University of Connecticut
School of Engineering

ANSYS

Module 1.8: Bending Moment Loading of a 3D Cantilever Beam

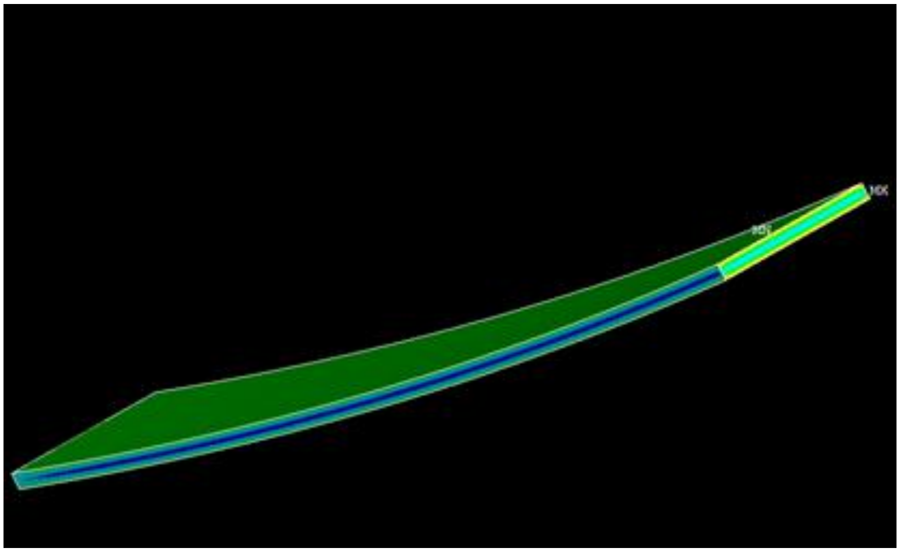
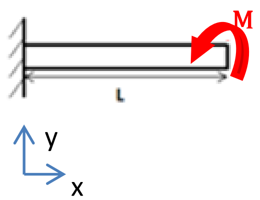
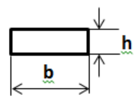


Table of Contents

Problem Description





Nomenclature:

L =110m	Length of beam
b =10m	Cross Section Base
h =1 m	Cross Section Height
M=70kN*m	Applied Moment
E=70GPa	Young's Modulus of Aluminum at Room Temperature
$\boldsymbol{\nu}$=0.33	Poisson's Ratio of Aluminum

In this module, we will be modeling an Aluminum cantilever beam with a bending moment loading about the the z-axis with three dimensional elements in *ANSYS Mechanical APDL*. Since the exact solution to this problem is numerical, we will be using beam theory and mesh independence as our key validation requirements. The beam theory for this analysis is shown below:

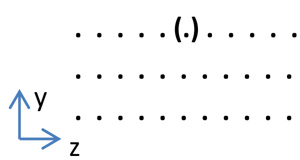
Theory

Von Mises Stress

Assuming plane stress, the Von Mises Equivalent Stress can be expressed as:

$$\sigma' = (\sigma_x^2 - \sigma_x\sigma_y + \sigma_y^2 + 3\tau_{xy}^2)^{\frac{1}{2}} \qquad (1.8.1)$$

During our analysis, we will be analyzing the set of nodes through the top center of the cross section of the beam:



Due to symmetric loading about the yz cross sections, analyzing the nodes through the top center will give us deflections that approximate beam theory. Additionally, since the nodes of choice are located at the top surface of the beam, the shear stress at this location is zero.

$$(\tau_{xy} = 0,\ \sigma_y = 0). \qquad (1.8.2)$$

Using these simplifications, the Von Mises Equivelent Stress from equation 1 reduces to:

$$\sigma' = \sigma_x \qquad (1.8.3)$$

Bending Stress is given by:

$$\sigma_x = \frac{M(x)c}{I} \tag{1.8.4}$$

Where $I = \frac{1}{12}bh^3$ and $c = \frac{h}{2}$. From statics, we can derive:

$$M(x) = M \tag{1.8.5}$$

$$\boldsymbol{\sigma_x = \frac{6M}{bh^2} = 42\text{KPa}} \tag{1.8.6}$$

Beam Deflection

As in module 1.1, the beam equation to be solved is:

$$\frac{d^2y}{dx^2} = \frac{M(x)}{EI} \tag{1.8.7}$$

Using Shigley's Mechanical Engineering Design, the beam deflection is:

$$\boldsymbol{\delta = \frac{Mx^2}{2EI}} \tag{1.8.8}$$

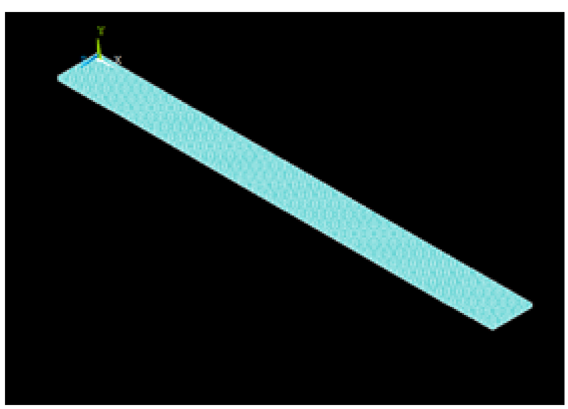
With Maximum Deflection at:

$$\boldsymbol{\delta = \frac{ML^2}{2EI} = 7.26\text{mm}} \tag{1.8.9}$$

WARNING: We are using beam theory as a **REFERENCE** gage for the solutions ANSYS will provide. The real driver behind a correct solution will be **mesh independence**

Geometry

1. Go to **Utility Menu ->** **Open ANSYS File** and open *3D Cantilever.db* that you created in module 1.7. To refresh your memory, this was a 1x10x110m Aluminum cantilever beam with two SOLID 185 elements through the thickness. The file that opens should look as follows:

