

ANSYS - Semi-monocoque shell - Step 4

Problem Specification

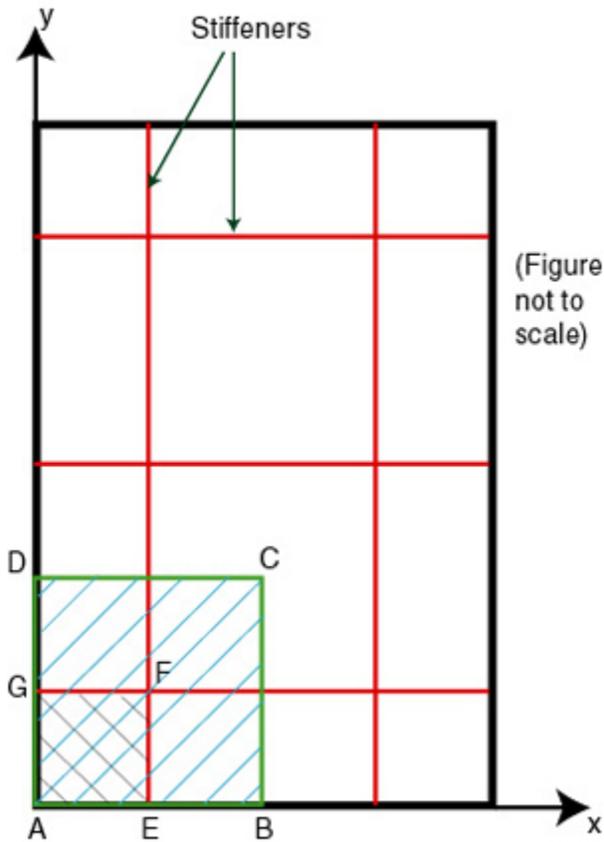
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. **Specify geometry**
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results

Step 4: Specify geometry

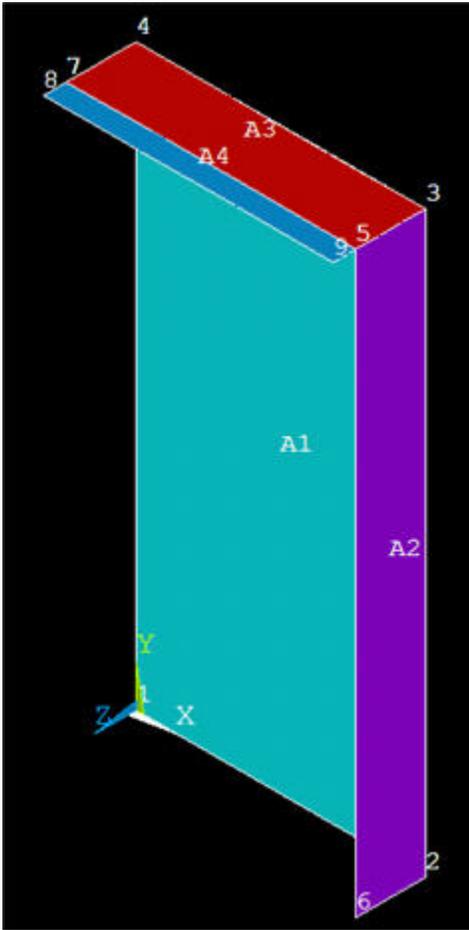
Strategy for Geometry Creation

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 *AEFG* 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 *AEFG* 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

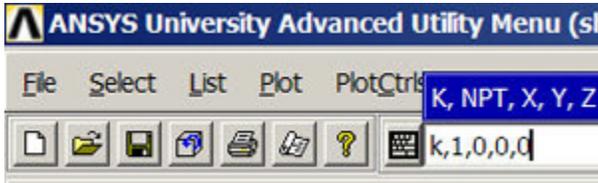
In previous tutorials, we have defined keypoints (i.e. vertices) using menus. Here, we will instead use the command line to create keypoints. The two methods are equivalent but the latter is faster for more experienced users. Let's check the documentation to find the command for creating keypoints. In 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.

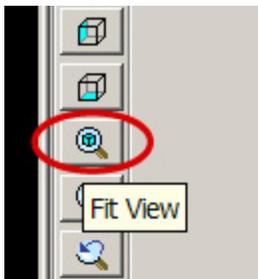


Similarly, create keypoints #2-#4 in the above table by entering the following at the command prompt:

```
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.

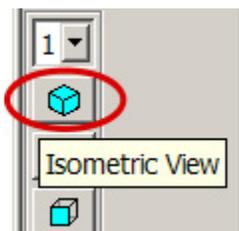


To create an area from keypoints #1-#4, enter `a, 1, 2, 3, 4` at the command line. Note that ANSYS automatically assigns the label *A1* to this first area. This area corresponds to the plate.

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**.

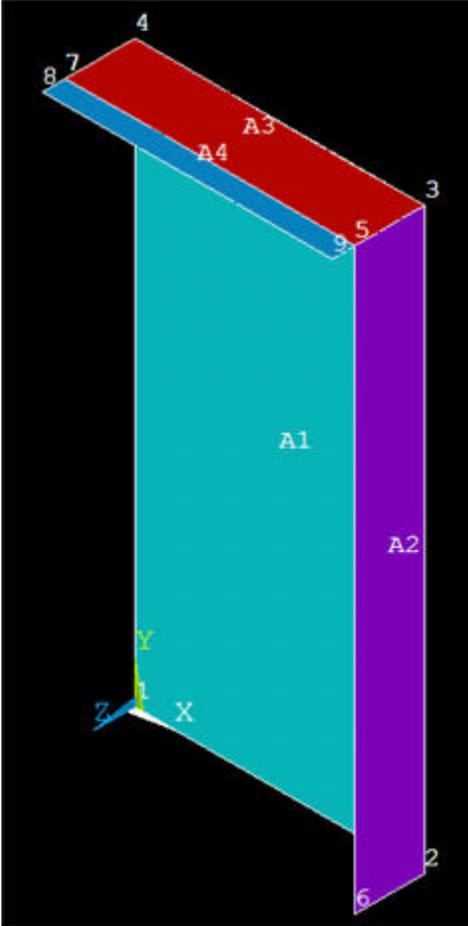


Now that we have gotten the hang of this business, we can create the remaining keypoints and areas in our sleep (remember to wipe the drool off the keyboard):

```
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](#)