FEMAP Tutorial 4

Consider an axial bar that is simply supported on the left end with an axial traction applied too the right end. In addition there is now a hole through the bar close to the right end. Using the same geometry as the previous tutorials we will now analyze a model that has a discontinuity.

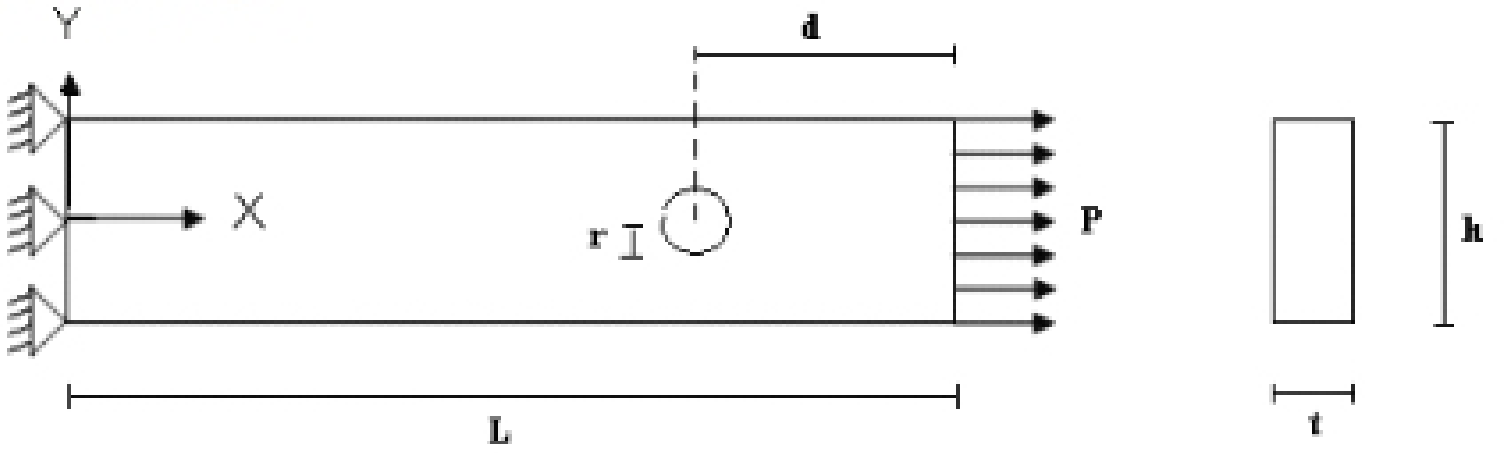


Figure 1: Bar with defined dimensions

The length (L) of the beam is 6 inches, the height (h) is 1 inch, the thickness is 0.25 inches, and the applied traction is 20000 psi. This traction represents a total force (P) of 5,000 lbs. The circle is centered vertically; a distance (d) of 2.4 inches from the right end, and has a radius (r) of 0.2 inches. 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:

1. The axial load will be applied as equal distributed pressure on the right boundary. (i.e. input pressure rather than point loads)

Follow the previous tutorials to produce the initial geometry, material and property of the bar. To model the hole:

Geometry.Curve – Circle.Center…
  Locate – Enter Location at Center of Circle…
    X (3.6)
    Y (0.5)
    Z (0.1)
    OK
  Radius of Circle…
    Radius (0.1)
    OK
  Cancel

In this tutorial, as in tutorial 2 and 3, use "membrane" elements.

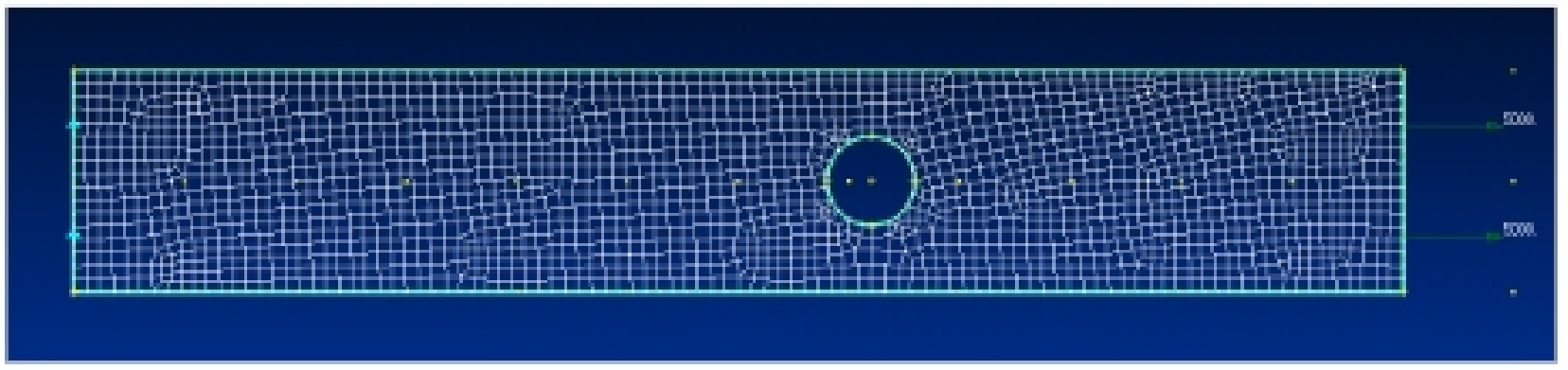
Because there is now a hole in the geometry the meshing process is different. In order to mesh this geometry, a surface must be created out of the existing curves, and then the mesh sizing must be set for the surface. Finally, a mesh can be applied on the created surface. By creating a surface, FEMAP will recognize the hole in the geometry when it applies the mesh to the model.