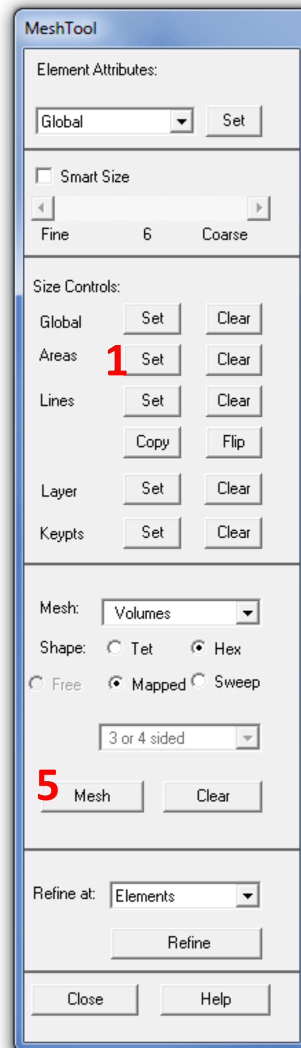


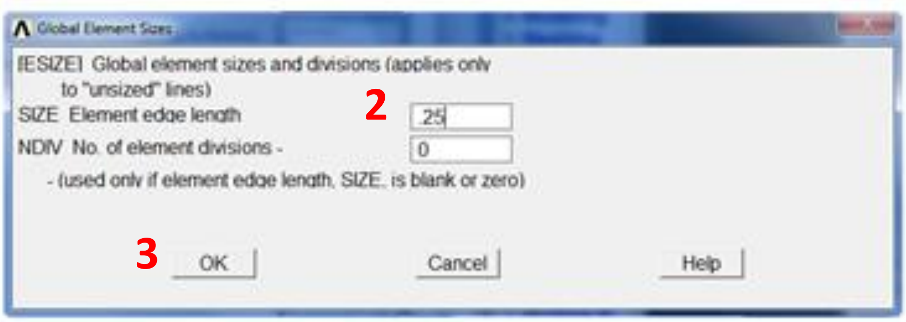
Preprocessor

Mesh

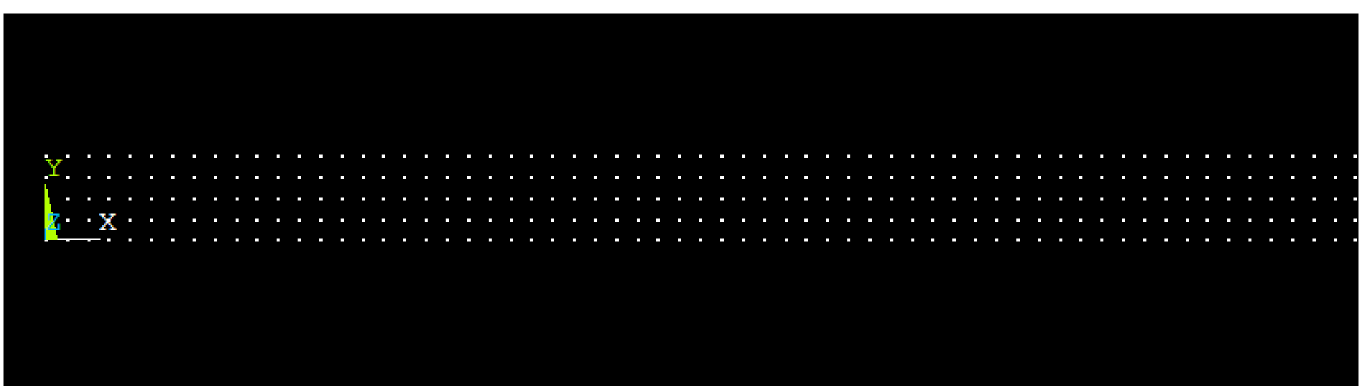
We are going to re-mesh this beam to have four elements through the thickness.

1. Go to **Main Menu -> Preprocessor -> Meshing -> Mesh Tool Size Controls : -> Global -> Set**
2. Set **Size Element Edge Length** to 0.25. This will map 4 elements through the thickness of the beam.
3. Click **OK**
4. Select Mesh: -> **Volumes -> Hex -> Mapped**
5. Click **Mesh**
6. Click **Pick All**

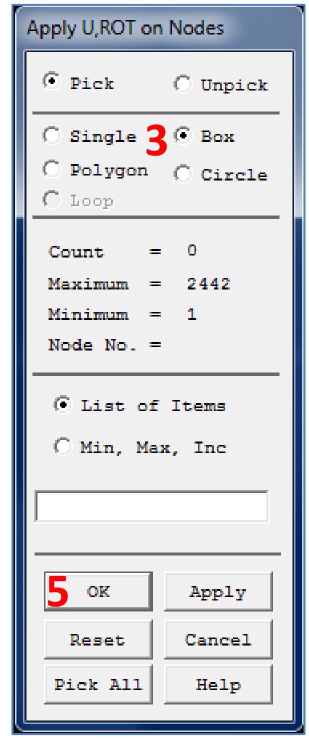




If you go to **Utility Menu -> Plot -> Nodes**, use the **Fit View** followed by the **Front View** and scroll in, the resulting mesh should look as follows:



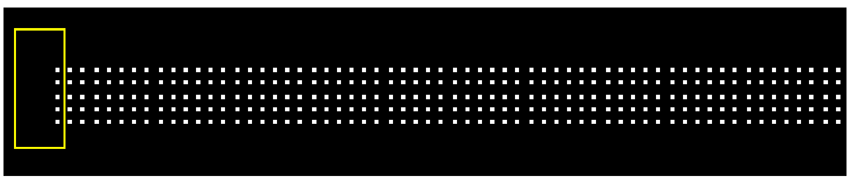
Loads



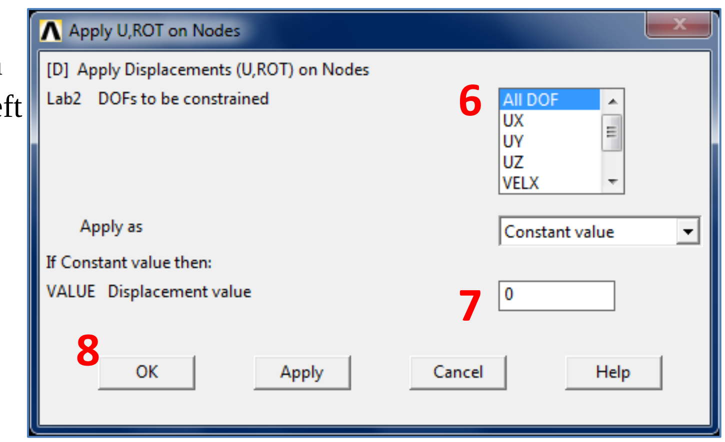
Displacement

1. The **Triad** in the top left corner blocks nodes.
   To get rid of the triad, type */triad,off* in **Utility Menu -> Command Prompt** followed by */replot*
2. Go to **Main Menu -> Preprocessor -> Loads -> Define Loads ->Apply ->Structural -> Displacement -> On Nodes**
3. Click **Pick -> Box**
4. With your cursor, drag a box around the first set of nodes on the far left side of the beam:

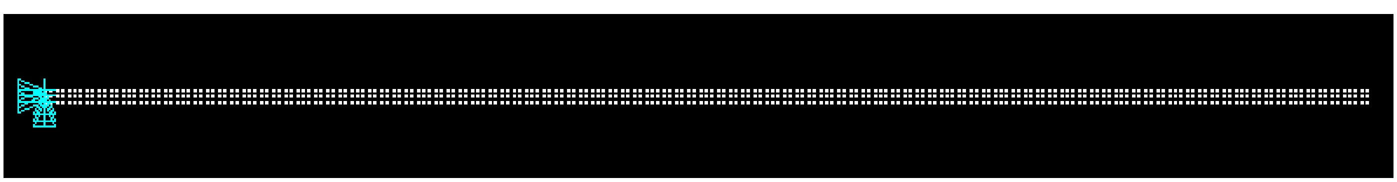
4



5. Click **OK**
6. Click **All DOF** to secure all degrees of freedom
7. Under **Value Displacement value** put 0. The left face is now a *fixed end.*
8. Click **OK**



The resulting graphic should be as shown:



