Finite Element Course

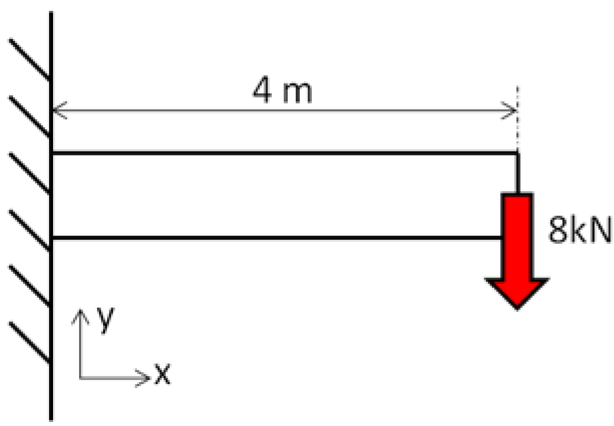
ANSYS Mechanical Tutorial

Tutorial 3 – Cantilever Beam

Problem Specification

Consider the beam in the figure below. It is clamped on the left side and has a point force of 8kN acting downward on the right end of the beam. The beam has a length of 4 meters, width of 0.346 meters and height of 0.346 meters (cross-section is a square). Additionally, the beam is composed of a material which has a Young's Modulus of 2.8x10^10 Pa. Using ANSYS, calculate the following:

1. Deformation of the beam
2. Maximum bending stress along the beam
3. Bending moment along the beam



Step 1: Start-up & preliminary set-up

Pre-Analysis

We'll start by carrying out hand calculations to predict the expected maximum bending stress and deformation. We'll later compare the ANSYS values to these hand-calculation results.

Calculate the Maximum Bending Stress at x=0m

The maximum bending stress is calculated using Euler-Bernoulli beam theory.

$$\sigma_{max} = \frac{M(x)Y}{I}$$
$$I = \frac{bh^3}{12}$$
$$\sigma_{max} = \frac{(8000N)(4m)\left(\frac{.346m}{2}\right)}{\frac{(0.346m)(0.346m)^3}{12}} = 4.635MPa$$

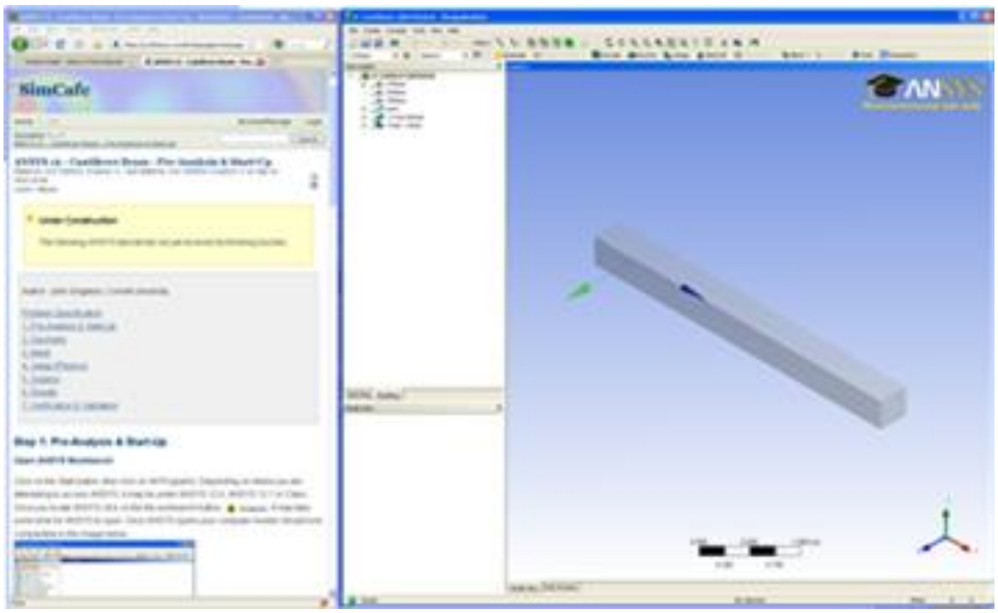
Calculate the Total Deformation at x=4m

$$u'' = \frac{M}{EI},\ u'(0) = 0,\ u(0) = 0$$
$$M = PL\left(1 - \frac{x}{L}\right)$$
$$u'' = \frac{PL}{EI}\left(1 - \frac{x}{L}\right)$$
$$u' = \int u''dx = \int \frac{PL}{EI}\left(1 - \frac{x}{L}\right)dx = \frac{PL}{EI}\left(x - \frac{x^2}{2L}\right) + C_1$$
$$u'(0) = 0\ \therefore\ u' = \frac{PL}{EI}\left(x - \frac{x^2}{2L}\right)$$
$$u = \int u'dx = \int \frac{PL}{EI}\left(x - \frac{x^2}{2L}\right)dx = \frac{PL}{EI}\left(\frac{x^2}{2} - \frac{x^3}{6L}\right) + C_2$$
$$u(0) = 0 \therefore\ u = \frac{PL}{EI}\left(\frac{x^2}{2} - \frac{x^3}{6L}\right)$$
$$u(4m) = \frac{(8000N)(4m)}{(2.8x10^{10})\left(\frac{(0.346m)(0.346m)^3}{12}\right)}\left(\frac{(4m)^2}{2} - \frac{(4m)^3}{6(4m)}\right) = 0.005103m$$

Start-Up

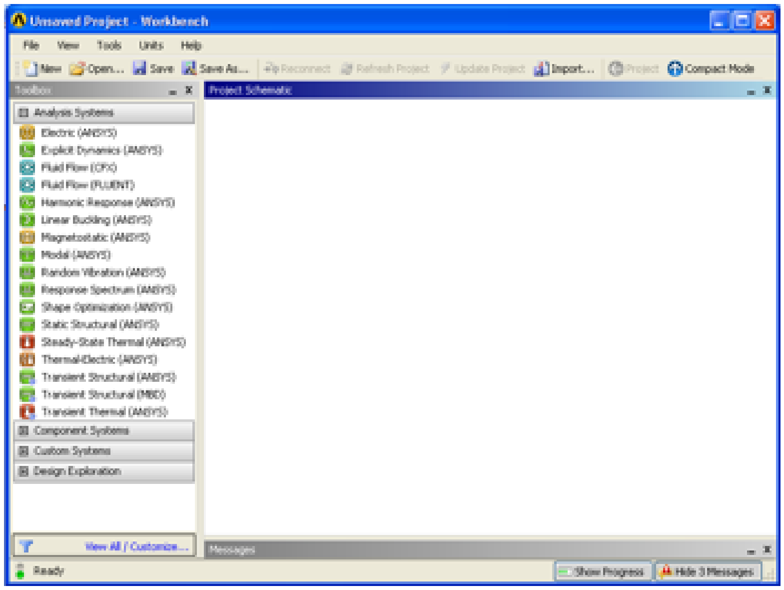
Optimizing Monitor Real Estate

This tutorial is specially configured, so the user can have both the tutorial and ANSYS open at the same time as shown below. It will be beneficial to have both ANSYS and your internet browser displayed on your monitor. Your internet browser should consume approximately one third of the screen width while ANSYS should take the other two thirds.



Open ANSYS Workbench

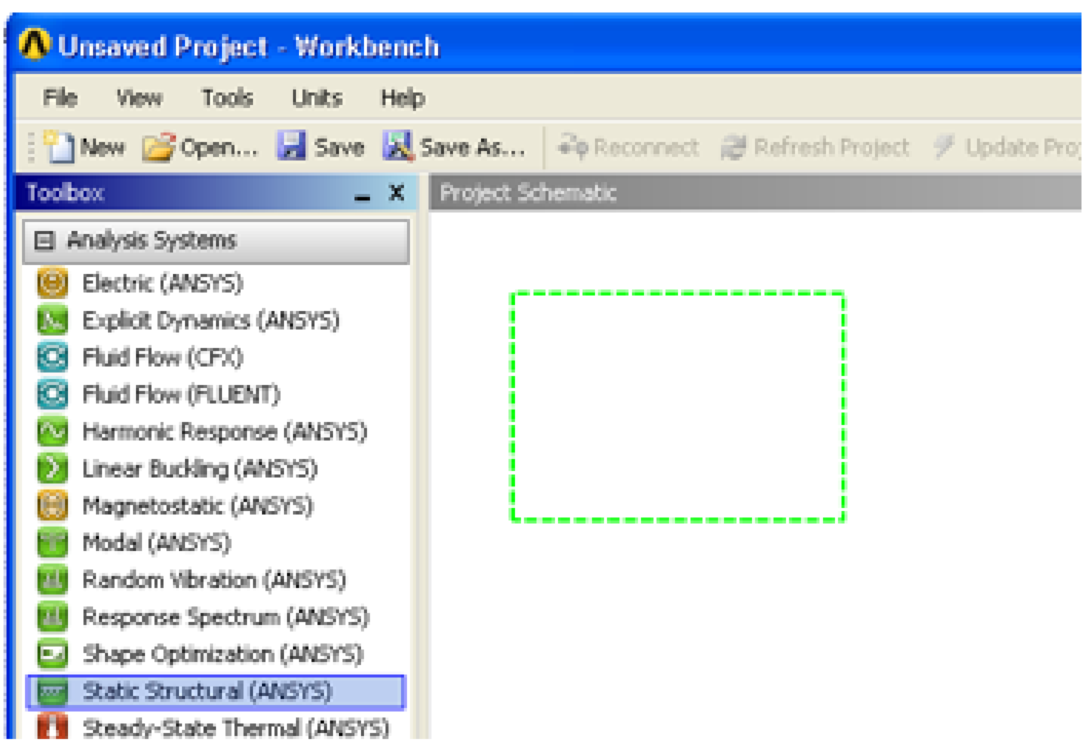
Click on the Start button, then click on All Programs. Depending on where you are attempting to access ANSYS, it may be under ANSYS 13.0, ANSYS, or Class. Once you locate ANSYS click on the the workbench button, Workbench. It may take some time for ANSYS to open. Once ANSYS opens your computer monitor should look comparable to the image below.



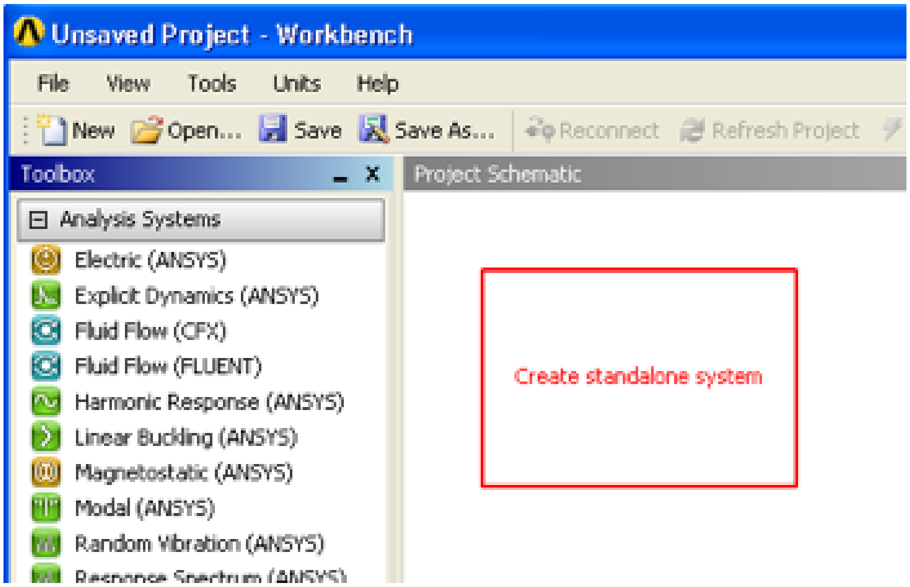
Static Structural Analysis System

The problem at hand is a static structural problem, so click and hold down the mouse button on the Static Structural (ANSYS) button, 

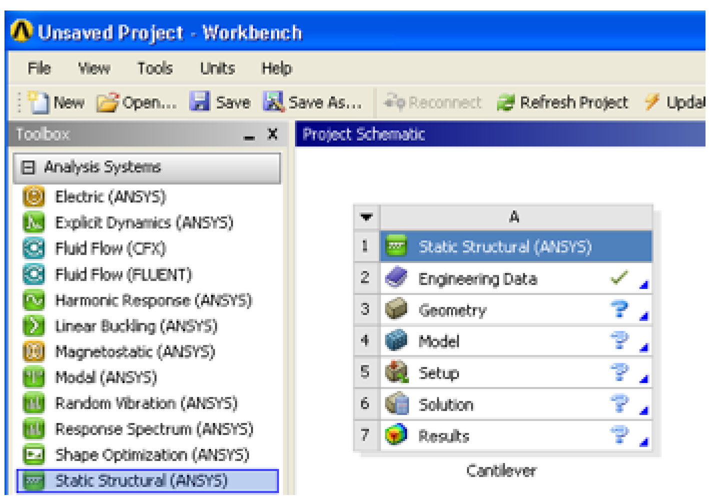
, and drag it over to the project schematic window. When you begin to drag the Static Structural (ANSYS) button over to the Project Schematic window a green dashed box should appear as seen in the image below.