To create the surface:
Geometry.Boundary Surface.From Curves…
    Select All
    OK
Cancel

To set the mesh size on the surface:
Mesh.Mesh Control.Size on surface…
    Entity Selection – Select Surface(s) to Set Mesh Size…
        Select All
        OK
    Automatic Mesh Sizing…
        Initial Sizing…
            Element size(0.06)
        Surface Interior Mesh Growth
            (Check) Growth (1)
        Curvature-Based Mesh Refinement…
            (Check) Refinement Ratio (0.1)
        OK
    Cancel

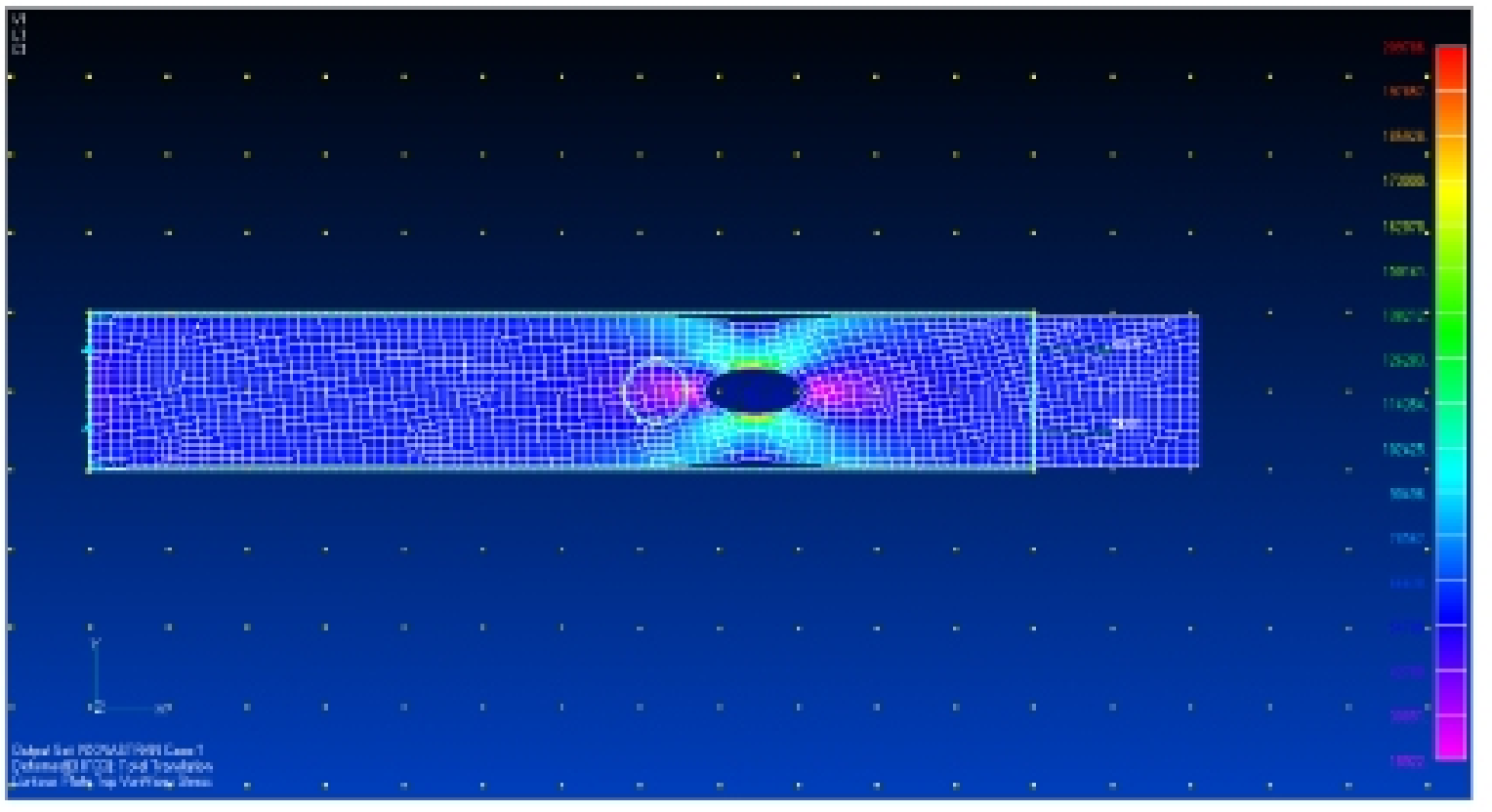
To mesh the surface:
Mesh.Geometry.Surface…
    Entity Selection – Select Surfaces to Mesh…
        Select All
        OK
    Automesh Surfaces…
        Property (Select ¼" Steel from list)
        OK

A new method can be used to constrain and load the model. After following the same steps as in previous tutorials to define load and constraint sets, use the "Constraint.On Curve" command to constrain the left most boundary. This can be used in place of constraining each node. Using your mouse you can select the curve simply by clicking on it. The load can also be applied by using the "Load On Curve" command.



Model Analysis

VonMises:



Sigma XX: