis.

■ **Analysis**

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

Load Step Creation...

Job Step Name:	**pull1**

Solution Type	**Static**

Apply

Cancel

Load Step Selection...

Selected Job Steps:	**pull1**

OK

Apply

When the job is done there will be a results file titled **pull.t16** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at ***pull.log*** and ***pull.sts*** as well as using the ***pull.out*** file or the **Analysis** option, **Monitor**.

Action:	**Monitor**
Object:	**Job**

View Status File...

A successful run should give the job exit number: **3004.**

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

◼ **Analysis**

Action:	**Read Results**
Object:	**Results Entities**
Method:	**Attach**

Select Results File ...

Available Files:	**pull1.t16**

OK

Apply

24. Adjust the viewport to post the default and fem_all groups.

Group/Post...

Select Groups to Post	**default_group** **fem_all**

Apply

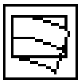
Cancel

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

■ **Results**

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

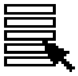
Before viewing the stresses change the **Deformation Attributes**

 **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**

Select Results Case:	**pull1, A1..., Time=1.0**
Select Fringe Results:	**Stress, Global System**

Position...(At Layer 1)

Quantity:	**von Mises**
Selected Deformation Results:	**Displacement, Translation**

Apply

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 3 different layers numbered 1 to 3, the bottom being the lowest number. Lets start with the middle layer.

Select Results Case: **pull1, A1..., Time=1.0**

Select Fringe Results: **Stress, Global System**

Position...(At Layer 2)

Option: **Average**

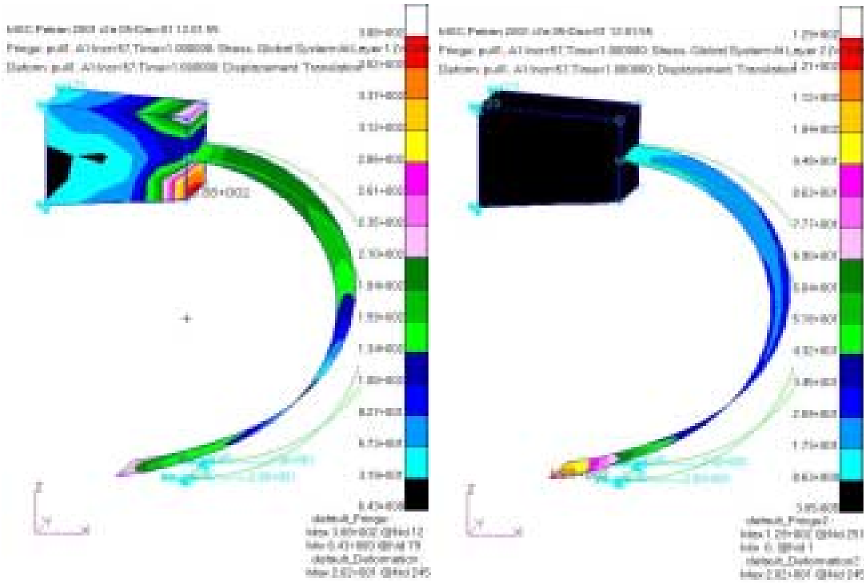
Close

Quantity: **von Mises**

Selected Deformation Result: **Displacement, Translation**

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: **pull1, A1..., Time=1.0**

Select Fringe Results: **Stress, Global System**

Position...(At Layer 3)

Option: **Average**

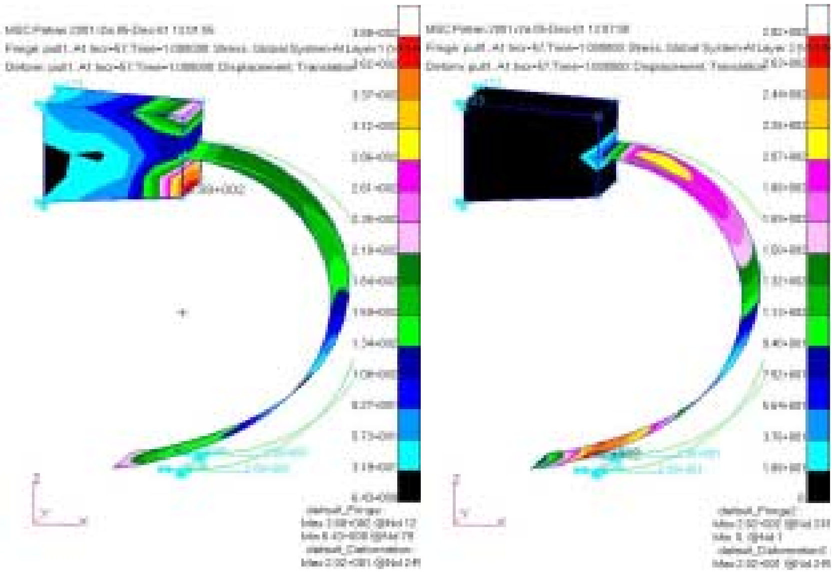
Close

Quantity: **von Mises**

Selected Deformation Result: **Displacement, Translation**

Apply

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



Shell theory in this case overestimated 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.

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

Delete the extra viewport by *first selecting the default viewport window*.

Viewport/Post

Post/Unpost Viewport(s) **default_viewport**

Apply

Cancel

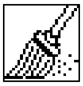
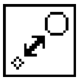
Viewport/Delete

Select a Viewport(s) **another**

Apply

Cancel

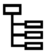
Next, reset the graphics and make the point size small again

 **Reset Graphics**  **Point Size**

Action: **Create**

Object: **Deformation**

Click on the **View Subcases** icon then the **Select Subcases** to bring up the *Select Result Case* form

Select Result Case(s): **pull1**

Filter Method **All**

Filter

Apply

Close

Select Deformation Result **Displacement, Translation**

■ **Animate**

Before hitting Apply select the Animation Options icon

 **Animation Options**

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(s)	all results should still be selected
Select Fringe Result	**Stress, Global System**
Quantity:	**Z Component**

■ **Animate**

Again select the Animation Options icon

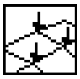
Animation Options

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



Display Attributes

Style	**continuous**

Apply

Another feature on the Animation Control form is Cycle and Bounce. Pause the animation and change it to Bounce.

This ends the exercise. You may close the database and quit PATRAN.