Drag the Static Structural (ANSYS) button into the green box until it turns red and has the text "Create standalone system" within it, then release the mouse button.

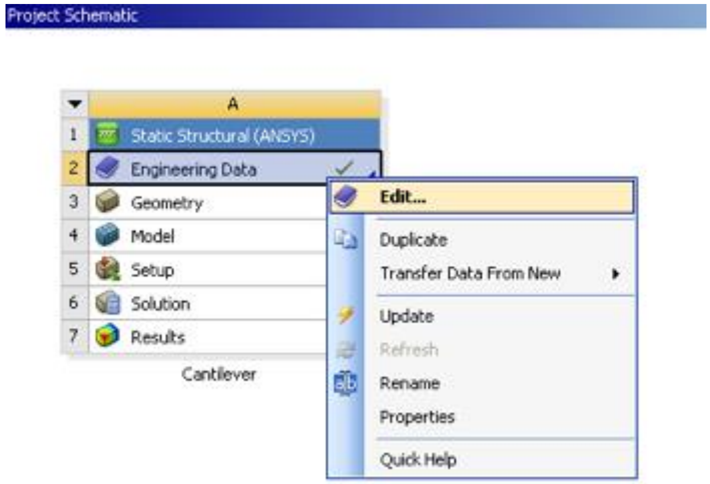


Change the name of the project to `Cantilever` and your workbench window should look similar to the image below.

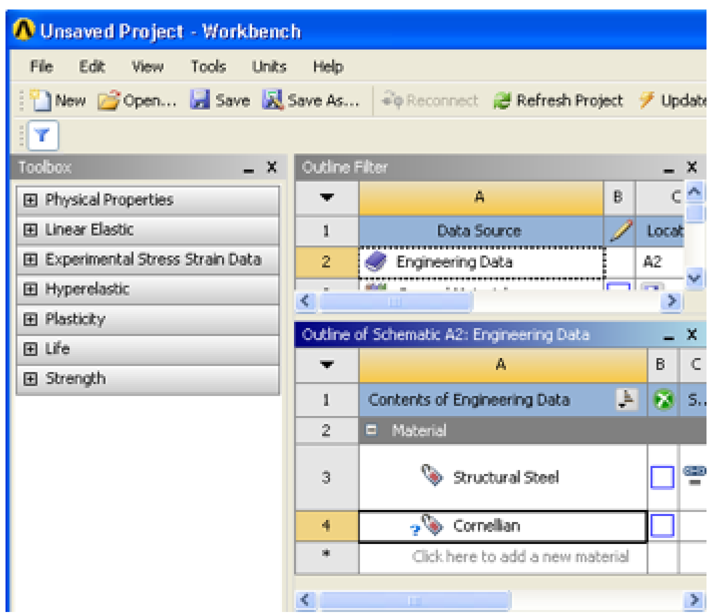


Engineering Data

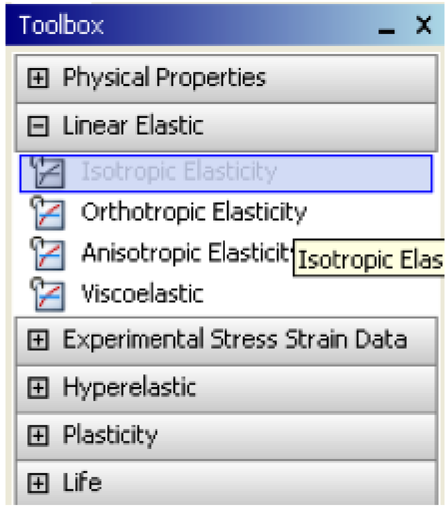
The specific properties of our material needs to be inputted into ANSYS. Start by right clicking on ***Engineering Data*** and then clicking on ***Edit..*** as seen below.



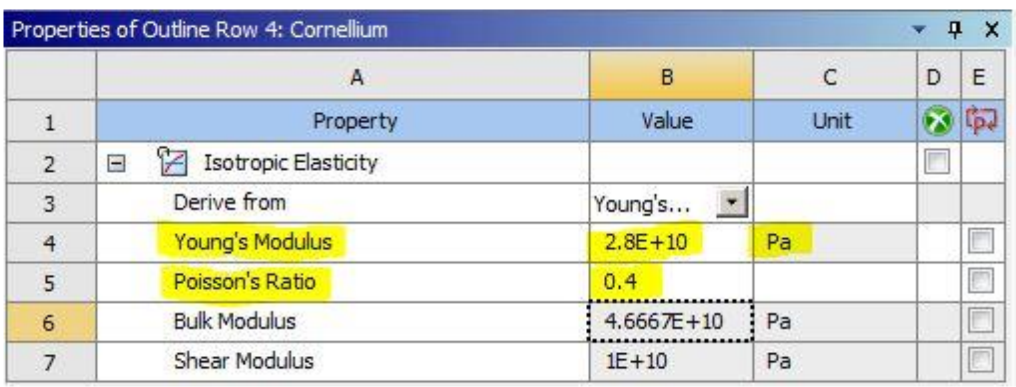
At this point a new window will open. Under *Outline of Schematic A2: Engineering Data* there will be a box with text inside that says "Click here to add a new material". We'll call our material "Cornellian". Click on that box and type in `Cornellian` and then press enter.



Now, expand the ***Linear Elastic*** tab on the left and double click on ***Isotropic Elasticity***.



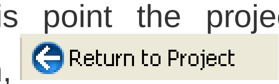
Now, the properties of "Cornellian" need to be entered. Set the Young's Modulus to`2.8e10` Pascals and set Poisson's Ratio to `0.4`. Note that the stiffness matrix for the beam element is independent of the Poisson's Ratio. Hence the solution will not change if a different Poisson's Ratio is used.



Properties of Outline Row 4: Cornellium

	A	B	C	D	E
1	Property	Value	Unit		
2	Isotropic Elasticity				
3	Derive from	Young's...			
4	Young's Modulus	2.8E+10	Pa		
5	Poisson's Ratio	0.4			
6	Bulk Modulus	4.6667E+10	Pa		
7	Shear Modulus	1E+10	Pa		

The material "Cornellian" will be assigned to our model in a later step. Here we have just input its Young's Modulus and Poisson's ratio.

At this point the project can be returned to. Click on the Return to Project button, 

at the top.

Saving

It would be of best interest, to save the project at this point. Click on the "Save As.." button, 

, which is located on the top of the Workbench window. Save the project as "Cantilever" in a suitable folder. When you save in ANSYS a file and a folder will be created. For instance if you save as "Cantilever", a "cantilever.wbpj" file and a folder called "cantilever_files" will appear. In order to reopen the ANSYS files in the future you will need both the ".wbpj" file and the folder. If you do not have BOTH, you will not be able to access your project.

Step 2: Geometry

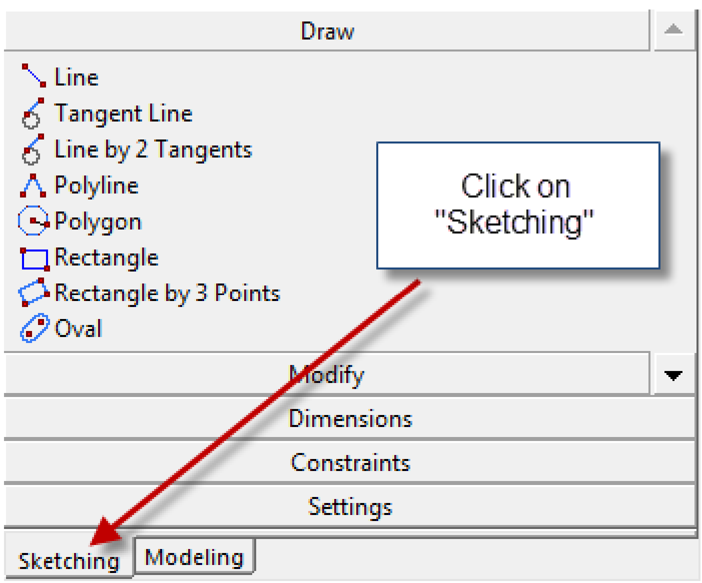
Enter the Preprocessor module

For users of ANSYS 15.0, follow the following procedures for turning on the Auto Constraint feature before creating sketches in DesignModeler.

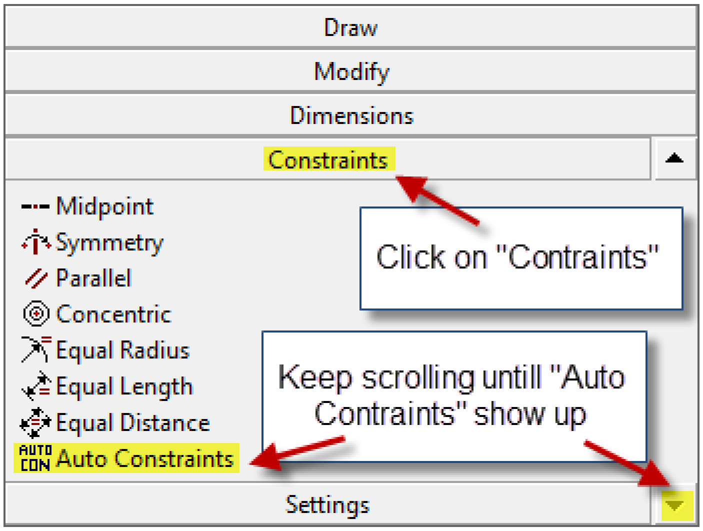
It is very important to check that the Auto Constraints feature is turned on before creating any sketches in DesignModeler. Otherwise, vertices and lines in your sketches will not be coincident with the coordinate axes. This can cause problems in your solution later on. The Auto Constraint feature is not turned on by default in ANSYS 15.0. This tip demonstrates how to turn on the Auto Constraint feature in DesignModeler.

Procedure

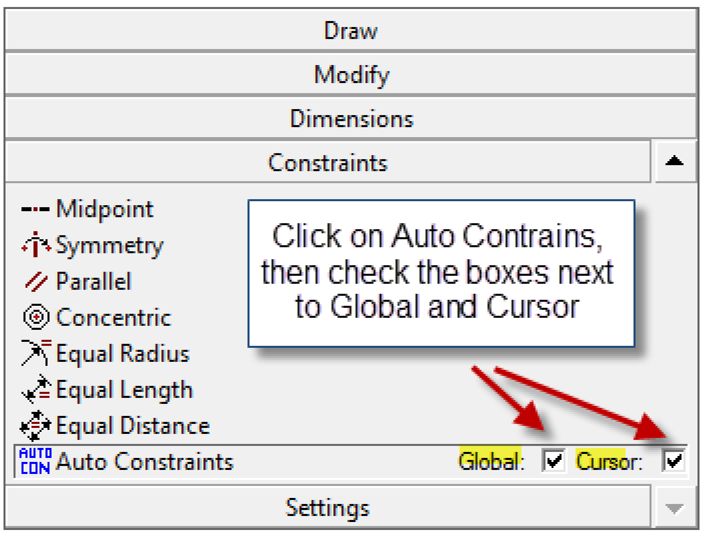
Before creating a sketch, click on the "sketching" tab.



Next, click on Contraints and keep scrolling untill Auto Contraints appear.



Finally, click on Auto Contraints and check the boxes next to Global and Cursor.



Okay, all set! Have fun sketching and modelling!

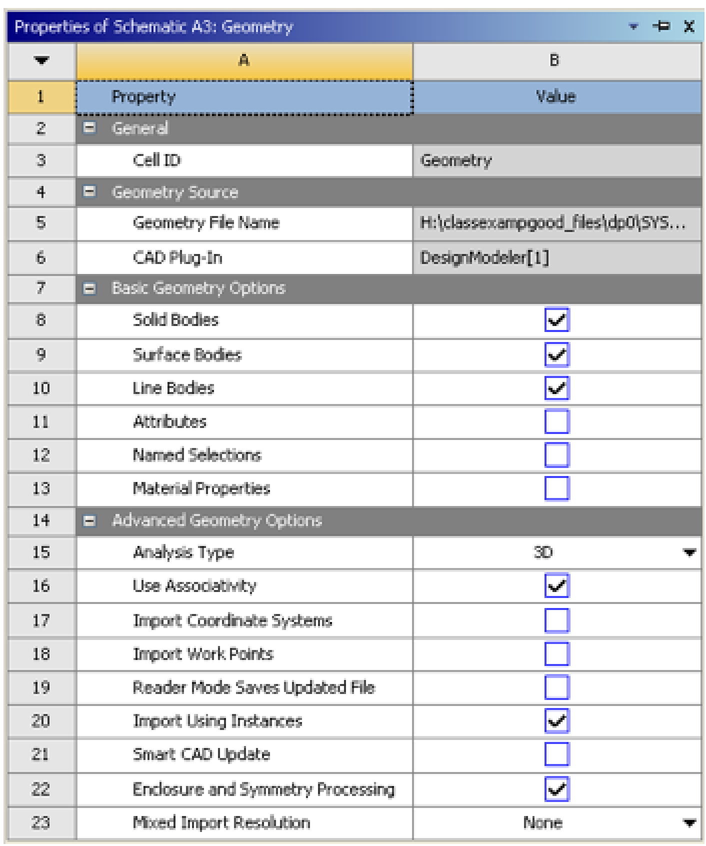
Overview

The process we'll follow is:

1. Sketch a line representing the undeformed neutral axis of the beam.
2. Turn this "line sketch" into a "line body". Only "bodies" can be meshed in ANSYS.
3. Define the beam cross-section and assign it to the "line body". ANSYS will then use the cross-section geometry to calculate the moment of inertia while forming the beam element stiffness matrices.

