

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](#)