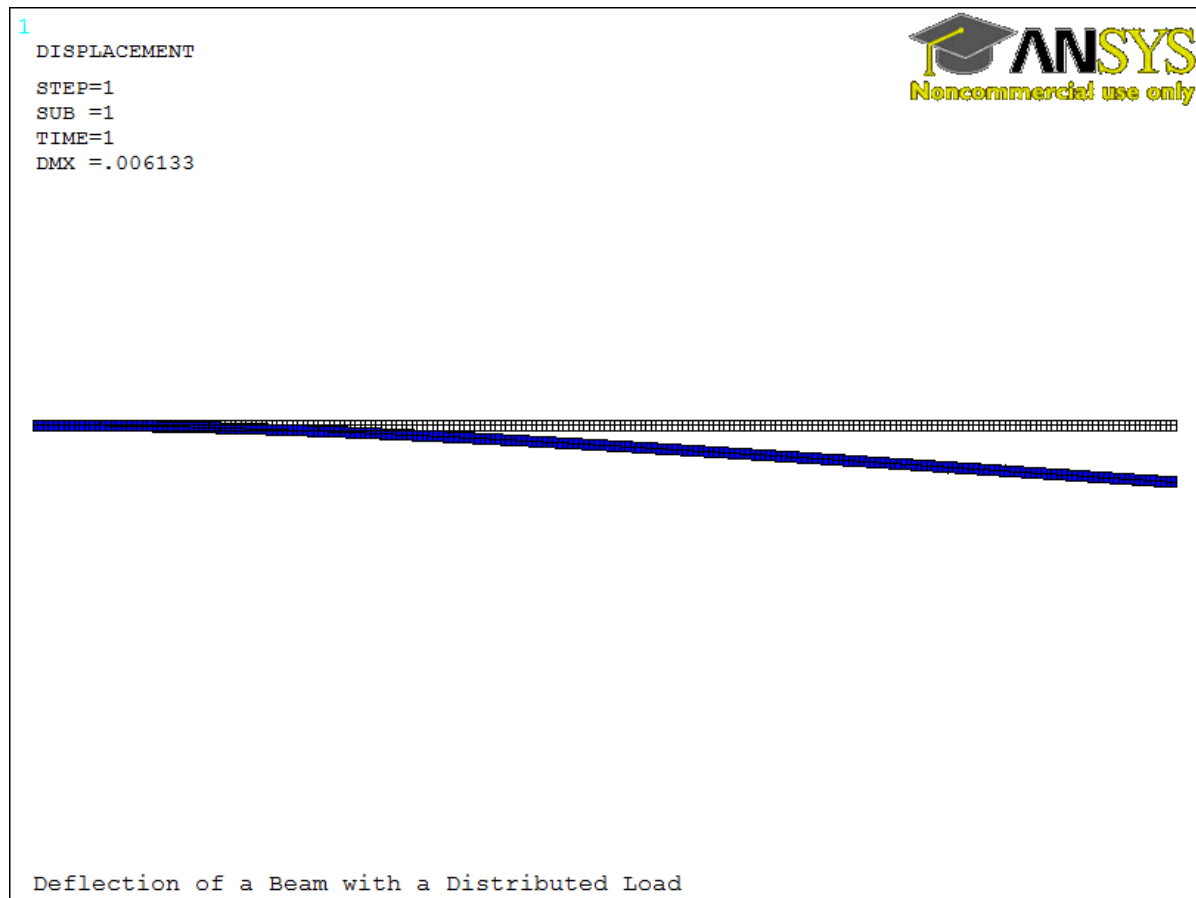
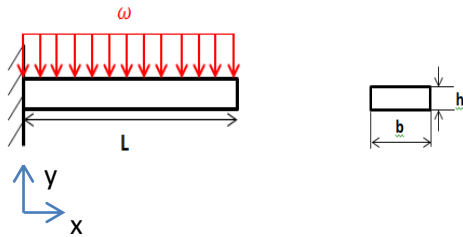


Module 1.9: Distributed Loading of a 3D Cantilever Beam



<u>Table of Contents</u>	Page Number
Problem Description	2
Theory	2
Geometry	4
Preprocessor	6
Element Type	6
Material Properties	7
Meshing	7
Loads	9
Solution	11
General Postprocessor	11
Results	14
Validation	15

Problem Description



Nomenclature:

L =110m	Length of beam
b =10m	Cross Section Base
h =1 m	Cross Section Height
w =20N/m	Distributed Load
E =70GPa	Young's Modulus of Aluminum at Room Temperature
ν =0.33	Poisson's Ratio of Aluminum

In this tutorial, we will be analyzing a cantilever beam with a distributed load. The left side of the cantilever beam is fixed while there is a distributed load of 20N/m. The objective of this problem is to find Von-Mises stress and total deflection throughout the beam. 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:

$$\sigma' = (\sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3\tau_{xy}^2)^{\frac{1}{2}} \quad (1.9.1).$$

Additionally, since the nodes of choice are located at the top surface of the beam, the shear stress at this location is zero.

$$(\tau_{xy} = 0, \sigma_y = 0). \quad (1.9.2)$$

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

$$\sigma' = \sigma_x \quad (1.9.3)$$

Bending Stress is given by:

$$\sigma_x = -\frac{Mc}{I} \quad (1.9.4)$$

Where $I = \frac{1}{12}bh^3$ and $c = \frac{h}{2}$.

From statics, we can derive:

$$M = -\frac{w}{2}(L-x)^2 = \frac{w}{2}(2Lx - L^2 - x^2) \quad (1.9.5)$$

Plugging into equation 1.6.4, we get:

$$\sigma_x = \frac{\omega(L-x)^2 c}{2I} \quad (1.9.6)$$

$$\sigma_x = \frac{3\omega(L-x)^2}{bh^2} = \mathbf{72.6kPa} \quad (1.9.7)$$

Beam Deflection

The equation to be solved is:

$$\frac{d^2 y}{dx^2} = \frac{M(x)}{EI} \quad (1.9.7)$$

Plugging in equation 1.6.5, we get:

$$EI \frac{d^2 y}{dx^2} = \frac{w}{2} (2Lx - L^2 - x^2) \quad (1.9.8)$$

Integrating once to get angular displacement we get:

$$EI \frac{dy}{dx} = \frac{w}{2} \left(L \frac{x^2}{2} - xL^2 - \frac{x^3}{3} \right) + C_1 \quad (1.9.9)$$

