

# 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](#)