Initial settings

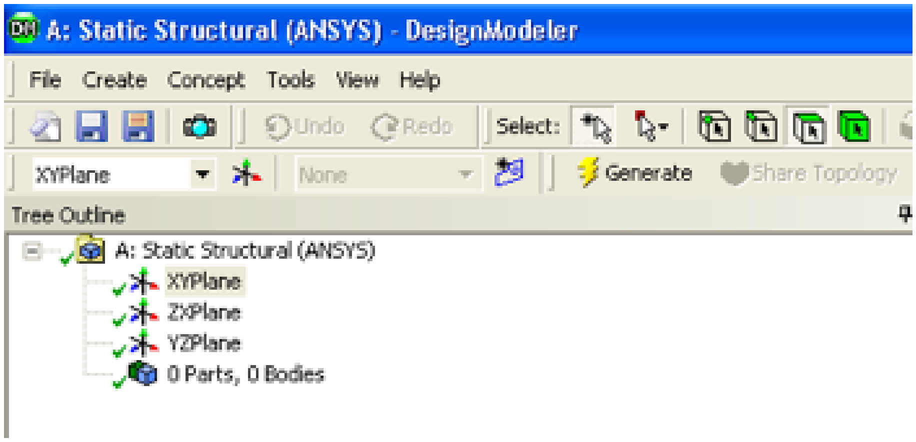
In order to make sure the geometry data gets transferred to the Model a couple of steps must be taken; First, right click on ***Geometry*** then click on ***Properties*** . Under *Properties of Schematic A3: Geometry* expand ***Basic Geometry Options*** and check the box to the right of ***Line Bodies*** as seen below. If you are using a later version such as ANSYS 15.0, you can skip this step.



Double-click on the geometry button, 

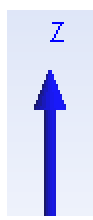
; in the Project Schematic area, which should launch the Design Modeler in ANSYS. A window should pop up asking for units. Units are in meters, so select ***Meters*** and press ***Ok*** . A folder called ***A: Static Structural***

(ANSYS) should be expanded in the tree outline of the Design Modeler; If it is not expanded, then expand it now.

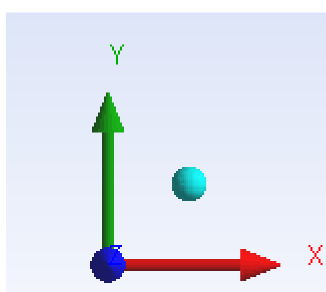


Proper Orientation

Click once on the XYPlane button, XYPlane ; Next, click once on the royal blue Z vector (displayed below) which should be in the bottom right section of the Design Modeler window.



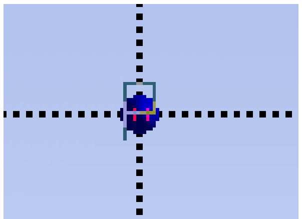
Now, you should be looking directly at the XY plane and the axes in the bottom right corner should be oriented as they are in the image below.



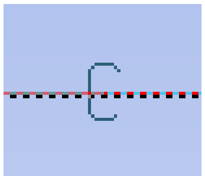
Line Sketching

First instinct is to make a rectangular solid as a model for our cantilever. This would create 3D elements which would be one way of modeling the beam. Here we will use a different modeling approach using 1D beam elements. In effect, we are only modeling the neutral axis of the beam and calculating its deformation directly. All other results such as bending stress and bending moment are derived from the deformation of the neutral axis.

Let's create a line corresponding to the undeformed neutral axis. Click once on the***Sketching*** tab, , which appears at the bottom of the Tree Outline. Click once on the ***Line*** button, , in the ***Draw*** tab, , that automatically appears. Then place the mouse cursor directly over the origin of the XY plane until a P appears (the *P* indicates that the cursor is co-incident with the Point at the origin). If you don't see the *P*, you need to turn on the Auto Constraint feature [as shown here](). **This step is necessary in version 15.0 as noted at the top of this page.** In other versions, the Auto Constraint feature is turned on by default.

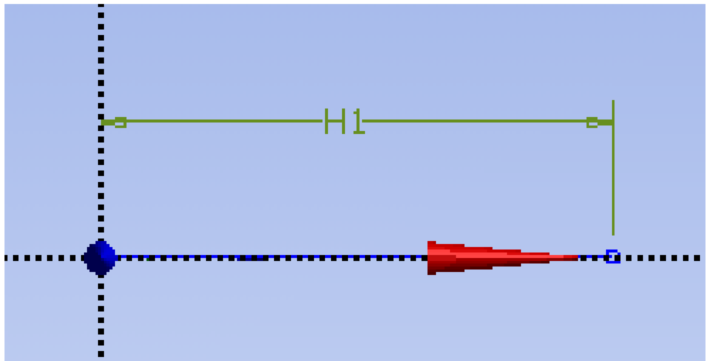


Once the P appears then click once on the mouse. Next, move the mouse over to the right so it lies somewhere on the positive x axis; Prior, to clicking again make sure that a C appears (the *C* indicates that the cursor is Co-incident with the horizontal axis).

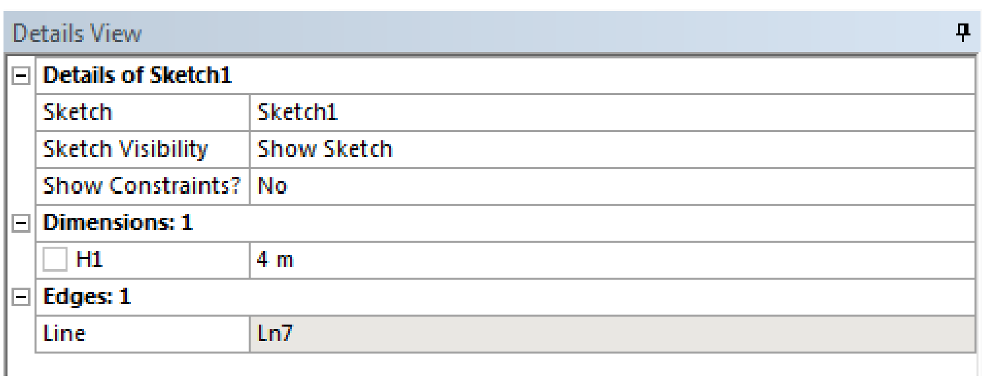


You should now have a line that starts at the origin and terminates somewhere on the positive axis.

At this point, the dimension of the line needs to be specified, so click once on the ***Dimensions*** tab, . Click on the line and place the dimension as shown below. You should see a dimension labeled H1 above the horizontal line as shown below. Note that there is an *Undo* button in the sketching mode that you can use if you make a mistake.

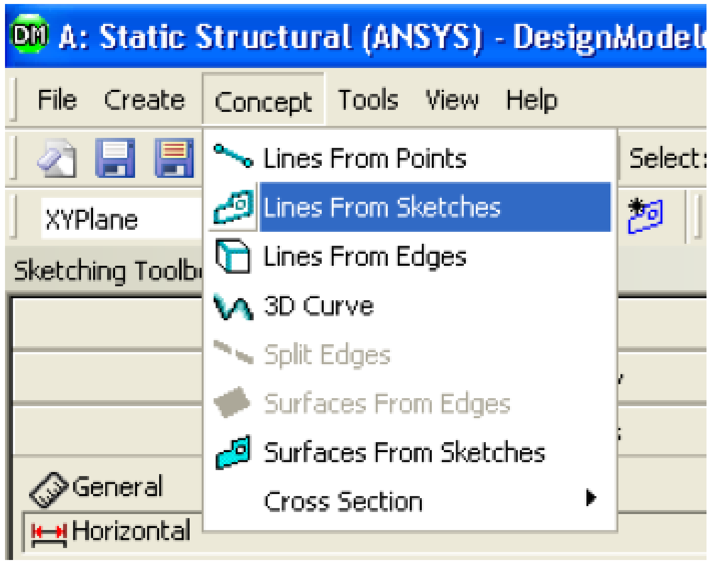


Now, the length of the line will be manually edited. Underneath the *Sketching Toolboxes* there will be a column called *Details View*. In *Details View* there is a subcategory called ***Dimensions: 1*** . Change the numerical value of H1 to 4 meters and press enter.

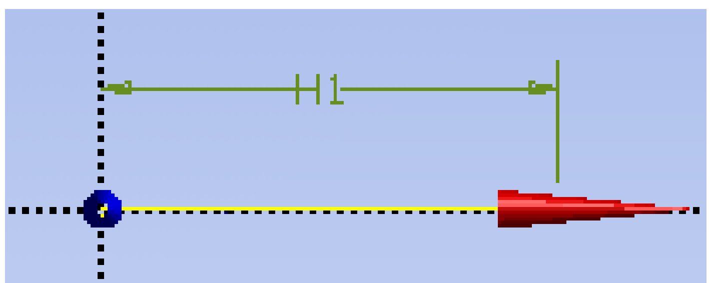


Line Body

The next step is to turn our "line sketch" into a "line body". In ANSYS, only "bodies" can be meshed. In order to do this click on **Concept** which will be on top of the Design Modeler window, then click on **Lines from Sketches**, as can be seen in the following picture.



Next, click on the blue horizontal line that you drew. The blue horizontal line should have changed from blue to yellow as can be seen below.

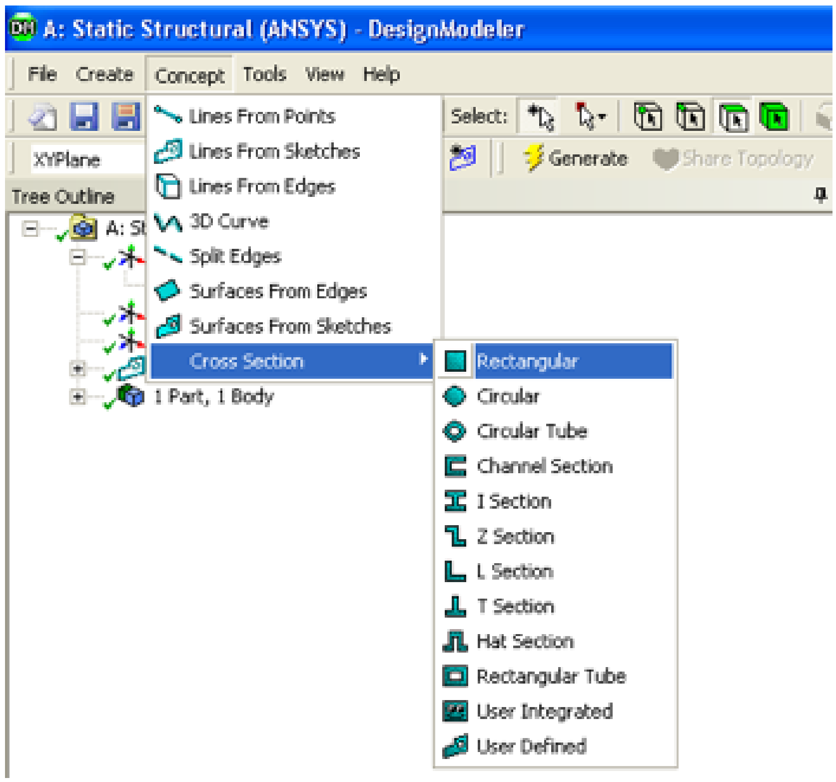


In the *Details View* column a yellow box to the right of ***Base Objects*** should be highlighted in yellow. Click on the yellow box and then click ***apply*** . Then, click on the***Generate*** button 

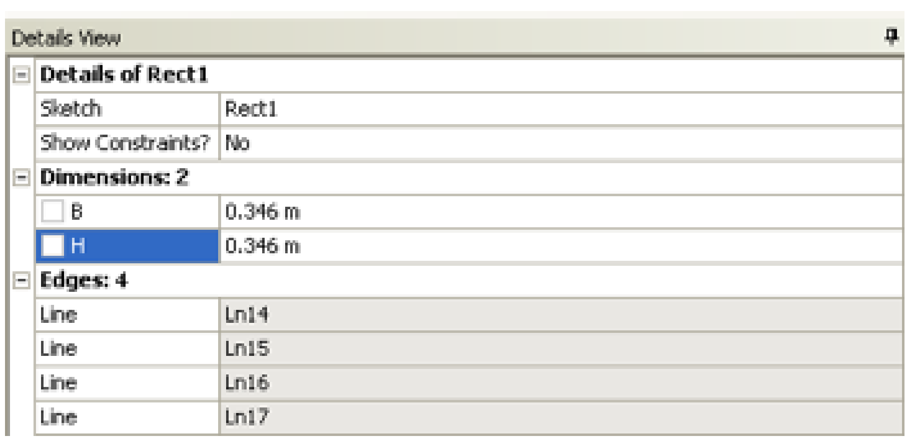
; it is located on the top left portion of the Design Modeler.