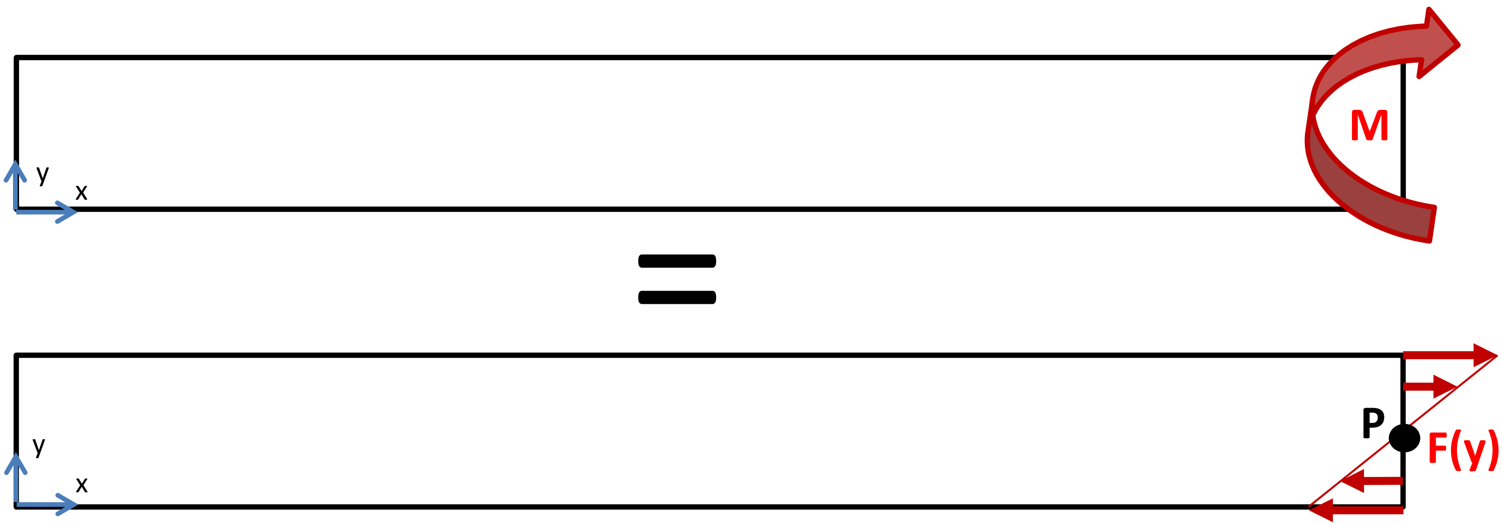
Force/Moment

In modules 1.2 and 1.5, we were able to apply an element's moment boundary condition to the end face of the beam to create the bending moment boundary condition. Unfortunately, 3D elements in ANSYS only have *3 Degrees of Freedom* and thus Bending Moment boundary conditions are not offered. We can overcome this hurdle however with a little bit of theory.

By definition:

$$M = rxF \qquad (1.8.10)$$

Where r is a perpendicular distance (lever arm) and F is a force applied. Thus we can see that moments are linearly related to forces. This means that we can model a moment as a linearly varying load across the end face of the beam:



In order to determine the forces along the distribution, we will take advantage of the linear relationship between force and distance when we sum moments about point P:

$$\sum_{i=1}^{N} M_p = M = \sum_{i=1}^{N} F_i x_i \qquad (1.8.11)$$

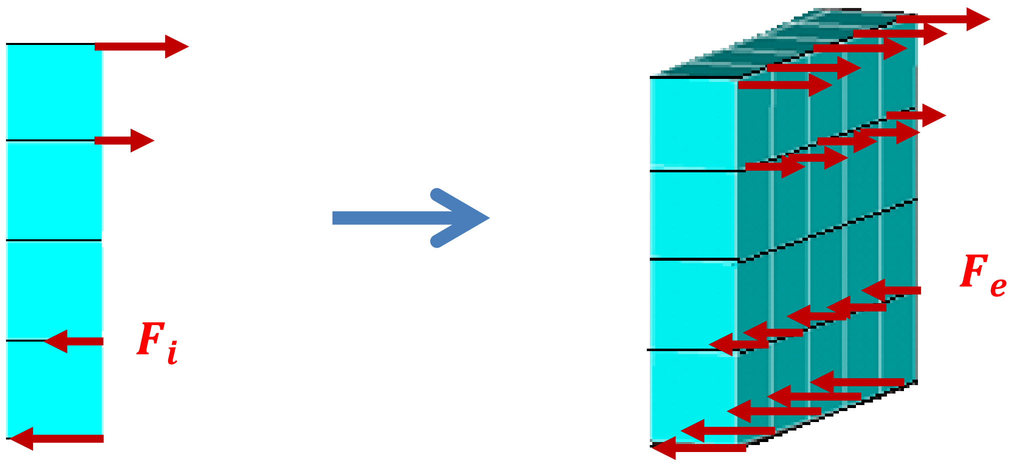
Where i is the i'th row of nodes up the y axis. The strategy here is to find the maximum force from equation 1.8.11 from which the other forces can be determined:

$$F_i = \frac{y_i}{\frac{h}{2}} * F_{max} \qquad (1.8.12)$$

Where y_i is the height of the i'th row of nodes above or below point P. Once F_i is determined, it is distributed evenly across the elements of the end face:

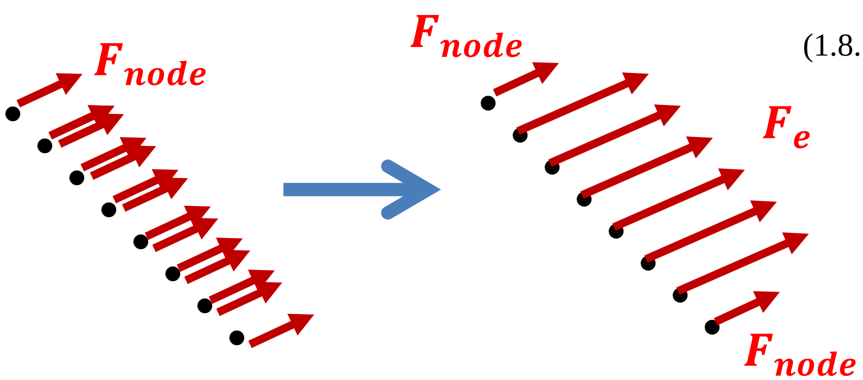
$$F_e = \frac{F_i}{n} \qquad (1.8.13)$$

where n is the number of elements in a row.

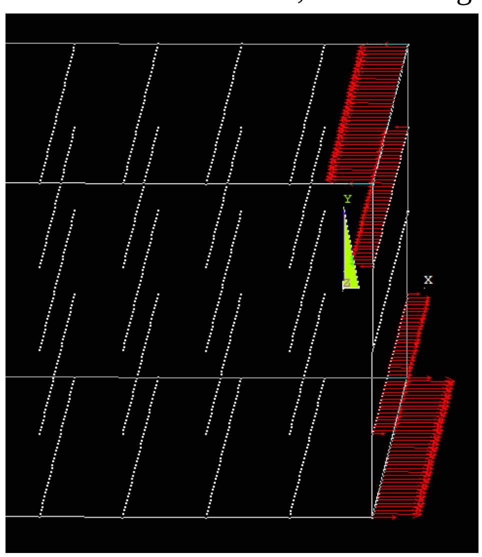


Now that we have calculated the forces to be distrubuted across the elements, we must calculate the forces that we will place at each node. Since F_e is constant across a given row of nodes and each element has two nodes in a given row:

$$F_{node} = \frac{F_e}{e} \tag{1.8.13}$$



With four elements through the thickness of the beam, the resulting boundary conditions are shown below:

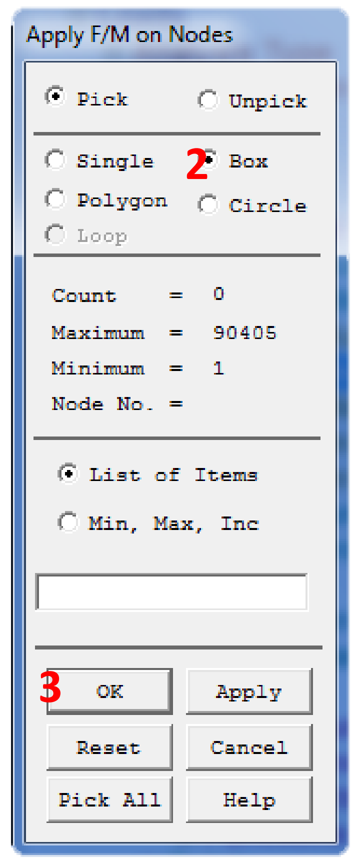


Using our model, we can calculate F_{max} from equation 1.8.11 since we have the moment load specified (70 kN M). By symmetry and using equation 1.8.12, we can derive:

