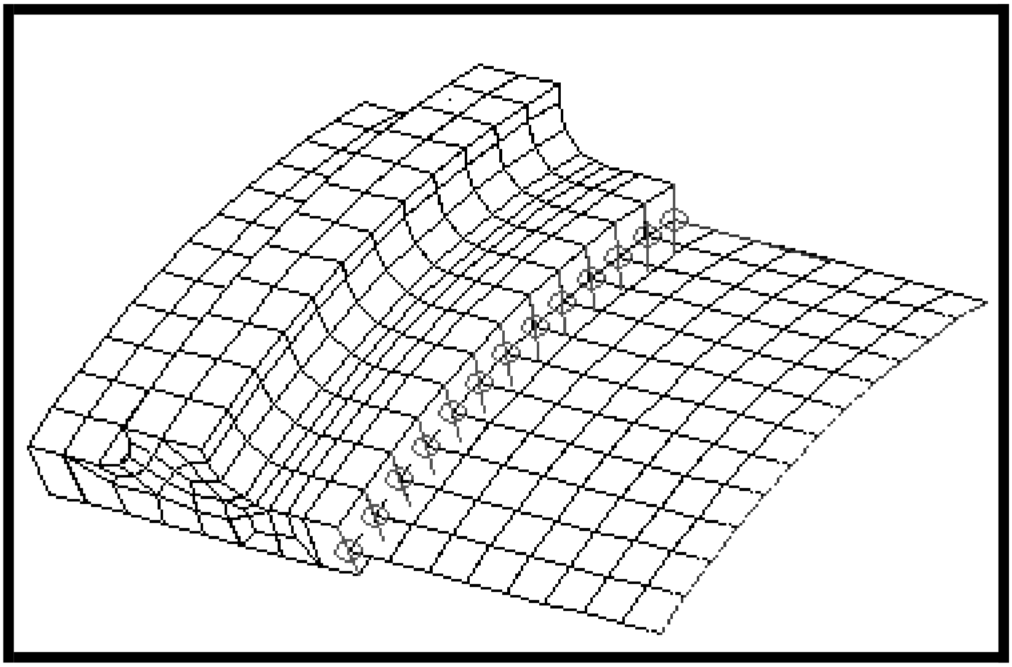
LESSON 10

Using Lists and Multi-Point Constraints



Objectives:

- Use Lists to isolate and group nodes of interest.
- Define MPC’s to convert solid translations into shell rotations.
- Create a cyclic symmetric boundary constraint along the 0 degree and 30 degree faces.

Model Description:

In this exercise you will create MPC's to attach the shell elements in your model to the solids. Shell and solid elements are incompatible. Solid elements provide stiffness in only the three displacement directions for each node, while plate elements in P3/FEA provide stiffness to three displacement directions and two rotations (in-plane rotation is undefined). To connect the two dissimilar elements together requires MPC elements to account for the rotational degrees of freedom defined by the shell element.

The MPC equations you will use will create relationships between the nodes on the shell elements, and those on the upper and lower edges of the solids. You will use Lists to group these nodes.

The MPC equations you will need to use and the justification for each are as follows:

First the translational degrees of freedom are linearly interpolated to obtain the translation at the mid-plane of the solid. The translations of the plate edge is then set equal to that value.

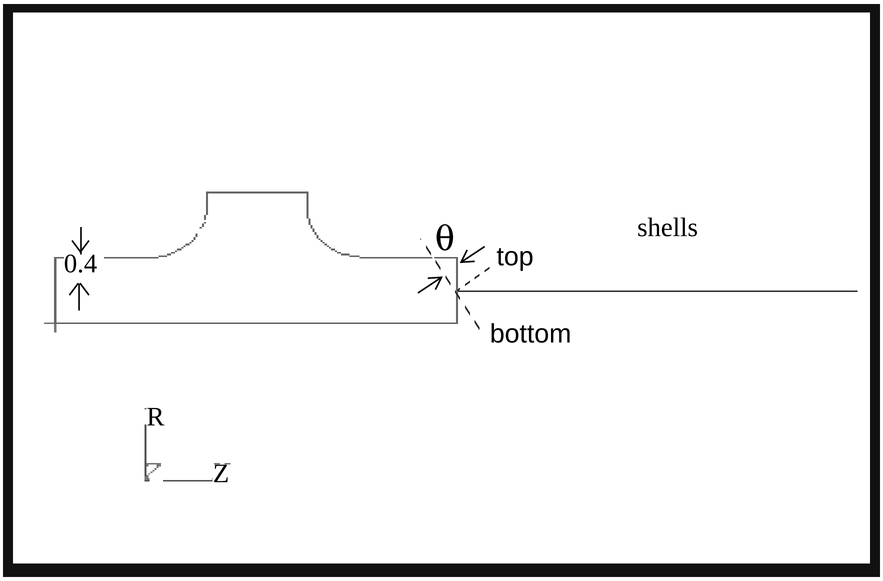


FIGURE 1.

