- **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**

Select in Main Menu:

## Solution > Solve > Current LS

Review the information in the /STAT Command window.

	s	0	L	U	T	I	0	N		0	P	τ	I	0	NS
PROBLEM DIMENSIONAL DECREES OF FREEDOM.	IT	۲.		•	•	ŵ	e.	•	÷	•					2-D
ANALYSIS TYPE	-	5	÷	÷			•		-	-			•		STATIC (STEADY-STAT
1	6 (		1	D	8	1	1	EP		0	) 1	1	r I	1	ONS
LOAD STEP NUMBER				÷.					-						1
NUMBER OF SUBSTEPS.			5		:	:	:	:	1	2	:	:	:	:	1.0000
STEP CHANGE BOUNDARS	1.0	200	(D)	ITI	ON	KS.									NO
PRINT OUTPUT CONTROL	LS.	ŵ,	2			٠	٠		-		٠	٠	٠		NO PRINTOUT
DRINBASE OUTPUT CON	1 160	76.6	•			٠	٠		-	٠	٠	٠	٠	•	FOR THE LAST SURSTE

Close this window.



Click OK in Solve Current Load Step dialog box\_.\_

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

## Save your work

Click on SAVE\_DB in the ANSYS Toolbar to save the database.

Go to Step 8: Postprocess the results

Go to all ANSYS Learning Modules