$$M = 2(F_{max} * \frac{h}{2} + \frac{1}{2}F_{max} * \frac{h}{4})$$

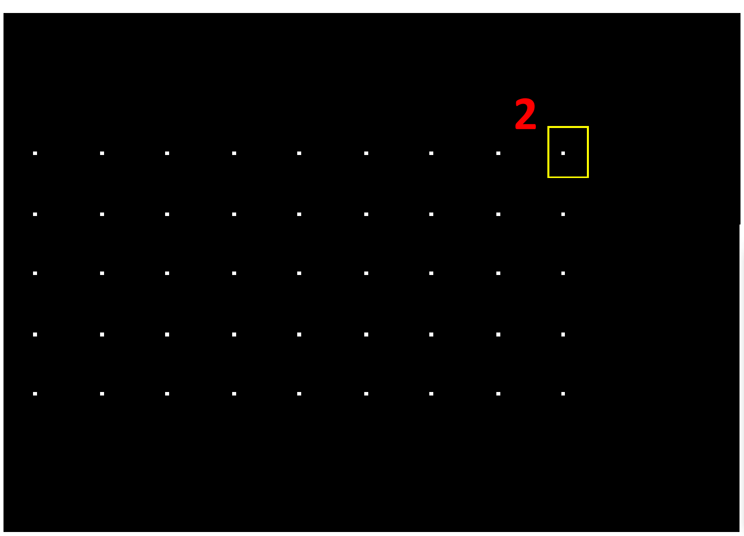
Solving for F_{max}, we get $F_{max} = 56000\ N$. Thus:

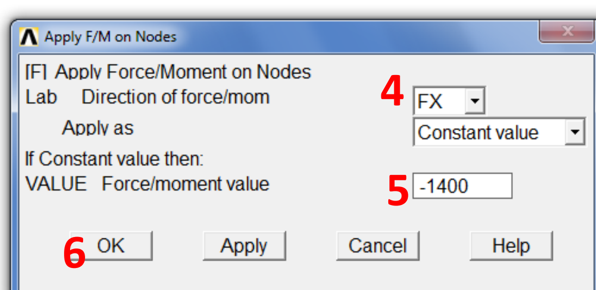
Row	F_i (N)	F_e (N)	F_{node} (N)
1	-56000	-1400	-700
2	-28000	-700	-350
3	0	0	0
4	28000	700	350
5	56000	1400	700



Picking Loads in ANSYS

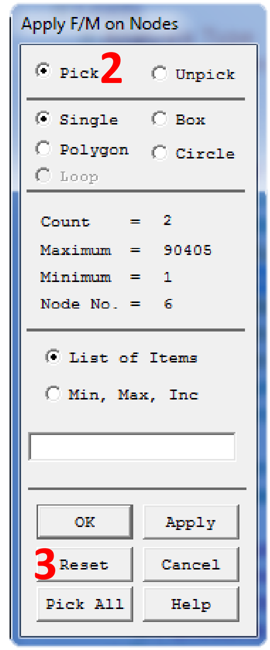
1. Go to **Main Menu -> Preprocessor -> Loads -> Define Loads -> Structural -> Force/Moment - > On Nodes**
2. Check **Pick -> Box** and box row 1 of the end face as shown below.
3. Click **OK**
4. Under **Direction of force/mom** select *FX*
5. Under **VALUE Force/moment value** enter *-1400*
6. Click **OK**
7. Repeat for rows 1-5. We have now assigned F_e to all the end face nodes.

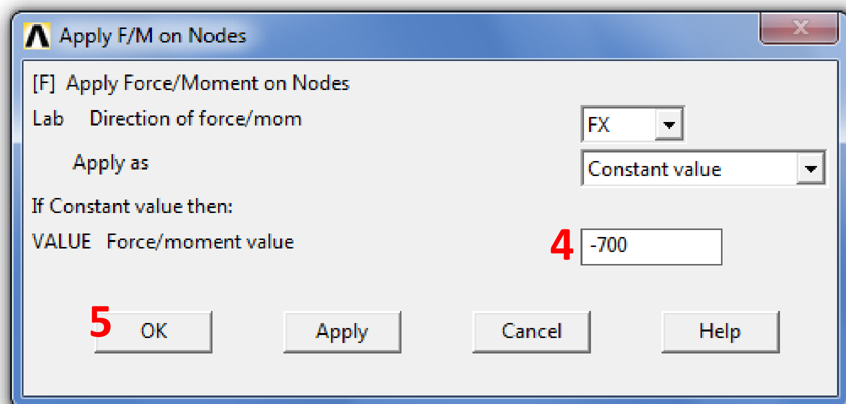


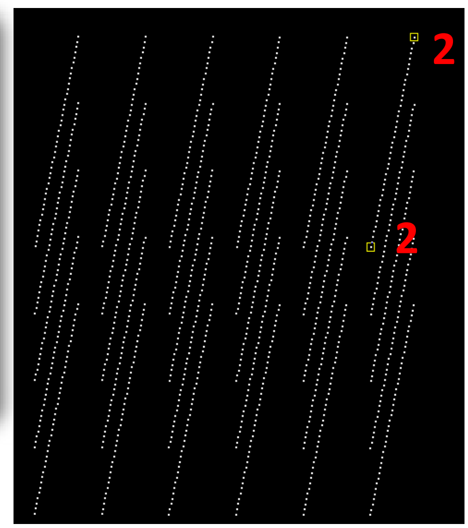


Now we must assign F_{node} to the relevant nodes.

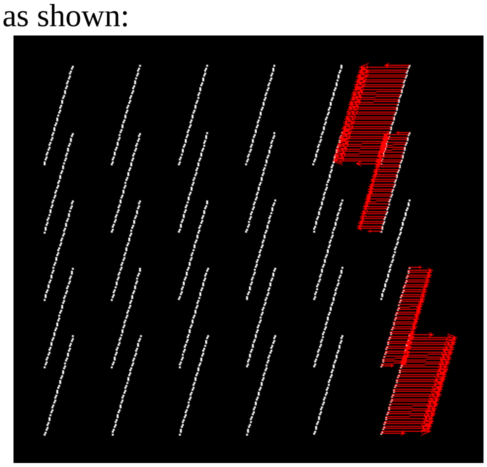
1. Go to **Main Menu -> Preprocessor -> Loads -> Define Loads -> Structural -> Force/Moment -> On Nodes**
2. Select **Pick -> Single** and select the nodes at the end of row 1.
3. Click **OK**
4. Under **VALUE Force/moment value** enter *-700*.
5. Click **OK**
6. Repeat Steps 1-5 for all F_{node} values for all rows of nodes







The resulting picture should look as shown:



Solution

Now that our boundary conditions have been specified, it's time to solve the problem.

