Cantilever Beam - Exercises

Author: John Singleton, Cornell University
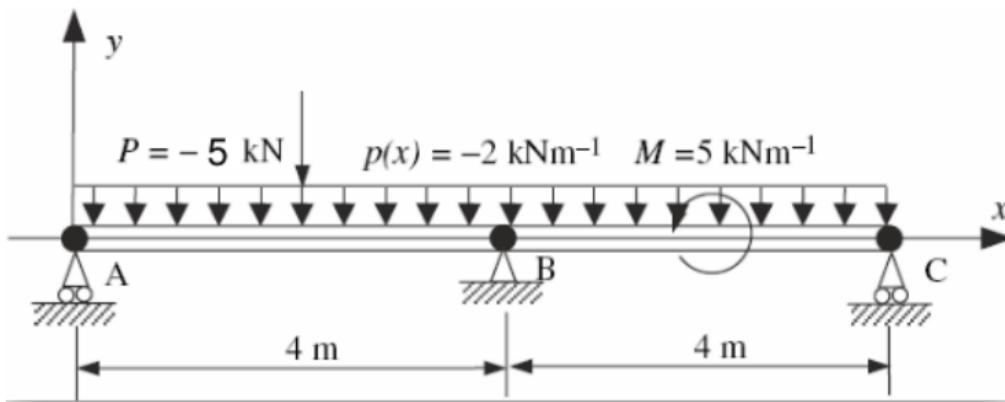
Exercises

Exercise 1

MATLAB and ANSYS beam analysis

Consider a two-span beam shown above. The beam is subjected to uniformly distributed loading, point force at x=2m and moment at x=6m about the Z-axis, as shown. The beam bending stiffness is EI=2 x 10^7 Nm^2.

Using ANSYS, plot the deflection, bending moment, and shear force distribution of the beam. How many elements are necessary to get a good solution?

Tips:

1. In ANSYS, you need to specify *E* and *I* separately. You can pick them independently as long as you get the desired *EI*. You specify *I* by specifying the cross-section as we saw in the preceding tutorial. You can pick E=2e11 (default) and calculate the equivalent square cross-section.
2. Model the geometry using four lines. You will need to have vertices where you will be applying forces, moments or displacement constraints.
3. Apply a distributed load using *Line Pressure* (see snapshot further down this page). In version 17, you can select all four lines when applying the line pressure. But in version 16 and prior versions, when you apply the line pressure, you can select only one line at a time. Since there are four lines in total, you'll need to apply line pressure four times.
4. Apply the simply supported constraints using *Supports > Displacement*. For example, the settings in the figure below can be used to apply the simply supported constraint at *A* or *C*. As we saw in the tutorial, ANSYS uses a generalized 3D beam formulation which includes *z* displacements. Since we don't have any displacement in the *z* direction, you can set the *z* displacement to zero. You really need to do this at two vertices only. This will prevent the beam from translating in the z-direction and rotating about the y-axis. Otherwise, the problem becomes ill-posed, the stiffness

matrix cannot be inverted and ANSYS will report a vague "solver pivot error".

5. You also need to add a constraint at one vertex to prevent rotation about the x axis as discussed in the truss tutorial in our free online ANSYS course at edx.org. This can be done by selecting *Supports > Fixed Rotation* as shown below.
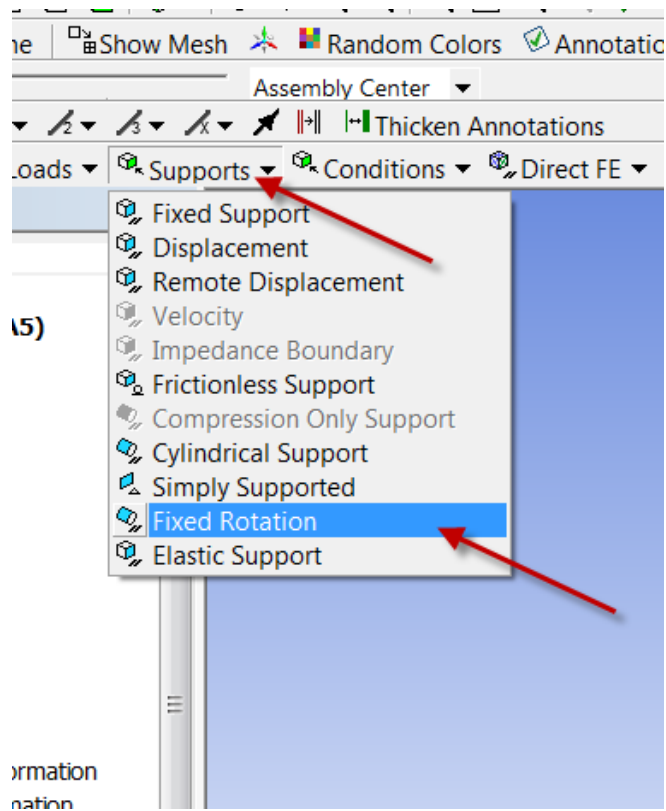
Exercise 2: Distributed Load