Cross Section

Now, the beam cross section will be defined. First go to **Concept** then click on **Cross Section** then finally click on **Rectangular**, as shown below.

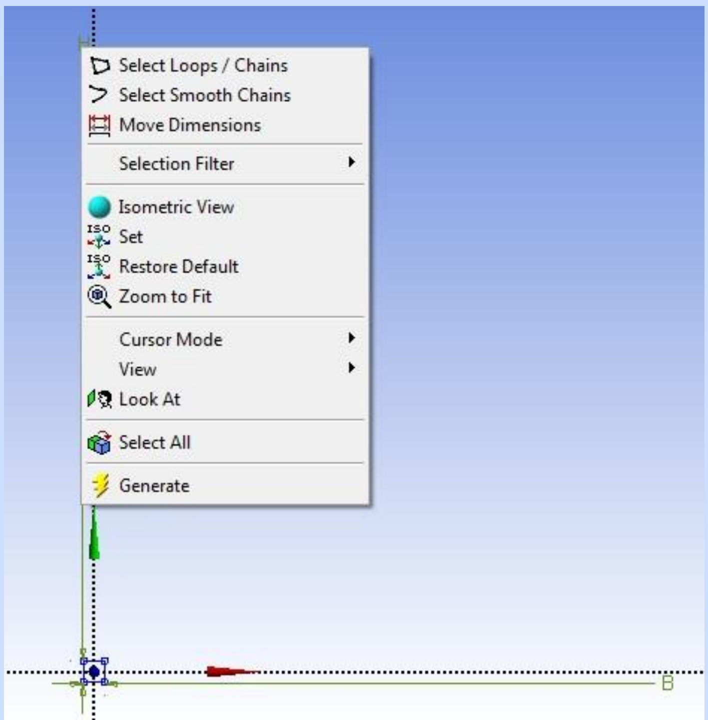


Now, the width and height of the cross section need to be defined; Under "Details View" set B to `0.346` meters and set H to `0.346 meters`, as can be seen below:



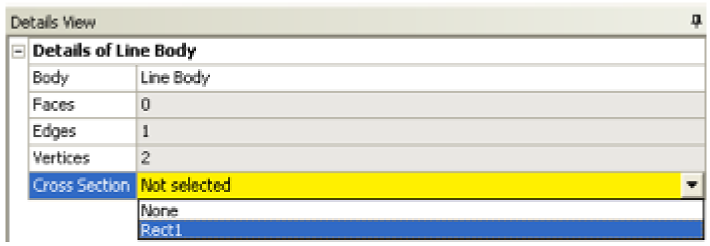
Then click on the Generate button, .

move dimensions
This is an optional step that will only change the way the cross-section is displayed, so you can choose to skip it. You can right click on the dimension and select ***Move Dimensions*** and move the dimensions closer to the cross section. The cross section will be easier to see if you click on the zoom to fit tool .



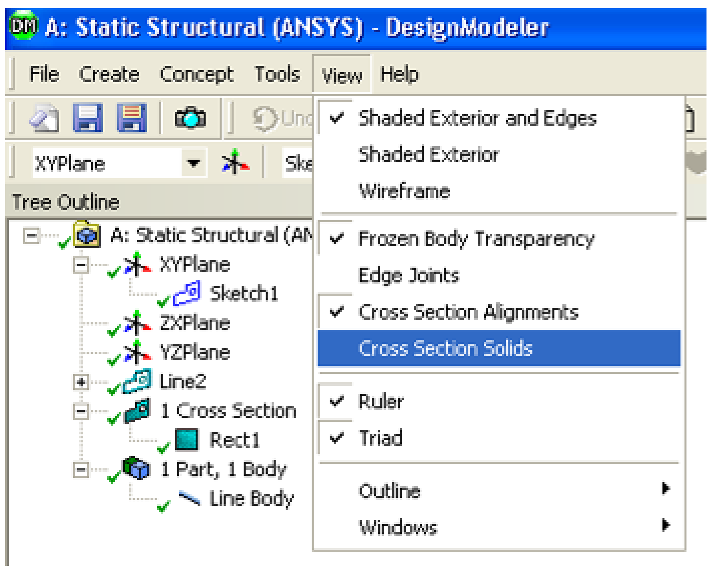
Assign the Cross Section to the Line Body

Now the defined cross section will be assigned to the line body. ANSYS will then use this cross-section to calculate the moment of inertia while forming the element stiffness matrices. First, expand "1 Part, 1 Body" which is located in the *Tree Outline*. Next, click on ***Line Body*** , and there should be a yellow box to the right of Cross Section under the *Details of Line Body*. Click on the yellow box and select ***rect1*** as seen below.



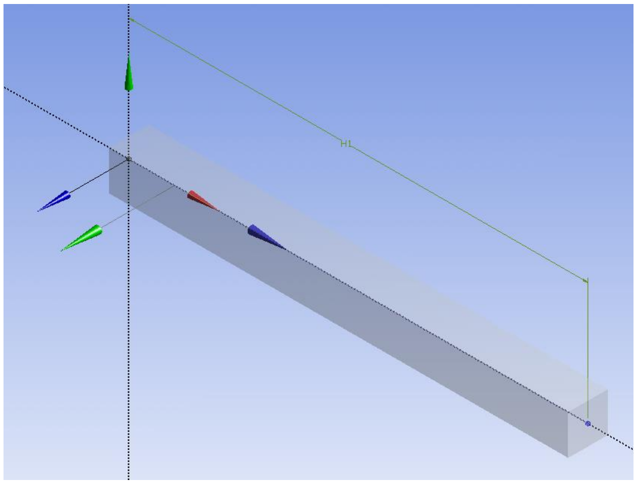
Verify Geometry

We can visualize the beam in 3D by getting ANSYS to wrap the cross-section around the line in the display. Click on **View > Cross Section Solids**, as shown below;



If you click on the ***1 Cross Section*** , 

, in the *Tree Outline* and then click on the light blue dot, , you should now see a three dimensionally rendered beam in an isometric view. Note that this is merely a visualization; our beam model is only a line.

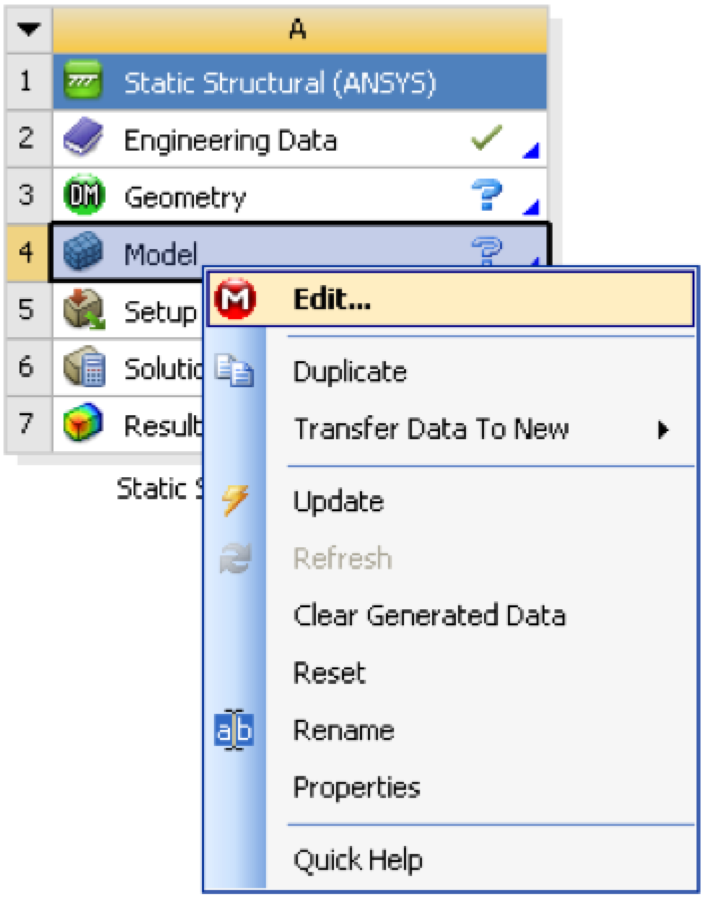


At this point, the Design Modeler window can be closed. Then, click on *Save*.

Step 3: Mesh

Open the Model

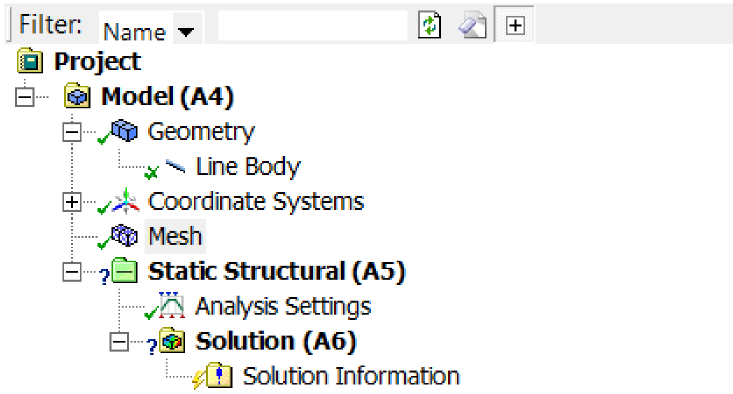
Right click on model button,  , in the Workbench window then click on ***Edit...*** as shown below.

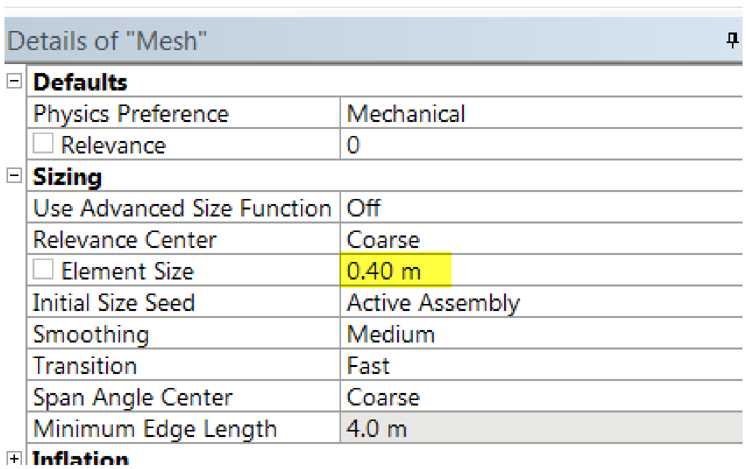


Expand "Model (A4)",  , if it is not already expanded.

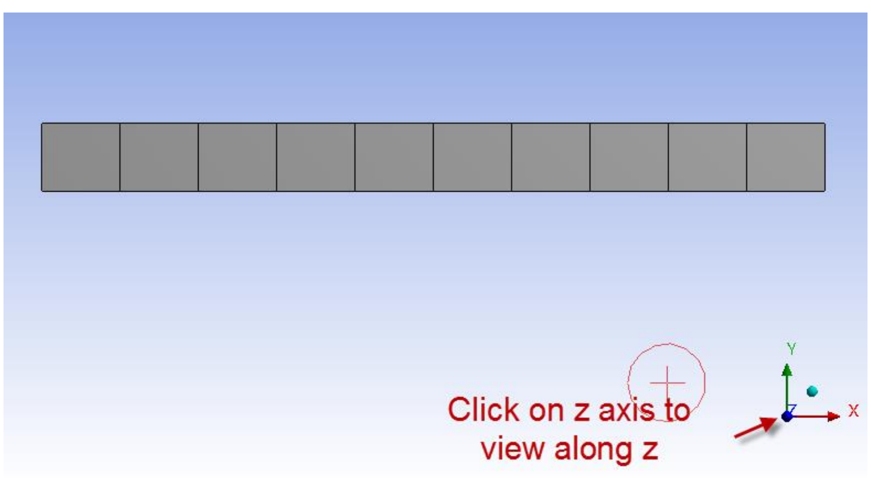
Specify the Element Size

Begin by clicking once on mesh,  . Next, expand sizing under *Details of "Mesh"* if it is not already expanded. To create ten elements along the beam, set the element size to 4m/10 = 0.4m. Then, click on ***Update***,  .