At the fixed end ($x=0$), $\theta(0) = \frac{dy(0)}{dx} = 0$, thus $C_1 = 0$

$$EI \frac{dy}{dx} = \frac{w}{2} \left(L \frac{x^2}{2} - xL^2 - \frac{x^3}{3} \right) \quad (1.9.10)$$

Integrating again to get deflection:

$$EI y = \frac{w}{2} \left(L \frac{x^3}{3} - \frac{x^2}{2} L^2 - \frac{x^4}{12} \right) + C_2$$

At the fixed end $y(0)=0$ thus $C_2 = 0$, so deflection ($\delta = y$) is:


$$\delta = \frac{wx^2}{24EI} (4Lx - 6L^2 - x^2) \quad (1.9.11)$$

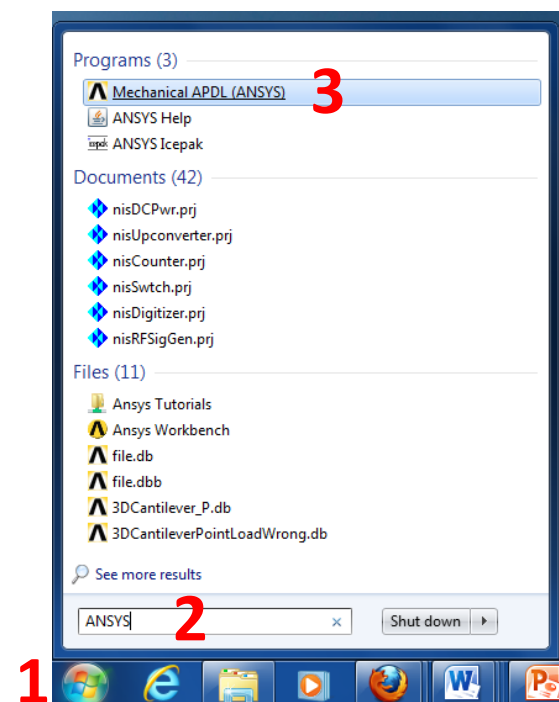
The maximum displacement occurs at the point load ($x=L$)

$$\delta_{max} = -\frac{wL^4}{8EI} = \mathbf{6.27mm} \quad (1.9.12)$$

Geometry

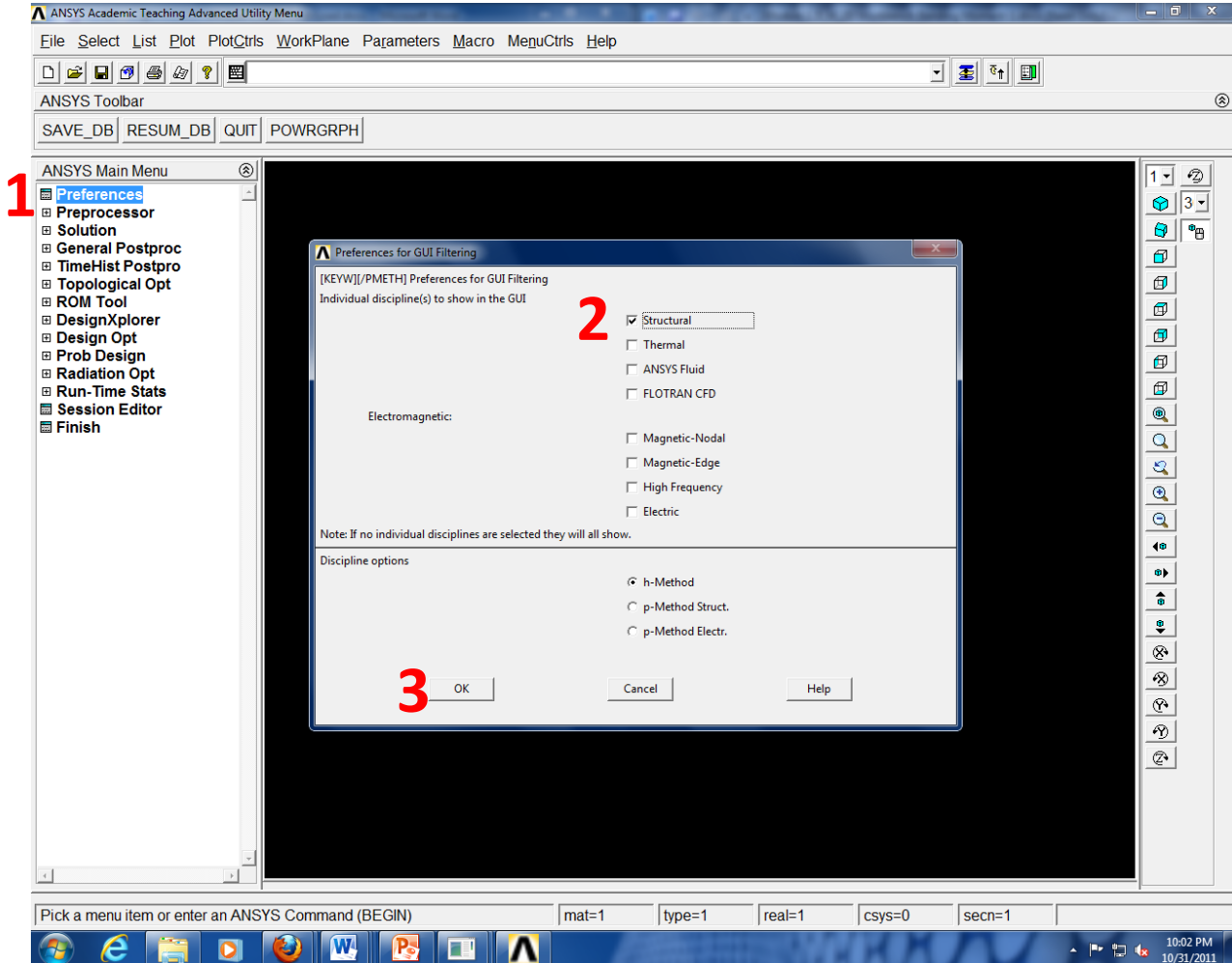
Opening ANSYS Mechanical APDL

1. On your Windows 7 Desktop click the **Start** button
2. Under **Search Programs and Files** type “ANSYS”
3. Click on  **Mechanical APDL (ANSYS)** to start ANSYS. This step may take time.



Preferences

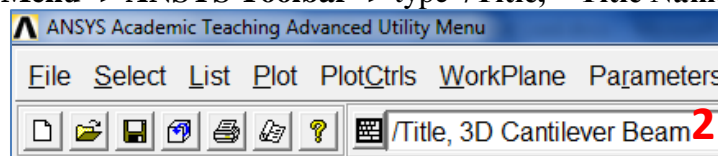
1. Go to **Main Menu -> Preferences**
2. Check the box that says **Structural**
3. Click **OK**



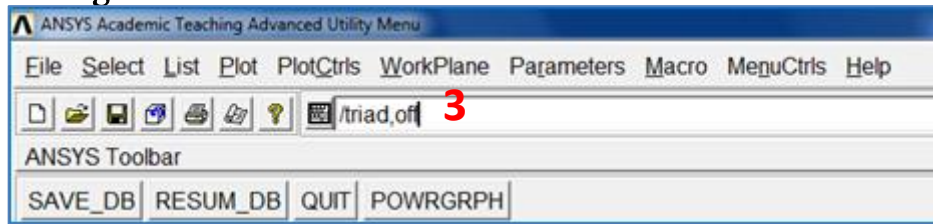
Title and Triad:

To add a title

1. **Utility Menu -> ANSYS Toolbar -> type /prep7 -> enter**
2. **Utility Menu -> ANSYS Toolbar -> type /Title, " Title Name" -> enter**

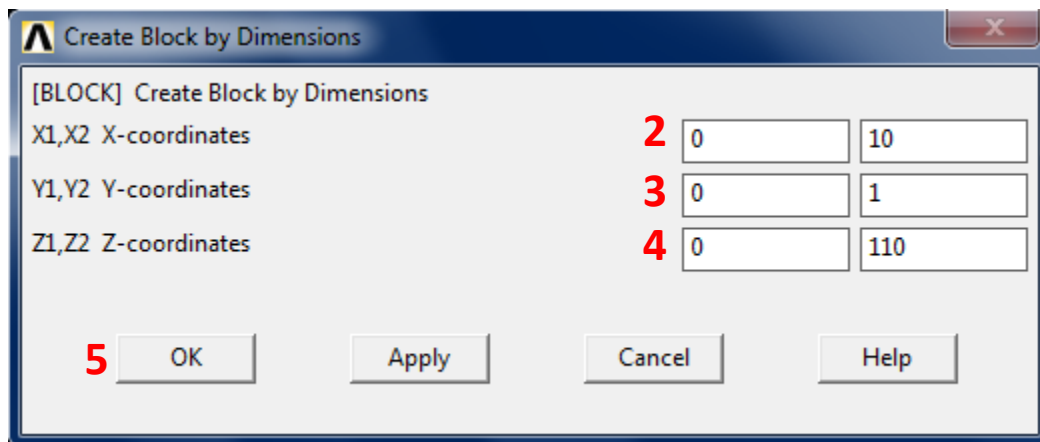


3. The **Triad** in the top left will block images along the way.
To get rid of the triad, type
/triad,off in **Utility Menu -> Command Prompt**



Beam:

1. Go to **ANSYS Main Menu -> Preprocessor -> Modeling -> Create -> Volumes -> Block -> By Dimensions**. This will open a new window, *Create Block by Dimensions*, where the Geometry will be created.
2. In **Create Block by Dimensions -> X1,X2 X-coordinates -> input 0 -> tab 2 input 10**
3. In **Create Block by Dimensions -> Y1,Y2 Y-coordinates -> input 0 -> tab 2 input 1**
4. In **Create Block by Dimensions -> Z1,Z2 Z-coordinates -> input 0 -> tab 2 input 110**
5. Then hit **Ok** to create the 3-Dimensional Cantilever Beam



This will generate a cantilever beam as shown:



SAVE_DB

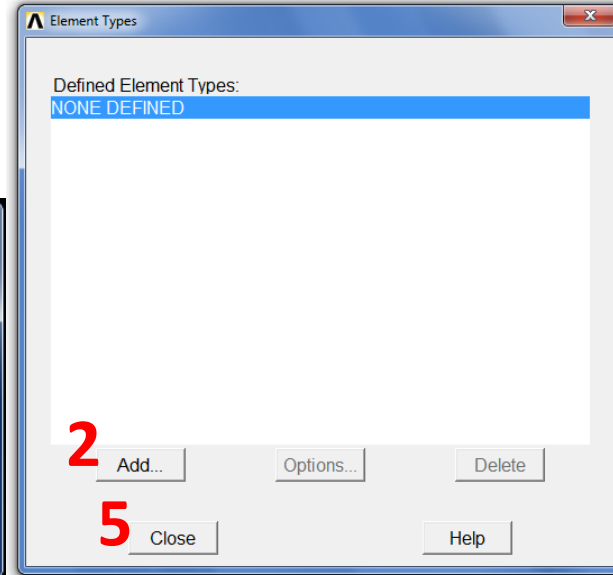
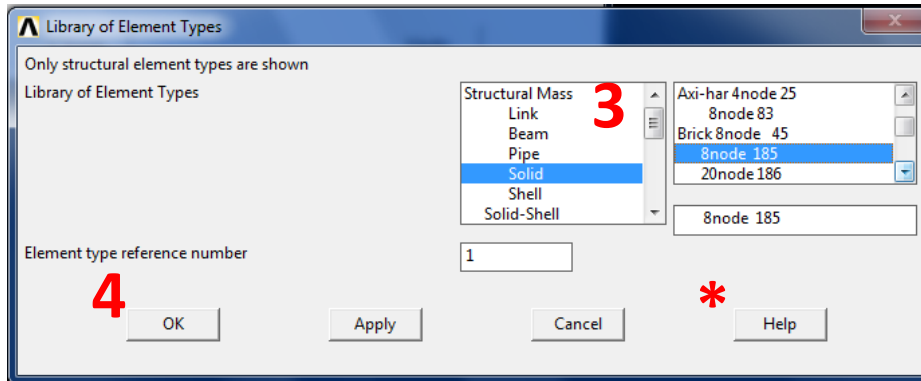
Since we have made considerable progress thus far, we will create a temporary save file for our model. This temporary save will allow us to return to this stage of the tutorial if an error is made.

