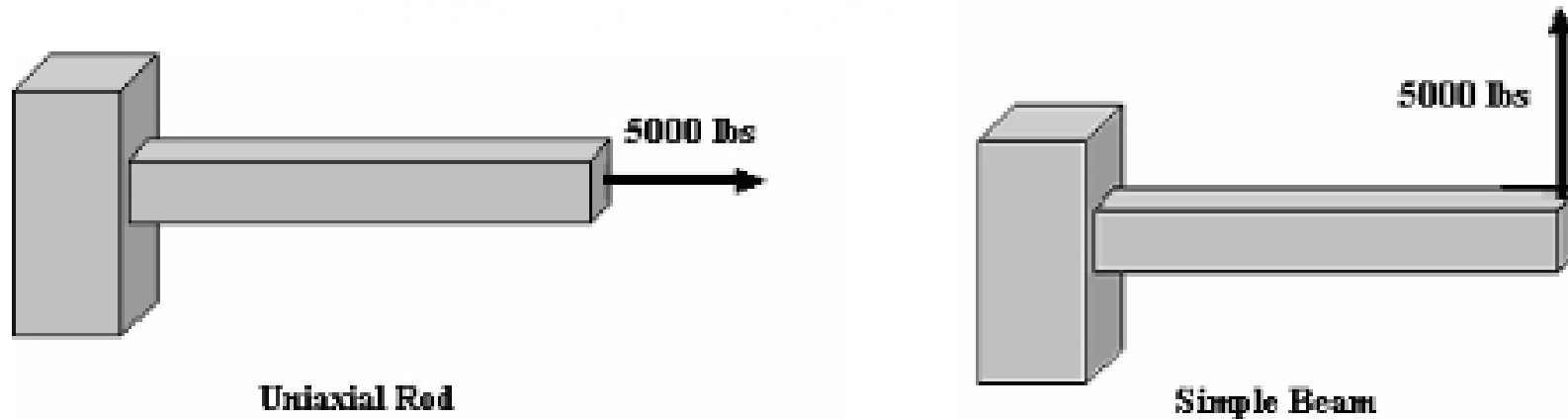


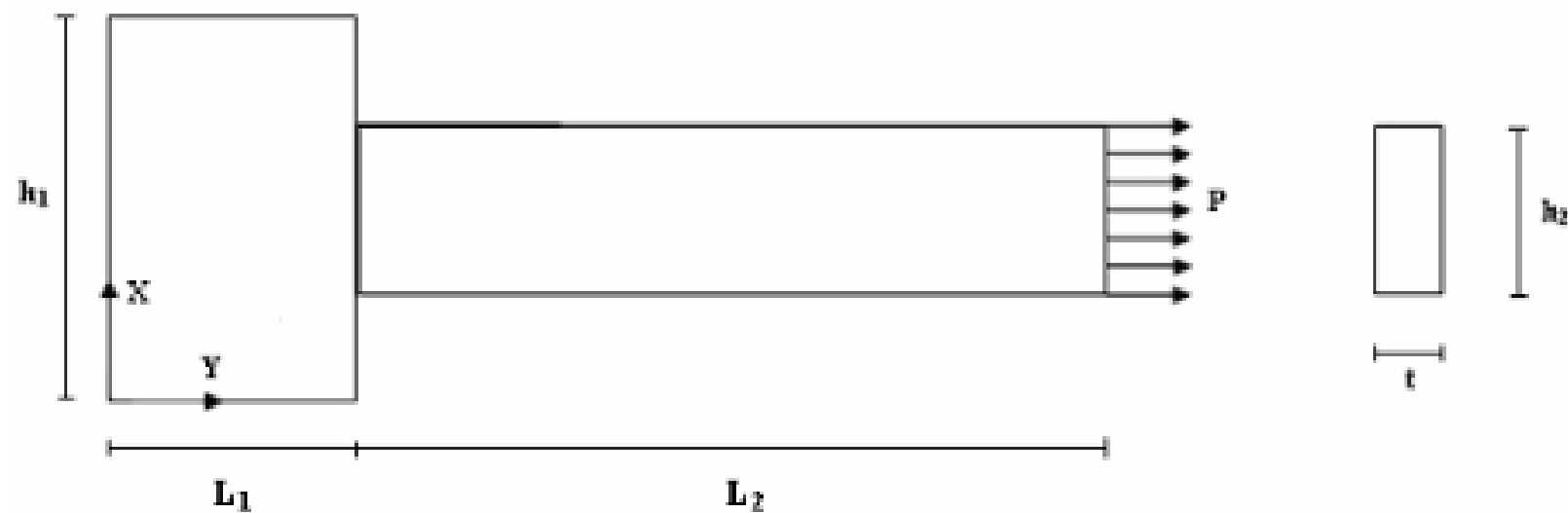
FEMAP Tutorial 3

In this tutorial, the uniaxial rod and simple beam will be modeled as elastic bodies, rather than using fixed supports or rollers on the left boundary. You will first construct the uniaxial rod, and then adapt this structure to represent a simple beam.