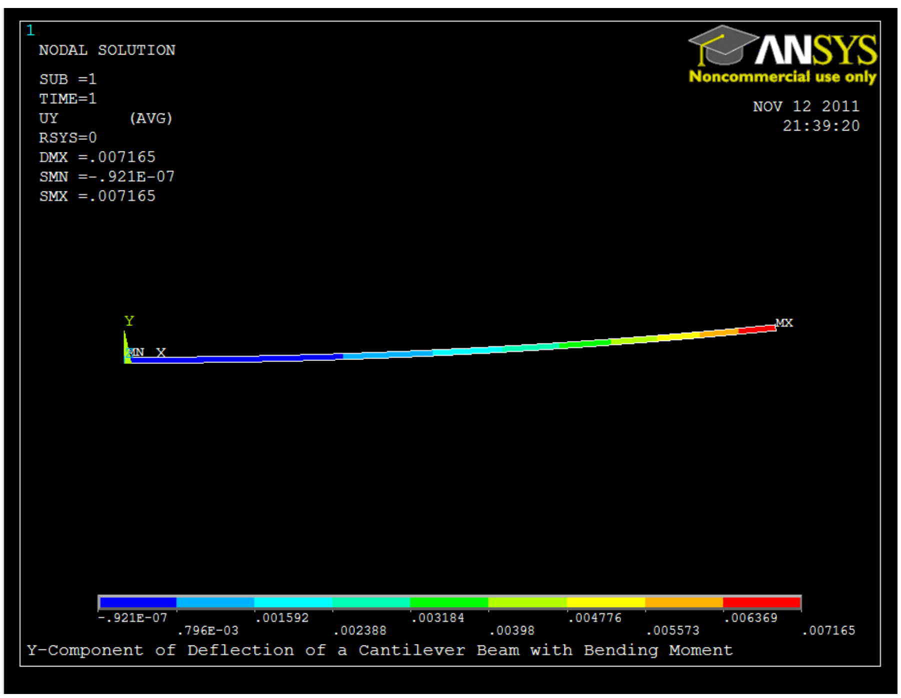
1. Go to **Main Menu -> Solution -> Solve -> Current LS**
2. Click **OK**

General Postprocessor

Now that ANSYS has solved the load step, lets create plots of Y Deflection and Equivalent Von Mises Stress.

Y Component of Displacement

1. Go to **Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solu**
2. Go to **Nodal Solution -> DOF Solution Y-Component of displacement**
3. Click **OK**
4. To give the graph a title, go to **Utility Menu -> Command Prompt** and type
 /title,Y-Component of Deflection of a Cantilever Beam with Bending Moment
followed by the return key and the command */replot*
The resulting graph should look as shown below:

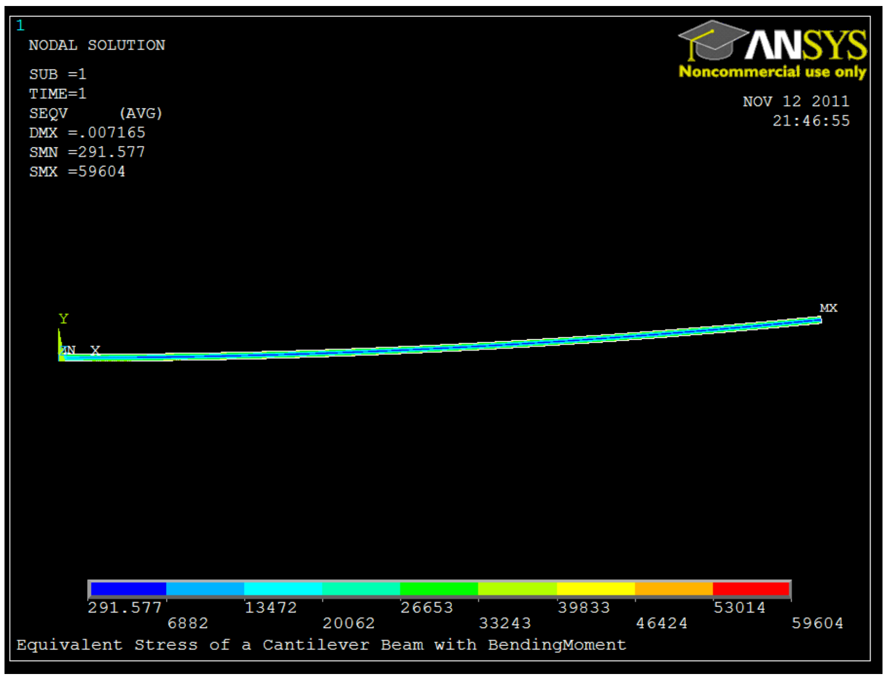


According to the graph, the max deflection is recorded as 0.007165 m.

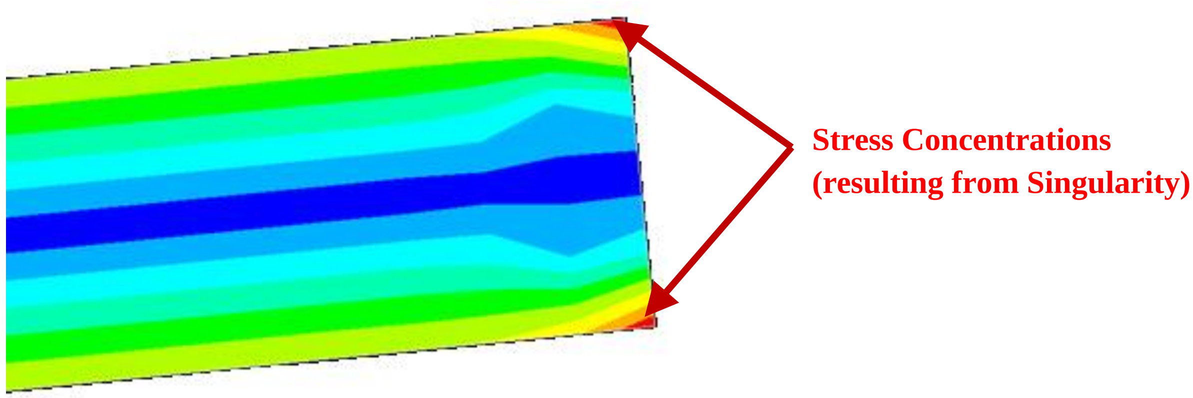
Equivalent (Von-Mises) Stress

1. Go to **Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solu**
2. Go to **Nodal Solution -> Stress -> von Mises stress**
3. Click **OK**
4. To give the graph a title, go to **Utility Menu -> Command Prompt** and type *⁄title,Equivalent Stress of a Cantilever Beam with BendingMoment* followed by the return key and the command */replot*

The resulting graph should look as shown below:



If you notice, the Maximum Equivalent Stress is much higher than the expected value. This is because of the localized stress concentration introduced in our model from the point load application at the end face:



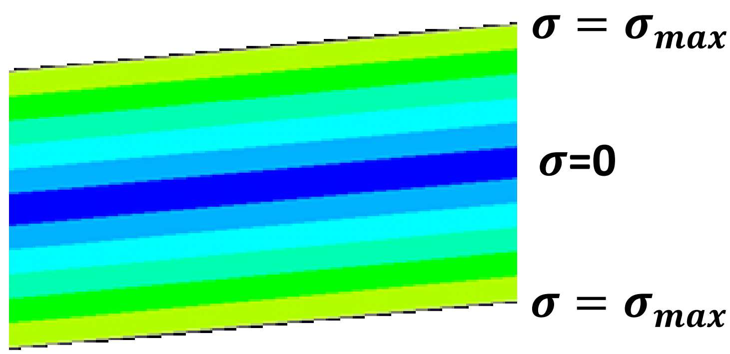
Point Load Singularity

By definition:

$$\sigma = \frac{F}{A} \tag{1.8.14}$$

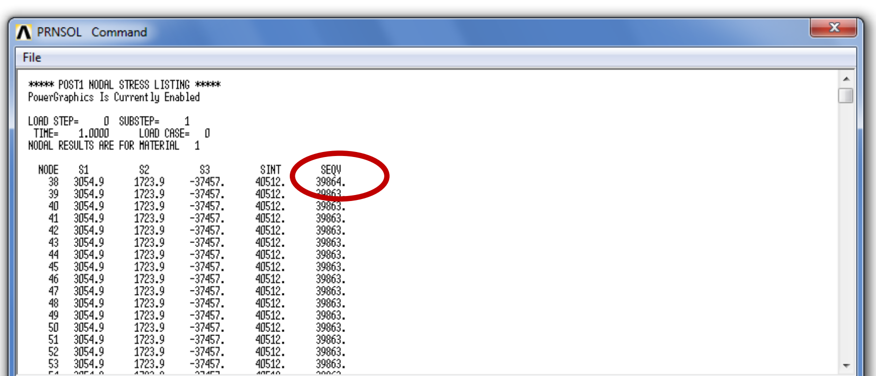
Since the point loads are applied at the nodes, there is effectively no area where the point load is applied. This creates a false stress concentration in the model. As the mesh is refined, this elevated stress level will approach infinity. In order to rid the model of this false stress, a **small pressure distribution can be applied at the element.** In the interest of time and simplicity, we will simply ignore this false stress in our analysis.

