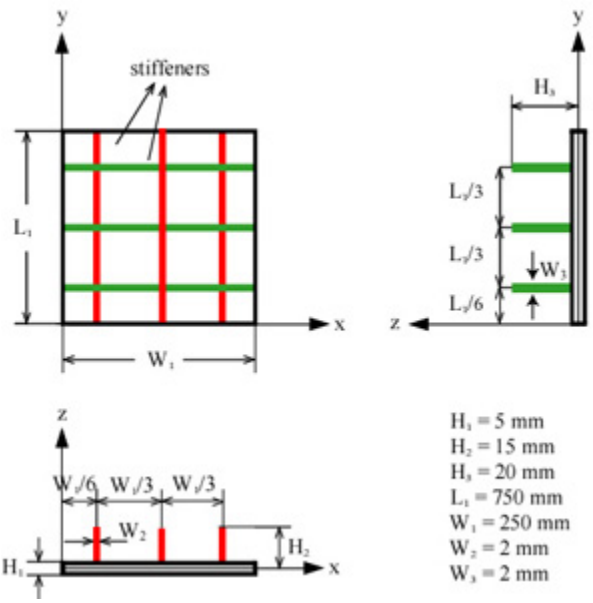
ANSYS - Semi-Monocoque Shell, Part 2

Author: Rajesh Bhaskaran, Cornell University

Problem Specification
1. Create input file
2. Modify and read input file

Problem Specification

Consider the semi-monocoque shell problem analyzed in a previous tutorial. Redo the problem changing only the number of stiffeners in the *x*-direction. For the quarter-plate, increase the number of x-stiffeners to **3** as shown below; previously, we had 2. Compare the displacement and stress values for the new and old cases and comment on the differences and/or similarities.



We'll play smart and avoid plodding through the solution procedure from scratch by using ANSYS capabilities for performing parametric studies.

Go to Step 1: Create input file

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Create input file
2. Modify and read input file

Step 1: Create input file

Generate Clean Version of Log File

Every action taken in the ANSYS GUI has a corresponding text command that is stored in the log file. At the end of step 6 for the original semi-monocoque shell problem, you were asked to create the log file for steps 1-6 by issuing the command **Utility Menu > File > Write DB log file**. We will create an input file from this log file. Copy this log file to your working folder.

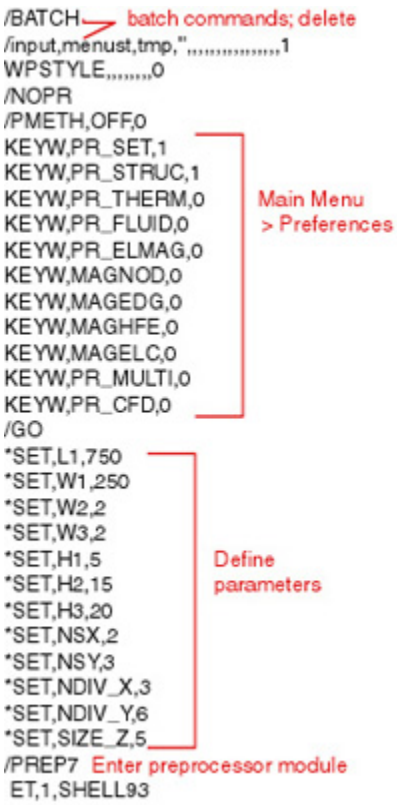
Your log file will contain the mis-steps you made while solving the problem. You don't want these mis-steps to be included in your input file since ANSYS will religiously run through these everytime you execute the input file. There are two ways around this:

1. You can go in and edit the log file to remove the mistakes. This takes an intimate familiarity with ANSYS commands and is recommended only for the power user.
2. You can start from scratch and go through steps 1-6 again being careful not to make any mistakes. The resulting log file will be a clean version without mis-steps.

Generate a clean version of the log fileshell_step6.lgw for steps 1-6. If you don't have the log file, I'll be nice and give it to you (right-click on this link to download).

