FEMAP Tutorial 2

Consider a cantilevered beam with a vertical force applied to the right end.
We will utilize the same geometry as the previous tutorial that considered an axial loading. Thus, this tutorial will consider the bending of a beam with the geometry, material properties and loading shown below.

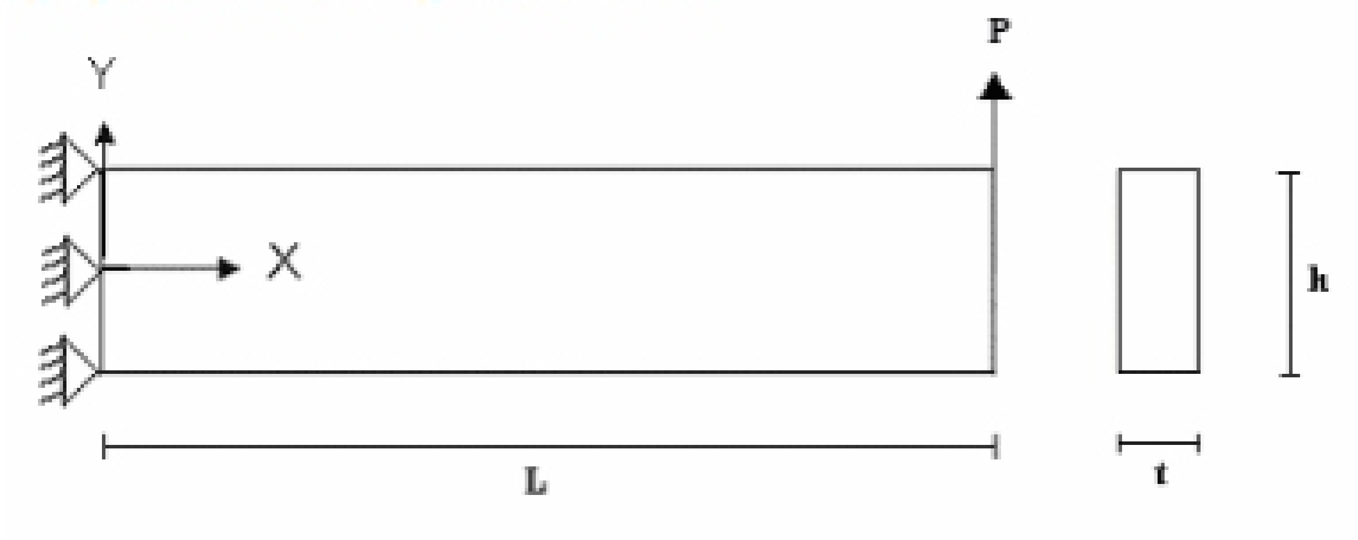


Figure 1: Bar with defined dimensions

The length (L) of the beam is 6 inches, the height (h) is 1 inch, the thickness is 0.25 inches, and the applied load (P) is 5,000 lbs. The bar is made of AISI 4130 steel which has material properties of Young's Modulus, $E = 29 \times 10^6$ psi, Poisson's ratio, $\nu = 0.32$, and weight density, $\rho = 7.33 \times 10^{-4}$ lb/in^3.

Boundary Condition cases:

1. Each node on the entire left boundary fixed

Loading cases (the load of 5000 lbs is idealized in three different ways):

1. Single load on center point of right boundary
2. Equal distributed point loads along the right boundary (where the sum equals P)
3. Total load P distributed to the right end nodes in a parabolic fashion (so the sum equals P)

At this point it is assumed that the reader has worked and understood FEMAP Tutorial 1. For steps addressing model geometry, materials, properties or boundary conditions, please refer to the previous tutorial. Note in this tutorial, when defining the property, make sure you choose "membrane" elements under Elem/Property Type in the define property window.

In this tutorial you will apply a mesh that has four times more elements than the previous example. As shown in class, the more elements that are used in the finite element method, the more accurate the results. This is also true for cases analyzed in FEMAP. Mesh the model with 80 elements along the length of the beam, and 16 elements along the height (this results in 81, and 17 nodes along each direction respectively).

