Ansys Lab — Constraints

Purpose

Learn to use constraint equations in ANSYS.

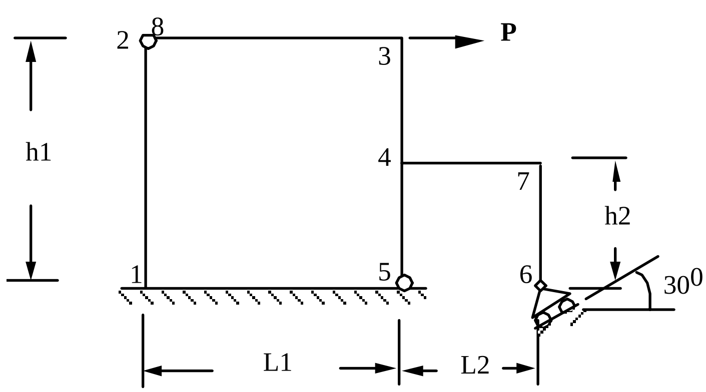
Background

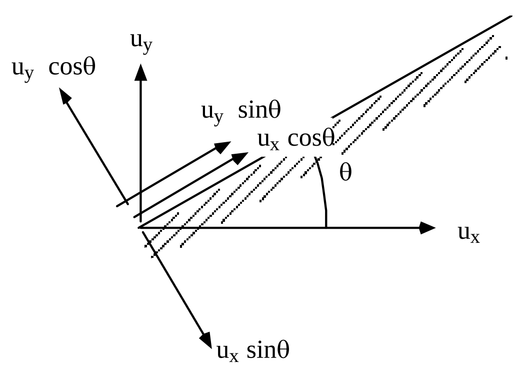
Consider analysis of the frame structure shown in the figure. The analysis procedure is similar to the one used for lab 3. The two complications are the inclined roller support and the presence of an internal hinge.

Specified displacements along global coordinate directions are handled by the standard boundary conditions commands in Ansys, as was done in the previous two labs. In the case of the inclined roller support both the x and y displacements are unknown. However the dispalcement normal to the inclined plane must be zero. For the given frame the roller support at node 6 can be modelled by defining a constraint equation as follows.

$$-u_{x6}\sin\theta + u_{y6}\cos\theta = 0$$

At the location of an internal hinge, the moment is zero. The elements joined at the hinge can have independent rotations but must have same horizontal and vertical displacements. We can use the constraint equations to model the internal hinges. We place two nodes at the location of an internal hinge and incorporate constraint equations that link the displacements of these two nodes but keep the rotations independent.





Numerical data (lbs-in units)

Load:	P=10000			
Dimensions:	L1=4000	L2=2500	h1=4500	h2=3000
Section properties:	A = 40000	I = 1600000000		h = 20
Material properties:	E = 200000			

