

ANSYS - Semi-monocoque Shell

- Problem Specification
- Step 1: Start-up and preliminary set-up
 - Create a folder
 - Start ANSYS
 - Set Preferences
 - Units
 - Enter Parameters
- Step 2: Specify element type and constants
 - Specify Element Type
 - Specify Element Constants
- Step 3: Specify material properties
- Step 4: Specify geometry
 - Strategy for Geometry Creation
 - Keypoints Coordinates
 - Create Keypoints and Areas
- Step 5: Mesh geometry
 - Set Mesh Size Along Edges
 - Generate Mesh for Plate
 - Generate Mesh for Vertical Stiffener
 - Generate Mesh for Horizontal Stiffener
 - Copy Mesh in x-Direction
 - Copy Mesh in y-Direction
 - Copy Sub-Section ABCD
 - Merge Coincident Entities
 - Compress Item Numbers
- Step 6: Specify boundary conditions
 - Apply Symmetry along AH
 - Apply Symmetry along AJ
 - Apply Clamped BC along HI
 - Apply Clamped BC along JI
 - Apply Pressure on Plate
 - Create Log File
- Step 7: Solve!
- Step 8: Postprocess the results
 - Plot Deformed Shape
 - Plot w Displacement
 - Deflection at Facesheet Center
 - xx for Facesheet
 - yy for Facesheet
 - xx for Stiffeners
 - yy for Stiffeners
- Step 9: Validate the results
 - Comparison With Plate Theory

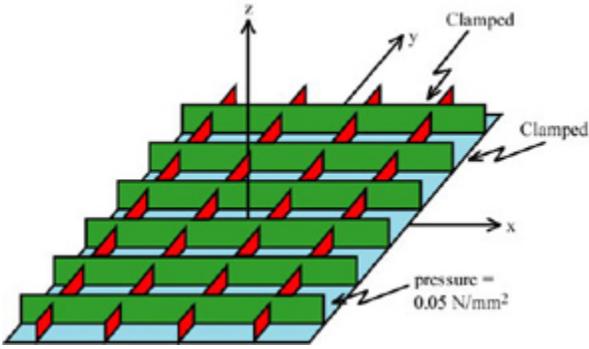
Author: Rajesh Bhaskaran, Cornell University

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

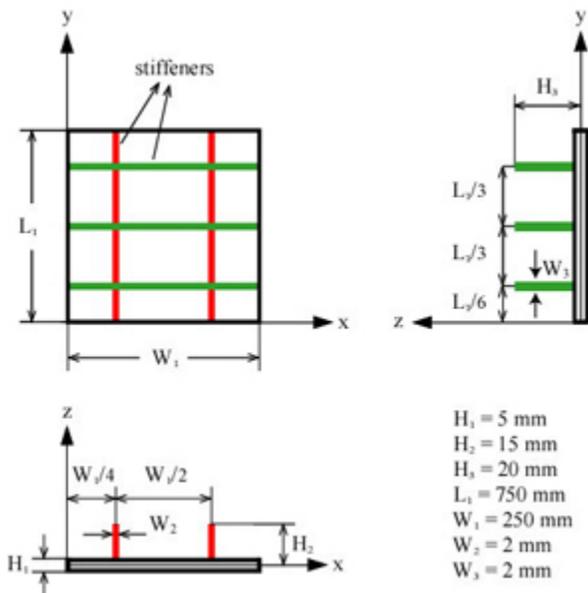
Problem Specification

In this tutorial, we'll analyze the plane shell with stiffeners shown in the figure below.



(Click for enlarged figure.)

All edges are clamped. A constant pressure of 0.05 N/mm^2 acts on the underside of the shell (i.e. the pressure acts in the $+z$ direction). Since the geometry and loads are symmetric about both the x and y -axes, we need to model only one-quarter of the structure. The dimensions of the plate and stiffeners are shown in the figure below which shows only one-quarter of the structure.



The Young's modulus $E = 7.3 \times 10^4 \text{ MPa}$ and the Poisson ratio is 0.33.

Print out this page and have it on hand so that you can easily refer to the geometry and problem specification as you go through the tutorial.

Go to [Step 1: Start-up and preliminary set-up](#)

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

[Go to all ANSYS Learning Modules](#)

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 1: Start-up and preliminary set-up

Create a folder

Create a folder called *shell* at a convenient location. We'll use this folder to store files created during the session.

Start ANSYS

On Windows systems, select the appropriate menu path. On my system, the path is

Start > Programs > ANSYS 10.0 > ANSYS Product Launcher

Enter the location of the folder *shell* that you just created as your Working directory by browsing to it.

Enter shell as your *Initial jobname*. Click on *Run*.

Resize windows as shown in [this snapshot](#) so that you can read instructions in your browser window and implement them in ANSYS.

Set Preferences

Main Menu > Preferences

In the *Preferences for GUI Filtering* dialog box, click on the box next to *Structural* so that a tick mark appears in the box. From now on, only the menu options valid for structural problems will be made available.

Units

ANSYS leaves it to us to use a consistent set of units. For convenience, we'll use the following set of units: mm for geometric dimensions; N for forces; and N/mm² for Young's modulus and pressures. The resulting stresses will be in N/mm² or MPa. Convince yourself that this is a consistent unit system; don't take my word for it.

Enter Parameters

We'll play smart and create scalar parameters corresponding to the plate and stiffener dimensions. This will later allow us to vary these parameters and perform optimization studies.

Utility Menu > Parameters > Scalar Parameters

Define a parameter for the plate length l_1 in mm:

```
L1=750
```

Click *Accept*. Similarly, define other parameters corresponding to the dimensions and click *Accept* after each (parameter names are *not* case-sensitive). Before you specify a parameter, refer to the [geometry specification](#) to remind yourself what dimension that parameter represents.

```
W1=250
```

```
W2=2
```

```
W3=2
```

```
H1=5
```

```
H2=15
```

```
H3=20
```

```
{{}}
```

We'll play smarter and also specify the number of stiffeners in each direction as parameters so that these too can be varied easily in tradeoff studies. We'll employ the labels *NSX* and *NSY* for the number of stiffeners in the *x* and *y* directions, respectively.

```
NSX=2
```

```
NSY=3
```

We'll use the above parameters when creating the geometry in [Step 4](#). We'll also define some parameters which we'll use in [Step 5](#) to set the mesh size along edges.

```
NDIV_X=3
```

```
NDIV_Y=6
```

```
SIZE_Z=5
```

Close the *Scalar Parameters* window.

Go to [Step 2: Specify Element Types and Constants](#)

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

Go to [all ANSYS Learning Modules](#)

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 2: Specify element type and constants

Since thin structures can be modeled efficiently as shells, we'll use shell elements to build the finite-element model. Shell elements can support membrane and bending loads consistent with classical shell theory (sorry, FEA doesn't let you off from understanding basic theory). As you can imagine, shell elements are appropriate when the thickness of the structure is small compared to the other dimensions. The computational savings come about because only the mid-surface of the structure is modeled; the thickness and other cross-sectional properties are incorporated into the element stiffness matrix and input as "real constants" in ANSYS. (This is analogous to modeling beams using beam elements where the beams are modeled as lines with thickness and other cross-sectional properties being "real constants"). Section 2.10 in the *ANSYS Element Reference* manual gives you a page of useful information on shell elements. Be sure to peruse it in the online documentation since it'll be on the final. 😊

Specify Element Type

Let's take a peek at the shell elements available in ANSYS. Bring up the ANSYS documentation window, select the **Search** tab, enter the phrase "pictorial summary" and click on **List Topics**. Then double-click on 3.2 *Pictorial Summary* in the left pane. At the top of the pictorial summary of element types in the right pane, click on **SHELL Elements**. This brings up the list of shell elements available in ANSYS including many with specialized capabilities. Perusing this list, you'll see that **SHELL63** (4-node elastic shell) is a basic shell element and a possible candidate for our problem. A close relative is **SHELL93** (8-node elastic shell) which has mid-side nodes in addition to the corner nodes. Since the mid-side nodes give greater accuracy, we'll use **SHELL93** for our problem. Click on **SHELL93** in the help and take a few minutes to peruse the manual page for this element. What are the "real constants" that we'll need to enter in the next step? Note that each node has six degrees of freedom: three translational and three rotational.

Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add...

Pick **Shell** in the left field and **Elastic 8node 93** in the right field. Click **OK** to select this element. The **SHELL93** element will now be available in the meshing step. Close the *Element Types* menu.

Specify Element Constants

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add

This brings up the *Element Type for Real Constants* menu. Click **OK** to specify the real constants for the **SHELL93** element.

When meshing, we'll have to assign three different thickness values: $H1$ for the plate; $W2$ and $W3$ for the stiffeners in the x and y directions, respectively. This means we'll have to create three real constant sets, one for each of these thickness values. According to the **SHELL93** help page, if the element has a constant thickness, only **TK(I)**, the shell thickness at the first corner node, needs to be input.

Create the first real constant set: make sure **Real Constant Set No.** is set to 1. For **TK(I)**, enter $H1$. Leave the other fields blank since they are not applicable to our problem. Click **Apply**.

