

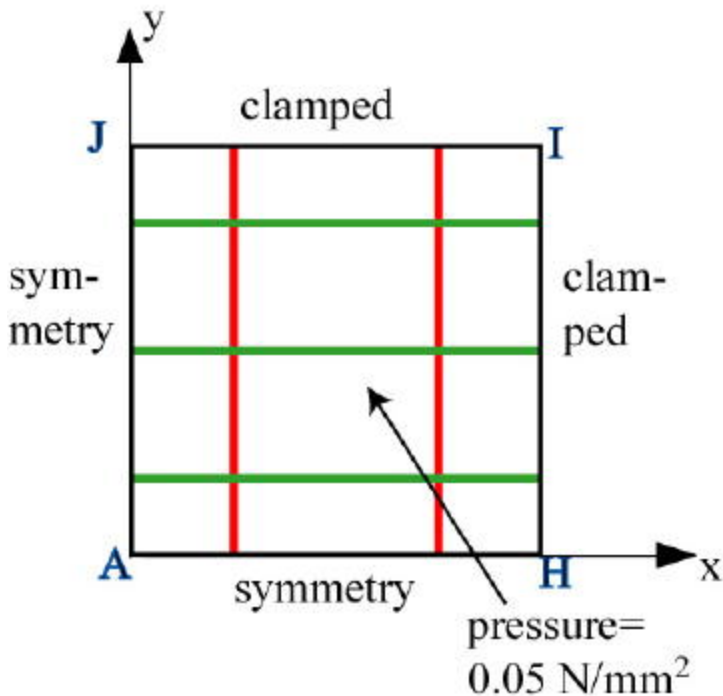
ANSYS - Semi-monocoque shell - Step 6

Problem Specification

1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
- 6. Specify boundary conditions**
7. Solve!
8. Postprocess the results
9. Validate the results

Step 6: Specify boundary conditions

The boundary conditions given in the [problem specification](#) are summarized in the schematic below. Keep in mind that the edge conditions need to be applied to the plate as well as the stiffeners.



Apply Symmetry along AH

We'll apply this BC in two steps:

1. Select edges along AH
2. Apply symmetry condition to the selected edges

Plot areas: **Utility Menu > Plot > Areas**

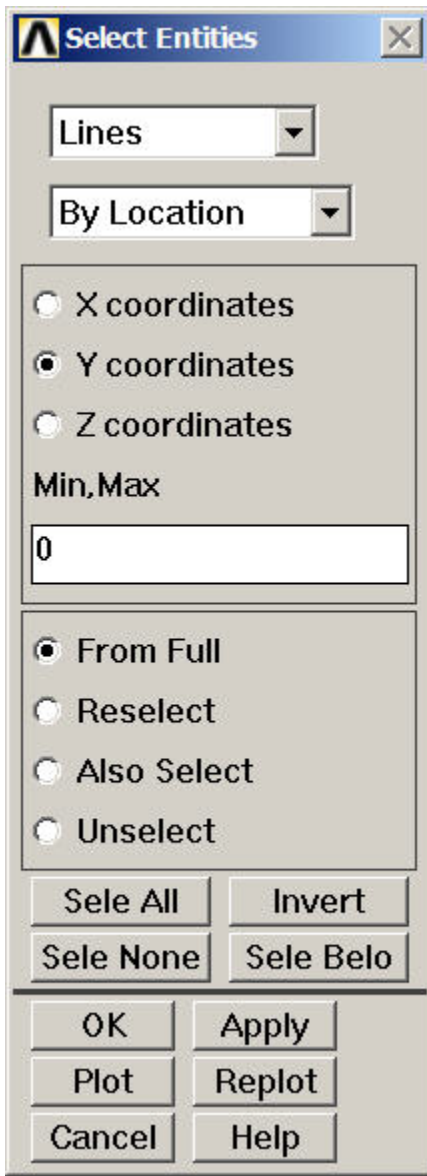
Adjust the display: Click on the *Isometric View* and *Fit View* icons in the rightmost part of the GUI.

