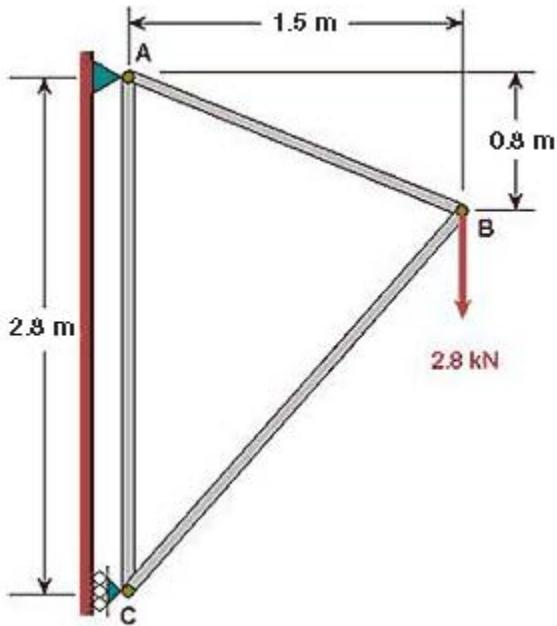


ANSYS - Truss 6 Specify boundary conditions Content

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.

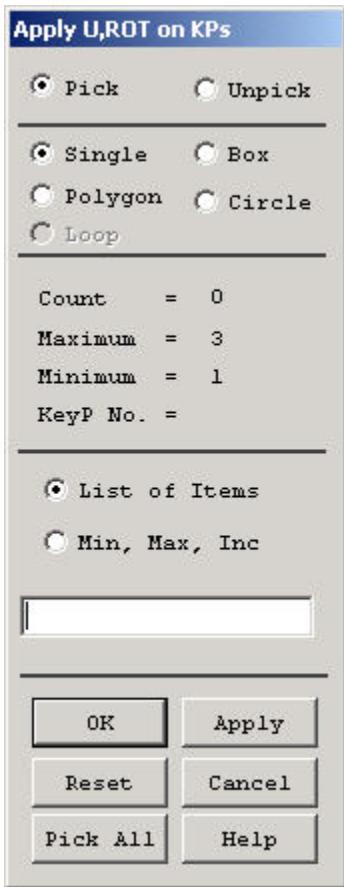


You can see from the diagram that the pin at A is constrained in x and y directions; or equivalently, keypoint 1 is constrained such that its *UX* and *UY* displacements are zero. Similarly, keypoint 3 is constrained such that its *UX* displacement is zero.

Apply Displacement Constraints

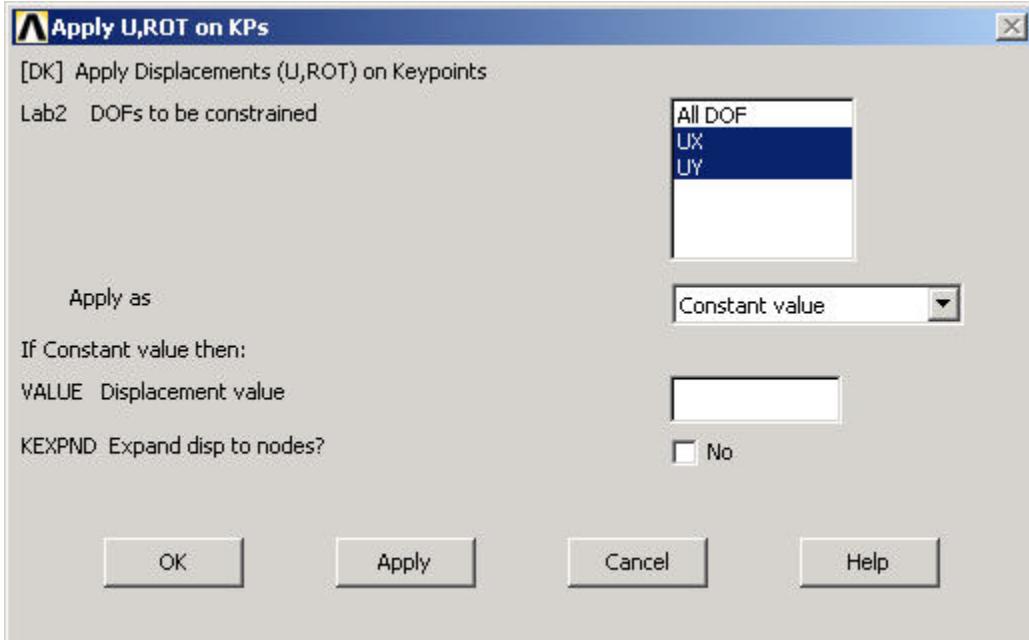
Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Keypoints

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.