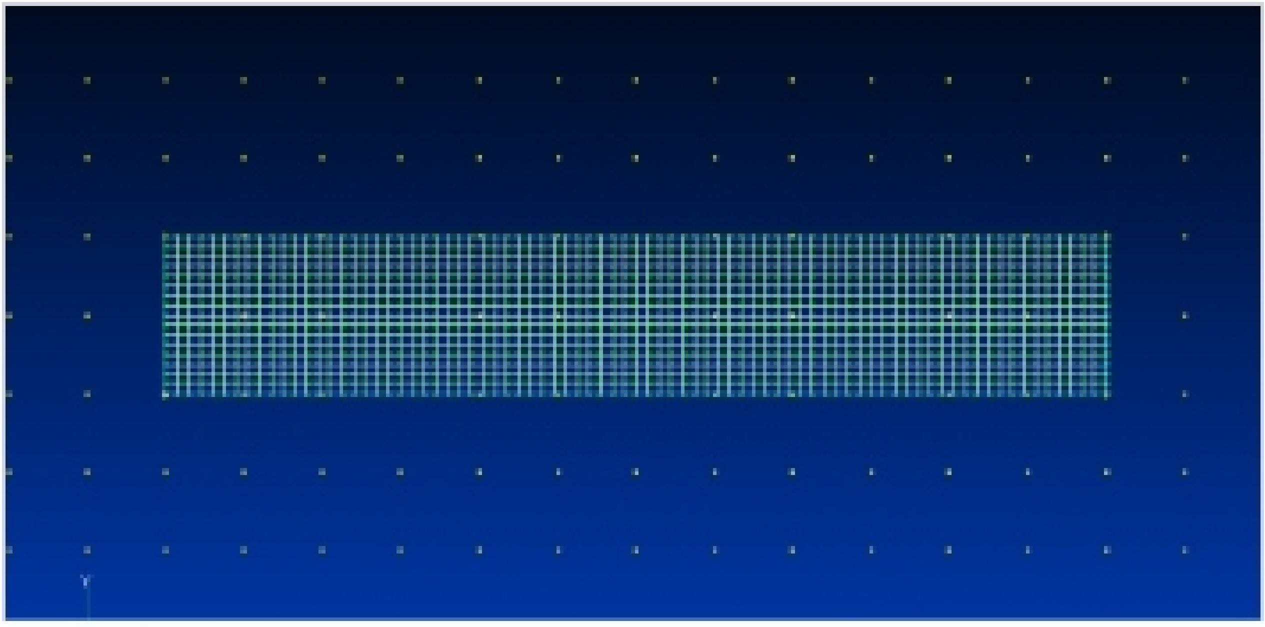


Figure 1: Meshed Bar, 80 x 16 Elements

Loads and Constraints

Now that there are significantly more nodes on the boundaries, it is important to utilize other methods of entering loads and constraints rather than selecting each node individually.

Create a load set

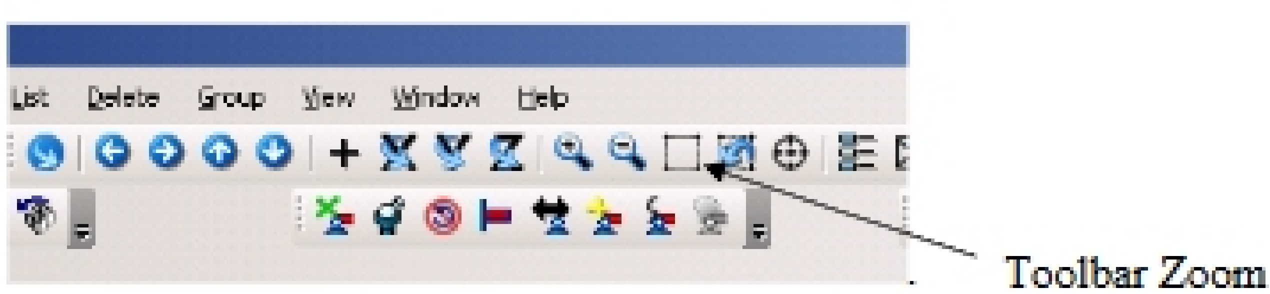
Model.Load.Set…(or **Control-F2**)
  Title(LoadSet1)
  OK

