ANSYS - Semi-monocoque shell - Step 5

Step 5: Mesh geometry

As discussed in the previous step, we'll first mesh the subsection *AEFG* 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
NDIV_X	3	No. of divisions for edges along *x*
NDIV_Y	6	No. of divisions for edges along *y*
SIZE_Z	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.

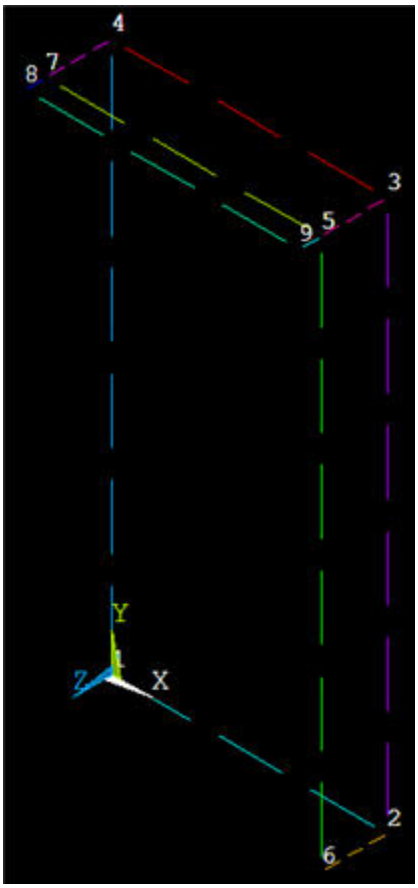
If you have trouble selecting the correct line below, **hold down** the left mouse button until the line is selected and then release the left button. If you want to **deselect** a line, right-click to go into deselect mode, left-click on the line to be deselected and right-click again to go back into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

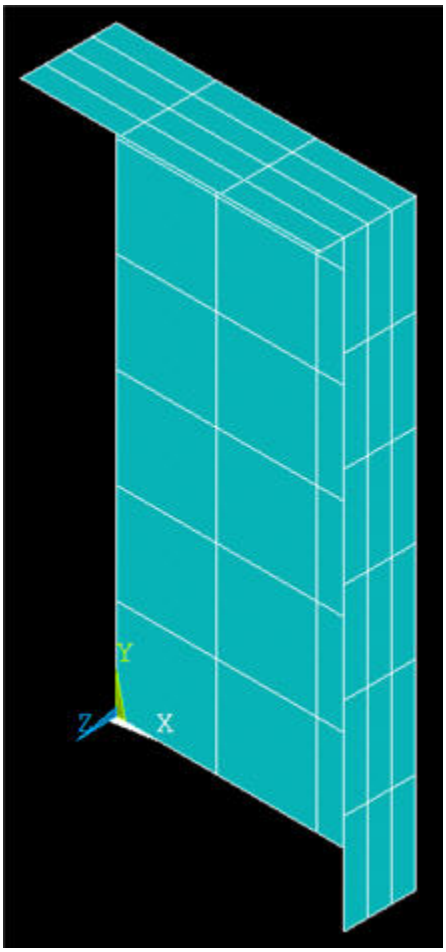
Click on Mesh in *MeshTool*. Pick area A2 and click OK. The resulting mesh for the plate is displayed.

Generate Mesh for Horizontal Stiffener

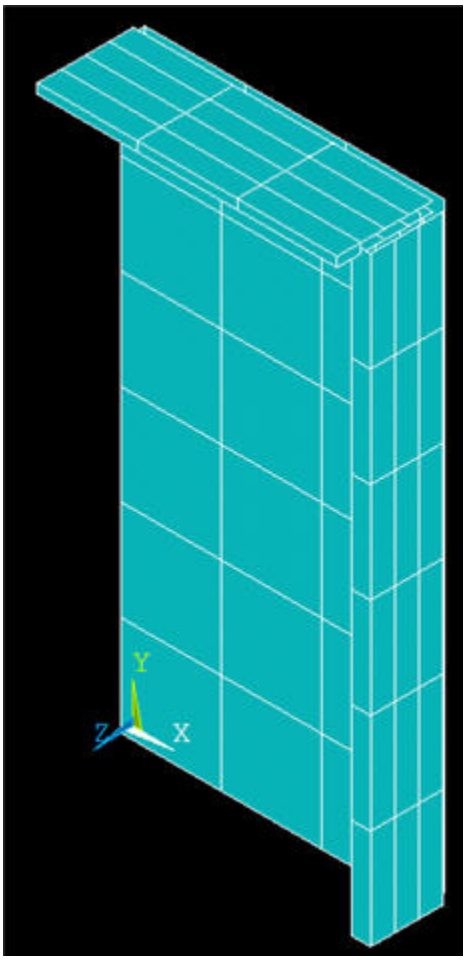
Under ***Element Attributes***, click on ***Set***. Set Real constant set number to 3 and click OK.