From our model we notice that far away from this false stress spike the beam displays more expected stress behavior, varying linearly from the max value at the top and bottom surfaces to effectively zero at the center of the beam:

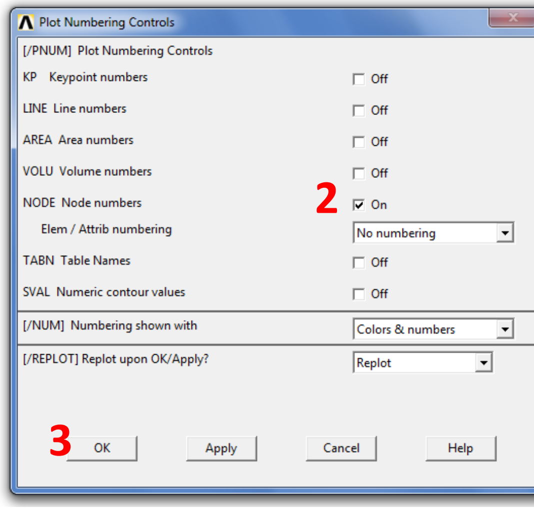


Since the chart values are skewed by the false stress concentration value, we will *list* the equivalent stress values at each node and determine σ_{max}.

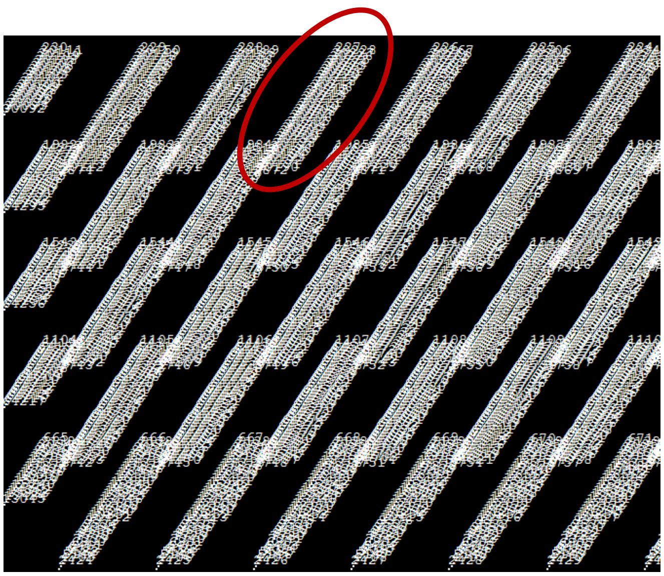
1. Go to **Utility Menu -> List -> Results -> Nodal Solution …**
2. Go to **Nodal Solution -> Stress -> von Mises stress**
3. Click **OK**
4. In the **PRNSOL Command** window, the column to the far right lists the Equivalent stress at each node (*SEQV*)

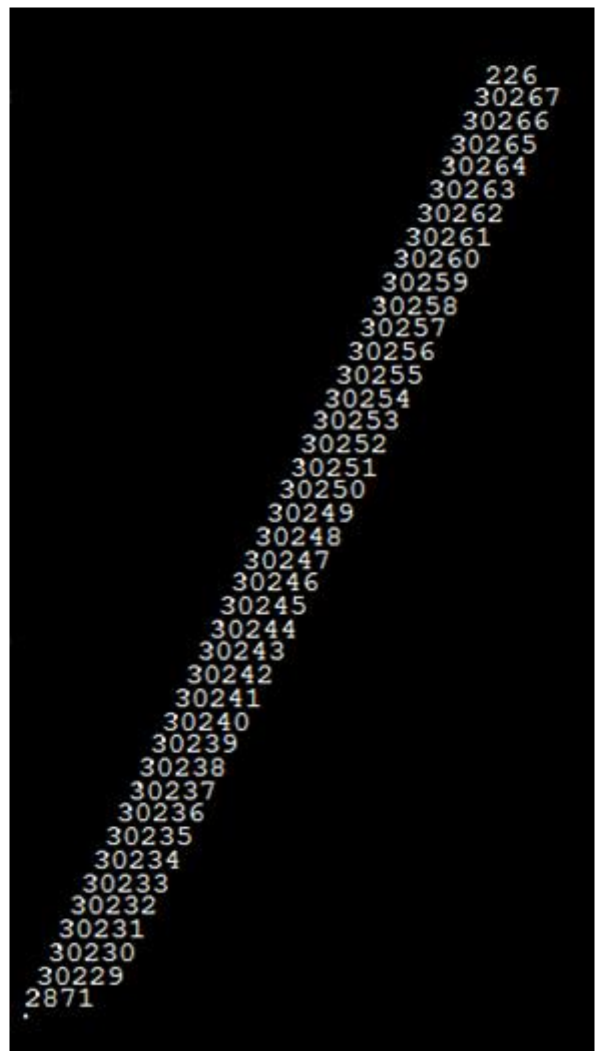


If there is any uncertainty pertaining to node numbers, you can plot the node numbers in a nodal plot of the beam.



1. Go to **Utility Menu -> Plot Ctrls -> Numbering …**
2. Check the box next to **NODE Node numbers**
3. Click **OK**
4. Go to **Utility Menu -> Plot -> Nodes**
5. In the graphics window, you can inspect a few node numbers in the top row of nodes. Once these node numbers are collected, look them up in the **PRNSOL Command** window to determine the stresses at the top of the beam





According to the PRNSOL Command Window, the stress At the top and bottom surfaces of the beam is 40106 Pa.

Results

Max Deflection Error

The percent error (%E) in our model max deflection can be defined as:

$$\%E = abs\left(\frac{\delta_{theoretical} - \delta_{model}}{\delta_{theoretical}}\right) * 100 = \mathbf{1.31\%} \tag{1.8.15}$$

This is a very good error baseline for the mesh considering eqation 1.8.9 *is quadratic with respect to displacement*. Since the 3D Elements we are using *linearly interpolate* between nodes, we can expect a degree of *truncation error* in our model. As we will show in our v*alidation* section, our model will converge to the expected solution as the mesh is refined.

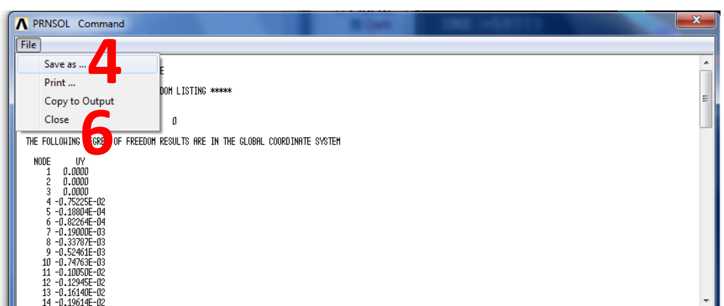
Max Equivalent Stress Error

Using the same definition of error as before, we derive that our model has **4.51%** error in the max equivalent stress. The error results from extrapolation of stress at the integration points of the elements to the nodal values. We will see later that if *higher order elements* are used in this problem, the error in equivalent stress will disappear!

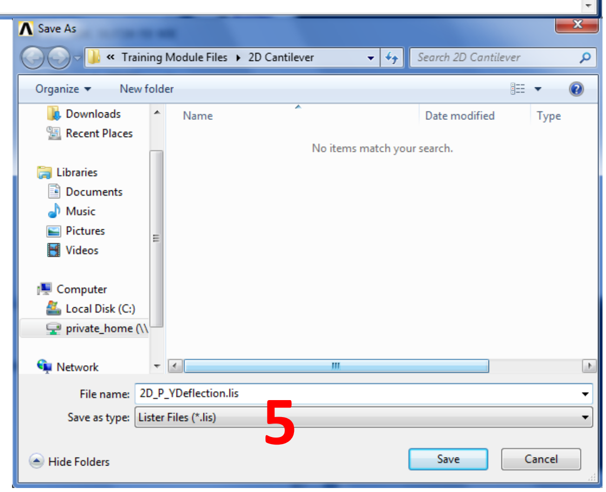
Further Analysis

In addition to this baseline data, we can export both the deflection and Von-Mises data to *Excel*.

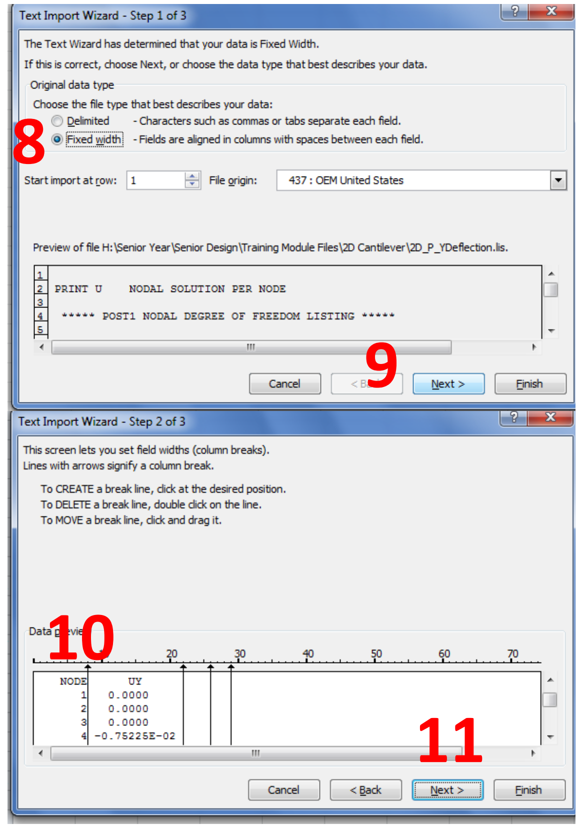
1. Go to **Utility Menu -> List -> Results -> Nodal Solution …**
2. Select **Nodal Solution -> DOF Solution -> Y-component of displacement**
3. Click **OK**



4. The list file should populate. Go to **PRNSOL Command -> File -> Save As …**
5. Save the file as *3D_M_YDeflection.lis* to the path of your choice
6. Go to **PRNSOL Command -> File -> Close**



7. Open *3D_M_YDeflection.lis* in *Excel*
8. Click **Fixed Width**
9. Click **Next >**



10. Click a location on the ruler between the *NODE* and *UY* columns. This will cause *Excel* to separate these columns into separate columns in the spreadsheet
11. Click **Next >**
12. Click **Finish**

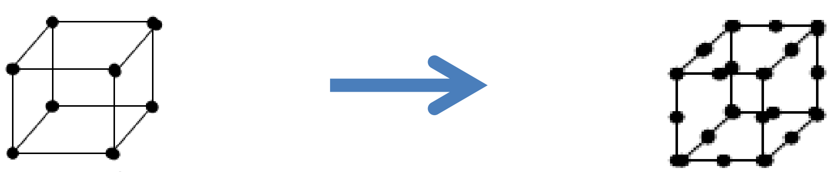
This process can be repeated for Von-Mises Stress data.

Remember, we want to *plot the centerline nodes* as this will give us the best representation of beam theory values.

<u>Higher Order Elements (Midsize Nodes)</u>

Mesh refinement is difficult to do with this model. If you are using the academic version of *ANSYS Mechanical APDL*, you will be limited to 256,000 equations the FEA code can solve. That means the software cannot handle more than *five elements* through the thickness of the beam. Since we have already specified a beam with four elements through the thickness for this tutorial, there is not much room for improvement.

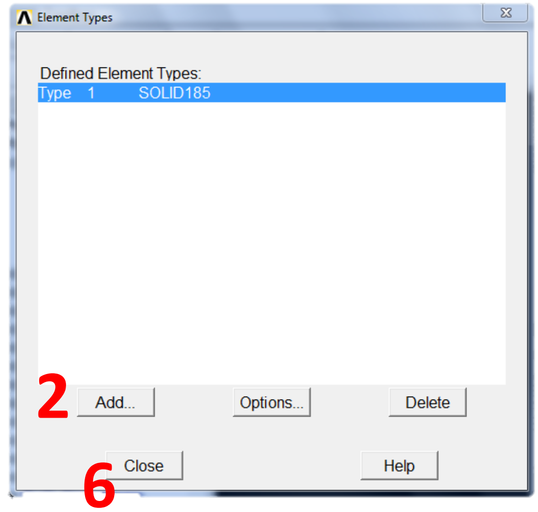
Up until this point, we have only used elements with *linear shape functions* between nodes. However, ANSYS provides a variety of higher order element types with *higher order* polynomial shape functions between nodes. This is achieved through the usage of *midsize nodes*, nodes that lie midway between the nodes of the typical linear element. In the case of 3D elements, a higher order 3D element such as *SOLID 186* will have 20 nodes instead of 8.