Two cases will be considered in FEMAP:

1. Axial load with equal forces on all nodes on the right boundary



2. Bending modeled with equal transverse forces on the right boundary



The length (L_1) of the support is 1.5 inches, and the length (L_2) of the bar/beam is 6 inches. The height (h_1) is 3 inches, and the height (h_2) 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/in³.

Follow the same procedures for geometry, materials, and properties as in previous tutorials. This analysis is going to utilize "membrane" elements again, and it will also use the same mesh sizing for the bar/beam (i.e. 80 elements along the length of the beam, and 16 elements along the height, or 81 and 17 nodes along each direction respectively). The support mesh size will have 20 elements along the length, and 48 elements along the height (i.e. 21 nodes, and 49 nodes respectively).

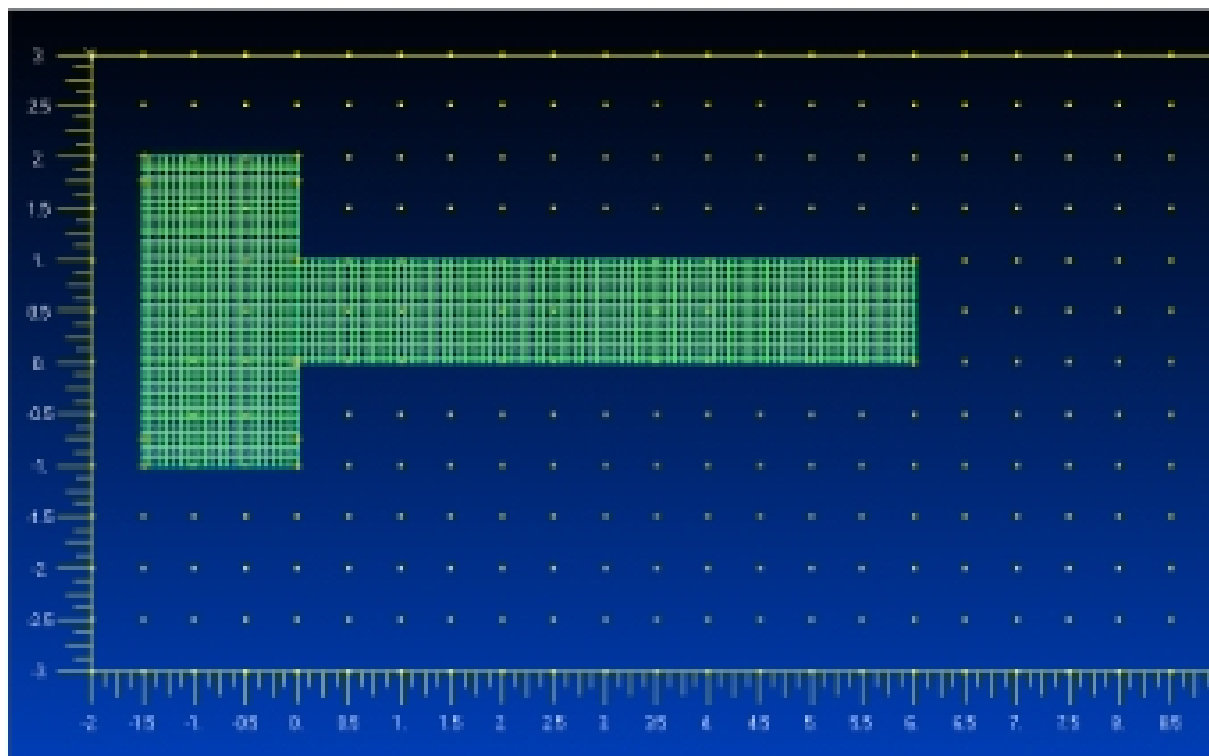
In tutorial 1 compatibility between nodes was mentioned. For additional information refer to tutorial 1. Remember that each super element has its own set of nodes. Therefore, the common boundary (line) between the two super elements will have two sets of nodes (one from each super element) that each occupies the same point in space. However these common nodes must be converted to one unique node point so that compatibility is achieved.

