

ANSYS - Truss Problem Set 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

Problem Set 1

Problem Set 2

Problem Set 1

Consider the case where the displacement constraints at A and C are interchanged i.e.

- at A, only UX is set to zero
- at C, both UX and UY are set to zero

1. How would you expect the reaction forces at the supports A and C to change?
2. What can you say about how the x-component of the forces in the truss will change?

Re-solve the truss problem with the modified constraints. From your ANSYS solution:

1. List the reactions. Note that you can save the reaction listing as follows: in the window that comes up with the listing of the reaction forces, click on:

File > Save As

2. Determine the force in each truss member and whether the member is in tension or compression.

Tips: Start from your ANSYS solution to the truss tutorial problem but use a different job name as follows:

- Start the ANSYS Product Launcher. Specify the same directory as for the tutorial but use a *different jobname*. Once ANSYS comes up, select: **Utility Menu > File > Resume from**. Choose *truss.db* and click **OK**.

Note that you can delete constraints using:

Main Menu > Preprocessor > Loads > Define Loads > Delete

It works similar to how you apply loads.

You can plot the displacement constraints in the Graphics window as follows:

Utility Menu > Pltctrls > Symbols

Select **All Applied BCs** for **Boundary condition symbol**. Click **OK**.

You might have to use :

Utility Menu > Plot > Replot

or

Utility Menu > Plot > Multi-plots

for the constraint symbols to appear in your plot.

Go to Problem Set 2

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

[Go to all ANSYS Learning Modules](#)