Specify the loading

For the first case we will approximate the load of 5000 lbs by applying a single point load on the middle node of the right boundary. In loading case 2 we will model the load as a series of equal point loads on each node of the right boundary. Finally, in case three we will apply the total load P by distributing it to the right end nodes in a parabolic fashion (so the sum equals P).

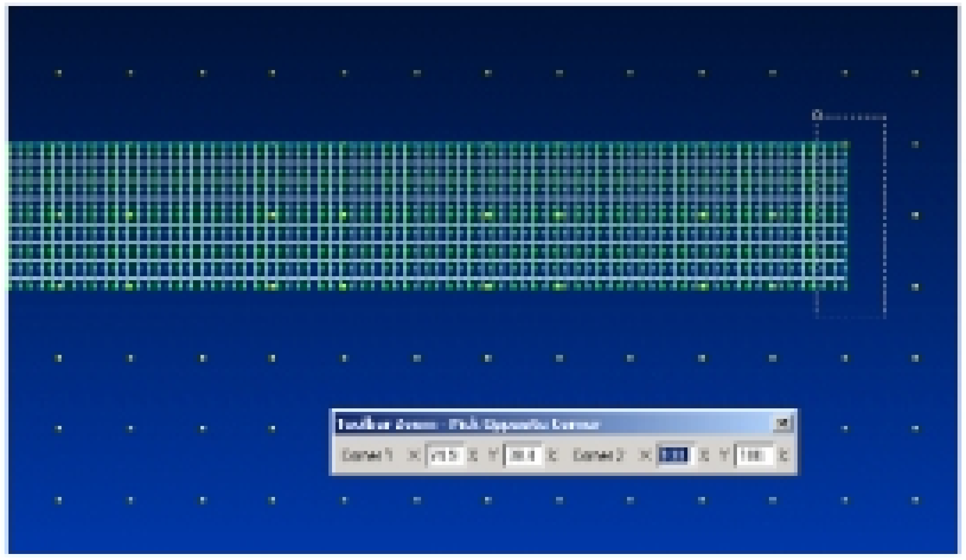
The loading method for case 1 is similar to the one used in the first tutorial.

The easiest method for case 2 is as follows:
Zoom in on the right boundary using the toolbar zoom tool. This is found next to the magnifying glass with the minus sign in it, it is a square box made up of dotted lines.



After clicking the icon a small Toolbar Zoom window will appear. Using your mouse drag a square box around the right end of the beam to select that area to zoom in on.

Once you have the box drawn, click once more and the window will resize with a larger view of the right end. This can also be done by going to View.Zoom (or F7).



Now that you have zoomed in on the area, the same method can be used to select the nodes that you want to apply a load to.

Model.Load.Nodal…
    Entity Selection…
        Select the Pick ^ button
            Select Box (use the same method as described above and draw a box around the desired nodes)
            Click on the screen when you are finished drawing the box to make your selection.
    OK
        Load.FY.Value
        OK
    Cancel

Enter a load that evenly distributes the 5000 lbs between the 17 nodes on the right boundary.

The method for case 3 is as follows:

From strength of materials (ENGR 214 and AERO 304), we know that the internal shear stress distribution is parabolic in the vertical direction (y). Therefore, it might be reasonable that the external load could be approximated by a parabolic distribution of point forces, rather than a uniform distribution. The parabolic force distribution to apply is as follows (note the nodes are symmetric about node 8):

Node	*Load in y direction (lbs)*
1 (top)	*0*
2	*110.29*
3	*205.88*
4	*286.77*
5	*352.94*
6	*404.41*
7	*441.18*
8	*463.24*
9 (center)	*470.59*