# **ANSYS - Truss Step 6**

**Problem Specification** 

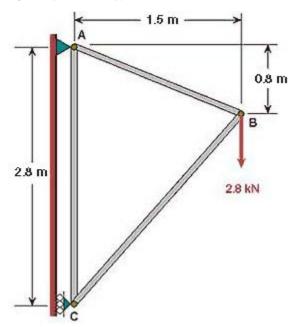
- 1. Start-up and preliminary set-up
- 2. Specify element type and constants
- 3. Specify material properties
- 4. Specify geometry
- 5. Mesh geometry
- 6. Specify boundary conditions
- 7. Solve!
- 8. Postprocess the results
- 9. Validate the results

Problem Set 1

Problem Set 2

## Step 6: Specify boundary conditions

Next, we step up to the plate to define the boundary conditions, namely, the displacement constraints and loads. Note that in ANSYS terminology, the displacement constraints are also "loads". We can apply the loads either to the geometry model or to the finite-element model (that is to the elements and nodes directly). The advantage of the former is that one doesn't have to re-specify the constraints on changing the mesh. So we'll apply the constraints to the geometry i.e. to the keypoints.

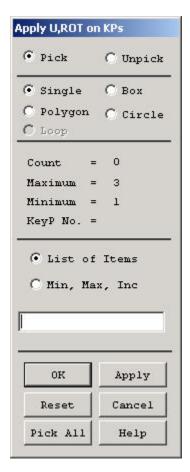


You can see from the diagram that the pin at A is constrained in x and y directions; or equivalently, keypoint 1 is constrained such that its *UX* and *UY* displacements are zero. Similarly, keypoint 3 is constrained such that its *UX* displacement is zero.

#### **Apply Displacement Constraints**

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoints

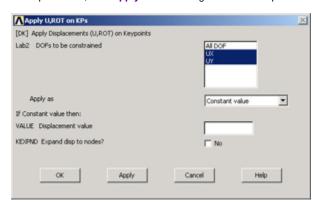
This brings up the Apply U, ROT on KPs pick menu.



In the Graphics window, click on keypoint 1;

This will draw a small square around keypoint 1 to indicate that it's been picked.

In the pick menu, click *Apply*. The following menu shows up.

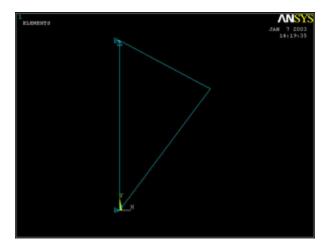


Since we want to constrain *UX* as well as *UY* to zero at keypoint 1, select both *UX* and *UY* from items in *DOFs to be constrained* list. Since the *Displace ment value* is zero by default, leave that field empty. Click on *Apply*.

You will see triangle symbols appear in the Graphics window indicating that both UX and UY DOFs are constrained at keypoint 1.

Next we apply the displacement constraint at keypoint 3. In the *Graphics* window, click on keypoint 3. In the pick menu, click *Apply*. Select only *UX* from items in *DOFs to be constrained* list. Click *OK*.

You will see a triangle symbol appear indicating that only the UX DOF is constrained at keypoint 3.



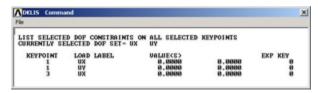
Close the Displacement and Apply menus.

#### **List Displacement Constraints**

You can verify the displacement constraints on the model by listing them.

#### Utility Menu > List > Loads > DOF constraints > On All Keypoints

This brings up a window with the constraint information.



If you made a mistake in applying a constraint, you can delete and reapply it. You can delete a constraint using

#### Main Menu > Preprocessor > Loads > Define Loads > Delete > Structural > Displacement > On Keypoints

Alternately, you can resume from your last save and continue from there.

Close the window listing the constraint information.

Save the database: Toolbar > SAVE\_DB

#### **Apply Loading**

We have only one load to apply on the structure, namely, the 2800 N force in the negative y-direction at keypoint 2 (see figure 1).

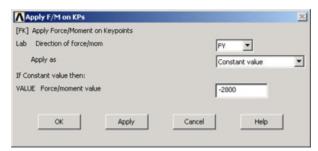
### ${\bf Main\ Menu> Preprocessor> Loads> Define\ Loads> Apply> Structural> Force/Moment> On\ Keypoints}$

This brings up the Apply F/M on KPs pick menu.

In the  ${\it Graphics}$  window, click on keypoint 2; then in the pick menu, click  ${\it OK}$ .

In the menu that appears, select FY for Direction of force/mom.

Enter -2800 for Force/ moment value. Click OK.



The negative sign for the force indicates that it is in the negative y-direction. You'll see a vector indicating the applied force in the *Graphics* window.

Close the Force/ Moment, Apply, Loads and Preprocessor menus.

Save the database: Toolbar > SAVE\_DB

Go to Step 7: Solve!

See and rate the complete Learning Module

Go to all ANSYS Learning Modules