Create the second set: For **Real Constant Set No.**, enter 2. For **TK(I)**, enter $\{W2\}$ and click **Apply**.

Create the last set: For **Real Constant Set No.**, enter 3. For **TK(I)**, enter $W3$ and click **OK**.

Save: **Toolbar > SAVE_DB**

Go to [Step 3: Specify material properties](#)

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

Go to [all ANSYS Learning Modules](#)

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 3: Specify material properties

Main Menu > Preprocessor > Material Props > Material Models

In the *Define Material Model Behavior* menu, double-click on **Structural, Linear, Elastic**, and **Isotropic**.

Enter Young's modulus **EX** in MPa: 7.3×10^4

Enter Poisson's Ratio **PRXY**: 0.33. Click **OK**.

Close the *Define Material Model Behavior* menu.

Save your work: **Toolbar > SAVE_DB**

Go to [Step 4: Specify geometry](#)

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

Go to [all ANSYS Learning Modules](#)

Problem Specification

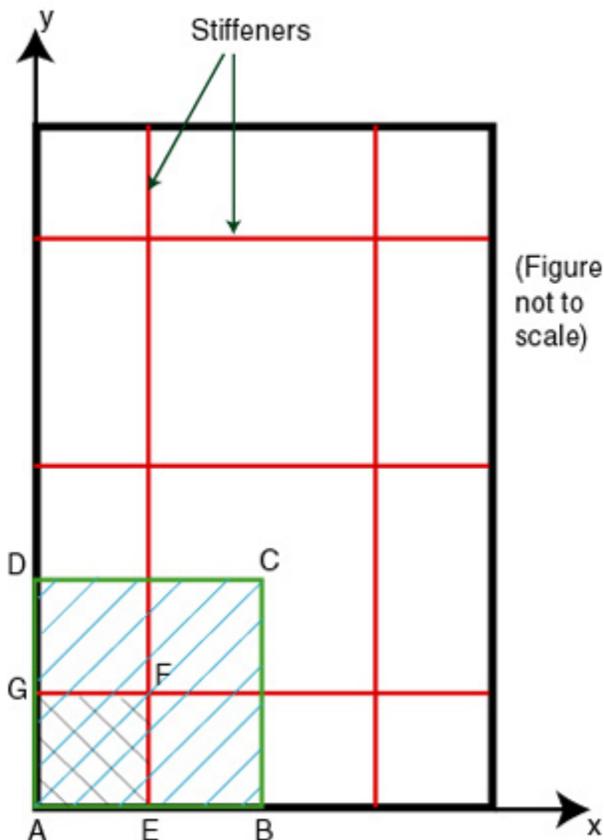
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
- 4. Specify geometry**
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results

Step 4: Specify geometry

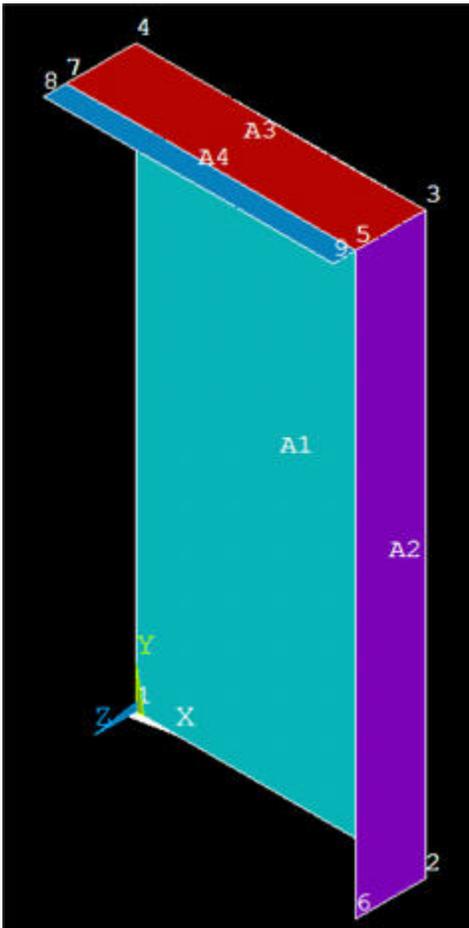
Strategy for Geometry Creation

The geometry of the structure has a repetitive pattern as can be seen in the schematic below. We'll exploit this feature in creating the finite-element mesh for the entire structure. In this step, we'll create the geometry for the sub-section *A EFG* only. In [Step 5](#), we'll mesh this sub-section and use it to generate the mesh for the entire structure in two steps:

1. We'll copy parts of *A EFG* to create the sub-section *ABCD*. (When we copy areas, the associated mesh is also copied.)
2. We'll copy sub-section *ABCD* in both directions to generate the mesh for the entire structure. Clever move, eh?



In sub-section *AEFG* above, the stiffeners cross each other at *F*. Modeling this correctly is tricky, so indulge me while I explain how we are going to proceed. At the end of the geometry step, the isometric view of *AEFG* will look as below, with the keypoint and area numbers turned on (different colors refer to different areas). If the stiffeners share keypoints 3,5 and 9, the stiffener crossing is modeled correctly and a load applied to one stiffener will be transferred appropriately to the other one. To this end, we will divide the horizontal stiffener into two areas (A3, A4). This will also yield a regular mesh as we'll see in the meshing step.



Note that the horizontal stiffener has to be split into two areas A3 and A4 because it has a larger width than the vertical stiffener. Instead, if the vertical stiffener is the wider one, it'd be the one that you'd have to split into two areas. This would change the topology of the model; something to keep in mind when you vary the stiffener dimensions in optimization studies.

Keypoints Coordinates

Using the figure in the problem specification and the one above, create a table of the keypoint coordinates required to create areas A1-A4, using the parameters defined in Step 1. Recall that areas correspond to the mid-surface of the structure. Are my keypoint coordinates below correct? || No. || x || y || z ||

1	0	0	0
2	$W1/(2*NSX)$	0	0
3	$W1/(2*NSX)$	$L1/(2*NSY)$	0
4	0	$L1/(2*NSY)$	0
5	$W1/(2*NSX)$	$L1/(2*NSY)$	H 2
6	$W1/(2*NSX)$	0	H 2
7	0	$L1/(2*NSY)$	H 2
8	0	$L1/(2*NSY)$	H 3
9	$W1/(2*NSX)$	$L1/(2*NSY)$	H 3

Create Keypoints and Areas

In previous tutorials, we have defined keypoints (i.e. vertices) using menus. Here, we will instead use the command line to create keypoints. The two methods are equivalent but the latter is faster for more experienced users. Let's check the documentation to find the command for creating keypoints. In the ANSYS documentation window, select the **Search** tab, enter "keypoint" as the search term and click on **List Topics**. Double-click on **5.2 Creating Your ... in the left pane**. Section 5.2 has useful information on creating your solid model. Scroll down to section 5.2.1. The table near the beginning of this section indicates that the command for creating keypoints is **K_**. Note that the corresponding menu path is what we have used in previous tutorials to create keypoints. The menu path also indicates that this command is part of the *Preprocessor* module. Click on **K** to see the format of this command.

Since it is useful to track the keypoint and area numbers as we create the geometry, turn on keypoint and area numbering: **Utility Menu > PlotCtrls > Numbering**

Turn on **Keypoint numbers** and **Area numbers** and click **OK**.

To access the **K** command, make sure the preprocessor module is open.

Create keypoint #1 at (0,0,0): at the command prompt, type `K,1,0,0,0` as below and hit **Enter**. Note that when you type the command, the format of the command pops-up as a reminder.

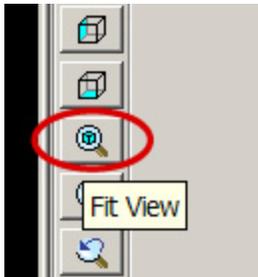


Similarly, create keypoints #2-#4 in the above table by entering the following at the command prompt:

```
K,2,W1/(2*NSX),0,0
K,3,W1/(2*NSX),L1/(2*NSY),0
K,4,0,L1/(2*NSY),0
```

ANSYS uses the *current* value of the parameters *W1*, *L1*, *NSX* and *NSY* when creating the keypoints. If you change the value of any of these parameters after you have created the keypoints, these keypoints will *not* be moved accordingly, i.e. the keypoint coordinates will still contain the old values of the parameters. It is important to remember this idiosyncrasy of ANSYS; otherwise, it could come back to bite you in uncomfortable parts of your anatomy.

Is it safe to presume that you are cheating like I would and cutting-and-pasting the command line inputs? That's OK as long as you do them one-by-one so that you can track the effect of each command. To fit all the keypoints in your current view, click on the *Fit View* icon in the rightmost part of the GUI (see snapshot below). You'll have to click on this button periodically to fit the entire geometry into your view.

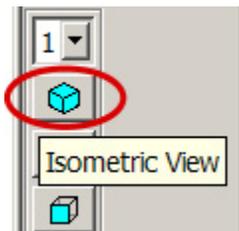


To create an area from keypoints #1-#4, enter `a,1,2,3,4` at the command line. Note that ANSYS automatically assigns the label *A1* to this first area. This area corresponds to the plate.