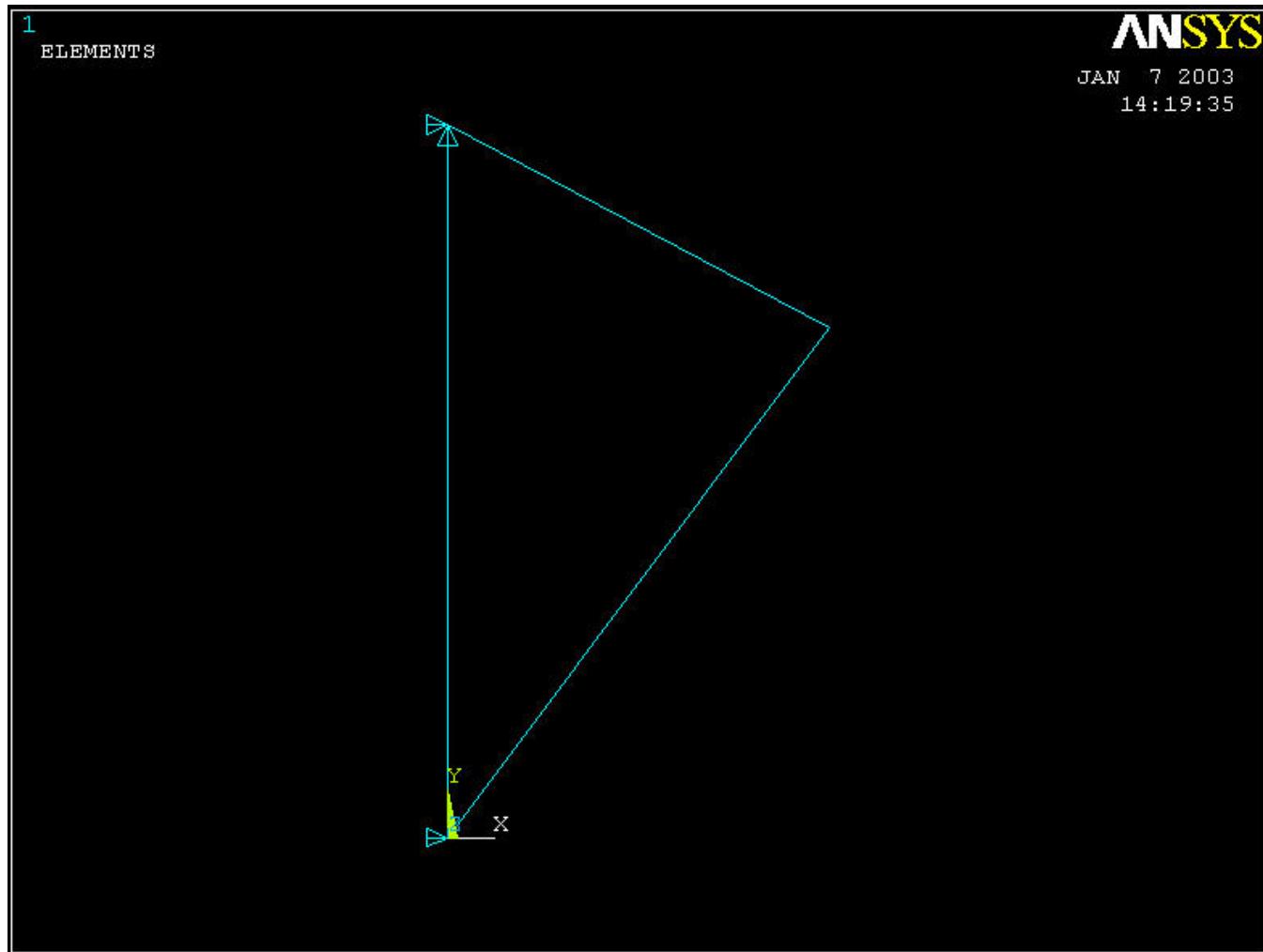


Since we want to constrain *UX* as well as *UY* to zero at keypoint 1, select both *UX* and *UY* from items in DOFs to be constrained list. Since the Displacement value is zero by default, leave that field empty. Click on *Apply*.

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.

LIST SELECTED DOF CONSTRAINTS ON ALL SELECTED KEYPOINTS CURRENTLY SELECTED DOF SET= UX UY					
KEYPOINT	LOAD LABEL	VALUE(S)		EXP	KEY
1	UX	0.0000	0.0000	0	
1	UY	0.0000	0.0000	0	
3	UX	0.0000	0.0000	0	

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).

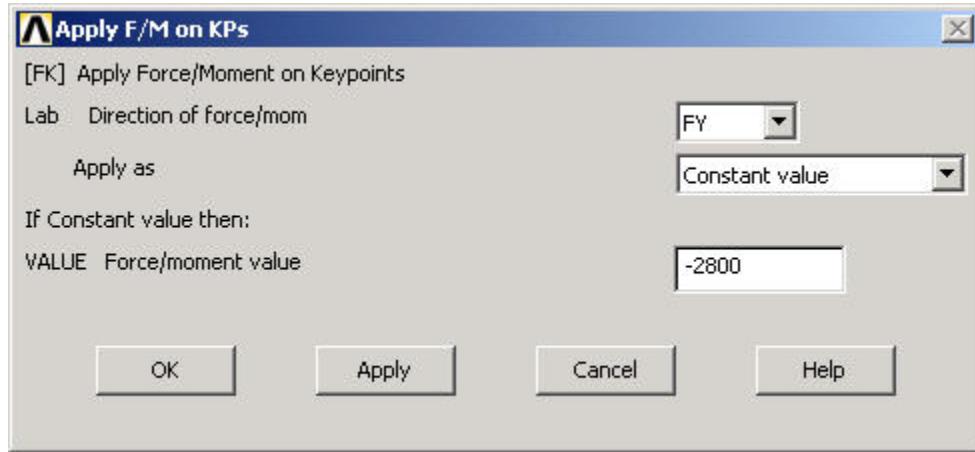
Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural >Force/Moment > On Keypoints

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

In the *Graphics* window, click on keypoint 2; then in the pick menu, click 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