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`

   ![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 Code: MSC/ADVANCED_FEA]

   Analysis Type: `Structural`

   ![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`

   ![Action: Create]

   Object: `Surface`

   ![Object: Surface]

   Method: `XYZ`

   ![Method: XYZ]

   Vector Coordinate List: `<40, 30, 3>`

   ![Vector Coordinate List: <40, 30, 3>]

   Origin Coordinate List: `[-40, 0, 34.93]`

   ![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**

   ![Isoview 3]

   Action: `Create`

   ![Action: Create]

   Object: `Surface`

   ![Object: Surface]
**Modeling a Shell to a Solid Element**

### Method:
- **XYZ**
- **<40, 30, -3>**
- **[-40, 0, 66.93]**

#### Apply

Now create the solid

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Solid</td>
</tr>
<tr>
<td>Method:</td>
<td>Surface</td>
</tr>
<tr>
<td>Starting Surface List</td>
<td>Surface 1</td>
</tr>
<tr>
<td>Ending Surface List</td>
<td>Surface 2</td>
</tr>
</tbody>
</table>

#### Apply

4. Delete the original surfaces

<table>
<thead>
<tr>
<th>Action:</th>
<th>Delete</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Surface</td>
</tr>
<tr>
<td>Surface List:</td>
<td>Surface 1 2</td>
</tr>
</tbody>
</table>

#### Apply

5. Create two points that will define the surface later

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Point</td>
</tr>
<tr>
<td>Method:</td>
<td>Interpolate</td>
</tr>
<tr>
<td>Starting Point List</td>
<td>Point 4 (see fig -1)</td>
</tr>
<tr>
<td>Ending Point List</td>
<td>Point 8</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Starting Point List</th>
<th>Point 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ending Point List</td>
<td>Point 7</td>
</tr>
<tr>
<td><strong>Apply</strong></td>
<td></td>
</tr>
</tbody>
</table>

**Figure 6-1**

![Diagram of a swept surface]

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

<table>
<thead>
<tr>
<th><strong>Action:</strong></th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Object:</strong></td>
<td>Surface</td>
</tr>
<tr>
<td><strong>Method:</strong></td>
<td>Revolve</td>
</tr>
</tbody>
</table>
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

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
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
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.

<table>
<thead>
<tr>
<th>Action:</th>
<th>Verify</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Elements</td>
</tr>
<tr>
<td>Type:</td>
<td>Normals</td>
</tr>
</tbody>
</table>

**◆ Draw Normal Vectors**

*Apply*

If the elements point inward then perform this step

<table>
<thead>
<tr>
<th>Action:</th>
<th>Modify</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Elements</td>
</tr>
<tr>
<td>Type:</td>
<td>Reverse</td>
</tr>
<tr>
<td>Element List</td>
<td>Select all Surface Elements</td>
</tr>
</tbody>
</table>

*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

```
Y X Z
```

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>MPC</td>
</tr>
</tbody>
</table>
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.

Define Terms ...

■ Create Dependent

Node List

Node 151:155

■ Create Independent

Node List

Node 126:150:6

Cancel

Node List

Node 6:30:6

Apply

Apply

Figure 6-2
17. Post the fem_sol group and create a Slider MPC

Action: 
Object: 
Type: 

Define Terms ...
- Create Dependent
  Node List
  Apply
- Create Independent
  Node List
  Apply
  Node List
  Apply
  Cancel
  Apply

Repeat this procedure twice more changing only the Dependent Node List to Node 66:90:6 and 36:60:6
Change the view back to **Iso_view 3**

![Diagram of a grid with labeled nodes]

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
Now the Surface

Action: Create

Dimension: 2D

Type: Shell

Property Set Name: sur

Material Name: m:aluminium_iso_SI_mm

Shell Thickness: f:thick

If you would like to view the surface thickness

Action: Show

Existing Properties: Shell Thickness

Display Method: Scalar Plot

Group Filter: fem_all

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

◆Loads/BCs
In order to select the appropriate solid faces, use the following entity select icon:

![Surface or Face Icon]

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

Select Geometric Entities: Solid 1.1 (opposite the surface)

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:

* Add
* OK
* Apply

Your Model should appear as below

20. Submit the model for analysis.

◆ Analysis

Action:

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

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

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*  

**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. Let's 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 let's look at the stress at the top of the model.

Select Results Case: \[\text{Step1, Total Time=1}\]
Select Fringe Results: \[\text{Stress, Components}\]
Position...((At SECTION_POINT_5))
Option: \[\text{Average}\]
Close
Quantity: \[\text{Von Mises}\]
Selected Deformation Results: \[\text{Deformation, Displacement}\]
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
Before hitting Apply select the Animation Options icon

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

Again select the Animation Options icon

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

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

This ends the exercise, you may quit Patran.