Next create keypoints #5 and #6 followed by the area corresponding to the stiffener in the *y*-direction. Enter

```
K,5,W1/(2*NSX),L1/(2*NSY),H2
K,6,W1/(2*NSX),0,H2
A,2,3,5,6
```

To bring up the isometric view, click on the *Isometric View* icon in the right part of the GUI (see snapshot below). The area is labeled *A2*. If the area disappears from the view, select **Utility Menu > Plot > Areas**.

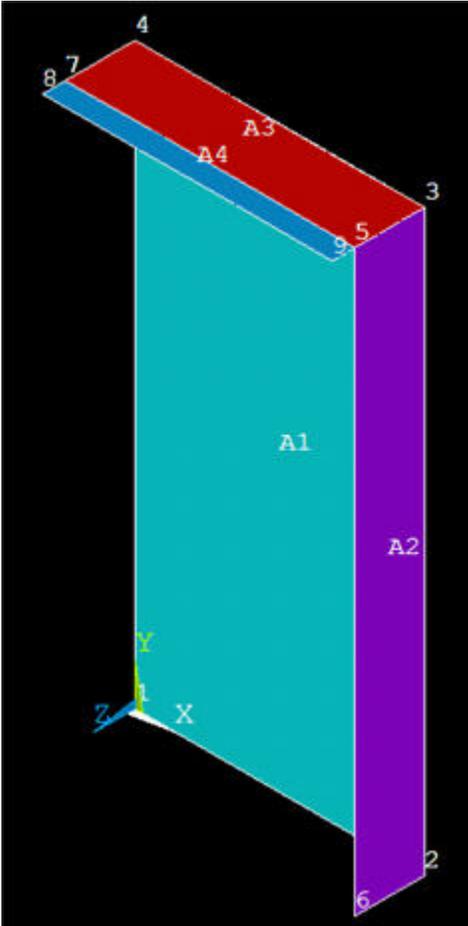


Now that we have gotten the hang of this business, we can create the remaining keypoints and areas in our sleep (remember to wipe the drool off the keyboard):

```
K,7,0,L1/(2*NSY),H2  
K,8,0,L1/(2*NSY),H3  
K,9,W1/(2*NSX),L1/(2*NSY),H3  
A,3,4,7,5  
A,5,7,8,9
```

Utility Menu > Plot > Areas

The geometry should look like this:



Save your work: **Toolbar > SAVE_DB**

Go to [Step 5: Mesh geometry](#)

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

Go to [all ANSYS Learning Modules](#)

- 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 5: Mesh geometry

As discussed in the previous step, we'll first mesh the subsection *A EFG* that we just created.

Set Mesh Size Along Edges

We'll use the [previously-defined parameters](#) *NDIV_X*, *NDIV_Y* and *SIZE_Z* to set the mesh size along edges. The table below describes what each parameter is and the value we have assigned to it.

Parameter	Value	Description
<i>NDIV_X</i>	3	No. of divisions for edges along x
<i>NDIV_Y</i>	6	No. of divisions for edges along y
<i>SIZE_Z</i>	5	Size of divisions for edges along z

Edges along z are of two different lengths, as should be apparent from the [geometry display](#). For instance, the edge between keypoints #5 and #9 is smaller than that between keypoints #3 and #5. By setting the element size rather than the number of divisions, we can use the same parameter for both edge lengths.

Bring up the *MeshTool*: **Main Menu > Preprocessor > MeshTool**

We are going to continually use the *MeshTool* to generate the mesh. So resize and rearrange the windows slightly so that you can access the *MeshTool*, ANSYS GUI, and tutorial simultaneously.

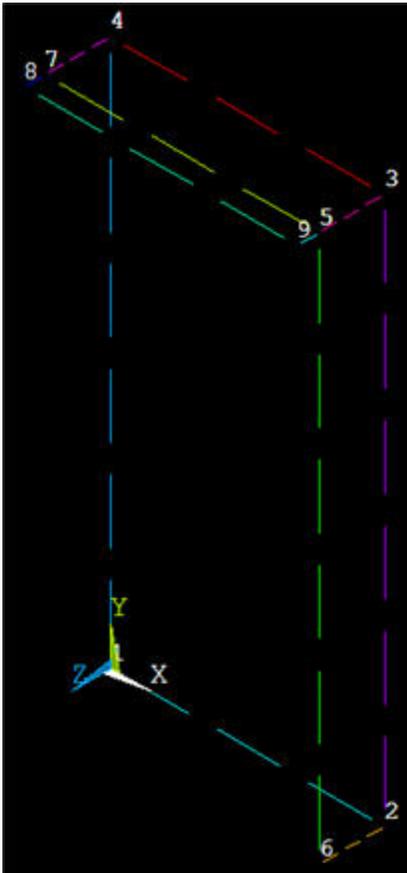
If you have trouble selecting the correct line below, **hold down** the left mouse button until the line is selected and then release the left button. If you want to **deselect** a line, right-click to go into deselect mode, left-click on the line to be deselected and right-click again to go back into select mode.

Under **Size Controls** and **Lines**, click **Set**. Pick all four lines in the x-direction. and click **OK** in the pick menu. Enter *NDIV_X* for **No. of element divisions** and click **Apply**.

Next, pick all three lines in the y-direction and click **OK** in the pick menu. Enter *NDIV_Y* for **No. of element divisions** and click **Apply**.

Last, pick all five lines in the z-direction and click **OK** in the pick menu. Enter *SIZE_Z* for **Element edge length**. Make sure **No. of element divisions** is blank. Click **OK**.

Plot lines to see the element divisions along edges and check that they have been set correctly: **Utility Menu > Plot > Lines**. If you made an error, repeat the above steps before saving.



Generate Mesh for Plate

Recall that the plate and the horizontal and vertical stiffeners have different thicknesses which have been assigned to different "real constant" sets. Jog your memory about which thicknesses have been assigned to which "[real constant](#)" sets in Step 2.

We'll first mesh the plate using real constant set #1. Under **Element Attributes** in *MeshTool*, click on **Set**. You will see that the element type and material number are already set and the default real constant set is #1. These are the options we need for the plate; so these don't need to be changed. Click **Cancel**.

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

Click on **Mesh** in *MeshTool*. Pick area A1 and click **OK**. The resulting mesh for the plate is displayed.

Generate Mesh for Vertical Stiffener

Under **Element Attributes** in *MeshTool*, click on **Set**. Set **Real constant set number to 2** and click **OK**.

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

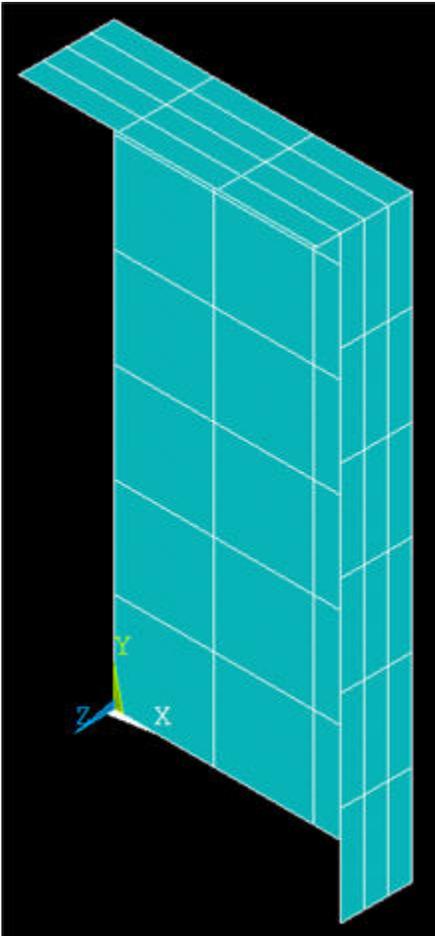
Click on **Mesh** in *MeshTool*. Pick area A2 and click **OK**. The resulting mesh for the plate is displayed.

Generate Mesh for Horizontal Stiffener

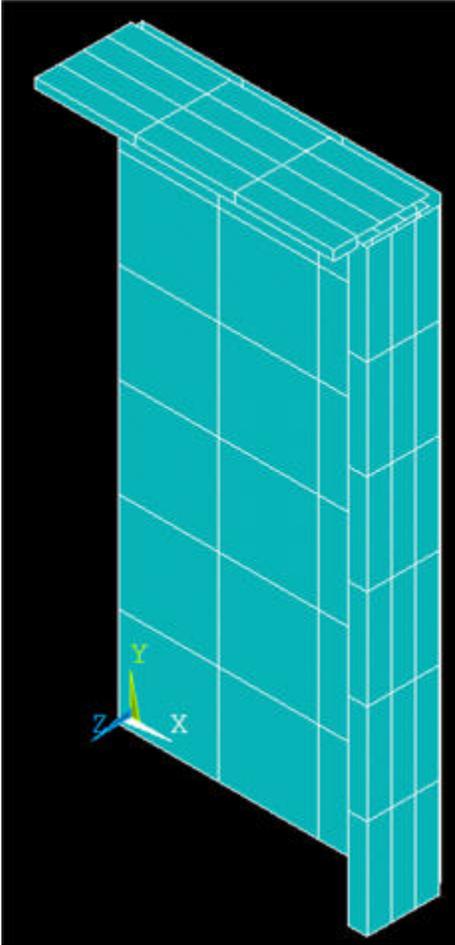
Under **Element Attributes**, click on **Set**. Set Real constant set number to 3 and click **OK**.