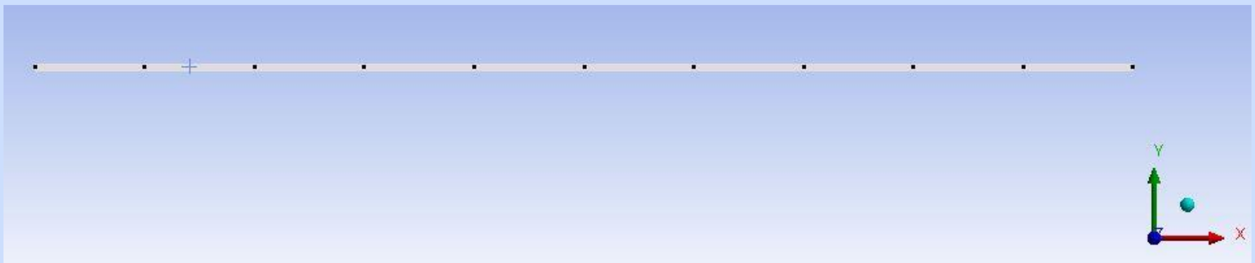
At this point you should see a similar image to the one below. The mesh is composed of ten elements.



The mesh has now been set.

Note

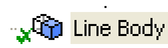
In the above mesh view, ANSYS is wrapping the cross-section around the line elements. To view the line elements and the corresponding nodes, click on ***View*** and uncheck ***Thick Shells and Beams***. ANSYS will calculate the displacements and slopes at the nodes shown in this view.



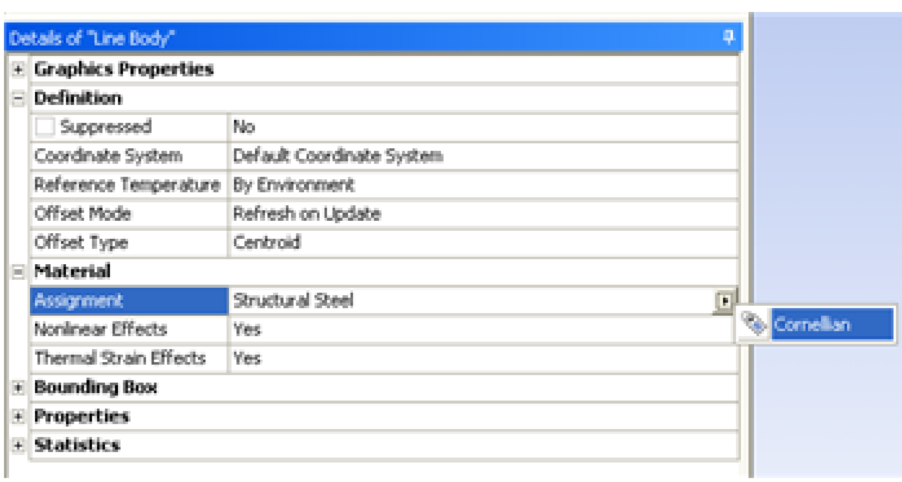
Step 4: Physics Setup

Assign Material Properties

The material Cornellian that was created earlier needs to be applied to the beam. In order to do so, expand ***Geometry***, 

. Next, click once on ***Line Body***, 

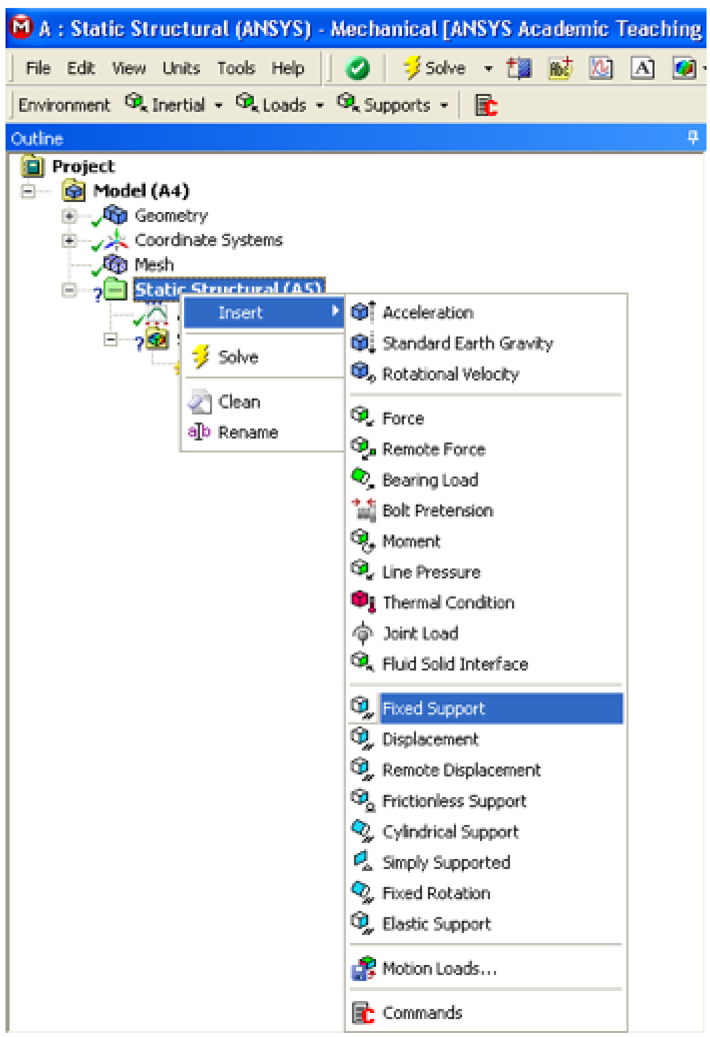
, which will appear underneath Geometry. Then expand ***Material*** which is located under *Details of Line Body*. Then click on the arrow on the far right and change the specified material to Cornellian as shown below. Now ANSYS will use the correct Young's Modulus while forming the element stiffness matrices.



Fix The Left Side of the Beam

First, click on the box Static Structural (A5) in the *Outline Window*. Next, click on the vertex pointer option, , which is on the top of the *Setup* window. Next, click on the left edge of the line body. A green box should appear on the left end of the line body as seen below.

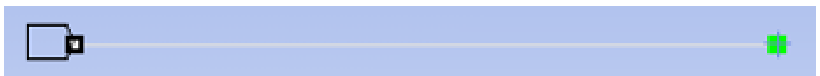
Now, right click on the Static Structural folder, then click ***Insert*** and then select ***Fixed Support*** as shown in the image below.



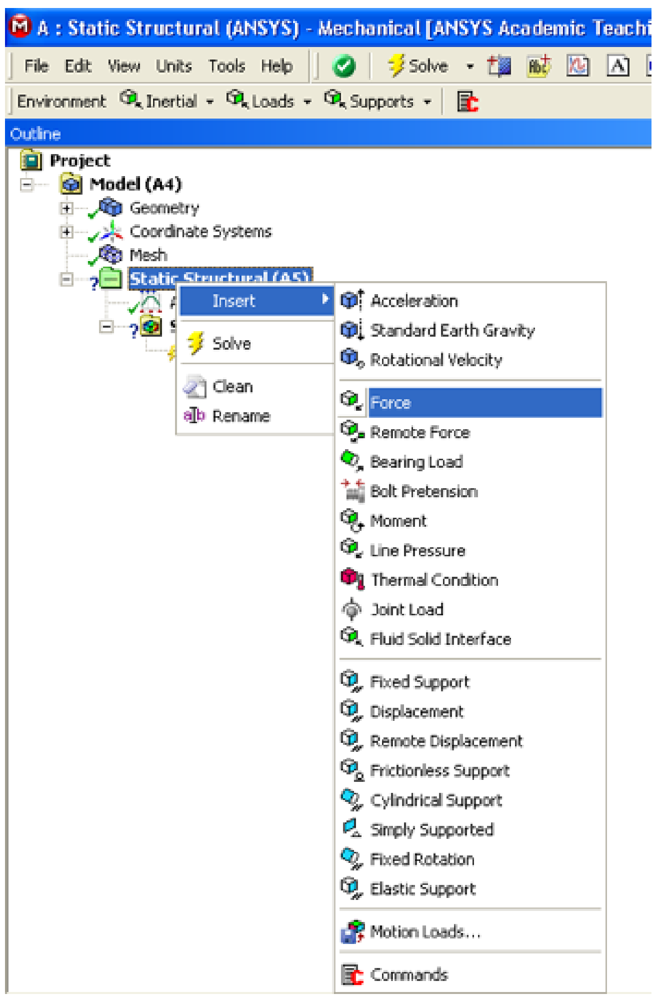
This will set the *x* and *y* displacements as well as the slope to zero for the node at the left end of the neutral axis.

Apply a Point Force to The Right Side of The Beam

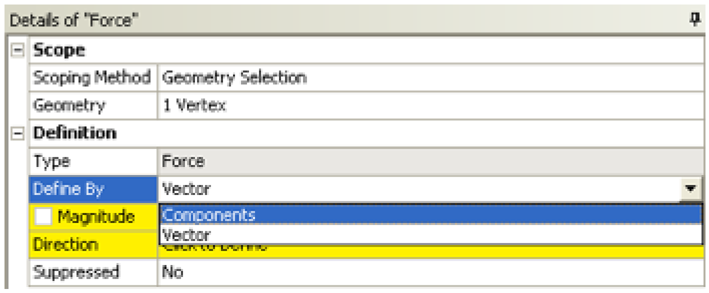
Once again, click on the vertex pointer option, , but now click the right edge of the line body. A green box should now appear on the right side of the line body as seen below.



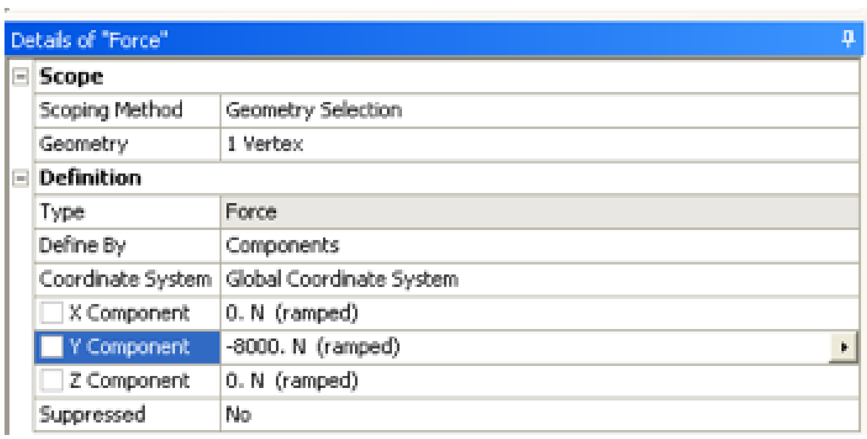
Right click on the Static Structural folder again, then click ***Insert*** and this time select ***Force*** as shown below.



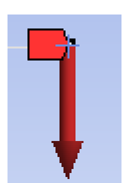
At this point, there should be a *Details of "Force"* window in the lower left corner of the Setup window. Expand ***Definition*** if it is not already expanded and then change***Define by*** to ***Components*** as seen below.



Now, click on the box to the right of Y Component without clicking the Y component button and change the force to `-8000N`. That is, you should NOT check the box to the left of Y Component. Your *Details of "Force"* window should now look very similar to the following image.



You should also see the following downward facing red arrow on the right side of the line body.