Utility Menu > PlotCtrls > Numbering: Turn *off* area and node numbering; turn *on* line numbering.

Select all edges along AH: **Utility Menu > Select > Entities**

We are going to continually use the *Select Entities* menu to apply the BC's. So resize and rearrange the windows slightly so that you can access this menu, the ANSYS GUI, and the tutorial simultaneously.

Select Entities menu: Select **Lines** from the pull-down menu at the top. Select below that. Choose **By Location Y coordinates**. Under **Min,Max**, enter 0. This will select all lines whose **centers** lie at $y=0$. Make sure **From Full** is selected so that we are selecting entities from the *full* model. Click **Apply**.



The image shows a 'Select Entities' dialog box with a blue title bar and a close button. It contains two dropdown menus: 'Lines' and 'By Location'. Below these are three radio buttons for 'X coordinates', 'Y coordinates' (which is selected), and 'Z coordinates'. A text field labeled 'Min,Max' contains the value '0'. Another set of radio buttons includes 'From Full' (selected), 'Reselect', 'Also Select', and 'Unselect'. At the bottom, there are two rows of buttons: 'Sele All' and 'Invert' in the first row, 'Sele None' and 'Sele Belo' in the second row, and 'OK', 'Apply', 'Plot', 'Replot', 'Cancel', and 'Help' in the third row.

Select Entities

Lines

By Location

☐ X coordinates

☒ Y coordinates

☐ Z coordinates

Min,Max

0

☒ From Full

☐ Reselect

☐ Also Select

☐ Unselect

Sele All Invert

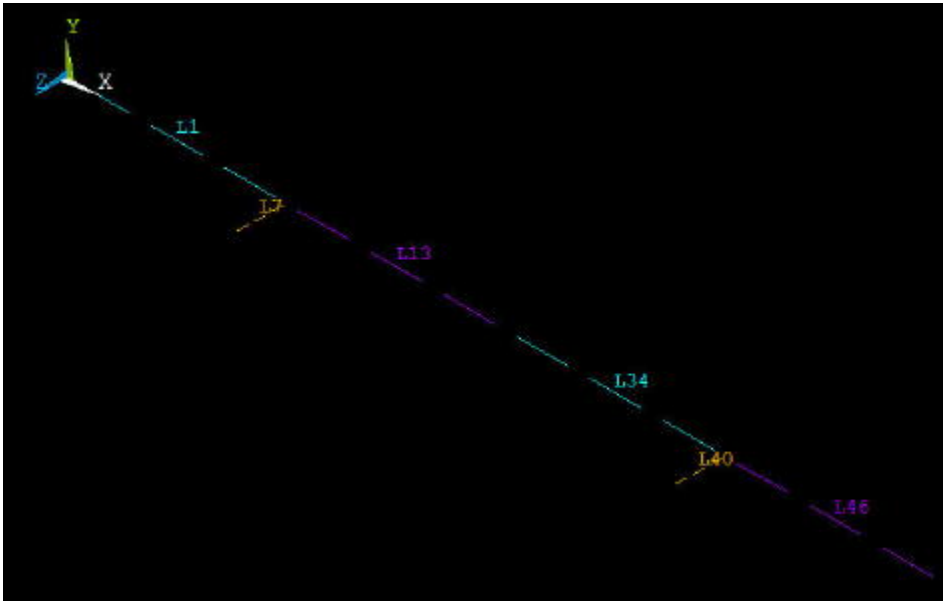
Sele None Sele Belo

OK Apply

Plot Replot

Cancel Help

Check which lines have been selected: *Select Entities* menu **>Plot**. You should see that the edges along AH have been selected.



Apply symmetry condition to the selected edges: **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > Symmetry B.C. > On Lines > Pick All**

This applies the symmetry condition to all the selected lines.

Select the entire model: Click **Select All** and then **Replot** in the *Select Entities* menu. You should see the S symbol along the edges where the symmetry BC has been applied.