Generate Input File from Log File

In your working folder, make a copy of the log file *shell_step6.lgw* and call it *shell_step6.inp*. The *inp* extension signifies that it is an input file. Bring up your pet text editor and load in *shell_step6.inp*. The top several lines, along with annotations, are shown below:



You can run ANSYS in the interactive mode or batch mode (in which you do not use the GUI). By default, ANSYS assumes you'll be running in batch mode; so it'll put a couple of batch-mode commands at the top of your log file. Since we are going to use the input file in the interactive mode, delete the first *two* commands above (/BATCH and /input) that pertain to the batch mode.

To see what the *KEYW* command does, type help,KEYW at the command prompt. At the bottom of the *KEYW* help page, you can see that this command is equivalent to the menu selection **Main Menu > Preferences**.

The **SET*command above is used to specify parameter values. To redo the original problem with a new set of parameter values, you just change the values of appropriate parameters and read in the input file. We'll do this shortly. As you can imagine, this is an efficient way to explore the effect of key parameters on your design.

In the snippet above, you enter the preprocessor module using the */PREP7* command; this is equivalent to selecting **Main Menu>Preprocessor i**n the GUI. Once you are in the *Preprocessor*, you can access its utilities such as specifying the element type (*ET* command). A key thing to remember is that you have to be in the appropriate module (*Preprocessor*, *Postprocessor*, etc) to access the corresponding utilities. If you are, say,in the *Postprocessor* module and try to access a *Preprocessor* command such as ET, ANSYS won't recognize the command. The *Utility* menu commands can be accessed from any module.

Go to Step 2: Modify and read input file

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

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.

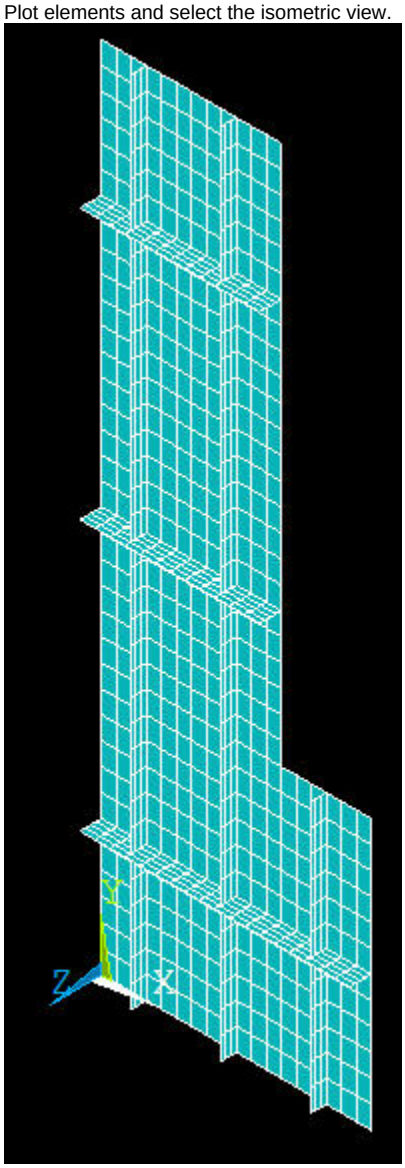
Enter shell2 as your ***Initial jobname***. Click on ***Run***.

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.



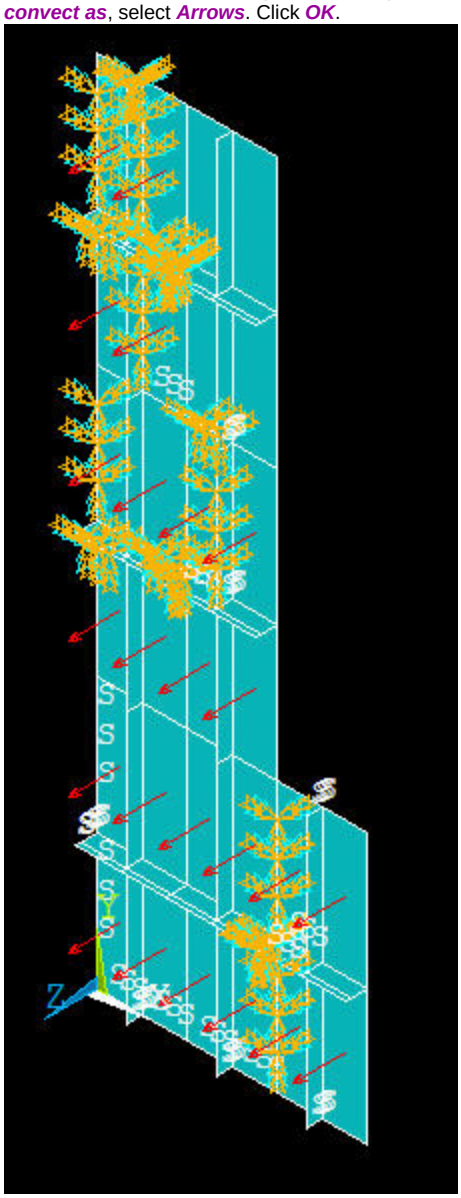
We see that there is a problem with the resulting geometry. There is a part of the geometry that is not copied in the *y*-direction. Why is this so? We'll look into this in just a minute.