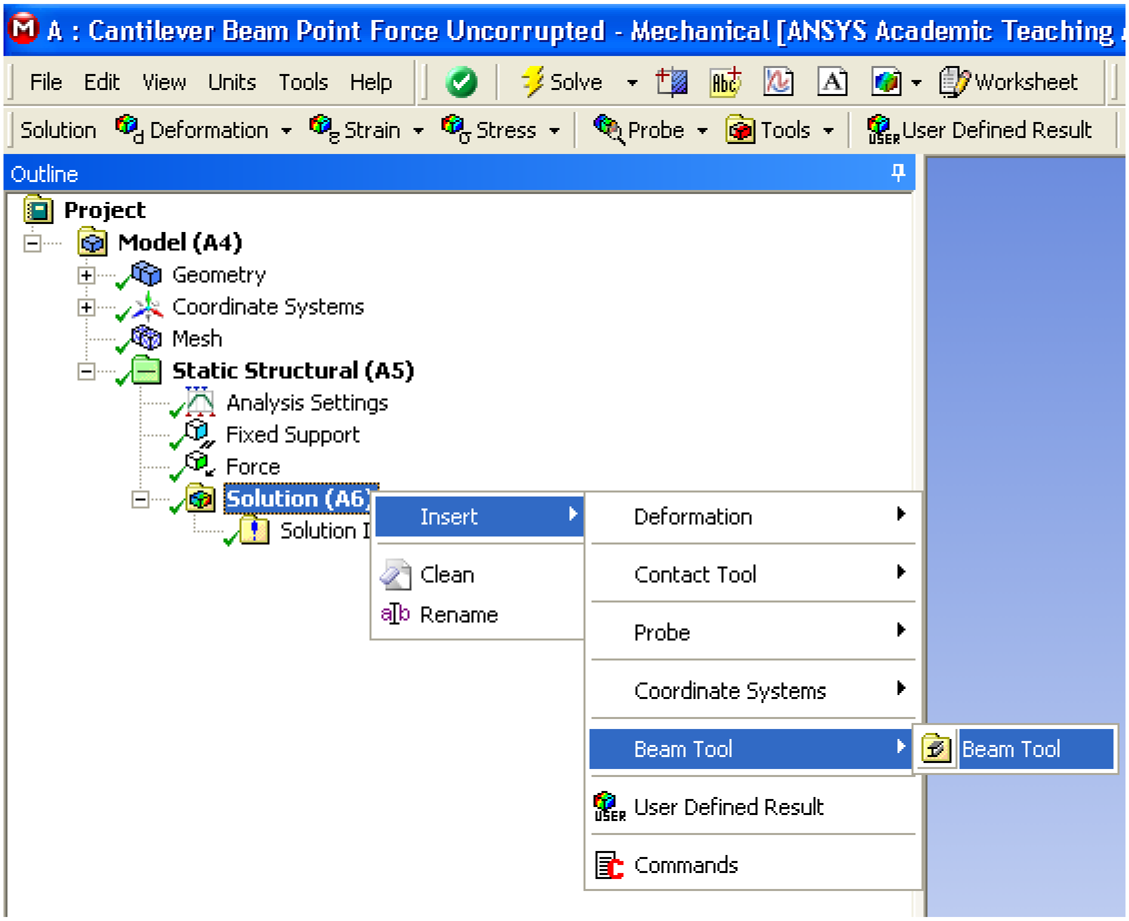


The fixed end and the point force have now been applied. Leave the Setup window open for the next step. Save the project.

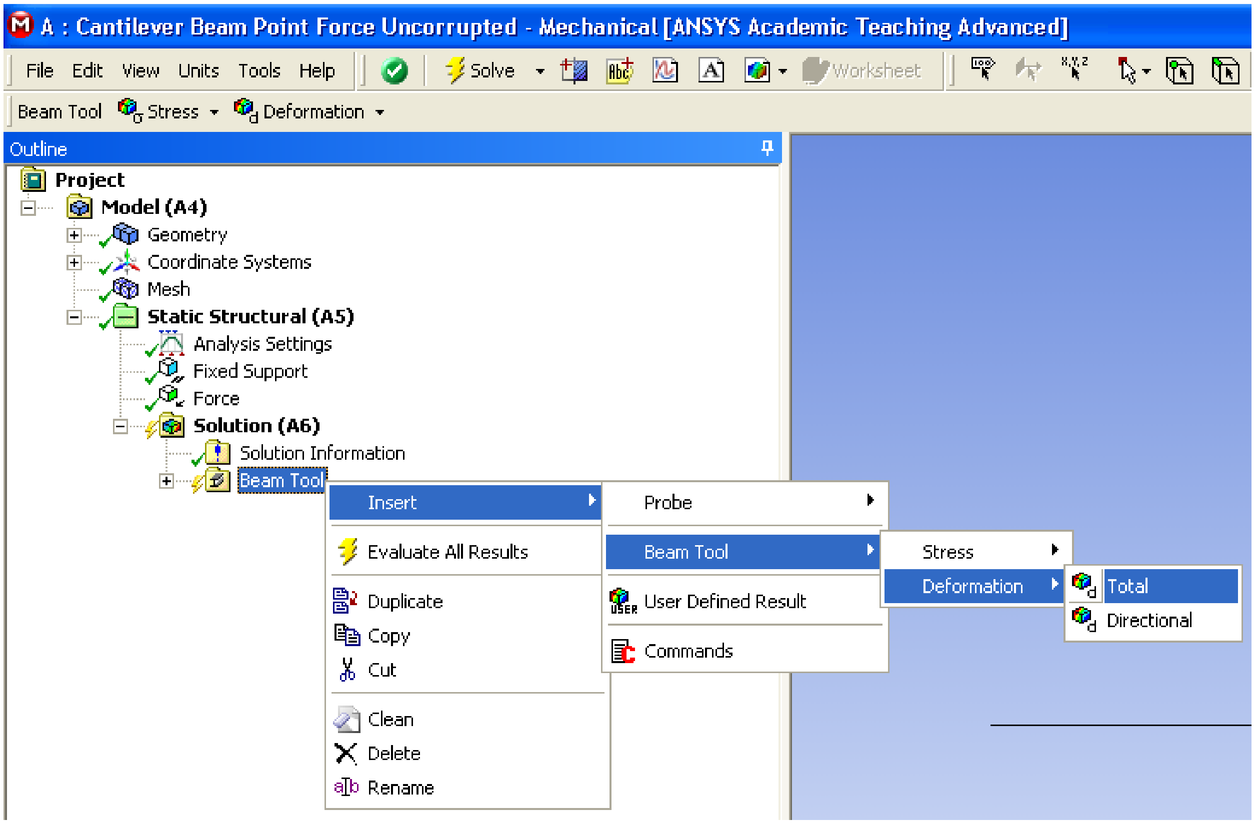
Step 5: Numerical Solution

Choosing Results

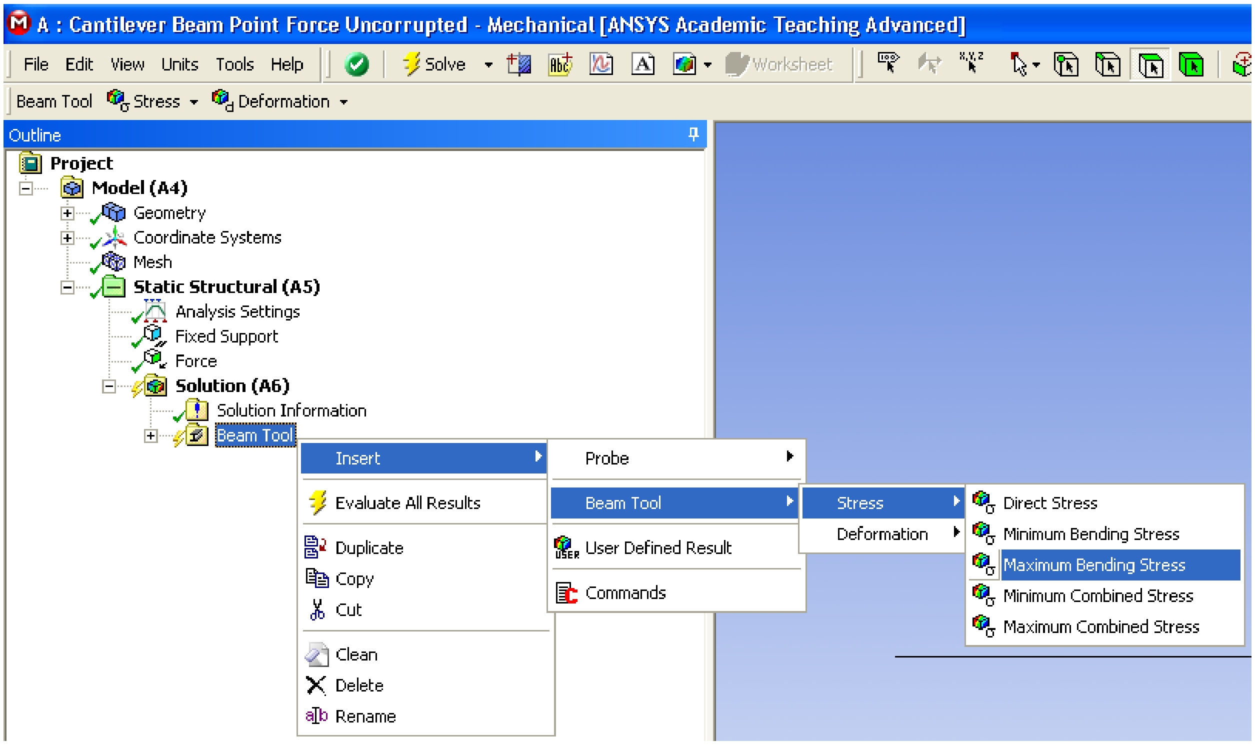
We next specify what results we'd like to look at. Note that these results can also be specified after we solve the model. First, click on the solution button, Solution, in the workbench window. Next, right click on the ***Solution (A6)*** folder, then click insert, then click ***Beam Tool*** and finally click ***Beam Tool*** as shown in the image below.



Then, right click on the ***Beam Tool*** folder that you have just added, then click on ***Insert***, then click on ***Beam Tool > Deformation > Total*** as displayed below.

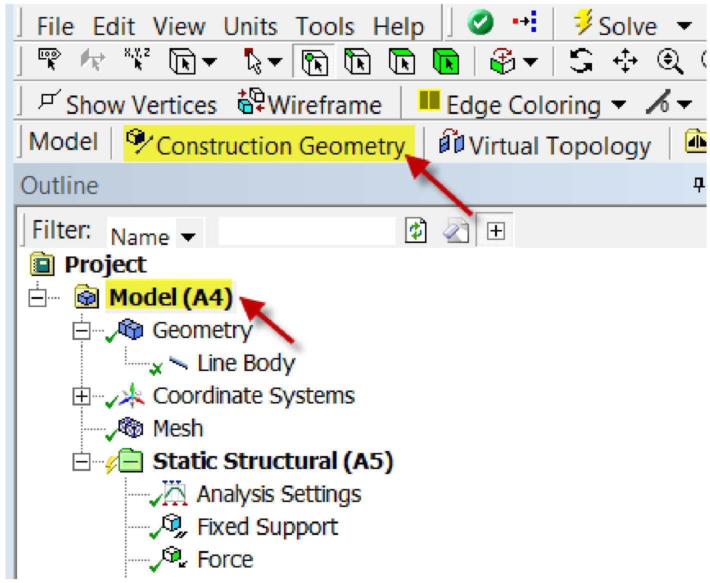


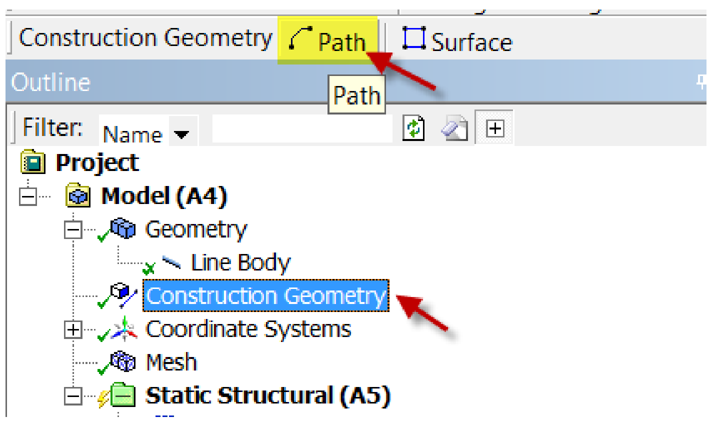
Next, right click on the ***Beam Tool*** folder, then click on ***Insert***, then click on ***Beam Tool > Stress > Maximum Bending Stress*** as shown below.



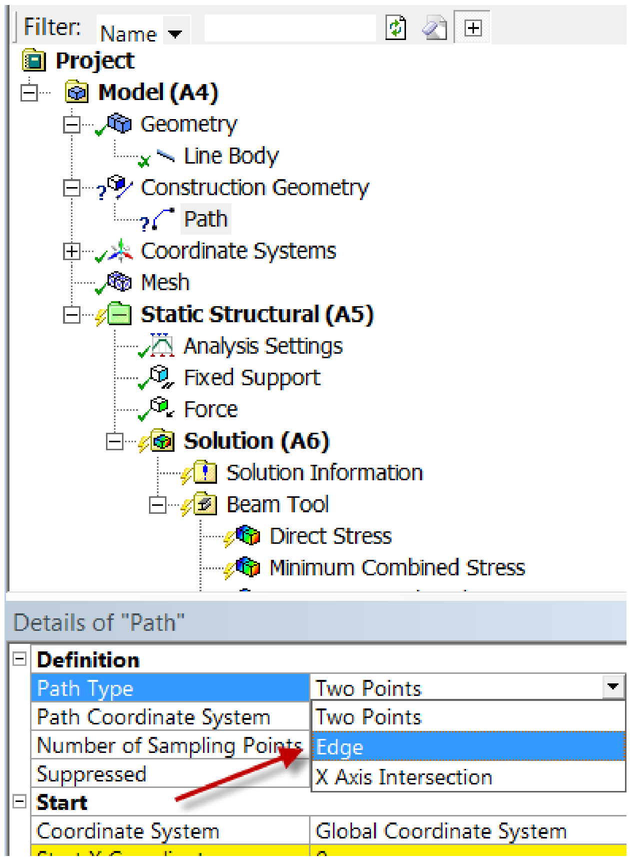
Bending Moment along the Beam

Now, we will set up a result object for the bending moment along the beam. We will do this by setting up a "path" along the line body. To set up a path, click on **Model (A4)** in the *Outline* window. This will launch the Model toolbar in the Menu Bar. In the Model toolbar, press Construction Geometry which will bring up the Construction Geometry Tool bar, then press Path to create a path.