$$U_r^{shell} = 0.5(U_r^{top} + U_r^{bottom}) \qquad \text{(EQ 1)}$$

$$U_{\theta}^{shell} = 0.5(U_{\theta}^{top} + U_{\theta}^{bottom}) \quad \text{(EQ 2)}$$

$$U_{z}^{shell} = 0.5(U_{z}^{top} + U_{z}^{bottom}) \quad \text{(EQ 3)}$$

To account for the rotations of the shell we apply the Kirchhoff-Love assumption which states that planes normal to the neutral plane remain planar during deformation. Figure 1 is a schematic of a deformation at the interface, showing a positive rotation. For small values of θ, $\sin \theta = \theta$, thus:

$$U_{rotr}^{shell} = 0 \quad \text{(EQ 4)}$$

$$U_{rot\theta}^{shell} = \frac{1}{0.4}(U_{z}^{top} - U_{z}^{bottom}) \quad \text{(EQ 5)}$$

$$U_{rotz}^{shell} = \frac{1}{0.4}(-U_{\theta}^{top} + U_{\theta}^{bottom}) \quad \text{(EQ 6)}$$

Suggested Exercise Steps:

- Make 3 lists: one for nodes attached to the edge of the shell elements at the surface to solid interface, one for nodes on the upper edge of the solid elements at that same interface, and one for the nodes on the lower edge of the solid elements.
- Create one MPC to tie together the translational degrees of freedom along the shell to solid interface. The motion of the nodes on the shell is dependent on the motion of the nodes on the solid.
- Create additional MPC's to assign rotation to the plate elements by coupling solid translations.
- Define a cyclic symmetry boundary constraints along the 0^{o} and 30^{o} faces of the model. These constraints are applied using the cylindrical coordinate system.

Files:

All the files used in this exercise are listed below. Each listing includes the file, where it originated, its format (text/binary) and summary information as to how it relates to this exercise.

File	Supplied/Created	Description
mpc.db	Created in ex2	This is a PATRAN database (binary) created in Exercise 2. The geometry for the model was created in Exercise 2. The mesh for the model was generated in Exercise 6. Finally, multi-point constraints will be created in Exercise 7.

Exercise Procedure:

1. Open database **mpc.db.**

2. Create a list of nodes along the edge of the shell elements at the solid to surface interface.

Creating Lists for interface nodes

Click on **Tools** in the *Control Panel*, select **List/ Create...** from the pull-down menu and then select the following:

Tools/List/Create...

Model:	**FEM**
Object:	**Node**
Method:	**Attribute**

The nodes along the edge of the shell are at a radius of 6.2 units and an axial location of 2 units. To group the nodes that share these coordinates, change the *Refer. Coordinate Frame* to the cylindrical coordinate frame in your model and search on the above listed coordinate values.

Attribute	**Coord Value**
Refer. Coordinate Frame	**Coord 1**
■ **R**	
■ **Z**	
R	**6.2**

Z **2**

Target List ◆ **“A”**

Apply

Creating Lists for Bottom Nodes

3. Create a list to reference nodes on the bottom of the solid.

Target List ◆ **“B”**

R **6.0**

Z **2.0**

Apply

You now have two lists but we need to use three to define the MPC’s. To create the third list, we need a list ‘c’.

Creating Lists for Top Nodes

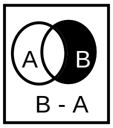
4. Create a list to reference nodes on the top of the solid.

Click on **Tools/ List** in the *Control Panel* and select **Boolean...** from the pull-down menu.

Tools/ List/Boolean...

You will make a list c that duplicates the current contents of list b. You will use list c when you need to reference nodes on the bottom of the solids. You will create a new list b referencing the nodes on the top edge of the solid for your third list. You will use list b when you need to reference nodes on the top of the solids.

