

ANSYS - Semi-monocoque shell - Step 1

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
9. Validate the results

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/mm² for Young's modulus and pressures. The resulting stresses will be in N/mm² 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](#)