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 `/esh ape, 1` and then select **Utility Menu > Plot > Replot**. When the */ESHAPE* command in issued, ANSYS uses the real constants associated with each element to determine its shape. (Note that commands are not case-senstive.)



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:

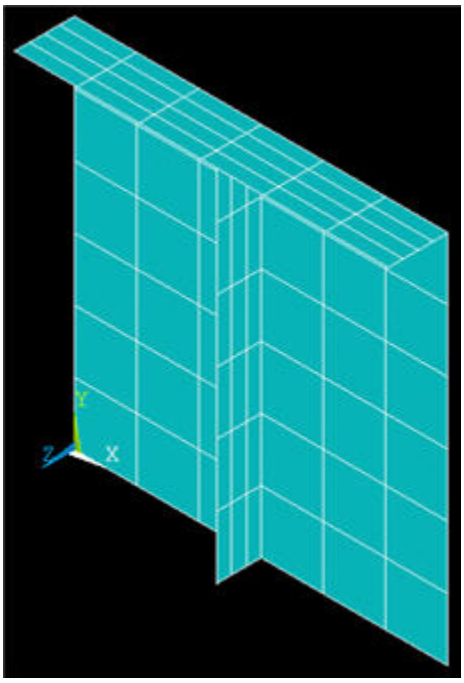
```
/eshape,0
/replot
```

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***.

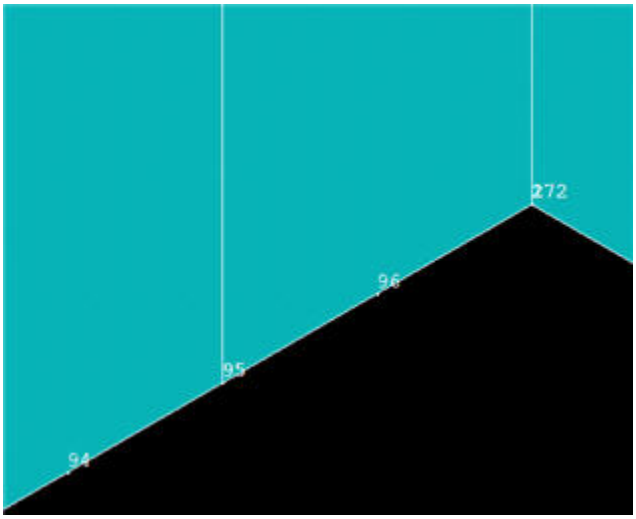
Since the associated keypoints are also copied over, you'll notice that there is a collection of two coincident but separate keypoints along shared boundaries where the original and copied entities overlap. It's key that we [merge these coincident keypoints|SIMULATION:Semi-monocoque shell page 5]; we'll 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-Plot Ctrls > 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 snaphsot 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.

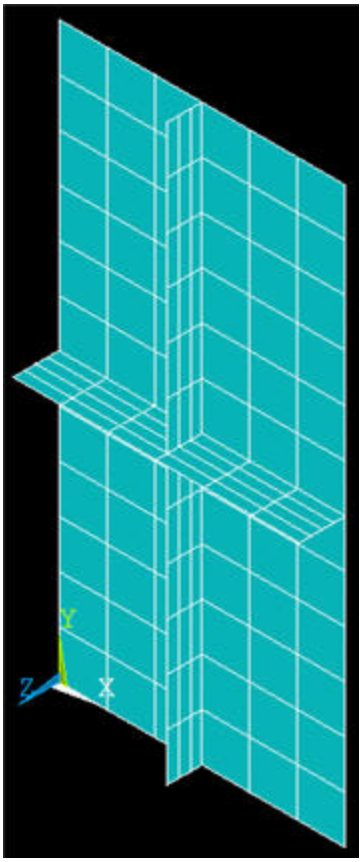
Copy areas: **Main Menu > Preprocessor > Modeling > Copy > Areas**

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.