Apply Symmetry along AJ

We'll first select all edges along AJ. Go back to *Select Entities* menu: Leave **Lines** and **By Location** in place. Choose **X coordinates**. Under **Min,Max**, retain 0. This will select all lines whose centers lie at $x=0$. Make sure **From Full** is selected. Click **Apply**.

Check which lines have been selected: *Select Entities* menu > **Replot**. You should see that only the edges along AJ are currently selected.

Let's apply the symmetry BC to these edges: **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > Symmetry B.C. > On Lines > Pick All**

Select the entire model: Click **Select All** and then **Replot** in the *Select Entities* menu.

Apply Clamped BC along HI

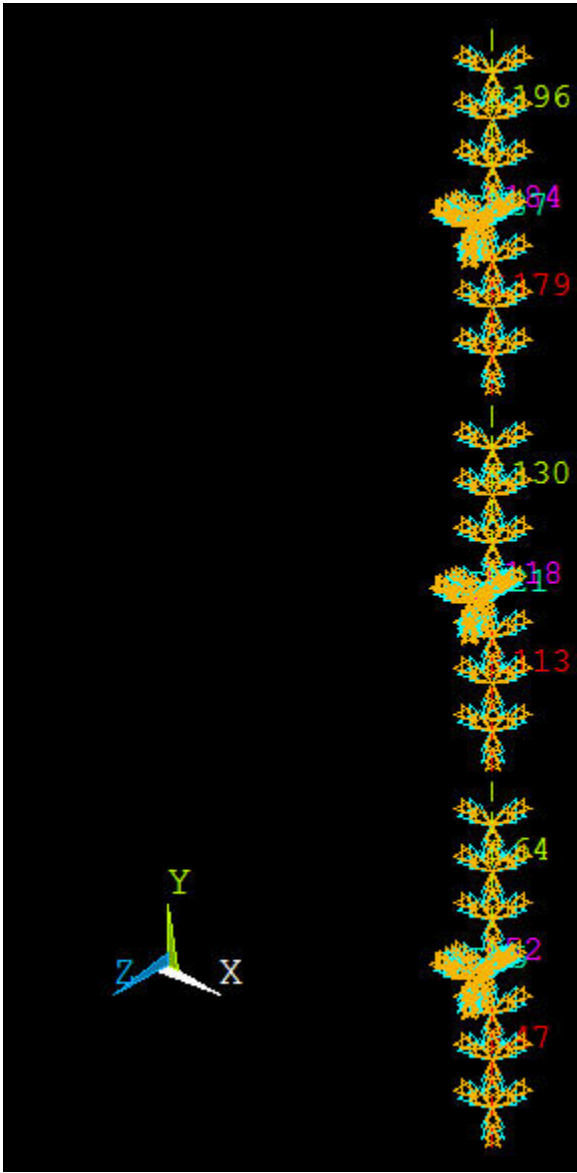
Now that we've gotten the hang of this boundary business, let's mop up Operation BC's in short order.

Select Entities menu: Leave **Lines**, **By Location** and **X coordinates** selections in place. Under **Min,Max**, enter $w1$. This will select all lines whose centers lie at $x=w1$. Make sure **From Full** is selected. Click **Apply**.

Select Entities menu > **Replot**

Constrain all six nodal degrees of freedom (DOF) for the selected edges: **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Lines > Pick All > All DOF > OK**

The (cluttered) display will show that all six DOF's have been constrained.



Select Entities menu: **Select All** and **Replot**

Apply Clamped BC along JI

Select Entities menu: Leave **Lines**, and **By Location** in place. Choose **Y coordinates**. Under **Min,Max**, enter 1.1. This will select all lines whose centers lie at $y=1.1$. Make sure From Full is selected. Click **Apply**.

Select Entities menu > **Replot**

Constrain all six nodal degrees of freedom (DOF) for selected edges: **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Lines > Pick All > All DOF > OK**

Select Entities menu: **Select All** and **Replot**

Save: **Toolbar > SAVE_DB**

Apply Pressure on Plate

Utility Menu > PlotCtrls > Numbering: Turn off line numbering.

