

ANSYS - Semi-Monocoque Shell, Part 2 - Step 1

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 file shell_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:

```
/BATCH batch commands; delete
Enter preprocessor module
ET,1,SHELL93
```

Main Menu
> Preferences

Define
parameters

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** in 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](#)