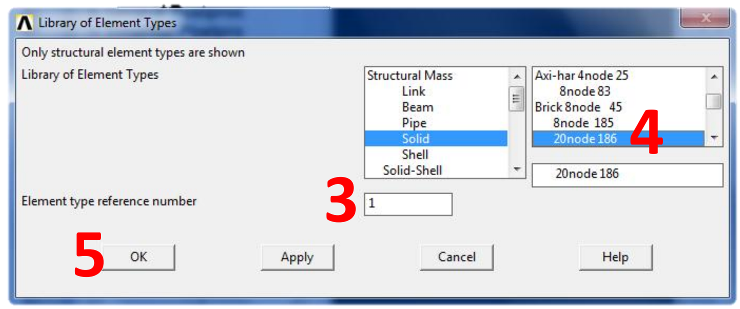


SOLID 185 **SOLID 186**

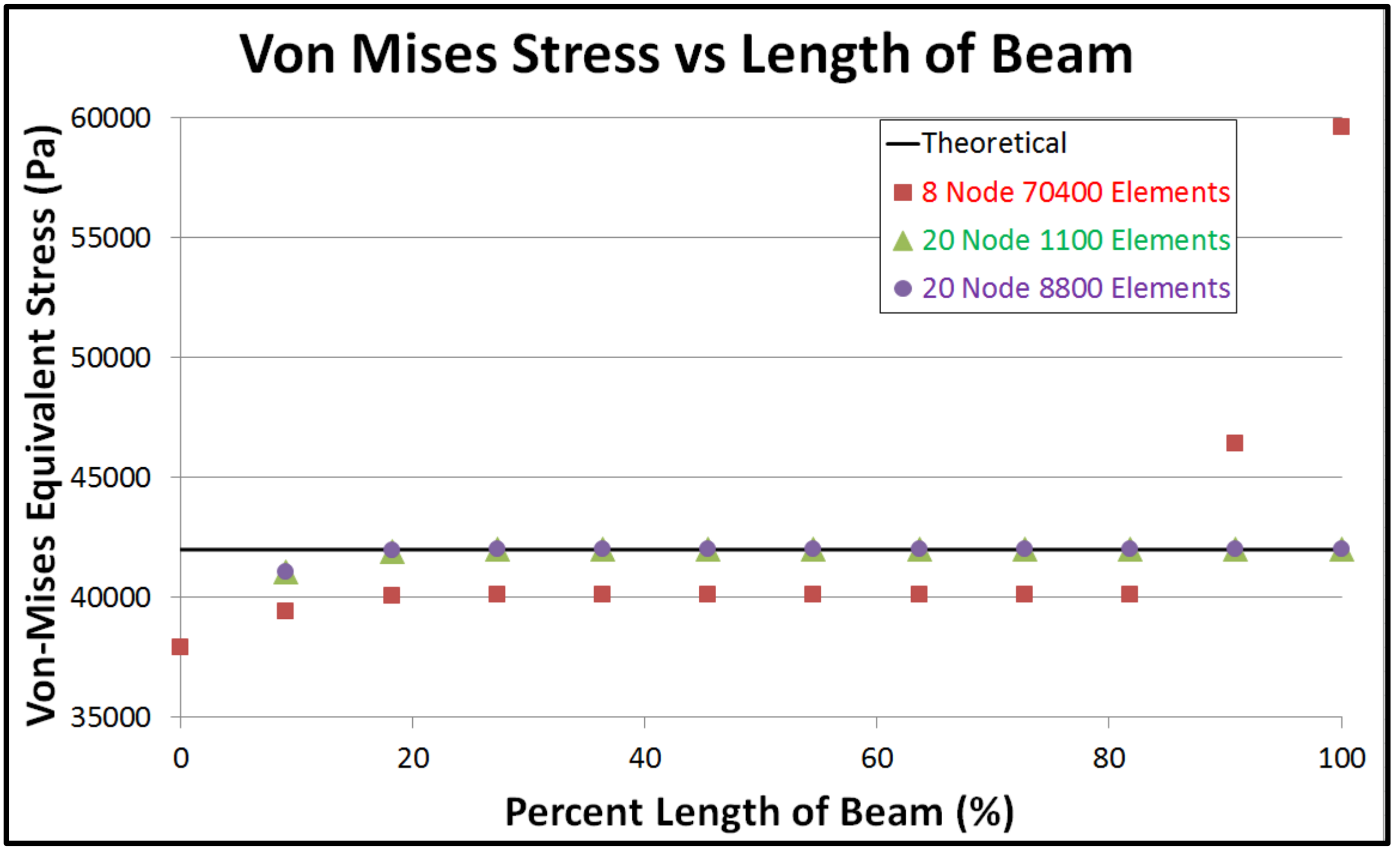
1. Go to **Main Menu -> Preprocessor -> Element Type -> Add/Edit/Delete**
2. Click **Add…**
3. Under **Element type reference number** enter 1
4. Select **Solid -> 20Node 186** this will override the current element settings.
5. Click **OK**
6. Click **Close**

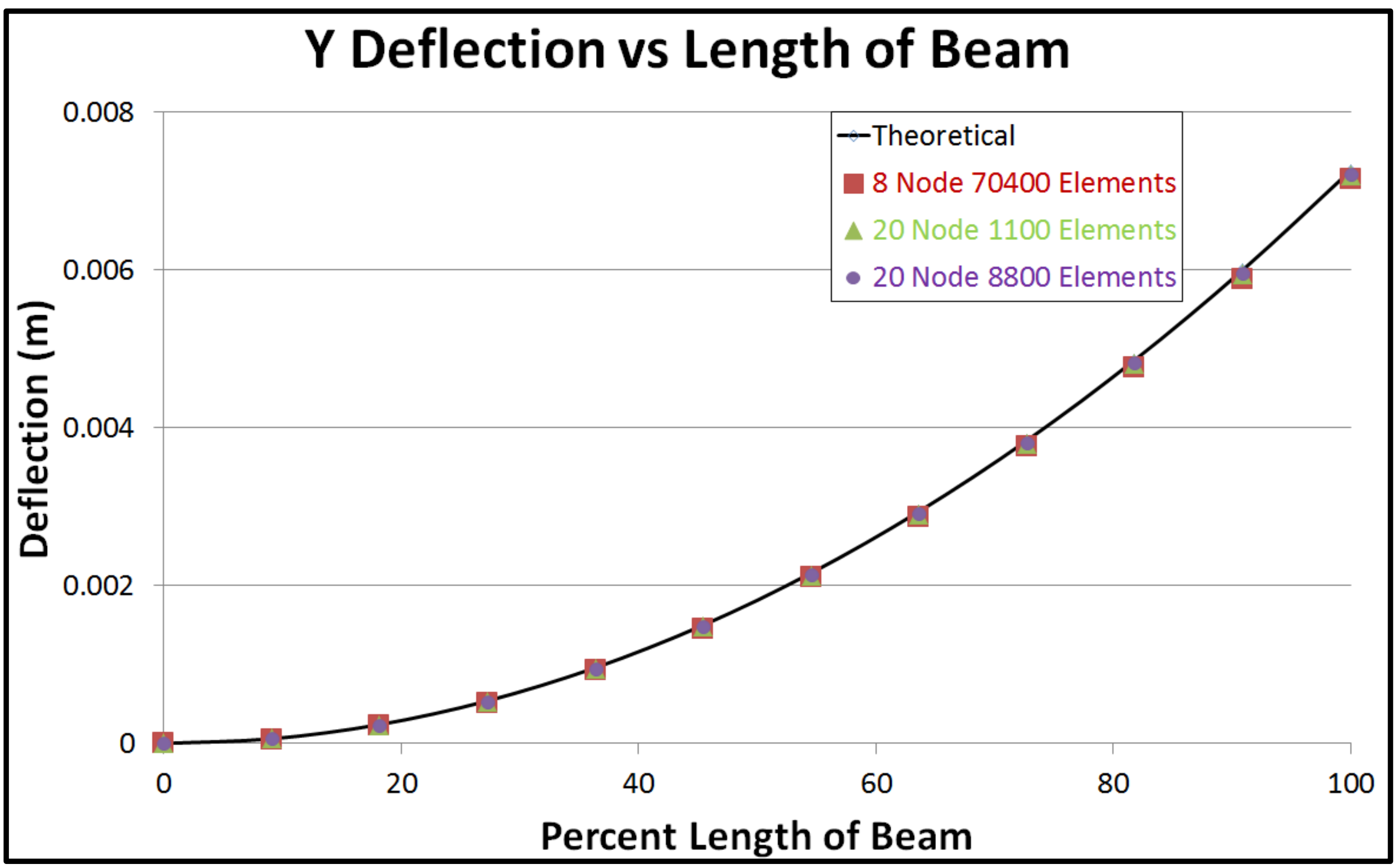
Now you are ready to re-mesh with 20 Node elements.





WARNING: Be careful when applying equation 1.8.13 to determine the loads across each row of nodes. Every row containing only midsize nodes will have **half the number of nodes**.


Von Mises Stress vs Length of Beam
Von-Mises Equivalent Stress (Pa)
Percent Length of Beam (%)
Theoretical
8 Node 70400 Elements
20 Node 1100 Elements
20 Node 8800 Elements
60000
55000
50000
45000
40000
35000
0
20
40
60
80
100


Y Deflection vs Length of Beam
Deflection (m)
Percent Length of Beam
Theoretical
8 Node 70400 Elements
20 Node 1100 Elements
20 Node 8800 Elements
0.008
0.006
0.004
0.002
0
0
20
40
60
80
100