Utility Menu > Plot > Areas

Choose areas corresponding to the plate: In the **Select Entities** menu, select **Areas** from the pull-down menu at the top. Leave **By Location** below that. Choose **Z coordinates** Under **Min,Max**, enter 0. This will select all areas whose centers lie at $z=0$. Make sure **From Full** is selected. Click **Apply**.

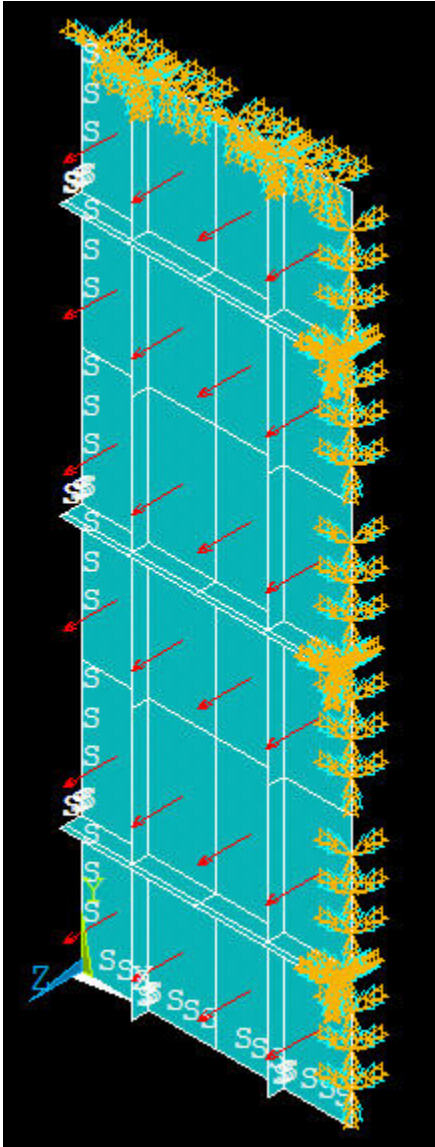
Check which areas are currently selected: **Select Entities menu > Replot**

Apply a pressure of 0.05 N/mm^2 on the plate in the +z direction: **Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Areas > Pick All**

For **VALUE**, enter 0.05. Click **OK**.

ANSYS will mark the faces where the pressure is applied. Let's instead plot the applied pressure using arrows to check its direction: **Utility Menu > PlotCtrls > Symbols**. For **Surface Load Symbols**, select **Pressures** and under **Show pres and convect as**, select **Arrows**. Click **OK**. Are the pressures acting in the right direction?

Select Entities menu: **Select All**, **Replot** and **Cancel**. You should now see the entire model. Review that all the BC's have been applied correctly.



Save: **Toolbar > SAVE_DB**

Create Log File

In parametric studies to be undertaken later, we'll start with the log file containing the commands from the first six steps that we just went through. To save this log file, select

Utility Menu > File > Write DB log file

Under **Write Database Log** to, enter the filename for the logfile: *shell_step6.lgw*. At the bottom of this menu, select **Write Essential Commands** only. Click **OK**. Review *shell_step6.lgw* by opening it in a text editor.

Write Database Log to	
<input type="text" value="shell_step6.lgw"/>	
<div>shell.lgw shell_step1.lgw shell_step3.lgw shell_step4.lgw shell_step5.lgw shell_step6.lgw</div>	<div>Directories: c:\...\shell_tutorial_test</div> <div>c:\ Documents and Settings rajesh My Documents home ansys zehnder shell_tutorial_test</div>
List Files of Type:	Drives:
<div>Database Log (*.lgw)</div>	<div>c:</div>
<div>Write essential commands only</div>	

Go to [Step 7: Solve!](#)

[See and rate the complete Learning Module](#)

Go to [all ANSYS Learning Modules](#)