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

Click on **Mesh** in *MeshTool*. Pick areas A3 and A4 and click **OK**.



Let's change the graphical display so that the geometry is displayed as a solid model with the shell thicknesses shown. At the command prompt, type `/eshape, 1` and then select **Utility Menu > Plot > Replot**. When the `/ESHAPE` command is issued, ANSYS uses the real constants associated with each element to determine its shape. (Note that commands are not case-sensitive.)



Do the relative thicknesses of different parts of the shell look correct? Feel free to manipulate the graphical display to check this.

You can access different model views such as front, right, isometric etc. using the buttons to the right of the graphics window. At the bottom of this row of icons is the *Dynamic Model Mode* . Clicking this icon allows you to manipulate the model using the mouse (alternatively, you can hold down the *Ctrl* key). To see the help page on this mode, right-click on the icon and select *Dialog Overview > Pan, Zoom, Rotate > Dynamic Mode: Model*.

Save: **Toolbar > SAVE_DB**

Copy Mesh in x-Direction

Turn off the element shapes to make the display less cluttered by entering the following commands:

```
/eshape,0  
/replot
```

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

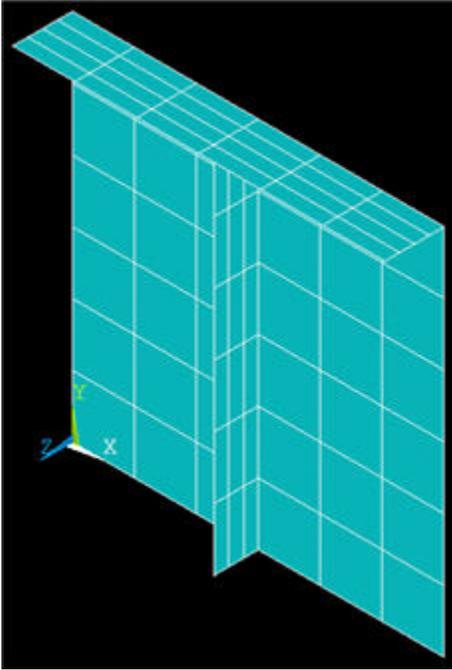
Copy areas: **Main Menu > Preprocessor > Modeling > Copy > Areas**

Pick **A1, A3, and A4** and click **OK**. (Hold down the left mouse button until the correct area is selected.)

Copy Areas menu: We want two copies including the original, so leave **Number of copies** as 2. For **X-offset**, enter $w1 / (2 * NSX)$, leave **Y-** and **Z-offsets** blank and click **OK**.

Since the associated keypoints are also copied over, you'll notice that there is a collection of two coincident but separate keypoints along shared boundaries where the original and copied entities overlap. It's key that we [\[merge these coincident keypoints\]](#) [SIMULATION:Semi-monocoque shell page 5]; we'll undertake this after we are finished with the copy step.

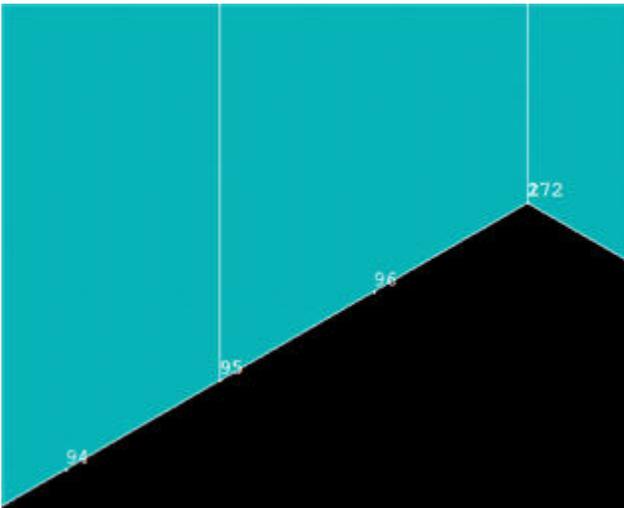
Have the elements associated with the areas also been copied? Check this: **Utility Menu > Plot > Elements**



Save: **Toolbar > SAVE_DB**

As with keypoints (and lines too), there is a collection of two coincident but separate nodes along the shared boundaries. Let's convince ourselves of this truism:

- Turn on node numbers (in addition to keypoint and area numbers) using **Utility Menu > PlotCtrls > Numbering**
- Display elements and nodes together: **Utility Menu > PlotCtrls > Multi-PlotCtrls > OK**. Leave only **Nodes** and **Elements** turned on and click **OK**. Select **Utility Menu > Plot > Multi-Plots**.
- Zoom into the bottom-center of the model as in the snapshot below. At the bottom of the symmetry plane, you should notice that there are actually two nodes, 2 and 172. Node 2 is associated with the original area and node 172 with the copy. We'll merge coincident nodes a little later. (Note that your node numbers may be different from mine since numbering can vary on different computer systems.)



Copy Mesh in y-Direction

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

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

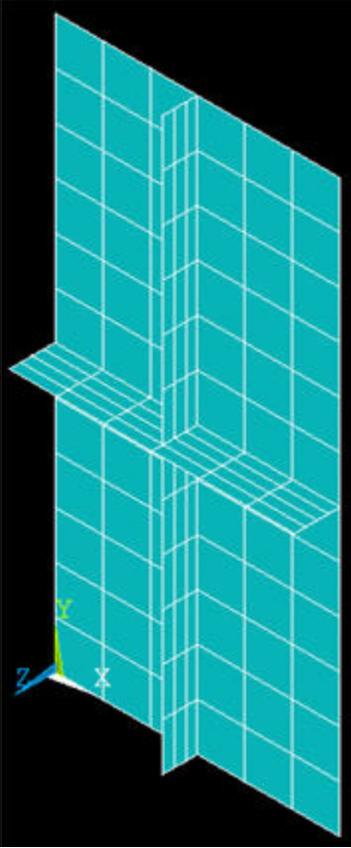
Copy areas: **Main Menu > Preprocessor > Modeling > Copy > Areas**

Pick A1, A2, and A5 and click **OK**.

Copy Areas menu: Leave **Number of copies** as 2. For **Y-offset**, enter $L1 / (2 * NSY)$; delete **X-offset**; leave **Z-offset** blank and click **OK**.

Check the resulting mesh for sub-section ABCD: **Utility Menu > Plot > Elements**

Turn-off node numbers to reduce clutter.



Copy Sub-Section ABCD

How many copies of sub-section ABCD do we need to make in the x- and y-directions? What are the values of X-Offset and Y-Offset in each case? Jot your answers down so that you can check the values below.

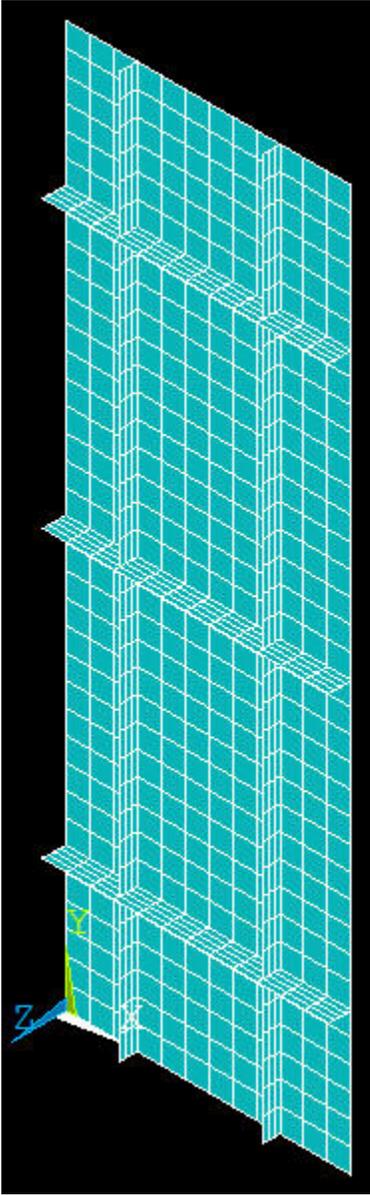
Copy areas: **Main Menu > Preprocessor > Modeling > Copy > Areas > Pick All**

Copy Areas menu: **Number of copies** = NSX; **X-offset** = W1/NSX; **Y-** and **Z-offsets** should be blank or zero. Click **Apply**.

Repeat copy in y-direction: Select **Pick All** again in the pick menu.

