

ANSYS - Semi-Monocoque Shell, Part 2 - Step 2

Problem Specification

1. Create input file

2. Modify and read input file

Step 2: Modify and read input file

Modify Input File

We need to change the number of stiffeners in the x-direction. In the input file, look for the line where the value of NSX is specified. Set $NSX=3$ as shown below and save the input file.

```
*SET, H3, 20
*SET, NSX, 3
*SET, NSY, 3
```

Read Input File

Create a folder called *shell2* at a convenient location. Start ANSYS:

... > ANSYS 10.0 > ANSYS Product Launcher

Enter the location of the folder *shell2* that you just created as your *Working directory* by browsing to it.

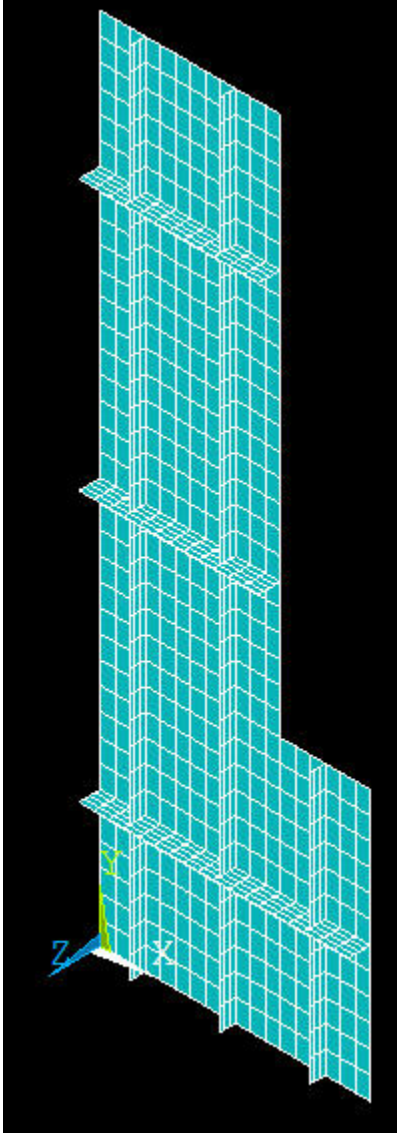
Enter *shell2* as your *Initial jobname*. Click on *Run*.

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

Utility Menu > File > Read Input from

Select the *shell_step6.inp* file and click **OK**. ANSYS will run through the commands in *shell_step6.inp* i.e. repeat steps 1-6 with $NSX=3$.

Plot elements and select the isometric view.



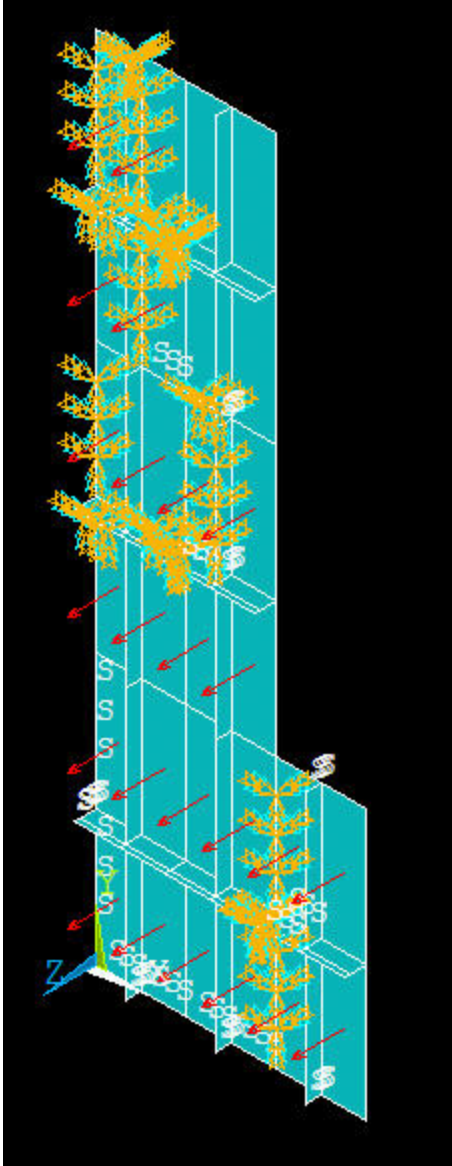
We see that there is a problem with the resulting geometry. There is a part of the geometry that is not copied in the y -direction. Why is this so? We'll look into this in just a minute.

Let's check what BC's have been applied.

Utility Menu > Plot > Areas

Utility Menu > PlotCtrls > Symbols

Symbols menu: For **Boundary condition symbol**, select **All Applied BCs**. For **Surface Load Symbols**, select **Pressures** and under **Show pres and connect as**, select **Arrows**. Click **OK**.



We see that the symmetry and clamped BC's haven't been applied at the correct locations. The pressure has been applied only over part of the facesheet. We'll fix these mistakes below.

Fix Geometry

Recall that originally we made *two* copies of subsection ABCD. Now we are making *three* copies of ABCD. However, the input file still insists on making only two copies; this is inherited from the original case. Let's dig a little further into this mystery and see how we can solve it.

When we made the copies in the y-direction, we selected the components to copy using "Pick All" in the pick menu. Go to near the end of your input file and find the corresponding statement:

```
AGEN,NSY,P51X,,,L1/NSY,,,0
```

The *P51X* in the above statement corresponds to our "Pick All" selection. It turns out that when we increase the number of components to be selected, "Pick All" doesn't pick up the **&\$\$#\$*** additional components (pardon my language). We can get around this by using "ALL" instead of "Pick All" in the command. Modify the AGEN statement in the input file as follows:

```
AGEN,NSY,ALL,,,L1/NSY,,,0
```

Save the input file. This will fix the problem with the geometry.

