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](image)

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).
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).

![Image](image_url)

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

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):

<table>
<thead>
<tr>
<th>Node</th>
<th>Load in y direction (lbs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (top)</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>110.29</td>
</tr>
<tr>
<td>3</td>
<td>205.88</td>
</tr>
<tr>
<td>4</td>
<td>286.77</td>
</tr>
<tr>
<td>5</td>
<td>352.94</td>
</tr>
<tr>
<td>6</td>
<td>404.41</td>
</tr>
<tr>
<td>7</td>
<td>441.18</td>
</tr>
<tr>
<td>8</td>
<td>463.24</td>
</tr>
<tr>
<td>9 (center)</td>
<td>470.59</td>
</tr>
</tbody>
</table>
The center node (8) is the only node with that particular force, but the node above and below the center node will have the same force applied. This means as you are selecting which node to apply the loads to you can choose two nodes at a time and apply the same load. The procedure for applying the loads is the same as the previous tutorial. Note that the sum of the nodal forces applied on the right side equals to 5,000 lbs. but follows a parabolic distribution.

Model.Load.Nodal…
Entity Selection. Select the nodes corresponding to the load you are entering.
OK
Load.FY.Value(Specific load for that node)
OK
Cancel

Create a constraint set

Model.Constraint.Set…(or Shift-F2)
Title(ConstraintSet1)
OK

Use the same method to zoom in and select the nodes as used in case 2 of the loading descriptions. Remember that the beam is cantilevered on the left boundary so that all nodes on the left boundary will be fixed from all motion.

Model.Constraint.Nodal…
Entity Selection – Enter Node(s) to Select
Select the Pick button
Select Box (use the same method as described above and draw a box around the desired nodes on the left boundary)
Click on the screen when you are finished drawing the box to make your selection.
Create Nodal Constraints/DOF…
Select (Fixed)
OK
Cancel

This step has now fixed the entire left boundary. This is the same result as selecting each node individually in the previous tutorial.

Make sure that you have constrained the left boundary, and saved the model before you send the model to NX Nastran for analysis.

Before you complete the analysis steps, take time to determine what classical beam theory will predict for each of the stress components in the x-y plane ($\sigma_{xx}$, $\sigma_{yy}$, $\sigma_{xy}$). $\sigma_{xx}$ max (top or bottom surface) should be 720,000 psi, $\sigma_{yy} = 0$ everywhere, and $\sigma_{xy} = 60,000$ psi at the centerline of the beam (and zero at the top and bottom).
Model Analysis

The results for the first loading case (with a single point load) should look similar to these:

VonMises:

In the above contour plot for VonMises stress, note the stress concentration in the vicinity of the applied point load.

Sigma XX:

Note from the above $\sigma_{xx}$ contour that the axial stress is compressive on the top boundary and tensile on the bottom boundary (towards the left end).
**Sigma YY:**

Note from the above contour of $\sigma_{yy}$ that $\sigma_{yy}$ is almost zero except for the location where the point load is applied and at the corners of the left boundary where the structure is constrained.
From the above contour plot for $\sigma_{xy}$, you can see that the shear stress $\sigma_{xy}$ is a maximum at the centerline of the beam, and is zero at the top and bottom surfaces. This is as expected. From strength of material, the shear stress is given by an equation similar to

$$
\sigma_{xy}(x, h) = \frac{V_y(x)}{I_{zz}(y)} Q(h) \quad \text{where} \quad Q(h) = \int_{h}^{c} y \ t(y) \ dy
$$

For a rectangular cross-section, the $Q$ term becomes:

$$
Q(y = h) = \int_{h}^{c} y dy = h \int_{h}^{c} y dy = \frac{h}{2} (c^2 - h^2)
$$

From the above, you can see that since $Q(y=y)$ varies as a quadratic function of position $y$ (or $h$), then the shear stress is also quadratic. The equation also shows that the shear stress will be zero at the top and bottom ($y=c$ or $-c$), and that the shear stress will be a maximum at $y=h=0$. 

Sigma $XY$:
Loading Case 2 (Point load applied as set of distributed equal nodal forces)

The second loading cases represents the 5,000 lb point load by equal point forces of 294.12 lbs at each node on the right boundary. The results for this second loading case results should look similar to these:

VonMises:

![VonMises Diagram]

Sigma XX:

![Sigma XX Diagram]
Sigma YY:

Sigma XY:
Loading Case 3 (Point load applied as set of nodal forces which form a quadratic distribution)

The third loading case results should look similar to these:

VonMises:

Sigma XX:
Sigma YY:

Sigma XY:
Questions:

1. How does the idealization of the point load affect the internal stress distributions?
2. Do each of the stress components agree reasonably well with classical beam theory ($\sigma_{xx}, \sigma_{yy}, \sigma_{xy}$)?
3. What information does the VonMises stress component give you that the other components do not?
4. How do end conditions affect the stress components? Are the stress concentrations predicted by classical beam theory?
5. Comparing the classical beam theory results, and the finite element results (using plane stress/membrane elements), which provides the most “exact” solution.