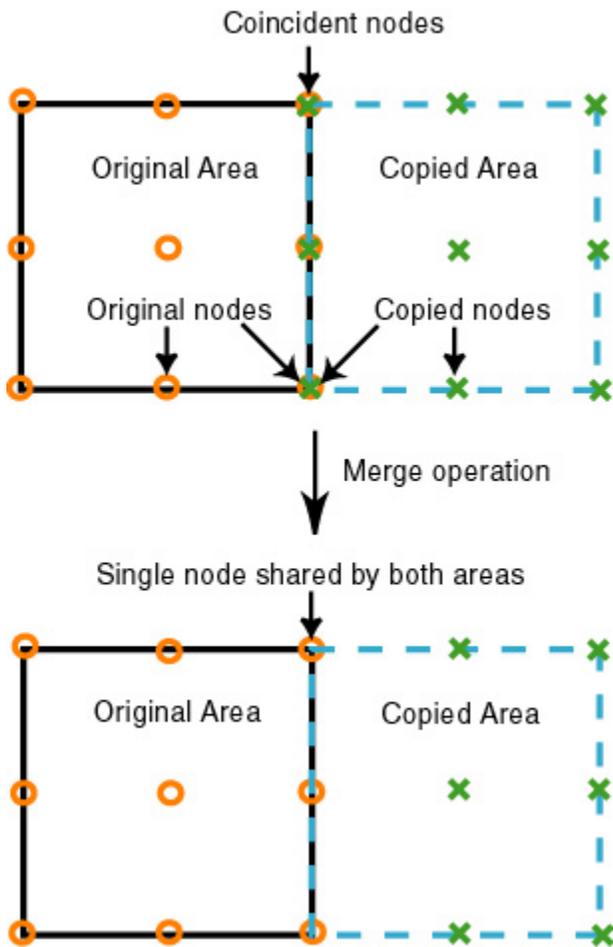
Copy Areas menu: **Number of copies** = NSY; **Y-offset** = L1/NSY; **X-** and **Z-offsets** should be blank or zero. Click **OK**.

Check the resulting mesh: **Utility Menu > Plot > Elements**



Merge Coincident Entities

We saw earlier that the copy operations resulted in coincident nodes, keypoints, etc. along shared boundaries. We need to merge these coincident items so that separate portions of the model are combined into one. Otherwise, the various portions will live as independent entities and loads applied to one portion will not be transferred to its neighbors. The effect of the merge operation is shown schematically below.



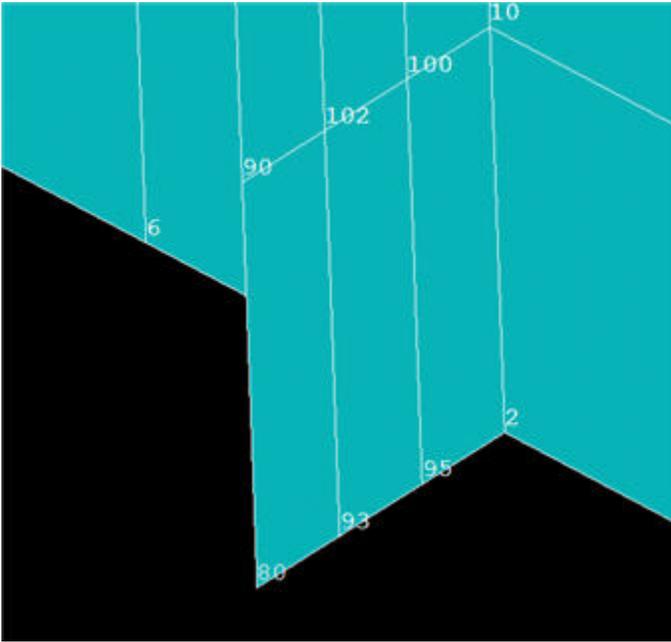
View documentation about the merge utility: [ANSYS Help > Contents > Modeling and Meshing Guide > Number Control and Element Reordering > Number Control](#)

This section has useful information about merging entities. It indicates that one can either merge keypoints, nodes, etc. individually or merge *all* coincident entities at once (with ANSYS ensuring that they are merged in the proper sequence). Since we'd like to palm off as much work as possible to ANSYS, we'll use the latter option.

Select **Main Menu > Preprocessor > NumberingCtrls > Merge Items**

For *Type of item to be merged*, select **All** and click **OK**. This will merge all coincident entities.

Earlier we saw that [nodes 2 and 172 were coincident](#). Zoom in on this region again with node numbers turned on. What do you see?



If you performed the merge operation correctly, you'll see that the higher numbered node has been deleted and the two coincident nodes have been replaced by a single node.

Save: **Toolbar > SAVE_DB**

Compress Item Numbers

If you scroll down the [help section on numbering](#) that we've been peeking at, you'll find a useful spiel on *Compressing Item Numbers*(section 11.1.2): "As you build your model, you might, by deleting, clearing, merging, or performing other operations, create unused slots in the numbering sequence for various items. These slots will remain empty for some items (such as elements) but will be filled in for other items (such as keypoints) as new items are created. To save data storage space (by eliminating otherwise empty numbers) or to preserve desired sequencing (by forcing newly-created items to be assigned numbers greater than those of existing items), you can eliminate these gaps by "compressing" your numbering".

Check the range of node numbers before compressing the numbering: **Utility Menu > List > Nodes > OK**

Compress numbering for all items (nodes, elements, etc.): **Main Menu> Preprocessor> Numbering Ctrl> Compress Numbers**

For *Item to be compressed* , select **All** and click **OK**.

Re-check the range of node numbers after compressing. You should find that the range of node numbers is reduced since there are now no gaps in the numbering.

Close *MeshTool*.

Save: **Toolbar > SAVE_DB**

Go to [Step 6: Specify boundary conditions](#)

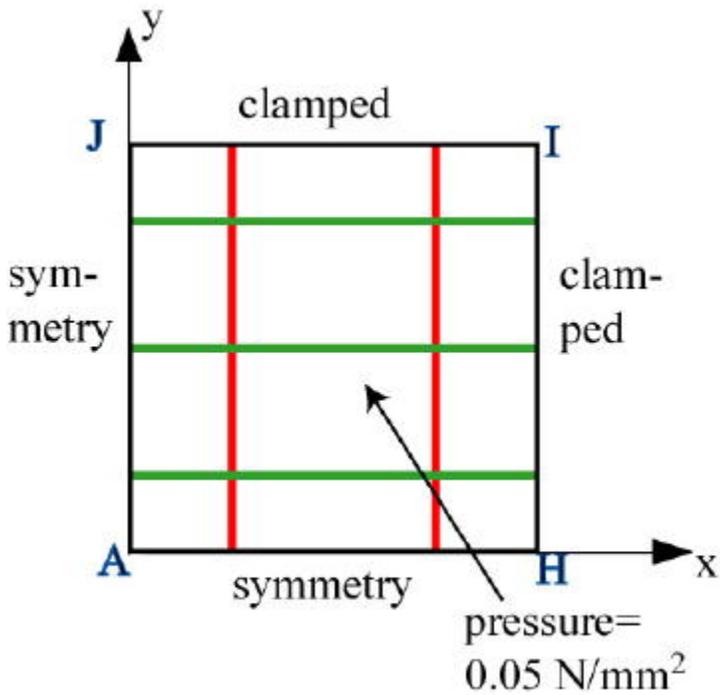
[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)

- 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**

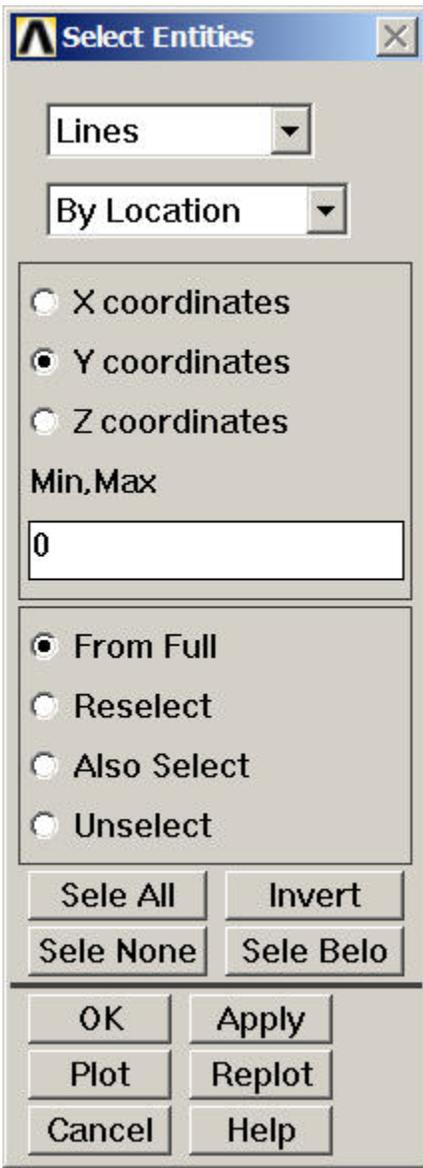
Adjust the display: Click on the *Isometric View* and *Fit View* icons in the rightmost part of the GUI.

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

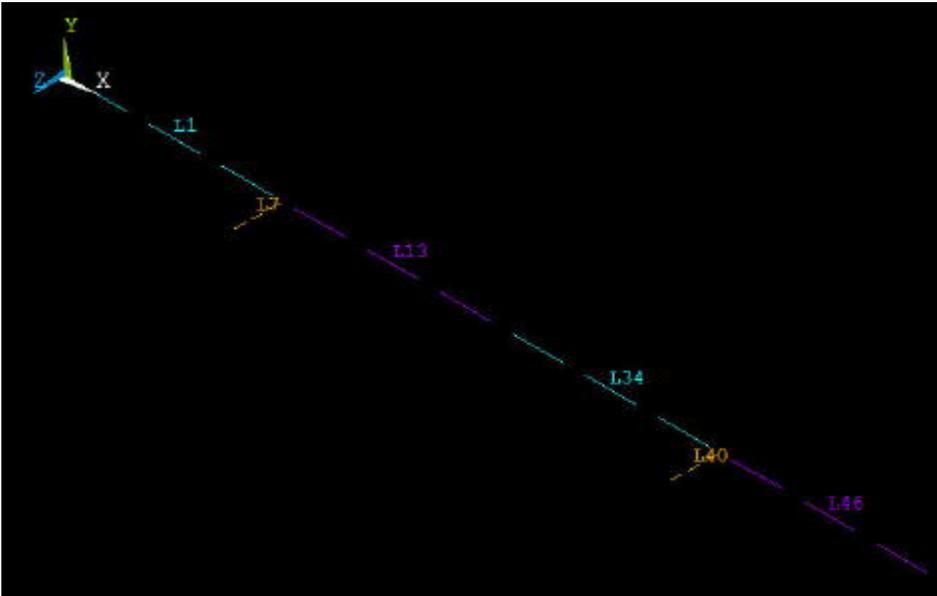
Select all edges along AH: **Utility Menu > Select > Entities**

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**.



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

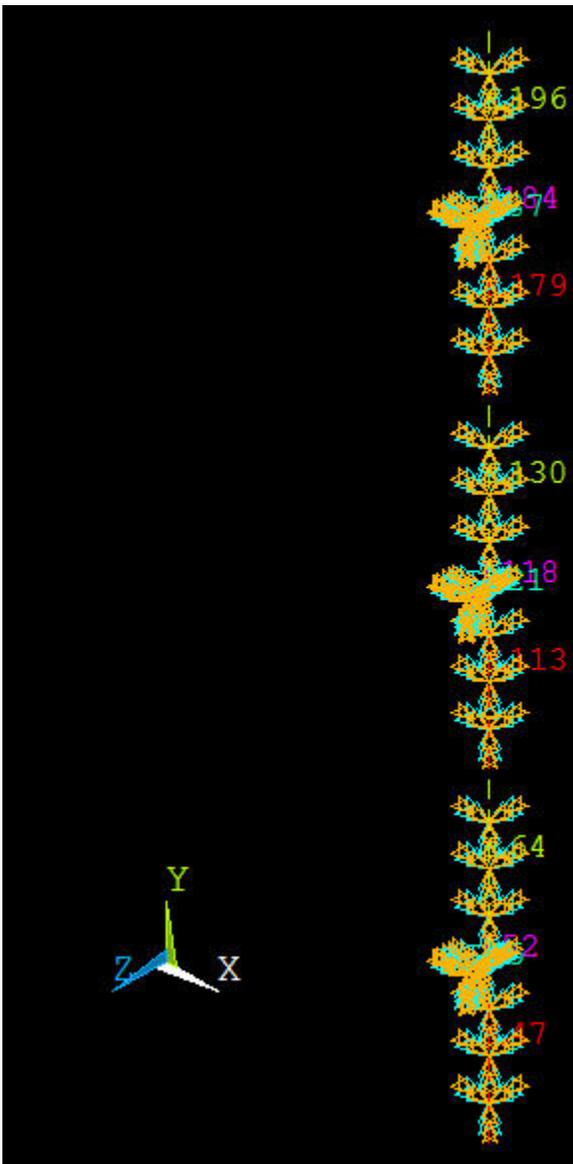
Now that we've gotten the hang of this boundary business, let's mop up Operation BC's in short order.

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**.

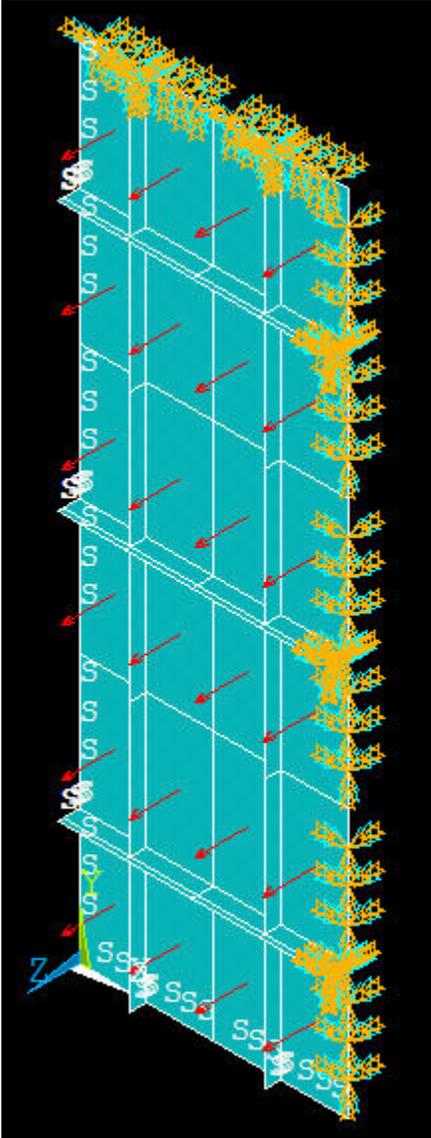
Check which areas are currently selected: **Select Entities menu > Replot**

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.



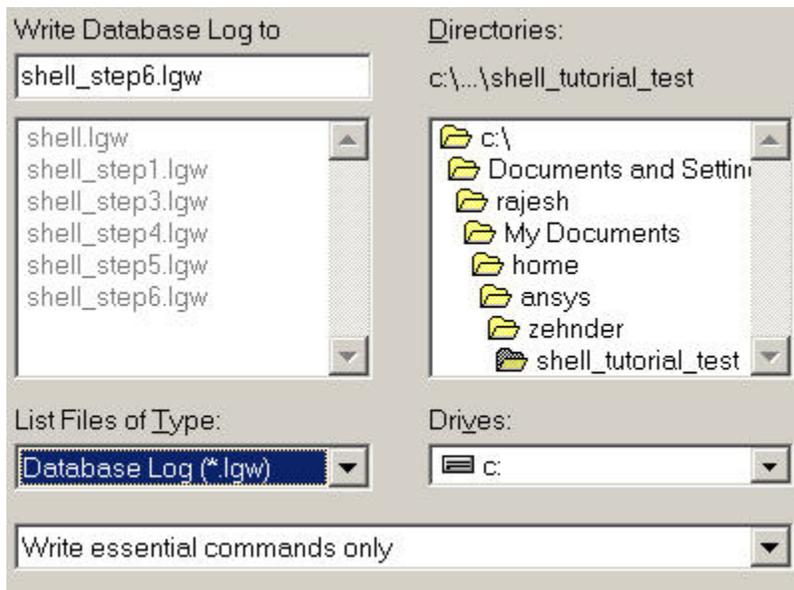
Save: **Toolbar > SAVE_DB**

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.



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

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

Go to [all ANSYS Learning Modules](#)

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 7: Solve!

Enter Solution Module: **Main Menu > Solution**

Enter check in the *Input* window. If the problem has been set up correctly, there will be no errors or warnings reported. You should see the following message in the *Output* window: The analysis data was checked and no warnings or errors were found.

Main Menu > Solution > Solve > Current LS

Review the information in the */STATUS Command* window. Close this window.

Click **OK** in *Solve Current Load Step* menu.

ANSYS performs the solution and a window should pop up saying "Solution is done!". Close this window.

Save: **Toolbar > SAVE_DB**

Go to [Step 8: Postprocess the results](#)

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

Go to [all ANSYS Learning Modules](#)

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 8: Postprocess the results

Plot Deformed Shape

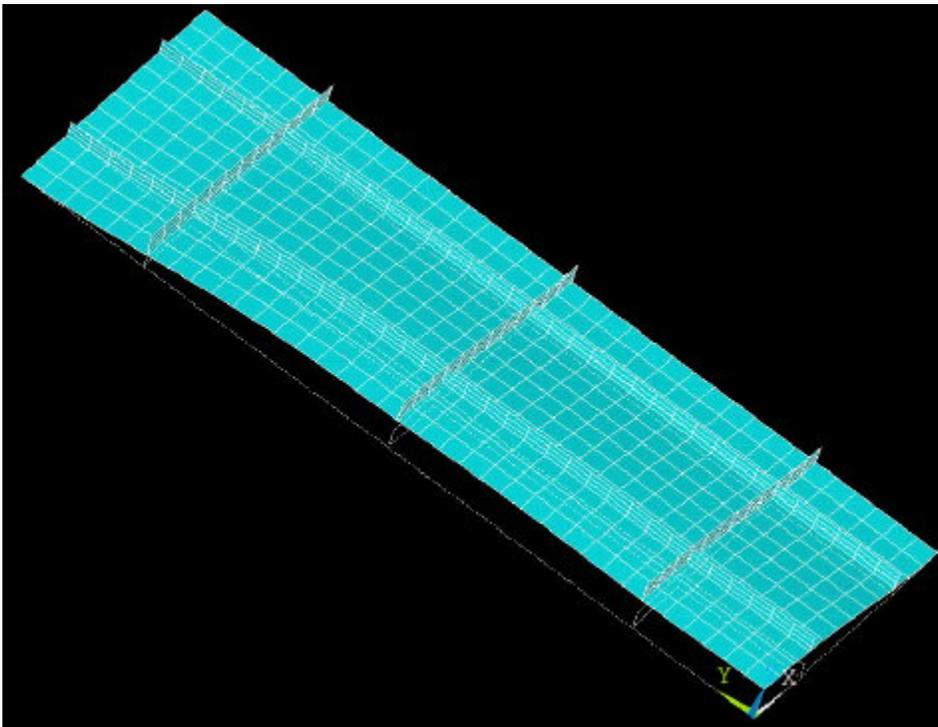
The *Results Viewer* provides a convenient way to review results.

Main Menu > General Postproc > Results Viewer

The title bar for the *Results Viewer* indicates that the results are being read from the *shell.rst* file. Rearrange the windows slightly so that you can access the *Results Viewer*, ANSYS GUI and tutorial simultaneously.

Under **Choose a result item**, select **Nodal Solution > DOF Solution > Displaced structure**. Click on the **Plot Results** icon.

Hold down the *Ctrl* key and use the mouse to **manipulate the model view**; check that the BC's and loads have been imposed correctly. The snapshot below shows one view that is useful in checking the BC's. Are the symmetry and clamped conditions satisfied at the corresponding edges for the facesheet as well as the stiffeners? Is the model deformation as you'd expect for a uniform pressure on the facesheet in the +z direction?



The maximum displacement $DMX=3.88$ mm. Where do you think this occurs? We'll check this in a minute.

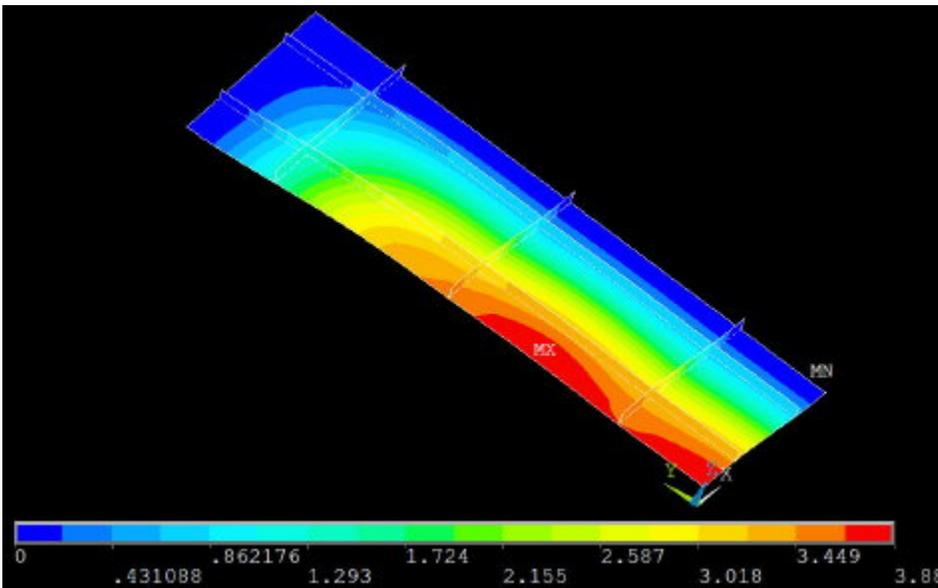
Animate the deformation and obsessively recheck the BC's:

Results Viewer > Animate Results > Deformed Shape > OK > Def + undef edge > OK

Select **Forward Only** in the *Animation Controller*.

Plot w Displacement

In the *Results Viewer*, in place of **Displaced structure**, choose **Nodal Solution > DOF Solution > Z-Component of displacement**. Click on the **Plot Results** icon.



The location of maximum deflection w_{max} is labeled *MX*. This occurs not at the facesheet center but on the edge $x=0$, at approximately $y=L/3$. As expected, the deflection is zero along the clamped edges.

To save a copy of this plot, select **Results Viewer > Image Capture > Capture to file > OK**.

For filename, enter `w_contours`. This creates the file `_w_contours.png_` in your working directory. Check that you can open this file using your pet image viewer.

Deflection at Facesheet Center

To query the value of the deflection at the center of the facesheet, select the **Query Results** icon in the *Results Viewer*.

Pick the node at $X=0, Y=0$. The coordinates of the picked node and the corresponding value of *UZ* (i.e. w displacement) are displayed in the *Query Subgrid Results* menu. The deflection at the center of the facesheet is 3.8 mm.

Cancel the *Query Subgrid Results* menu.

σ_{xx} for Facesheet

For clarity, let's first plot σ_{xx} for just the facesheet. Select the elements for the facesheet in two steps:

1. Select areas corresponding to the facesheet.
2. Select elements attached to these previously selected areas. Close the *Results Viewer*.

Utility Menu > Plot > Elements

Utility Menu > Select > Entities

Select Entities menu: Select **Areas, By Location** and **Z coordinates**. Under **Min,Max**, enter 0. Make sure **From Full** is selected. Click **Apply**.

Following this, select **Elements, Attached to, Areas** and **From Full**. Click **Apply** and then **Replot**. You should see that only elements for the facesheet are currently selected.

Main Menu > General Postproc > Results Viewer > Nodal Solution > Stress > X-Component of stress

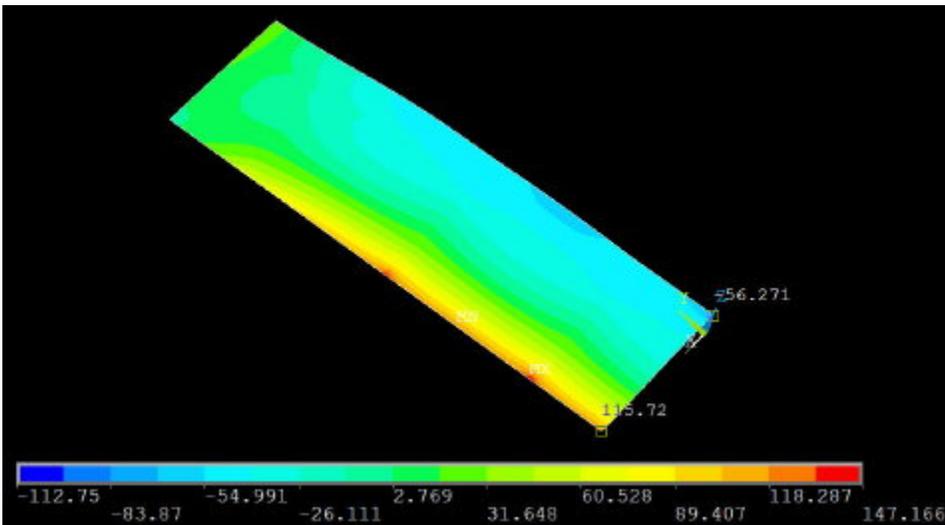
Click **Plot Results** icon.

Shell theory provides the stresses through the thickness of the shell. This is illustrated in the help page for SHELL93 element. If, like me, you are looking at the top of the facesheet, then the contours values correspond to σ_{xx} values for the top face. Check the σ_{xx} value for the top face at the facesheet center:

Results Viewer > Query Results

Pick the node at $X=0, Y=0$. I get $\sigma_{xx}=51$ MPa at the center-top of the facesheet.

Hold down the **Ctrl** key and flip the model over so that you are looking at the *bottom* of the facesheet as in the snapshot below. Now the contours values correspond to σ_{xx} values for the facesheet *bottom*. Use **Query Results** to determine σ_{xx} at the center-*bottom* of the facesheet. I get -56 MPa. What do you get? The negative sign indicates compression.



(For an enlarged view, click and maximize the browser window)

Thus, at the facesheet center, σ_{xx} varies from a tensile stress of 51 MPa at the top to a compressive stress of 56 MPa at the bottom.

Using **Query Results**, I get $\sigma_{xx}=116$ MPa at the edge point $x=W1, y=0$ for the bottom of the facesheet (see snapshot above). What's the value you finagled out of ANSYS?

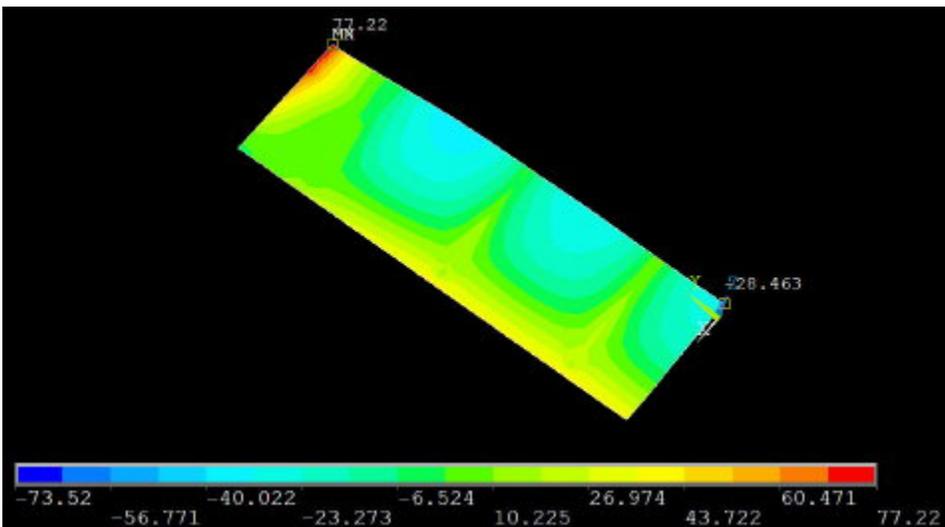
Do these values look plausible? We'll investigate this in **Step 9** when we compare the center and edge stress values to plate theory.

Cancel the **Query Subgrid Results** menu.

σ_{yy} for Facesheet

Results Viewer > Nodal Solution > Stress > Y-Component of stress

Click **Plot Results** icon. The snapshot below shows the σ_{yy} contours.



Using **Query Results**, I get the following values for the bottom of the facesheet (see snapshot above):

1. At the center, $\sigma_{yy} = -28$ MPa.
2. At the clamped edge location ($x=0, y=L1$), $\sigma_{yy} = 77$ MPa. What are your corresponding values?

σ_{xx} for Stiffeners

Currently, the facesheet is selected and the stiffeners are unselected. To plot stress contours for the stiffeners, we'll unselect the facesheet and select the stiffeners i.e. reverse the selection. This can be done using the **Invert** operation:

Select Entities menu > Invert

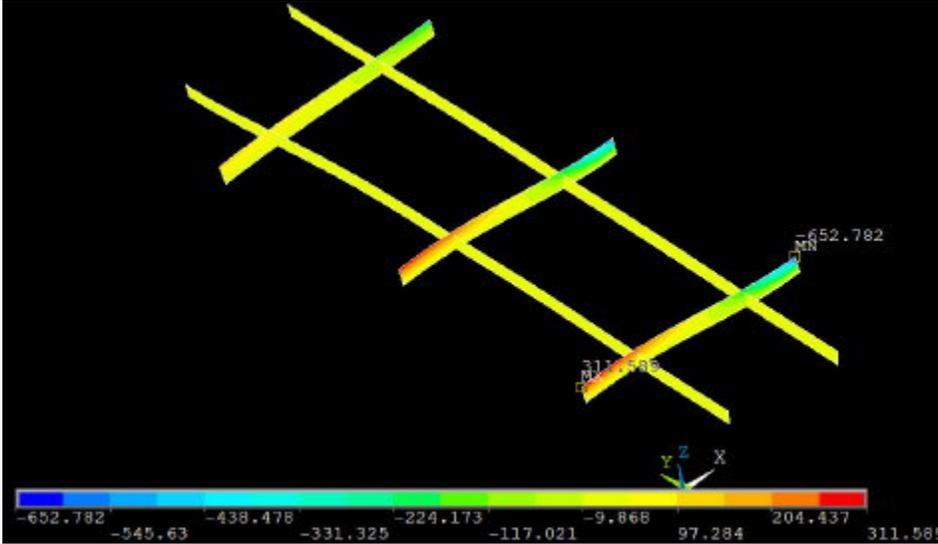
Select Entities menu> Replot

(In MS-Windows, you can cycle through windows on the desktop by holding down the *Alt* key and repeatedly pressing the *Tab* key. In case the *Select Entities* menu has disappeared, exit the *Results Viewer*, bring up the *Select Entities* menu and re-enter the *Results Viewer*.)

Results Viewer > Nodal Solution > Stress > X-Component of stress

Click *Plot Results* icon.

Manipulate the view so that you are looking down on the top of the stiffeners as in the snapshot below. You might have to flip the model over.



The σ_{xx} values for the stiffeners in the y-direction is virtually zero; confirm this using *Query Results*. This means that, as expected, the load in the x-direction is carried by the stiffeners in that direction.

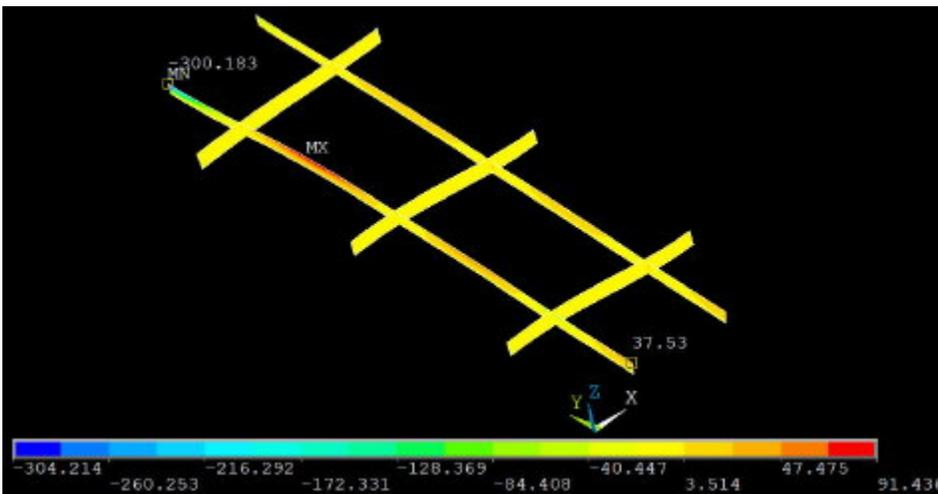
Let's determine σ_{xx} values at the top of the x-stiffener closest to the center; we'll compare these values to plate theory in [Step 9](#). Using *Query Results*, I get the following values at the top of this stiffener (see snapshot above):

1. At $(x=0, y=L_1/6)$, $\sigma_{xx}=312$ MPa.
2. At the clamped edge location $(x=W_1, y=L_1/6)$, $\sigma_{xx}=-653$ MPa. What are your corresponding values?

σ_{yy} for Stiffeners

Results Viewer > Nodal Solution > Stress > Y-Component of stress

Click Plot Results icon.



The σ_{yy} values for the stiffeners in the x-direction is virtually zero; confirm this using *Query Results*. This means that, as expected, the load in the y-direction is carried by the stiffeners in that direction.

Let's determine σ_{yy} values at the top of the y -stiffener closest to the center which we'll compare to plate theory in [Step 9](#). Using **Query Results**, I get the following values at the top of the stiffener (see snapshot above):

1. At $(x=W_1/4, y=0)$, $\sigma_{yy} = 38$ MPa.
2. At the clamped edge location $(x=W_1/4, y=L_1)$, $\sigma_{yy} = -300$ MPa. What are your corresponding values?

Save: **Toolbar > SAVE_DB**

Go to [Step 9: Validate the Results](#)

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

Go to all [ANSYS Learning Modules](#)

- 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 9: Validate the results

We checked in the previous step that the BC's have been applied correctly and the deflection looks plausible.

Comparison With Plate Theory

An FEA analyst displays his mettle by going head-to-head with theory. The theoretical solution for a clamped plate subjected to pressure is presented by Timoshenko and Woinowsky-Krieger, page 202. This solution can be modified to account for the presence of stiffeners using the parallel axis theorem. This calculation and the resulting stress and displacement values have been generously provided by Prof. Alan Zehnder and are summarized in [this pdf document](#). Do take the time to review [this document](#).

The head-to-head comparison of the analytical values with the FEA values from [Step 8](#) at the center of the structure is presented in the following table.

Entity	Location	FEA	Theory
w		3.8 mm	3.6 mm
σ_{xx}	Bottom	-56 MPa	-50 MPa
σ_{xx}	Top	312 MPa	350 MPa
σ_{yy}	Bottom	-28 MPa	-20 MPa
σ_{yy}	Top	38 MPa	105 MPa

The deflection at the center is about 8% larger than the theoretical value due to shear deformation in the FEA model that is not accounted for in the analytical result. Correlation of stresses in the center is reasonably good. It might improve if solution for anisotropic plate were used. Also, the stiffeners are not at the centerline of the model, while maximum stresses in theory are calculated at the center.

The head-to-head comparison of the *edge* stresses is given in the following table.

Entity	Location	FEA	Theory
σ_{xx}	Bottom	116 MPa	100 MPa
σ_{xx}	Top	-653 MPa	-704 MPa
σ_{yy}	Bottom	77 MPa	78 MPa
σ_{yy}	Top	-300 MPa	-403 MPa

There is good correlation of the edge stress on the bottom of facesheet but the edge stress at the top of the stiffeners is overestimated by the theory. This is possibly due to the stiffeners not being at the centerline of the model while the maximum stresses in theory are calculated at the center.

Back to [Problem Specification](#)

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

Go to all [ANSYS Learning Modules](#)