In order to move the contents of the current list b into list c, on the *Boolean List* form, click on the icon for **B-A**.



At this point, ‘*listc*‘ contents and ‘*listb*‘ contents should be identical: the nodes on the bottom of the solids.

On the *List B* form, click **Clear.**

Clear

On the *Create List* form, enter the following to create list b:

R	**6.4**
Z	**2.0**

Apply

You now have 3 lists: list 'a' references the nodes at the solid/shell edge of the shell elements, list b references the nodes at the top edge of the solids along the interface, and list 'c', the nodes along the bottom edge of the solids.

Creating MPC's

5. Create a MPC to tie together the translational degrees of freedom along the shell to solid interface.

The first MPC you will create will tie together the translational degrees of freedom along the shell to solid interface as shown in Equation 1 through Equation 3 on page -4 . The dependent nodes will be those in list a, which are associated with the quad elements.

◆ **Finite Elements**

Action:	**Create**
Object:	**MPC**
Method:	**Explicit**
Constant Term	**0**

Click on **Define Terms...** and a *Define Terms* Menu will show up on the screen. Enter the following (remember to enclose PCL variables in *backquote* (i.e. " **'** ") as opposed to single quote (" **'** ") or double quotes (" **"** ") when entering the *Node List*):

Define Terms

◆ **Create Dependent**

Node List	**'lista'**
DOFs	**UX** **UY** **UZ**

Apply

Next, you will create the independent terms, which will reference the translations of the nodes on the solids.

◆ **Create Independent**

Coefficient	**0.5**
Node List	**'listb'**
DOFs	**UX** **UY** **UZ**

Apply

Finally, edit the *Node List* databox as follow:

◆ **Create Independent**

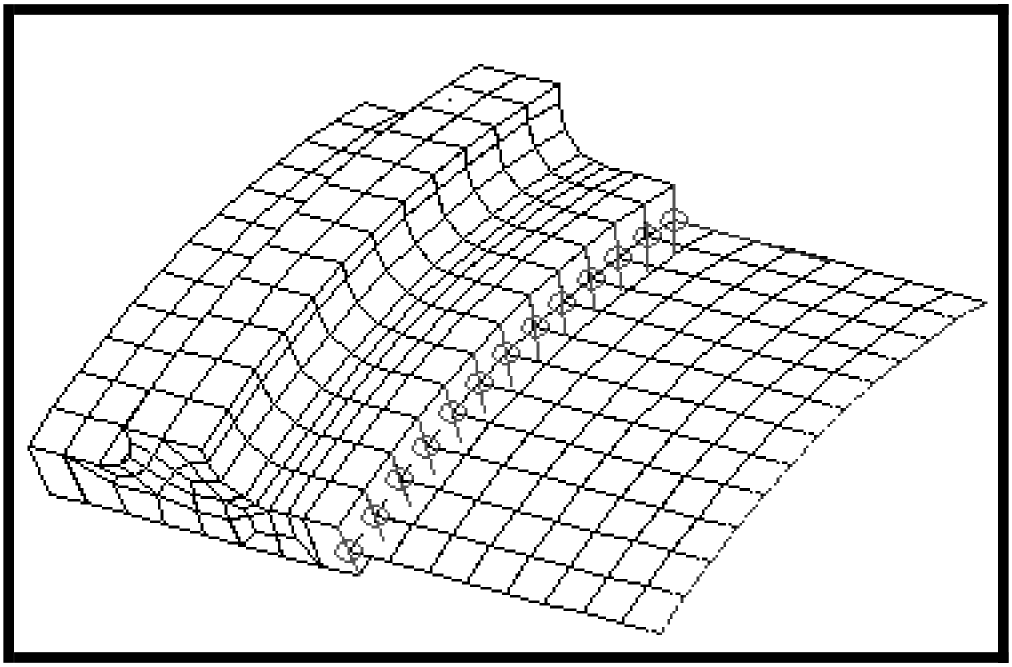
Coefficient	**0.5**
Node List	**'listc'**
DOFs	**UX** **UY** **UZ**

Apply

Now you have defined all the terms to relate the translations on the solids to the shell. To create the MPC, click **Apply** on the *Finite Elements* form.

Apply

Display your model in **Element Fill** style and it should appear as shown below.