1. Go to **Utility Menu -> ANSYS Toolbar -> SAVE_DB** This creates a save checkpoint
2. If you ever wish to return to this checkpoint in your model generation, go to **Utility Menu -> RESUM_DB**

Preprocessor

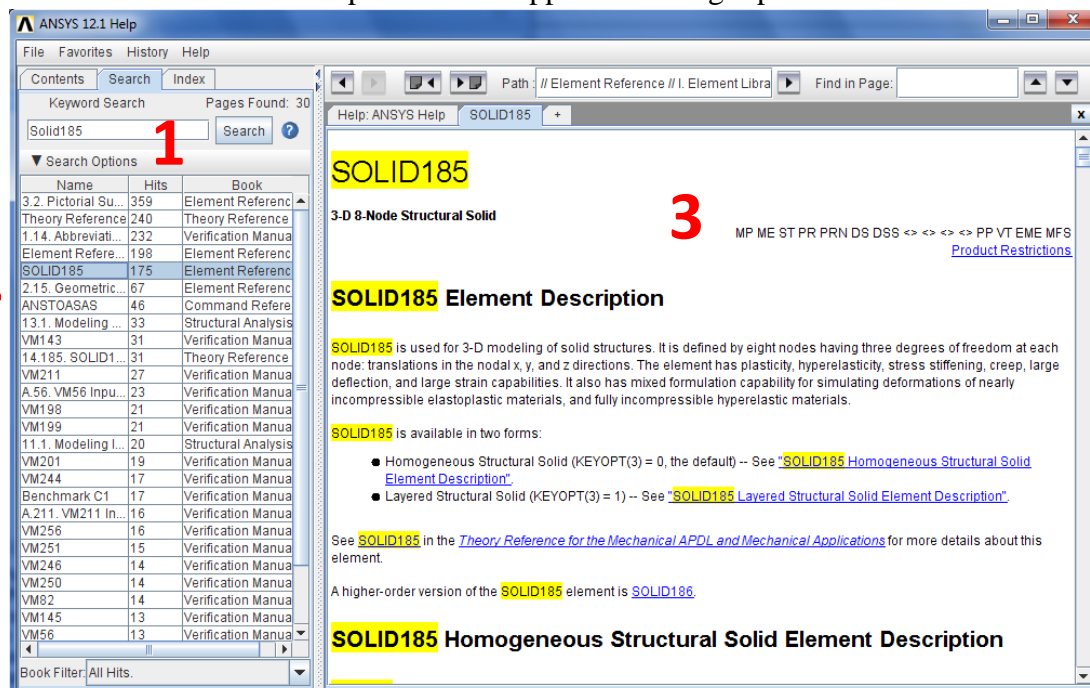
Element Type

1. Go to **Main Menu -> Preprocessor -> Element Type -> Add/Edit/Delete**
2. Click **Add**
3. Click **Solid -> 8node 185**
4. Click **OK**
5. Click **Close**




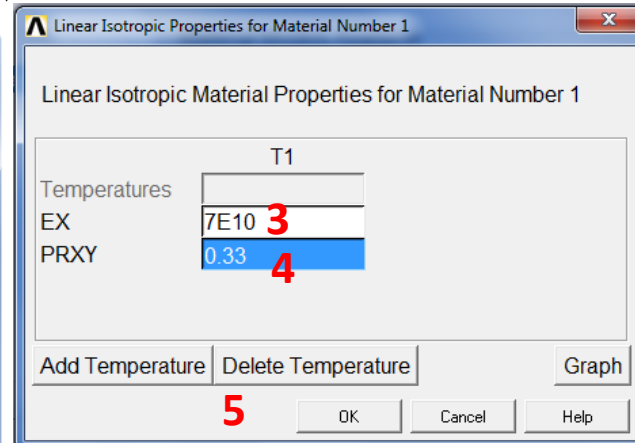
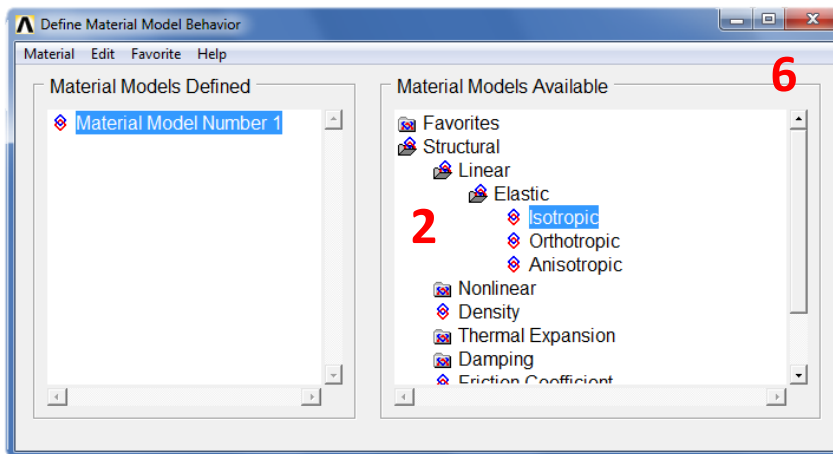
*For more information Solid185 click Help

1. Go to **ANSYS 12.1 Help -> Search Keyword Search -> type 'Solid185' and press Enter**
2. Go to **Search Options -> SHELL185**
3. The element description should appear in the right portion of the screen.



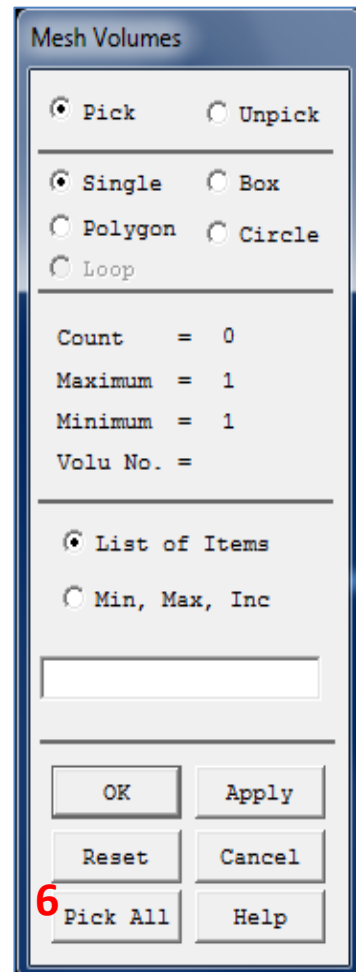
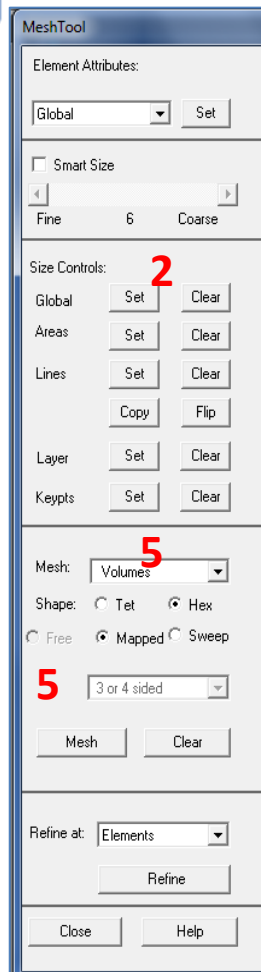
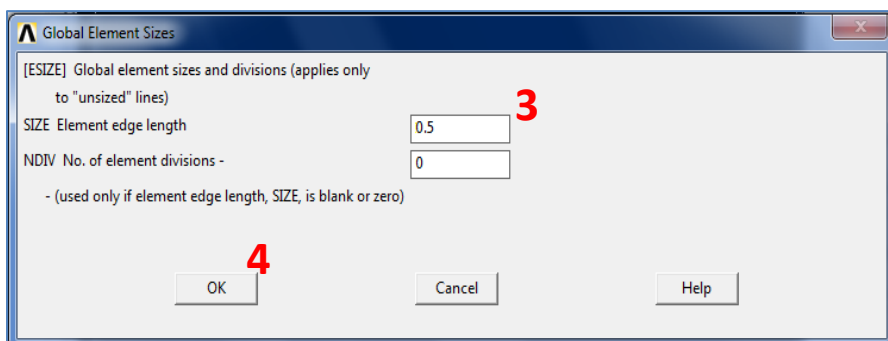
Material Properties

1. Go to **Main Menu -> Preprocessor -> Material Props -> Material Models**
2. Click **Material Model Number 1 -> Structural -> Linear -> Elastic -> Isotropic**
3. Input 7E10 for the Young's Modulus (Aluminum) in **EX**
4. Input 0.33 for Poisson's Ratio in **PRXY**
5. Click **OK**
6.  of **Define Material Model Behavior** window

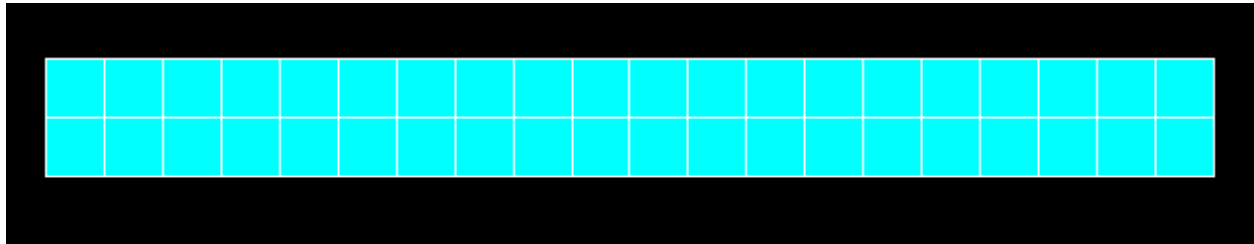






Meshing

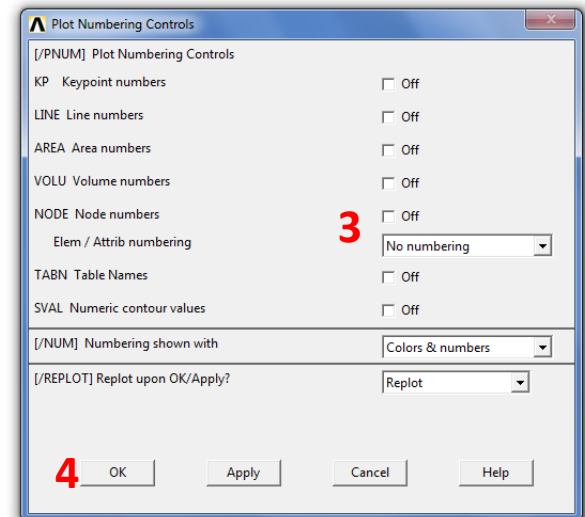
1. Go to **Main Menu -> Preprocessor -> Meshing -> Mesh Tool**
2. Go to **Size Controls: -> Global -> Set**
3. Under **SIZE Element edge length** put 0.5.
The **SIZE Element edge length** puts 1 element every distance you enter. This will do 2 elements every 1 meter.
4. Click **OK**
5. Click **Hex** followed by **Mesh**
6. Click **Pick All**



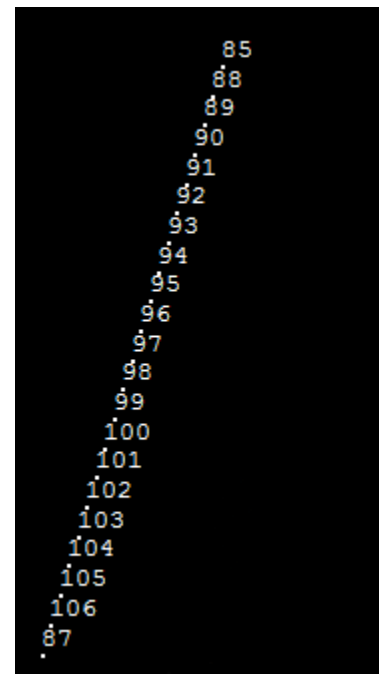
After meshing, your cantilever beam should look like the image below:



1. Go to **Utility Menu -> Plot -> Nodes**
2. Go to **Utility Menu -> Plot Controls -> Numbering...**
3. Check **NODE Node Numbers** to **ON**
4. Click **OK**
5. Click the  **Left View** to orient the cantilever beam horizontally down the z-axis
6. Shift the beam the left to view the far nodes more closely by pressing the **Pan Model Left**  button then zoom in on the far right nodes using the **Zoom in**  button or scrolling with the mouse
7. Use the  **Dynamic Model Mode** and right clicking slightly




The resulting graphic should be as shown:



You can follow the procedure above to remove node numbering.

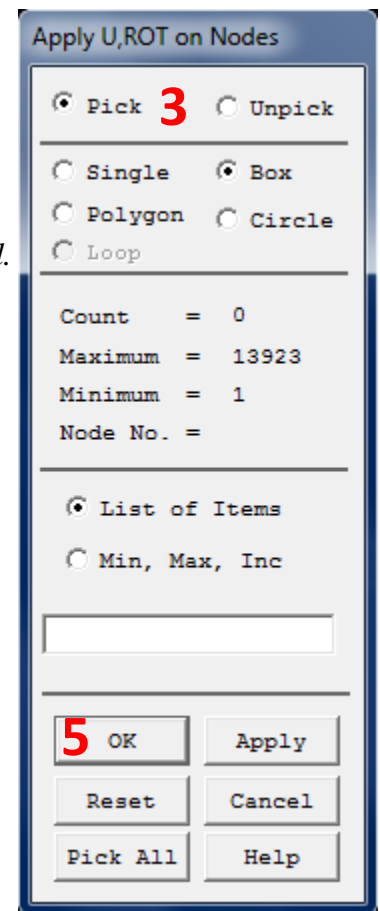
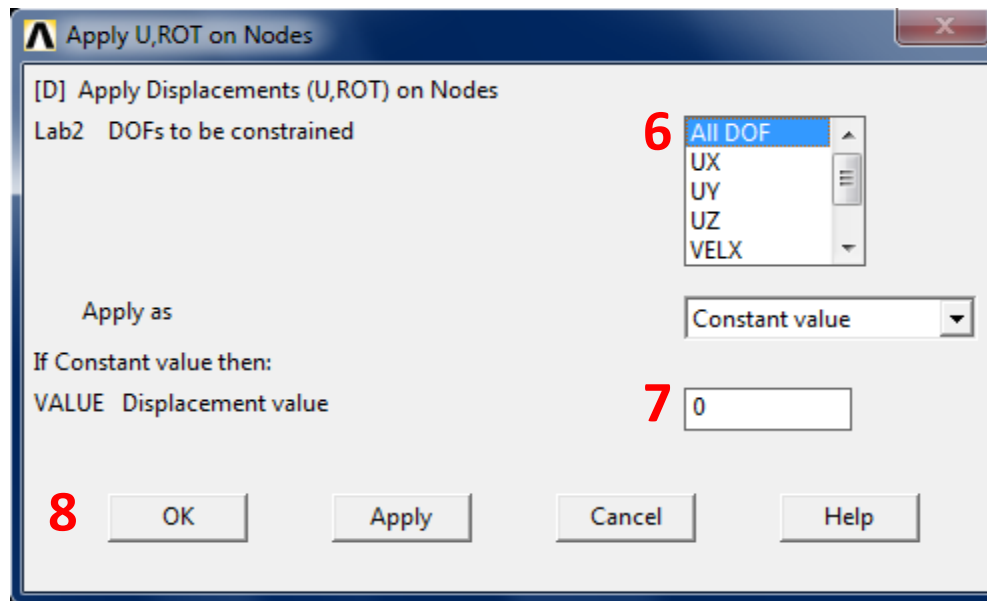
This is one of the main advantages of *ANSYS Mechanical APDL* vs *ANSYS Workbench* in that we can visually extract the node numbering scheme. This is one of the main advantages of *ANSYS Mechanical APDL* vs *ANSYS Workbench* in that we can visually extract the node numbering scheme. As shown, *ANSYS* numbers nodes at the left corner, the right corner, followed by filling in the remaining nodes from left to right.

Displacement (Fixed End)

1. Click the  **Left View** to see along the z-axis
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:




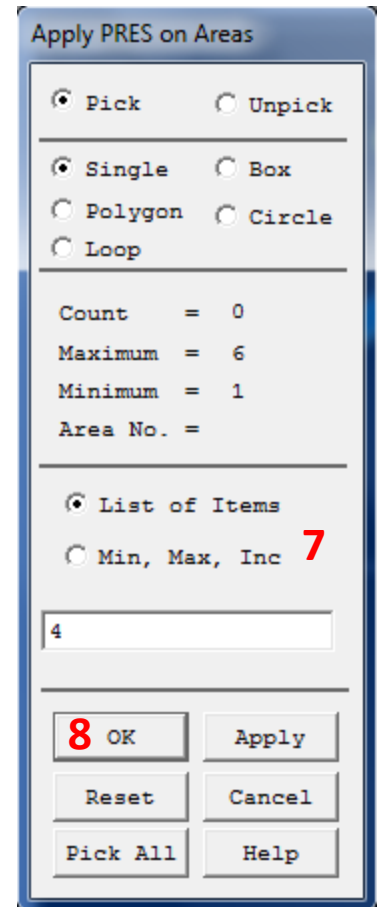
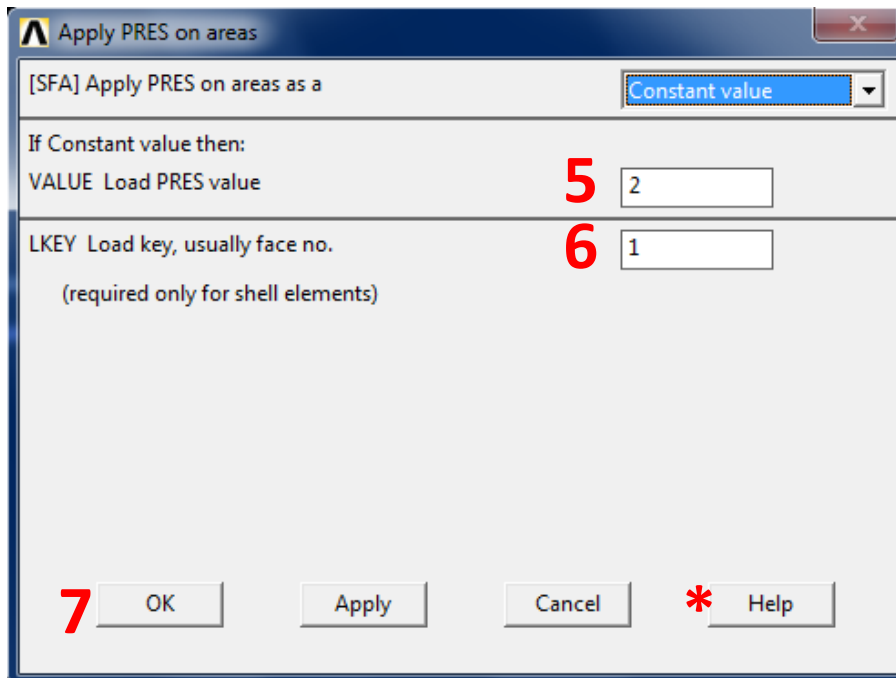
5. Click **OK**
6. Click **All DOF** to secure all degrees of freedom
7. Under **Value Displacement value** put 0. The left face is now a *fixed end*.
8. Click **OK**



WARNING: Selecting the wrong/wrong amount of nodes will result in a wrong answer; make sure the only nodes selected are only the end set as shown.

Distributed Load

1. Go to **Utility Menu -> Plot -> Areas**
2. Go to **Utility Menu -> Plot Controls -> Numbering...**
3. Check **AREA Area numbers** to **ON**
4. Click **OK**
5. Click the  **Isometric View** to view the cantilever beam angled. Here it is visible that **Area 4** is the top Area
6. Go to **Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Pressure -> On Areas**
7. Click **List of Items** and Input **4** to select area 4
8. Click **OK**
9. Under **VALUE Load PRES value** input 2
Since this is a distributed load of 20 N/m, 2 was found by dividing our moment by the width of the beam, 10.
10. Under **LKEY Load key, usually face no.** input 1
This justifies the face and direction that the pressure will be applied



11. Click **Ok**

Solution

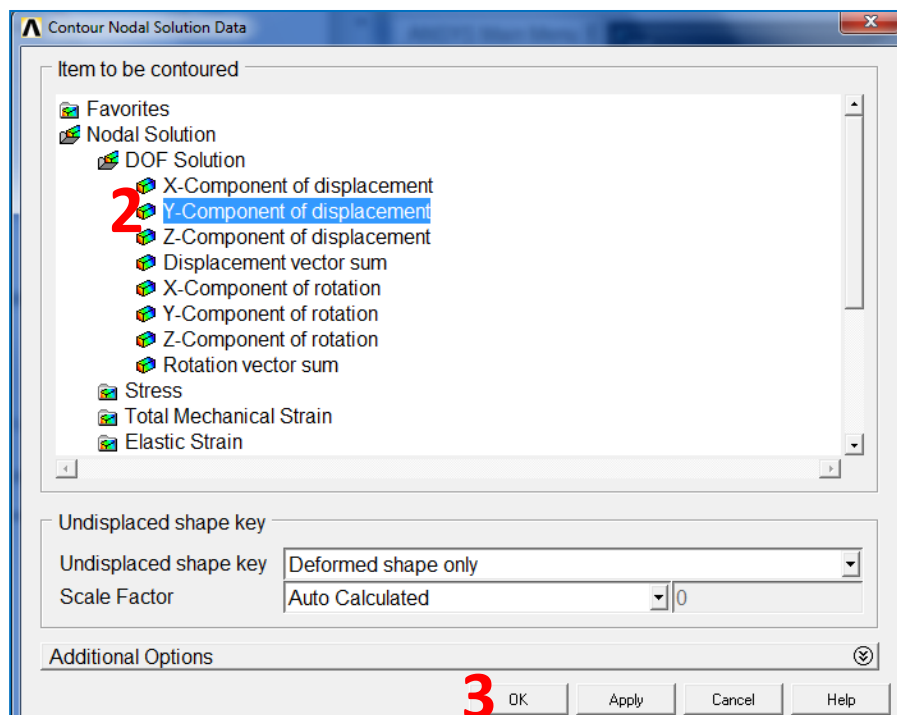
1. Go to **Main Menu -> Solution -> Solve -> Current LS** (*solve*). LS stands for Load Step.
This step may take some time depending on mesh size and the speed of your computer (generally a minute or less).

General Postprocessor

We will now extract the Displacement and Von-Mises Stress within our model.

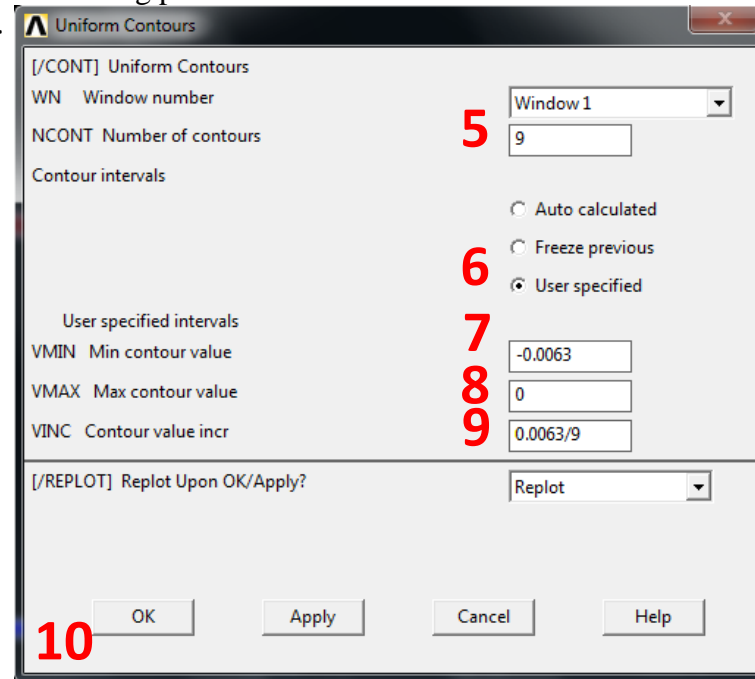
Displacement

1. Go to **Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solution**
2. Go to **DOF Solution -> Y-Component of displacement**
3. Click **OK**



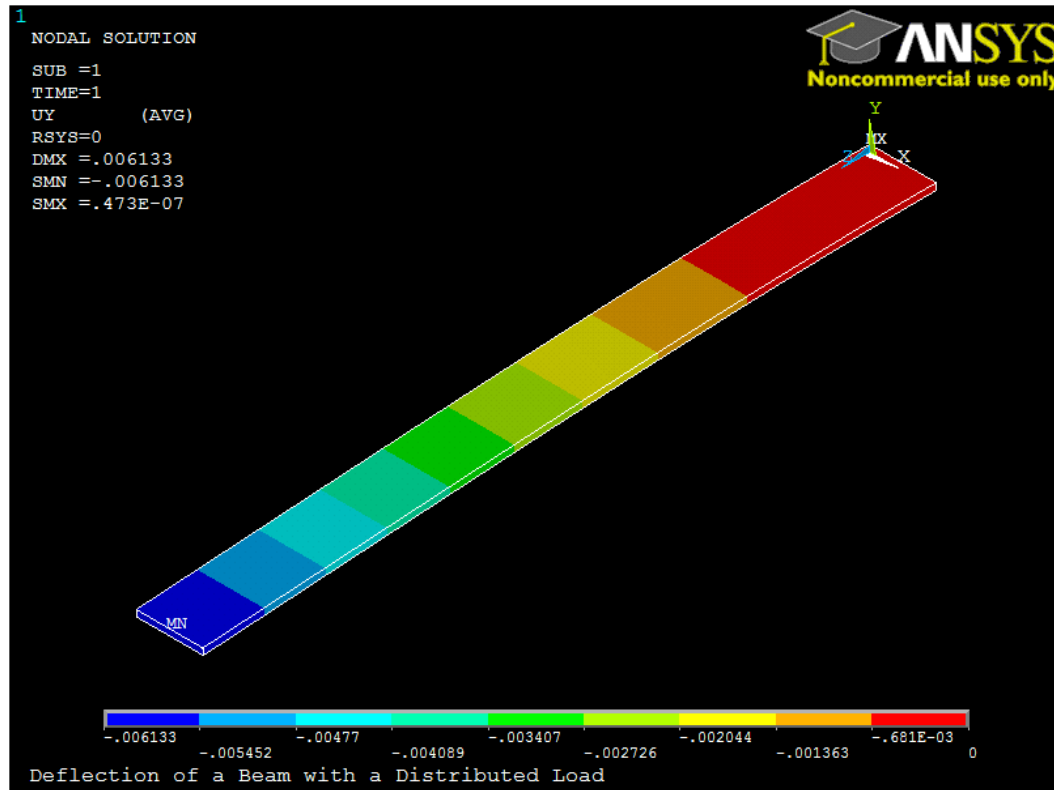
*Numbers 5-11 make alterations in the contour plot for viewing pleasure

