

ANSYS - Semi-monocoque shell - Step 7

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 7: Solve!

Enter Solution Module: **Main Menu > Solution**

Enter *check* in the *Input* window. If the problem has been set up correctly, there will be no errors or warnings reported. You should see the following message in the *Output* window: The analysis data was checked and no warnings or errors were found.

Main Menu > Solution > Solve > Current LS

Review the information in the */STATUS Command* window. Close this window.

Click **OK** in *Solve Current Load Step* menu.

ANSYS performs the solution and a window should pop up saying "Solution is done!". Close this window.

Save: **Toolbar > SAVE_DB**

Go to [Step 8: Postprocess the results](#)

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

Go to [all ANSYS Learning Modules](#)