The dependent nodes are circled, and lines are displayed which connect to each independent node.

Reset the *Render Style* back to **Wireframe**.

6. Next create an MPC to relate translations on the solids in the axial direction to rotations on the shells using Equation 5 on page 4 .

$$U_{rot\theta}^{shell} = \frac{1}{0.4}(U_z^{top} - U_z^{bottom})$$

On the *Define Terms* form, apply the following:

First, for the dependent term,

◆ **Create Dependent**

Node List	**‘lista‘**
DOFs	**RY**

Apply

Now, for the independent terms,

◆ **Create Independent**

Coefficient	**‘1/.4‘**
Node List	**‘listb‘**
DOFs	**UZ**

Apply

and

Coefficient	**‘-1/.4‘**
Node List	**‘listc‘**
DOFs	**UZ**

Apply

Now you have defined all the terms, click **Apply** on the *Finite Elements* form.

Apply

7. Create the final MPC using Equation 6 on page 4 .

$$U_{rotz}^{shell} = \frac{1}{0.4}(-U_{\theta}^{top} + U_{\theta}^{bottom})$$

First, for the dependent term,

◆ **Create Dependent**

Node List	**‘lista‘**
DOFs	**RZ**

Apply

Now, for the independent terms,

◆ **Create Independent**

Coefficient	**‘-1/.4‘**
Node List	**‘listb‘**
DOFs	**UY**

Apply

and

Coefficient	**‘1/.4‘**
Node List	**‘listc‘**
DOFs	**UY**

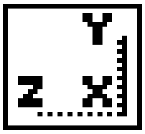
Apply

Now you have defined all the terms, click **Apply** on the **Finite Elements** form.

Apply

8. Change the view of your model as follows:

Select the side view icon from the toolbar



Construct Constraint in Theta Direction

9. Finally, construct the axisymmetric constraint. These constraints will be applied to the R-Z faces at Theta = 0 degrees and Theta = 30 degrees. In order to properly create a symmetric boundary condition, constraint of one displacement and the other two rotations.

First, construct the translation constraint in theta direction as follows:

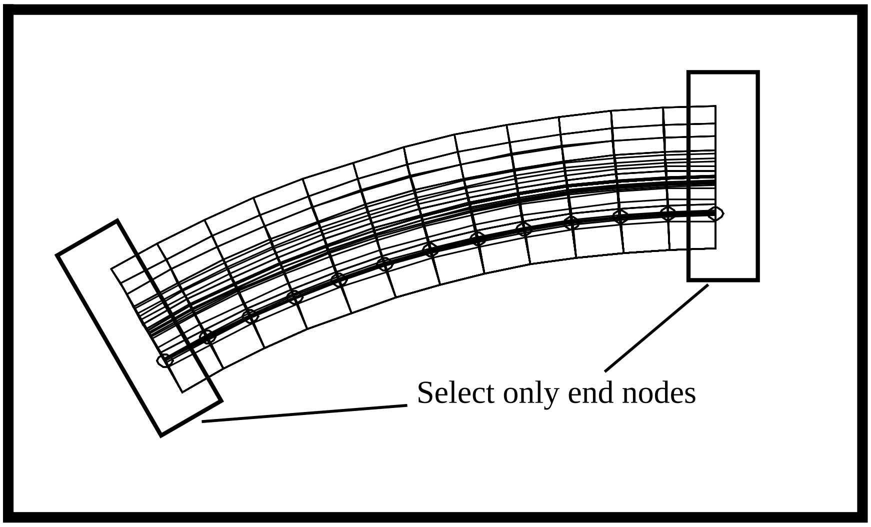
◆ **Load/BCs**

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

Click on **Input Data** to open up another menu and then enter the following:

Input Data	
Translations <T1 T2 T3>	**< ,0, >**
Rotations<R1 R2 R3>	**<0, ,0>**
Analysis Coordinate Frame	**Coord 1**
OK	

Click on **Select Application Region** to open up another menu and then select the nodes as shown below:



(You may want to use the polygonal click method to select the nodes at the left end.)

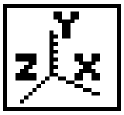
Select Application Region	
Geometry Filter	**FEM**
Select Nodes	**Node 1:52 54 56 57 69 169:176 684:729** **(The node entities may not be the same on your model)**
Add	
OK	

Press **Apply** in the *Load/BCs* form.