To perform this function, perform the following steps:

```
Tools>Check Coincident Nodes...
  Entity Selection>Select All
  OK
  Check/Merge Coincident...
    Maximum Distance to Merge (0.001)
    Options...
      Merge Coincident Entities (Check)
    OK
```

After you complete this step, a message should appear on the gray bar below the main window informing you how many nodes were merged (17 in this case)

At this point the model should look like:



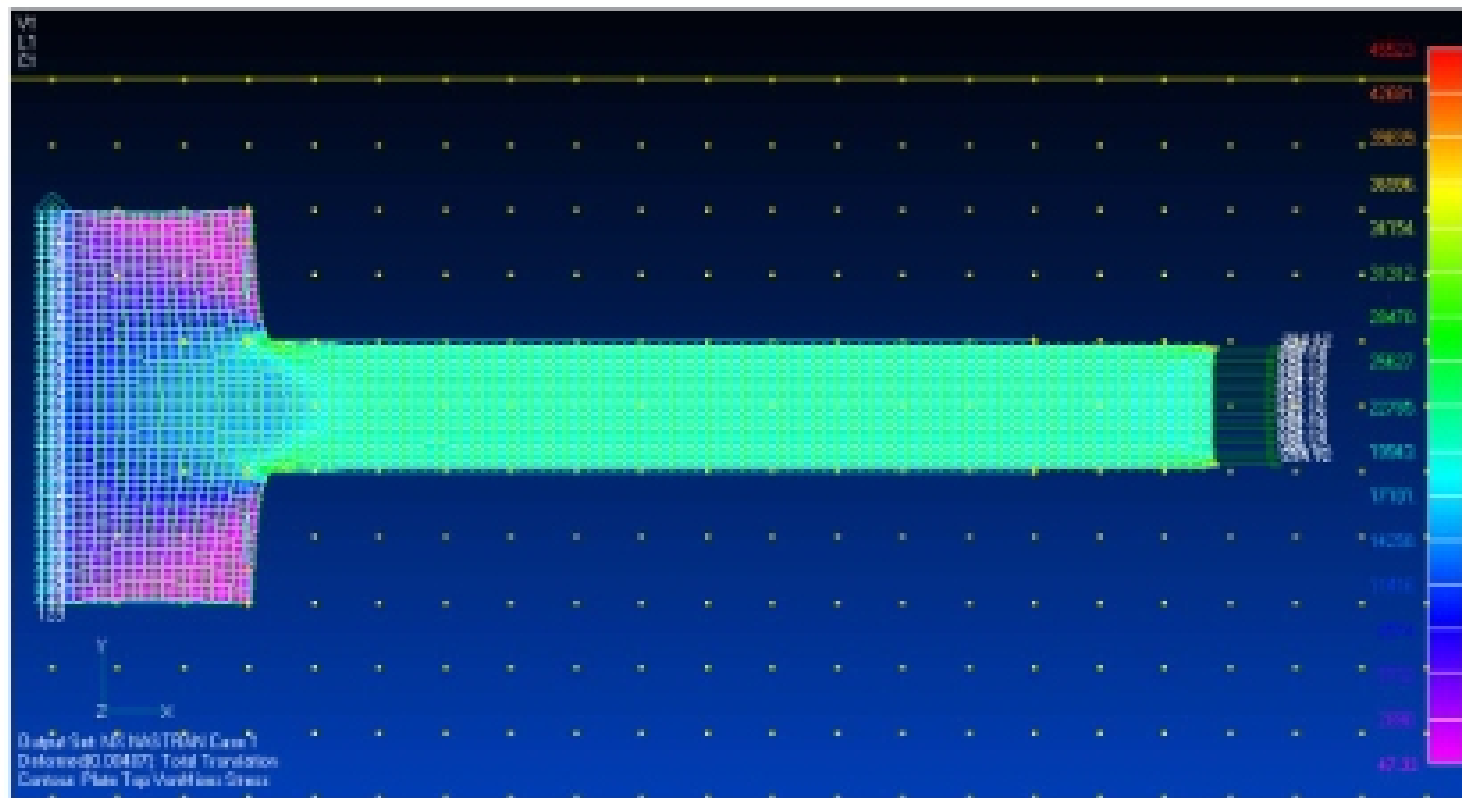
The loads will be applied using the same process as in previous tutorials. The axial loading case will have a 5000 lb load distributed evenly on the nodes of the right boundary in the 'x' direction. The transverse loading case will have a 5000 lb load distributed evenly along the right boundary in the 'y' direction. (This results in approximately 294.12 lbs per node.)

The constraints will be applied to the support's left boundary. Fix each node in the 'x' and 'y' directions, leaving the rotational directions free to move (Pinned).

Model Analysis

The first loading case results should look similar to these:

VonMises:



Sigma XX:

