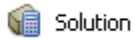
Cantilever Beam - Numerical Solution (OLD)

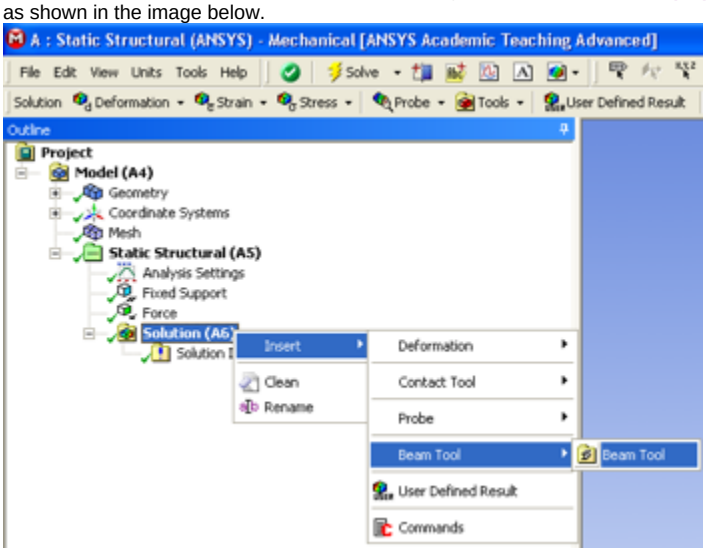
Author: John Singleton, Cornell University

Numerical Solution

Choosing Results

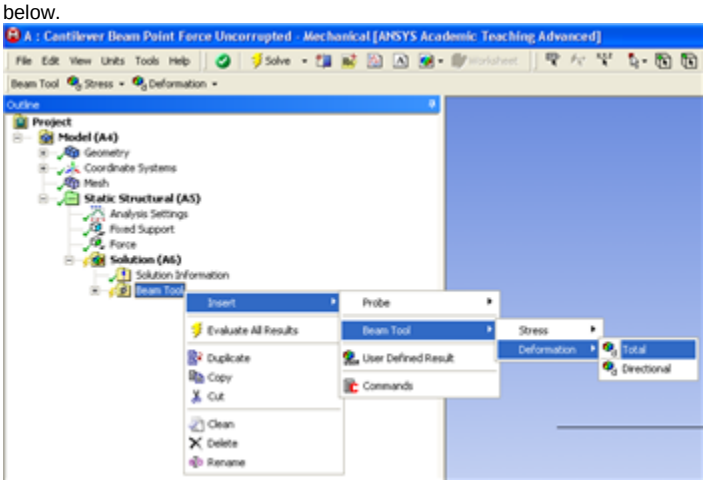
We next specify what results we'd like to look at. Note that these results can also be specified after we solve the model. First, click on the solution button, 

, 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.



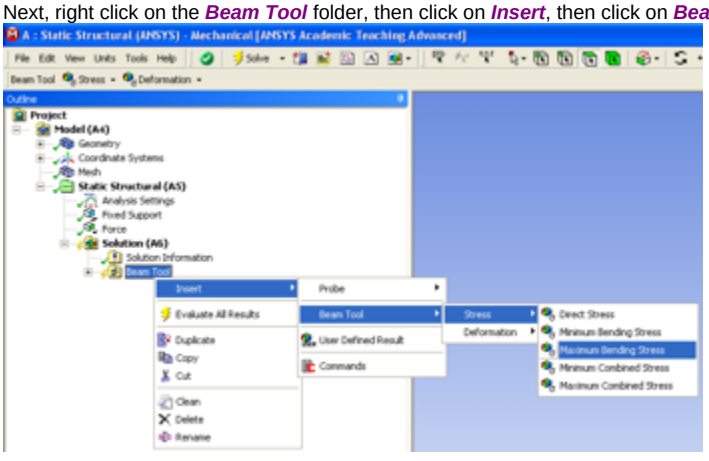
Click Here For Higher Resolution

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.



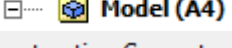
Click Here For Higher Resolution

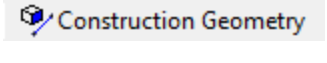
Next, right click on the ***Beam Tool*** folder, then click on ***Insert***, then click on ***Beam Tool > Stress > Maximum Bending Stress*** as shown below.