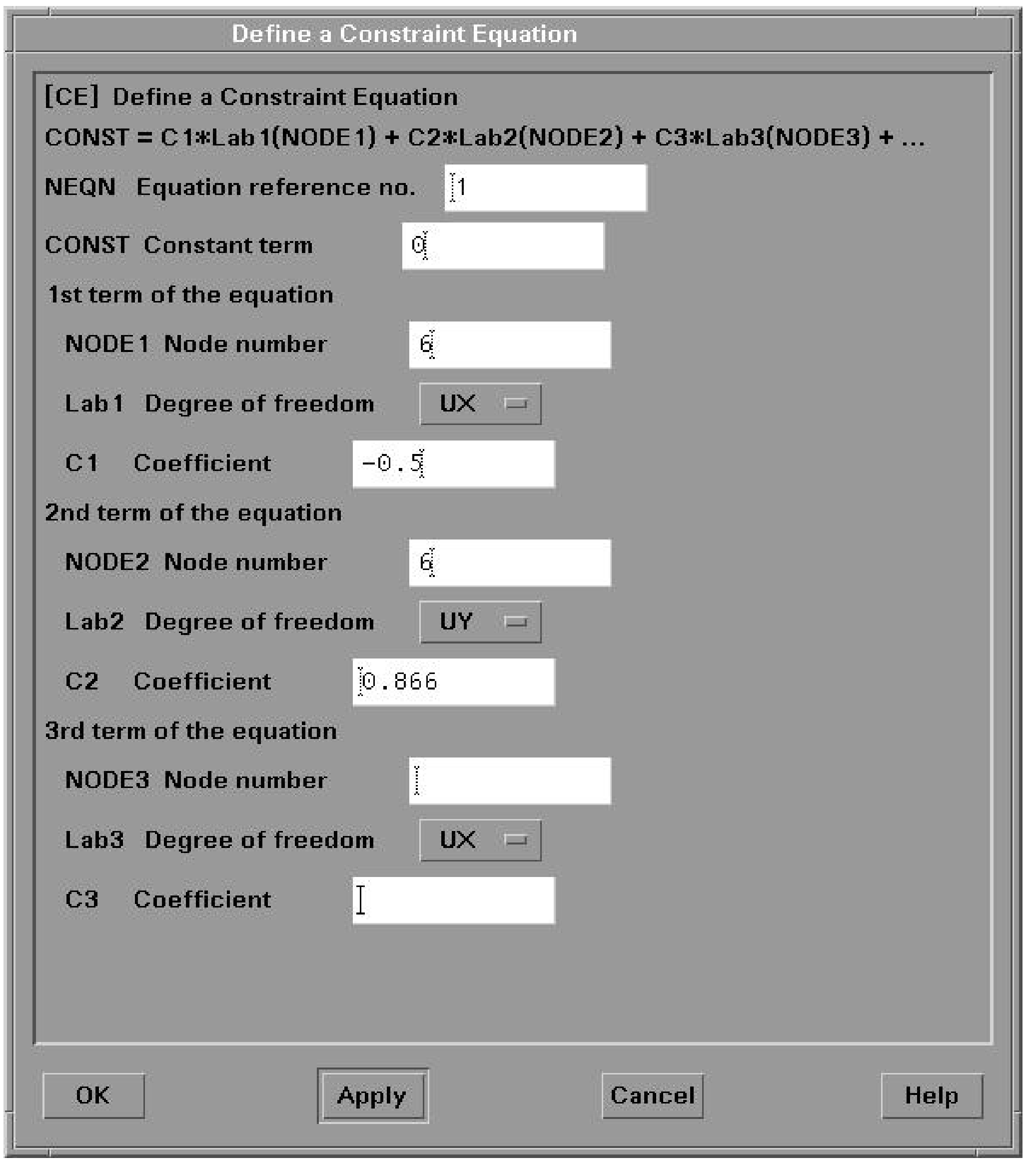
Linear constraints can be readily considered in Ansys. The general form of a linear constraint in Ansys is as follows.

$$\sum_{i=1}^{n} c_i u_i = c_0$$

where c_0 = constant, c_i = coefficient for the ith displacement. The path for defining constraint equations in ANSYS is as follow:

Main Menu > Preprocessor > Coupling/Ceqn > Constraint Eqn

A "Define a Constraint Equation" window will pop up and user need to fill in corresponding information. The following figure shows the data entered for the inclined support at node 6 of the example frame.



Lab exercise

Analyze the given frame by incorporating appropriate constraint equations. If you have entered everything correctly, you should get the following nodal displacements.

Nodal displacements

NODE	UX	UY	ROTZ
1	0.0000	0.0000	0.0000
2	0.2668	0.1202E-02	-0.8892E-04
3	0.2682	0.2370E-02	-0.3531E-04
4	0.1978	0.2770E-02	-0.5008E-04
5	0.0000	0.0000	-0.7386E-04
6	-0.2164	-0.1250	-0.1633E-03
7	0.1961	-0.1285	-0.8595E-04
8	0.2668	0.1202E-02	0.1809E-04

Lab Reports:

1) Provide the constraint equations. And how did you define them with Ansys
2) Specify all boundary conditions
3) Print out Nodal displacements list
4) Plot the deformed and unreformed body
5) Plot contour of von Mises stress