Fix BC's

Recall that, while applying the BC's, we selected the appropriate subset of components and applied the BC's to the selected components using "Pick All" in the pick menu. As above, "Pick All" mangles the selection when we change the number of components. We can again fix this by using "ALL" instead of "Pick All" in the BC commands.

In your input file, you'll find two instances of the statement:

```
DL,P51X, ,SYMM
```

This corresponds to the application of the symmetry condition along the appropriate edges. Modify this to

```
DL,ALL, ,SYMM
```

Similarly, you'll find two instances of the statement:

```
DL,P51X, ,ALL,
```

This corresponds to the application of the clamped condition along the appropriate edges. The "ALL" in the above statement implies that we are constraining all degrees of freedom at the clamped edges. Modify this to

```
DL,ALL, ,ALL,
```

At the very end of your input file, you'll see the statement that imposes a pressure on the faceplate:

```
SFA,P51X,1,PRES,0.05
```

You know the routine now; modify this to

```
SFA,ALL,1,PRES,0.05
```

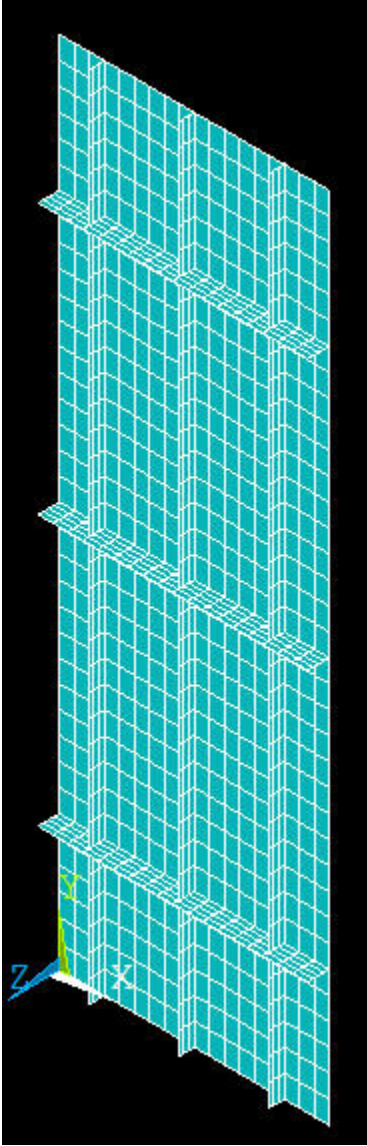
Save the input file.

Re-read Input File

Main Menu > File > Clear & Start New > OK

Utility Menu > File > Read Input from > *shell_step6.inp* > OK

Select the *shell_step6.inp* file and click **OK**. Plot elements and select the isometric view. Did ANSYS get it right this time?



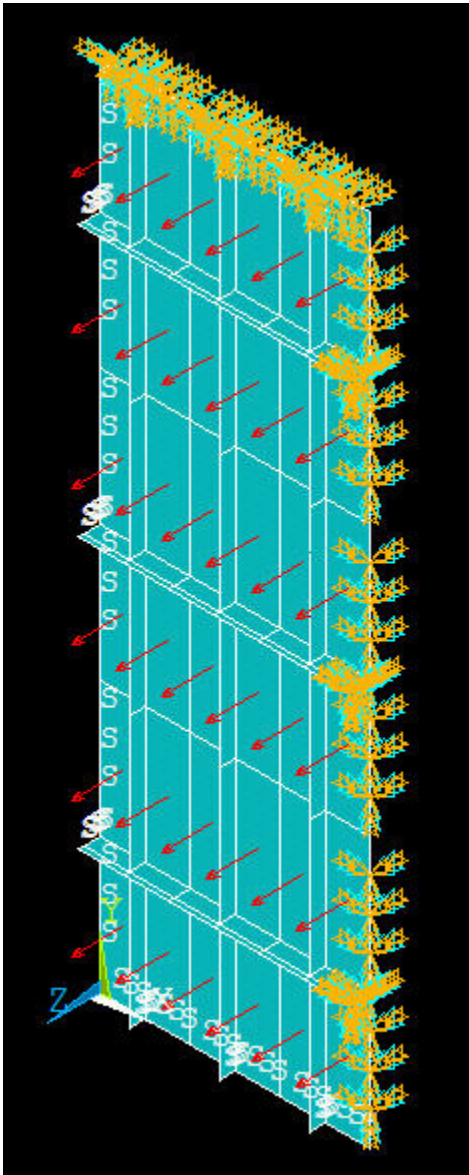
Check that the BC's have been applied correctly.

Utility Menu > Plot > Areas

Utility Menu > PlotCtrls > Symbols

Symbols menu: For **Boundary condition symbol**, select **All Applied BCs**. For **Surface Load Symbols**, select **Pressures** and under **Show pres and convect as**, select **Arrows**. Click **OK**.

Zoom into the sides of the structure and carefully check the BC's.



Now repeat steps 7-9 in the [original tutorial](#) and compare the results for $NSX=2$ and 3.

A tip from a battle-hardened user: you have to be careful about recycling the log file when the topology of the geometry changes. For instance, if you change $H2$ and $H3$ such that the y-stiffeners become wider than the x-stiffeners, the topology changes significantly. In this case, unless you are a power user, it might be advisable to use the log file only for steps 1-3 (as you look through the log file, you should be able to hone in on where step 3 ends and step 4 begins). Then, go through steps 4-6 in the GUI and generate clean new lgw/input files.

Save: Toolbar > SAVE_DB

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

Go to [all ANSYS Learning Modules](#)