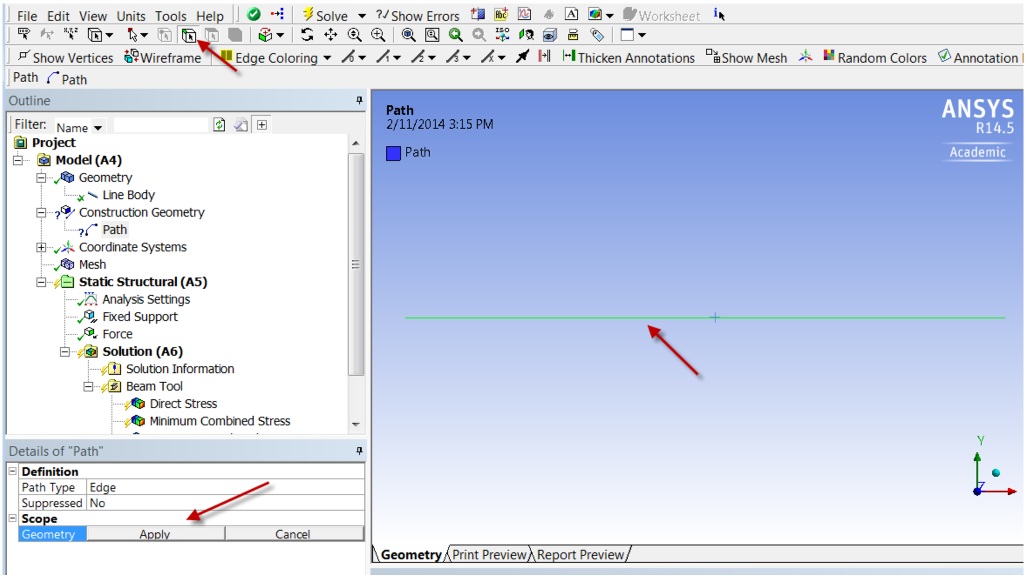

File Edit View Units Tools Help
Solve
Show Vertices
Wireframe
Edge Coloring
Model
Construction Geometry
Virtual Topology
Outline
Filter: Name
Project
Model (A4)
Geometry
Line Body
Coordinate Systems
Mesh
Static Structural (A5)
Analysis Settings
Fixed Support
Force


Construction Geometry
Path
Surface
Outline
Path
Filter: Name
Project
Model (A4)
Geometry
Line Body
Construction Geometry
Coordinate Systems
Mesh
Static Structural (A5)

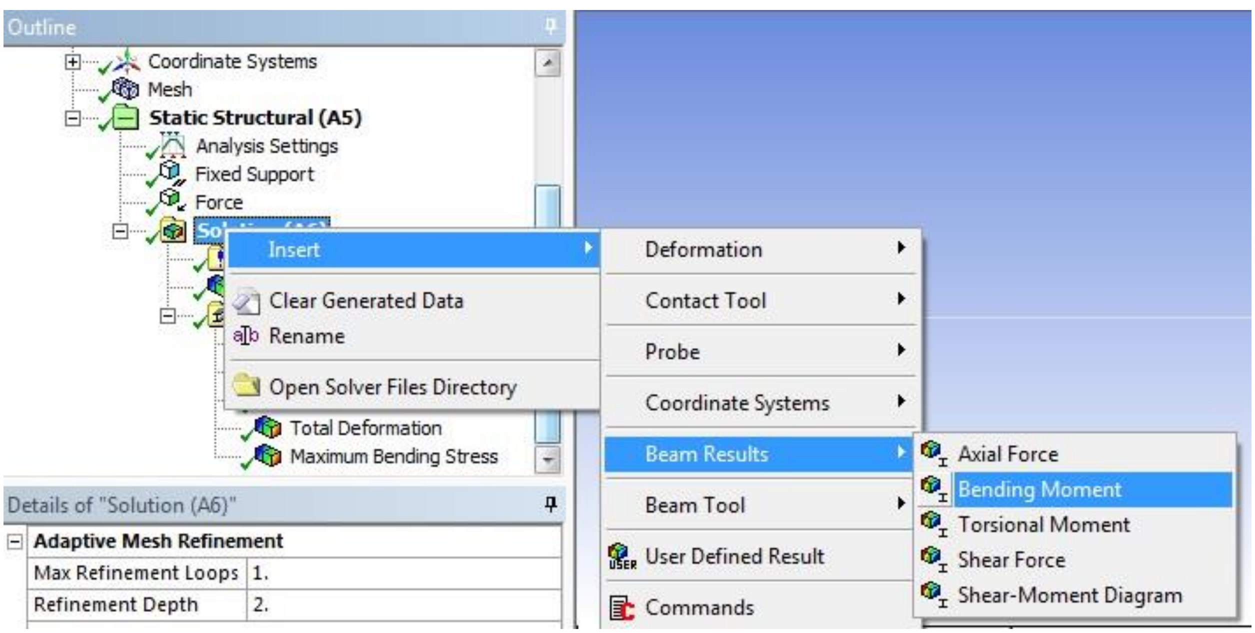
In the *Details* window, notice that the default path type is ***Two Points***. We need to change that to ***Edge***.



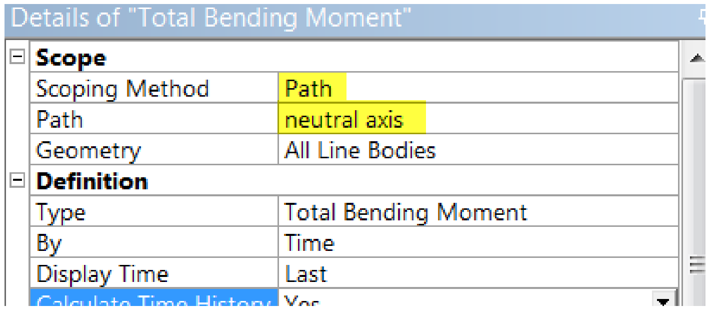
Next, using the ***Edge Selection filter***  select the line body in the *Graphics*window. Back in the *Details of "Path"* window, select ***Geometry > Apply***. Rename it "neutral axis".



Now that we have created the path, we need to create a solution object that gives the bending moment along the path. Click on Solution in the *Outline* window to bring up the solution menu, then select **Beam Results > Bending Moment**.



In the *Details of "Total Bending Moment"* window, change ***Scoping Method*** to ***Path***. Next, define the ***Path*** parameter to ***neutral axis*** (the path we created).

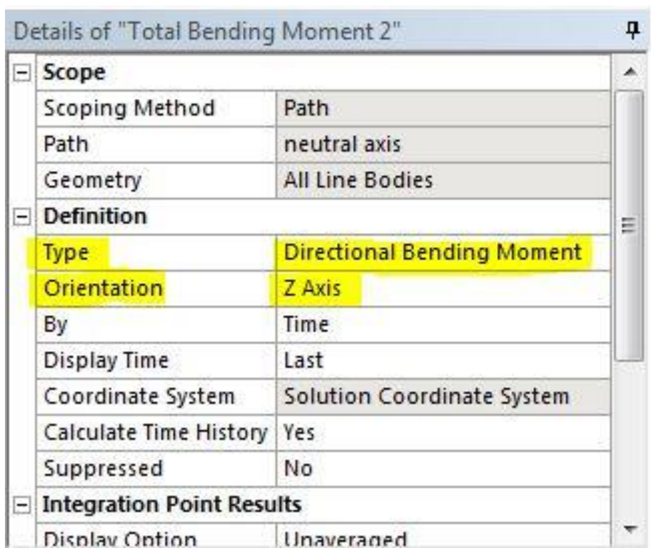


Details of "Total Bending Moment"

Scope	
Scoping Method	Path
Path	neutral axis
Geometry	All Line Bodies
Definition	
Type	Total Bending Moment
By	Time
Display Time	Last
Calculate Time History	Yes

Directional Bending Moment

We would also like to look at the bending moment in a specific direction. Repeat the above steps to setup another bending moment results object. In the details window of the new bending moment, change the type to ***Directional Bending Moment***instead of the default ***Total Bending Moment***. Change the orientation to ***Z axis***.



Details of "Total Bending Moment 2"

Scope	
Scoping Method	Path
Path	neutral axis
Geometry	All Line Bodies
Definition	
Type	Directional Bending Moment
Orientation	Z Axis
By	Time
Display Time	Last
Coordinate System	Solution Coordinate System
Calculate Time History	Yes
Suppressed	No
Integration Point Results	
Display Option	Unaveraged

Solve

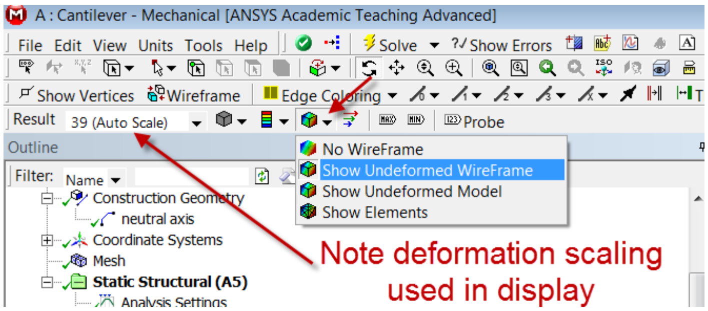
In order to solve, click on the solve button, 

, which is located near the top of the Setup window. ANSYS will obtain the numerical solution where the ANSYS solver will form the stiffness matrix for each beam element, assemble them into the global stiffness matrix and invert it to get the nodal displacements and slopes. It will then extract the requested results and populate the results objects in the tree.

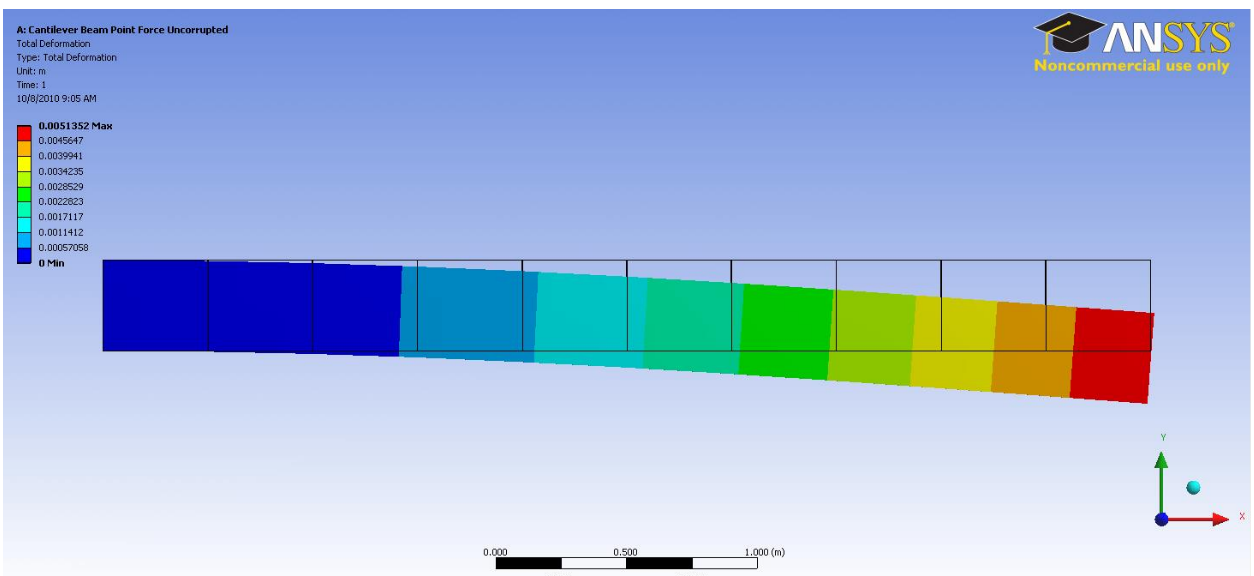
Step 6: Numerical Results

Total Deformation

First, examine the total deformation by clicking on the Total Deformation object Total Deformation in the tree. Turn on the *Undeformed Wireframe* as shown below.



With 10 line elements, you should see the following output for the total deformation.



If you turn off *View > Thick Shells and Beams*, you will see the deformation of the line elements. The 3D beam view is constructed from this. The maximum deformation is 0.0051 m which matches the hand-calculation value from the Pre-Analysis.

When ANSYS displays the beam deformation, it just connects the displacements at nodes by straight lines. The display ignores the fact that we also have the slope at the nodes. So you'll see an unphysical-looking kinked line in the deformation display. This is a shortcoming of the *display*, not of the underlying beam element formulation. You'll see the displayed deformed shape getting smoother as you refine the mesh.

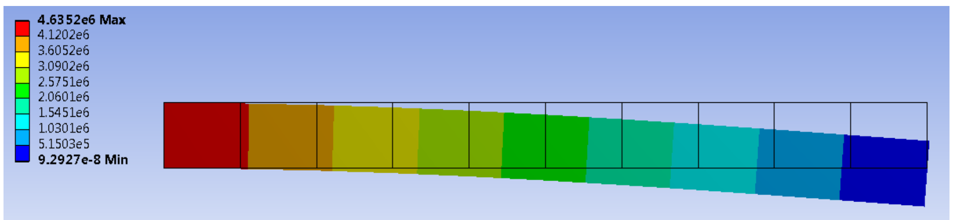
The beam deformation can be animated by clicking on the play button, , which is located underneath the beam deformation results. This will interpolate between the initial undeformed and final deformed configurations.

Maximum Bending Stress

In order to examine the maximum bending stress first expand the Beam Tool folder, 

, which is located under "Solution(A6)". Next, click on the Maximum Bending Stress button, 

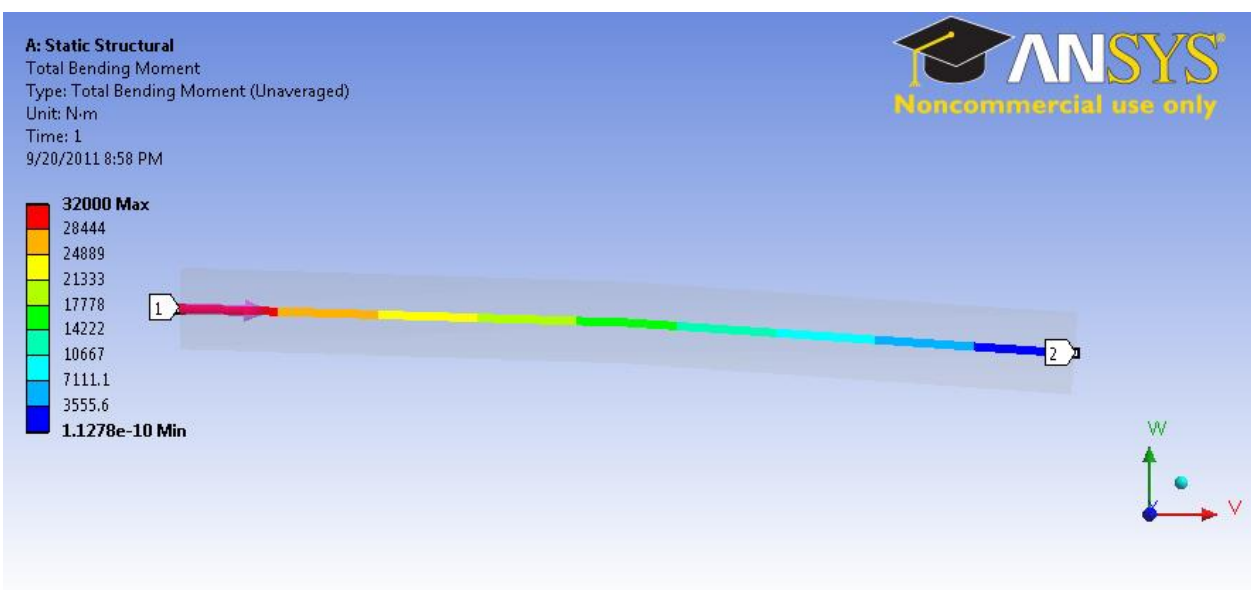
.



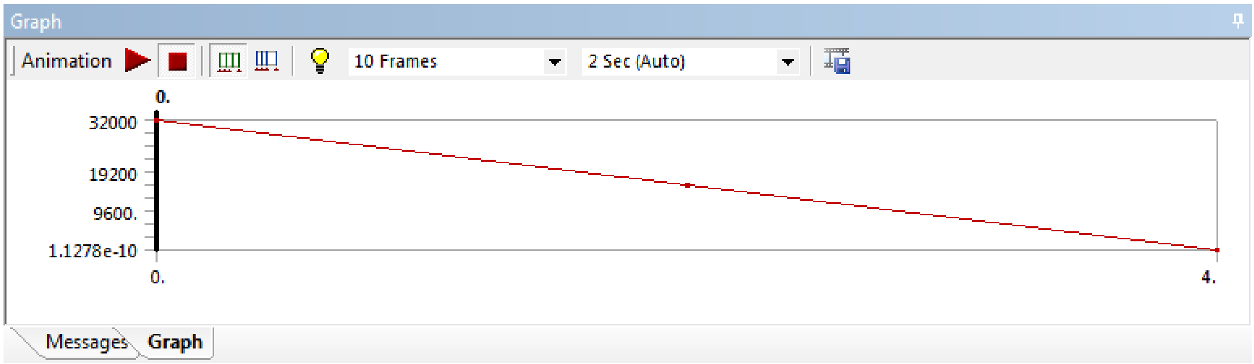
Note that in this display, ANSYS shows the same value across the cross-section. This visualization is misleading. The maximum bending stress occurs only at the top fiber. The value that ANSYS reports is 4.635 MPa which matches the value from the Pre-Analysis exactly.

Bending Moment

To view the bending moment along the beam, click ***Total Bending Moment*** in the *Outline* window. You should see the following in the graphics window.



Also notice that the values were plotted in a graph in the *Graph* window and also displayed in a table. The values can be exported into a Excel or text file by right-clicking on the table.

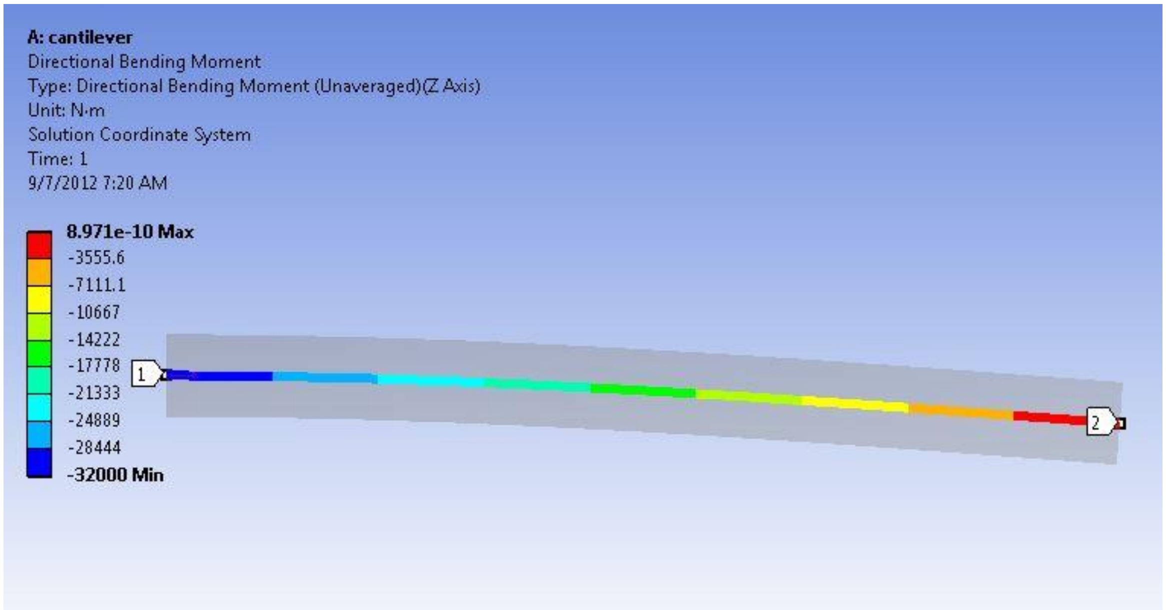


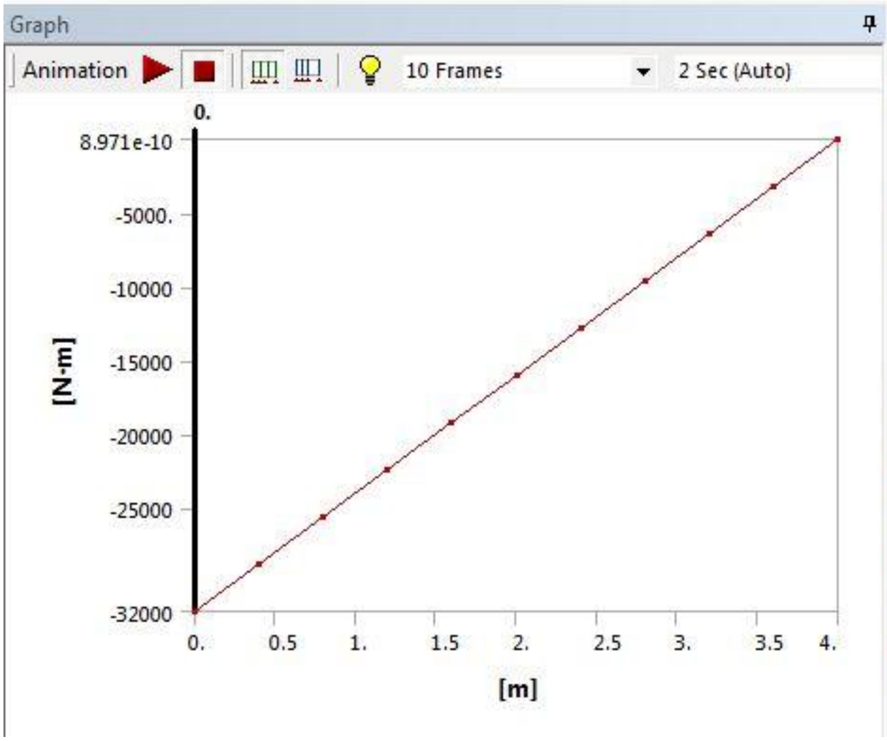
In the above, pay close attention to maximum and minimum values of the bending moment. At the left end, the bending moment is 32000 Nm; the calculation for moment is

So this checks out. We also notice that the minimum moment 1.1278E-10 Nm. Because this value is over 1E-14 smaller that the largest value, it can be assumed to be zero to machine precision.

Directional Bending Moment

To view the directional bending moment along the beam, click ***Directional Bending Moment*** in the *Outline* window. You should see the following in the graphics window. The Directional Bending Moment gives us the sign along with the magnitude.





Tabular Data

	Length [m]	Value [N·m]
1	0.	-32000
2	0.4	-28800
3	0.4	-28800
4	0.8	-25600
5	0.8	-25600
6	1.2	-22400
7	1.2	-22400
8	1.6	-19200
9	1.6	-19200
10	2.	-16000
11	2.	-16000
12	2.4	-12800
13	2.4	-12800
14	2.8	-9600.
15	2.8	-9600.
16	3.2	-6400.
17	3.2	-6400.
18	3.6	-3200.
19	3.6	-3200.
20	4.	8.971e-010

Step 7: Verification and Validation

Verification of Maximum Bending Stress and Maximum Total Deformation

We have already noted that the ANSYS results compare well to our hand calculations obtained using Euler-Bernoulli beam theory in the pre-analysis. The ANSYS simulation gave 4.6352MPa for the maximum bending stress and the calculation in the pre-analysis yielded 4.635 MPa. The ANSYS simulation gave 0.005135m for the total deformation of the beam at x=4 while the calculation from the pre-analysis yielded 0.005103m. The ANSYS results closely match the hand calculations from the pre-analysis. This is one way to verify the solution.

Mesh Refinement

Another way to verify the solution of a numerical method is to examine the convergence of the solution as the mesh is refined. Generally, the numerical solution should converge to the exact solution as the mesh is refined. In order to refine the mesh, first click on the ***Mesh*** tab, Mesh, in the tree outline. Next, expand ***Sizing*** in *Details of Mesh*. The mesh will be refined by adjusting the ***Element Size***. The length of the beam is 4 m so if you want **n** elements then you will need to set the "Element Size" to (4m/**n**). For instance, if you wanted **20** elements the "Element Size" should be set to (4m/20)=0.2m . After you have changed the ***Element Size*** to your preference, click on the ***Solve*** button, 

, to recalculate the solution with the new mesh.

The table below displays the outputs of the ANSYS simulation for a mesh of 2 elements and a mesh of 10 elements.

	Total Deformation (m)	Maximum Bending Stress Pa
Theory Values	0.005103	4.635x10^6
2 Element FEA	0.0051352	4.6352x10^6
10 Element FEA	0.0051352	4.6352x10^6

As one can see from the table above the results do not change as the mesh is refined. The reason that the results do not change is as follows: the exact solution for cantilever beam deformation is cubic and for this setup ANSYS uses element BEAM 188 which also uses cubic interpolation. Thus, for the simple cantilever beam setup the numerical method converges very quickly.