Using ANSYS, find the deformation and maximum bending stress of the beam when the load is applied as a distributed load. The 8 kN load is applied uniformly across the span of the beam. Thus, the line force is 8kN/4m=2kN/m.

$$w = \frac{-2kN}{m} \hat{y}$$

4m
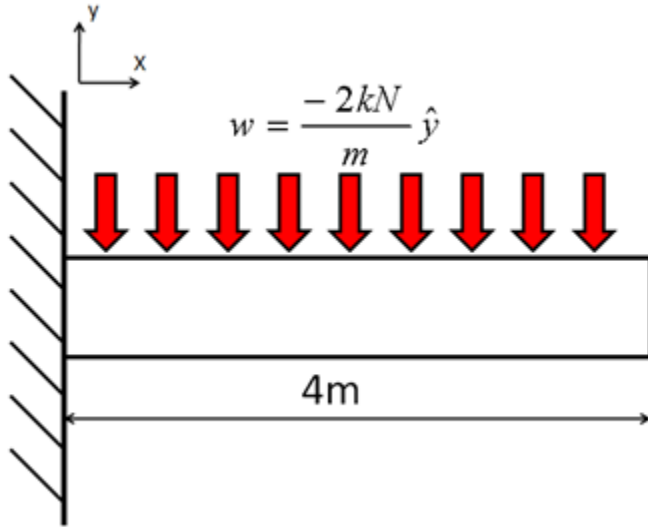
Hint: In order to carry out this exercise you can duplicate your original Static Structural project and then modify the duplicate. In the modifications you need to delete the original transverse force and apply the distributed load. In order to apply the distributed load right click on Static Structural, then select Insert and then click on Line Pressure as shown in the image below.
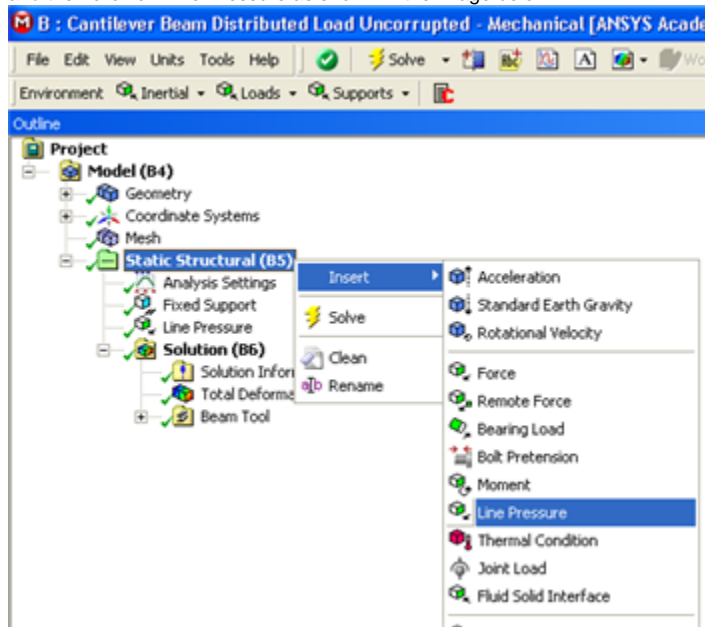
You will then need to apply the line pressure to the line, specify the magnitude and direction of the line pressure and lastly have ANSYS solve the system.

Exercise 3: Axial & Transverse Loading

In ANSYS, find the deformation and the stress of the beam when it is experiencing axial and transverse loading. Compare these results to those of the original problem.

$$P_1 = 80kN\hat{x}$$

$$P_2 = -8kN\hat{y}$$

4m
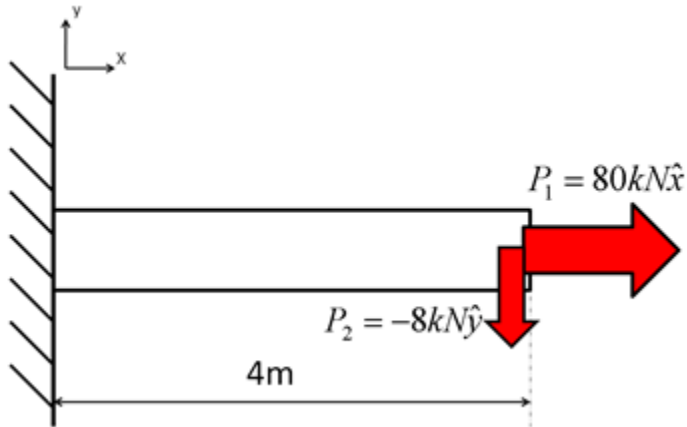
Hint: Once again, you can make a duplicate of the original Static Structural and then modify the duplicate. In order to carry out the computations you will need to add the axial force to the beam and then have ANSYS solve the system.

Exercise 4

Modeling a Cantilever Beam with 1D, 2D, and 3D elements

Problem Specification (pdf file)

Quick Video Summary:

- creating new coordinate systems
- positioning coordinating systems to create construction geometry
- determine displacements at any location
- how to create a mid-beam surface to determine the stresses along the midplane of the beam

Go to Comments

Go to all ANSYS Learning Modules