ANSYS - Semi-monocoque shell - Step 1

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/mm^2 for Young's modulus and pressures. The resulting stresses will be in N/mm^2 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