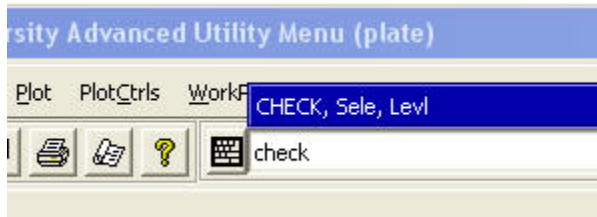


# ANSYS - Crank 7 Solve Content

## Step 7: Solve!

Before we start the solution, we should check our model for errors. Enter check in the *Input* window and press Enter.



All warnings and errors found will be displayed in the Output Window. You might see warnings regarding the size of several mesh elements. These warnings can be ignored due to the large overall number of elements in our model.

Now we're finally ready to kick back and let ANSYS do some of the work.

Main Menu > Solution > Solve > Current LS

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



Again ignore any warnings that may pop up. ANSYS should cheerfully report "Solution is done!"

Verify that ANSYS has created a file called *crank.rst* in your working directory. This file contains the results of the (previous) *solve*.