Apply

Change the view of your model as follows:.

Click on the **Iso 1 View** icon in the toolbar



To get a better view of the boundary conditions on the model, change the display as follows:

Display/Finite Elements ...

◆ **Free Edges**

Apply

Cancel

and

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

Vectors/Filters...

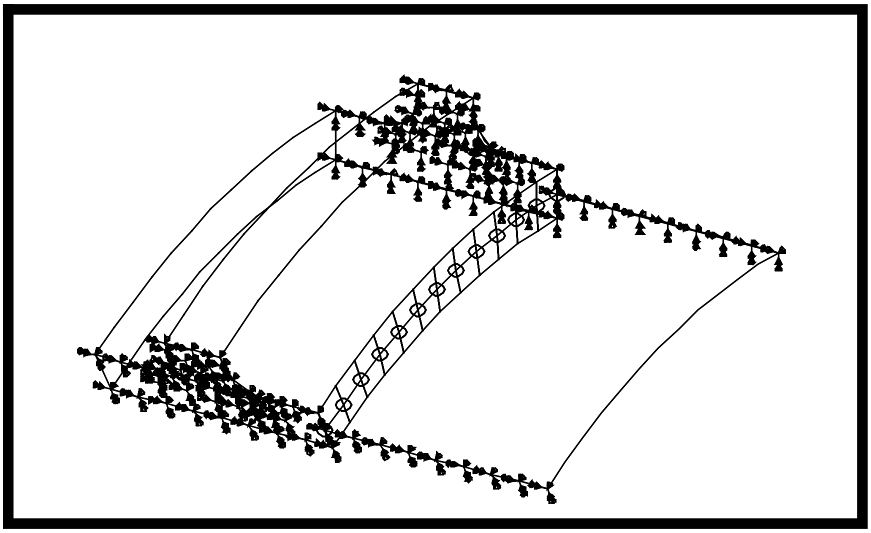
❒ **Show Result Values**

❒ **Show LBC/El. Prop Values**

Apply

Cancel

Your model should appear as shown below.



Construct Constraint in Z-Direction

Secondly, construct the translation constraint in z-direction to restrain the model from freely sliding in the axial direction as follows:

◆ **Load/BCs**

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

Click on **Input Data** to open up another menu and then enter the following:

Input Data

Translations <T1 T2 T3>	**< , ,0>**
Rotations <R1 R2 R3>	**< , , >**
Analysis Coordinate Frame	**Coord 1**

OK

Select Application Region

Geometry Filter	**FEM**

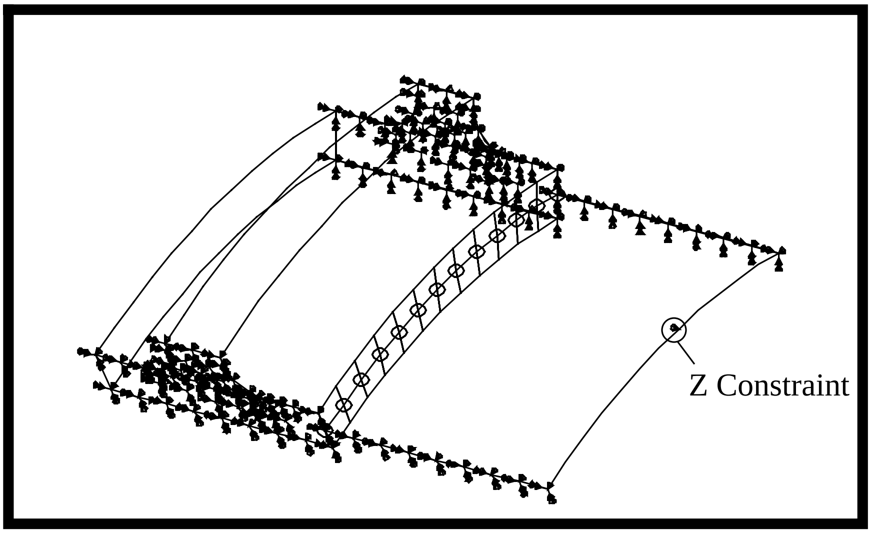
Select Nodes | **Node 66**
(Pick any node on the boundary of the model)

Add

OK

Apply

Your model should appear as follows:



10. Close the database and quit PATRAN to complete this exercise.

File/Quit