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 convect 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 approporiate 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

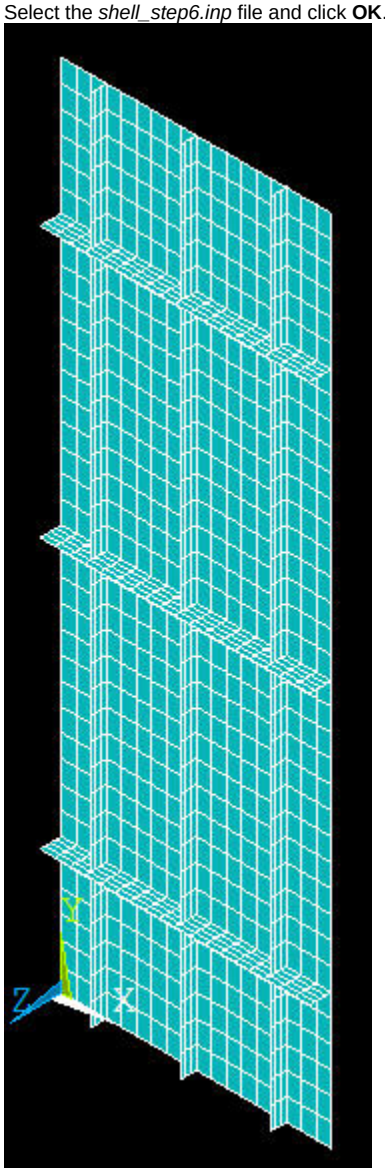
Save the input file.

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?



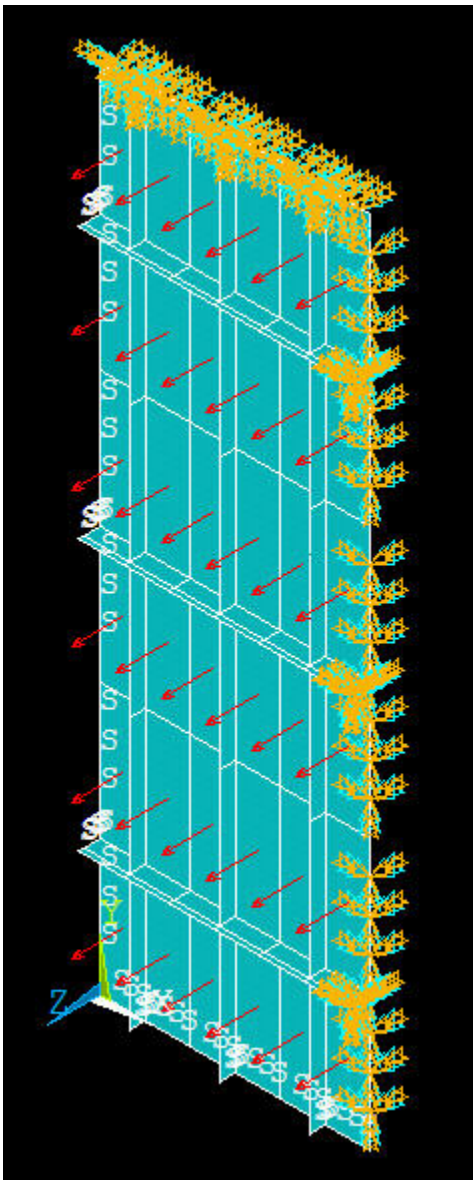
Check that the BC's have been applied correctly.

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