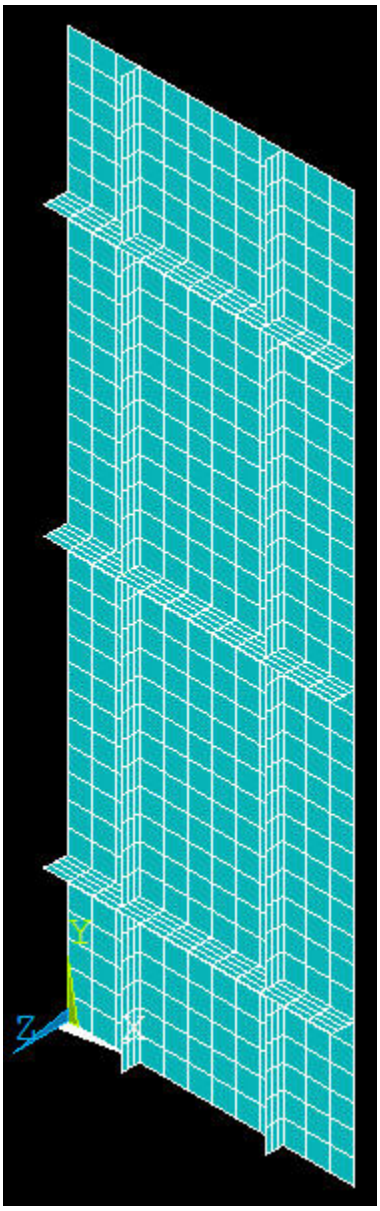
Copy areas: **Main Menu > Preprocessor > Modeling > Copy > Areas > Pick All**

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.

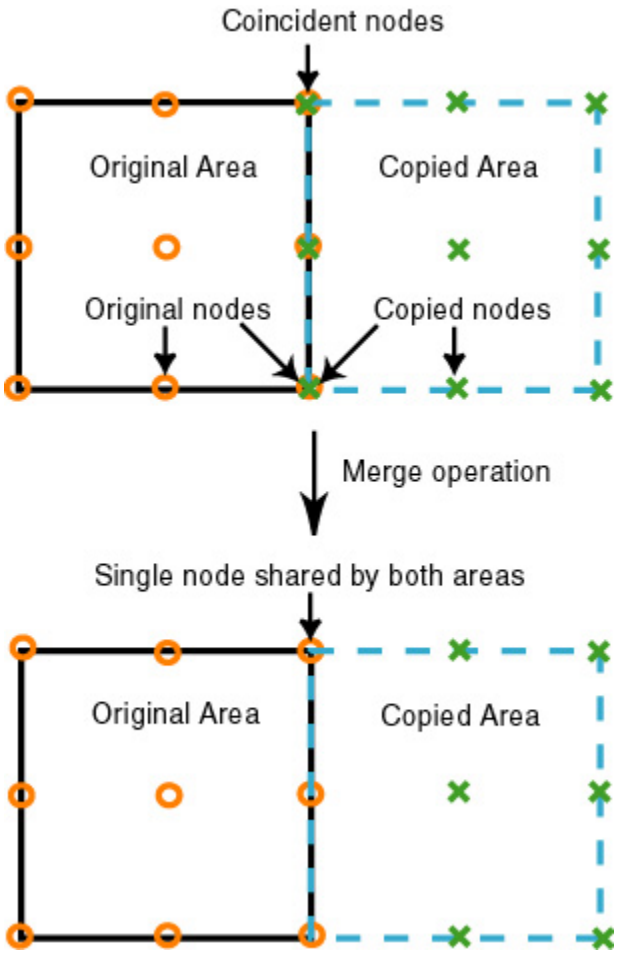
Copy Areas menu: ***Number of copies*** = `NSY`; ***Y-offset*** = `L1/NSY`; ***X-*** and ***Z-offsets*** should be blank or zero. Click ***OK***.

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.



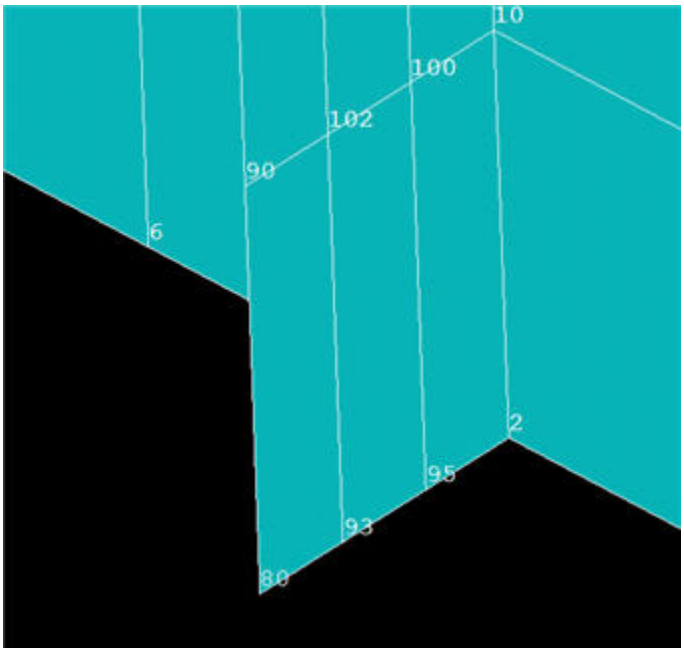
View documentation about the merge utility: **ANSYS Help > Contents > Modeling and Meshing Guide > Number Control and Element Reordering > Number Control**

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> Numbering Ctrls> 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 Ctrls> 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