4. Go to **Utility Menu -> PlotCtrls -> Style -> Contours -> Uniform Contours...**
5. Under **NCOUNT** enter 9
6. Under **Contour Intervals** click **User Specified**
7. Under **VMIN** enter -0.006133
The beam deflects in the $-Y$ direction so
The max deflection is treated as a minimum
8. Under **VMAX** enter 0
9. Since we will be using 9 contour intervals, we will enter $0.006133/9$ for **VINC**
10. Click **OK**



11. Let's give the plot a *title*. Go to **Utility Menu -> Command Prompt** and enter:
`/title, Deflection of a Beam with a Distributed Load`
`/replot`


The resulting plot should look like this:



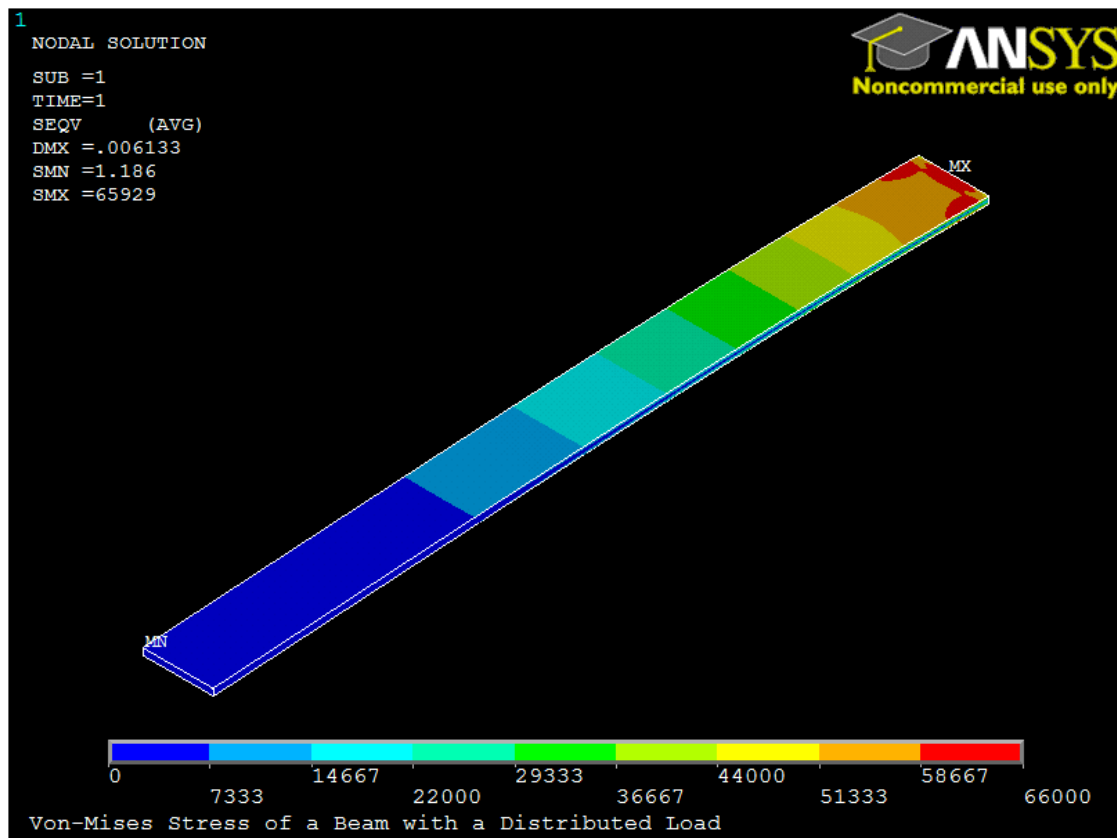
Equivalent (Von-Mises) Stress

1. Go to **Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solution**
2. Go to **Nodal Solution -> Stress -> von Mises stress**
3. Click **OK**
4. To get rid of the previous Plot Settings, go to **PlotCtrls -> Reset Plot Ctrls...**
5. Change the Title to “Von-Mises Stress of a Beam with a Distributed Load”
6. Go to **Utility Menu -> Plot -> Replot**

Aesthetics

1. Click the  **Isometric View** to see a better view of your cantilever beam.
2. Go to **Utility Menu -> PlotCtrls -> Style -> Contours -> Uniform Contours...**
3. Under **NCOUNT** enter 9
4. Under **Contour Intervals** click **User Specified**
5. Under **VMIN** enter 0
6. Under **VMAX** enter 66000
7. Under **VINC** enter 66000/9
8. Click **Ok**

Resulting Answer:



Results

Max Deflection Error

$$ANSYS \rightarrow \delta_{max} = 6.13mm$$

$$Theoretical \rightarrow \delta_{max} = 6.27mm$$

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

$$\%E = abs\left(\frac{\delta_{theoretical} - \delta_{model}}{\delta_{theoretical}}\right) * 100 = \mathbf{2.232\%} \quad (1.6.13)$$

Max Equivalent Stress Error

$$ANSYS \rightarrow \sigma_{max} = 65.9kPa$$

$$Theoretical \rightarrow \sigma_{max} = 72.6kPa$$

Using the same definition of error as before, we derive that our model has **9.28%** error in the max equivalent stress.

Using the same definition of error as before, we derive that our model has **9.28%** error in the max equivalent stress. The reason for the elevated stress level is singularity resulting from Poisson's effect at the fixed support. In the validation section, it is shown that with increased mesh size, the analytical answers for Max Equivalent stress are closely represented in nodes close to but not at the region where singularity occurs. The effect of singularity is also reduced with the implementation of higher order elements.

Validation