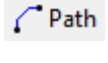


Click Here For Higher Resolution

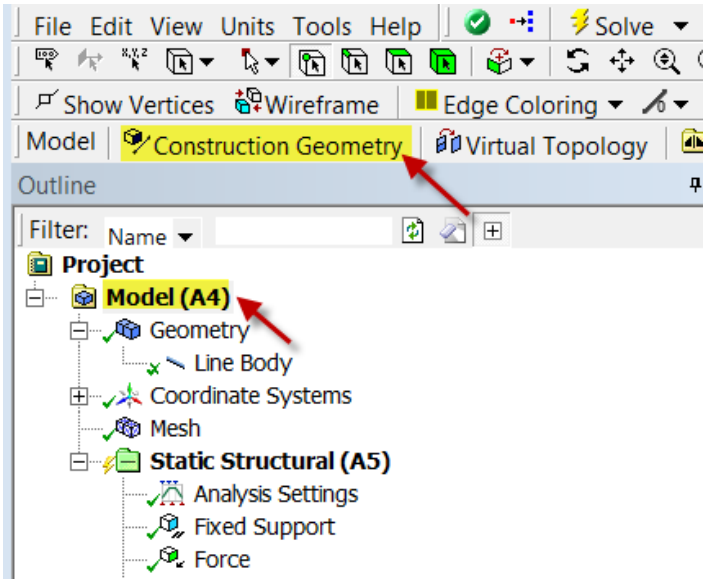
Bending Moment along the Beam

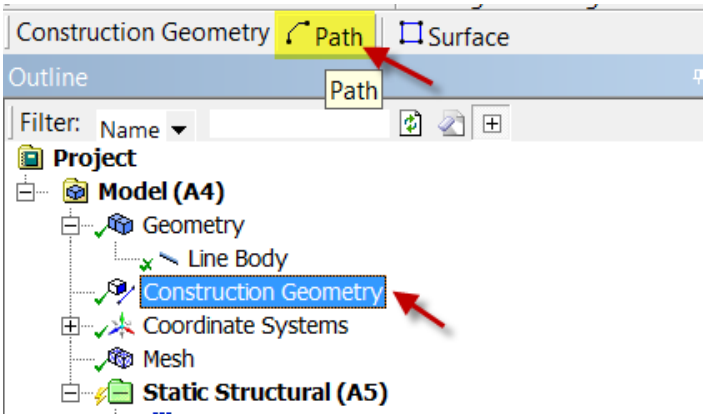
Now, we will set up a result object for the bending moment along the beam. We will do this by setting up a "path" along the line body. To set up a path, click on 

in the *Outline* window. This will launch the Model toolbar in the Menu Bar. In the Model toolbar, press 

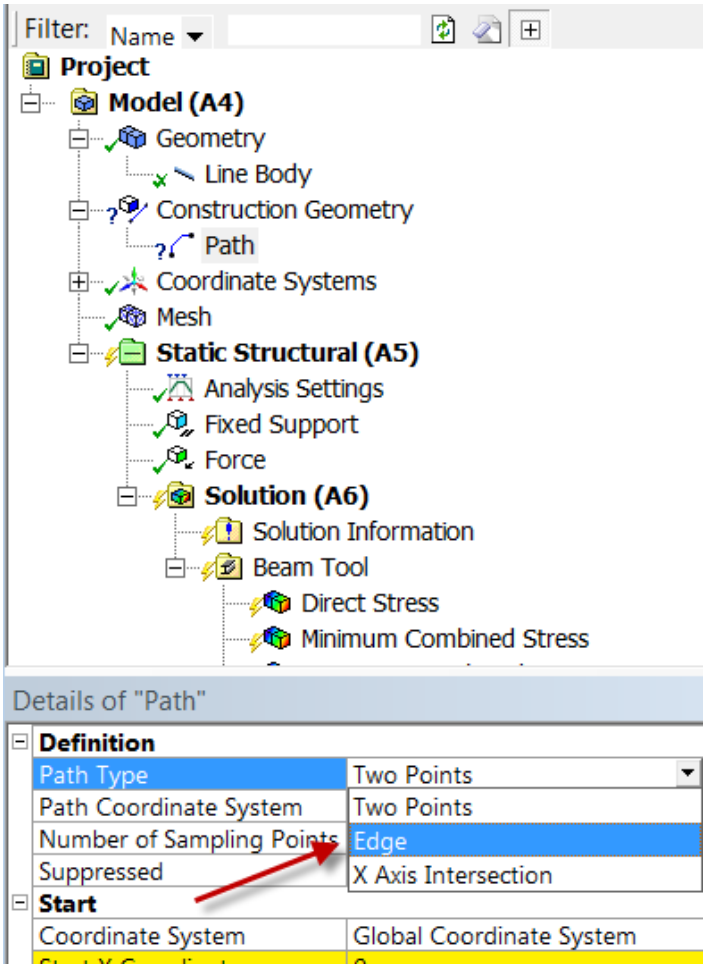
which will bring up the Construction Geometry Tool bar, then press 

to create a path.

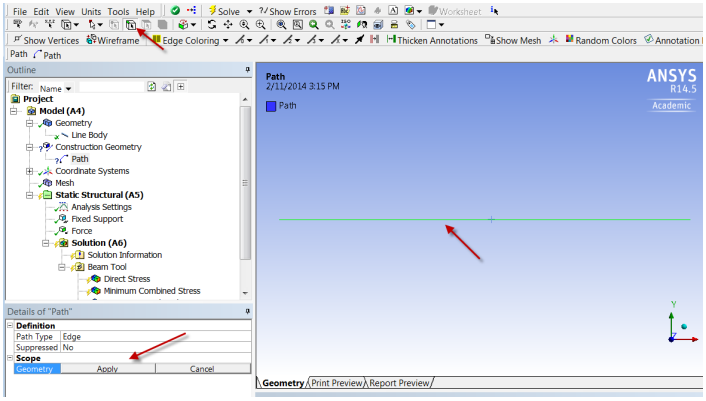

File Edit View Units Tools Help
Solve
Show Vertices
Wireframe
Edge Coloring
Model
Construction Geometry
Virtual Topology
Outline
Filter:
Name
Project
Model (A4)
Geometry
Line Body
Coordinate Systems
Mesh
Static Structural (A5)
Analysis Settings
Fixed Support
Force

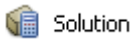


In the *Details* window, notice that the default path type is ***Two Points***. We need to change that to ***Edge***.

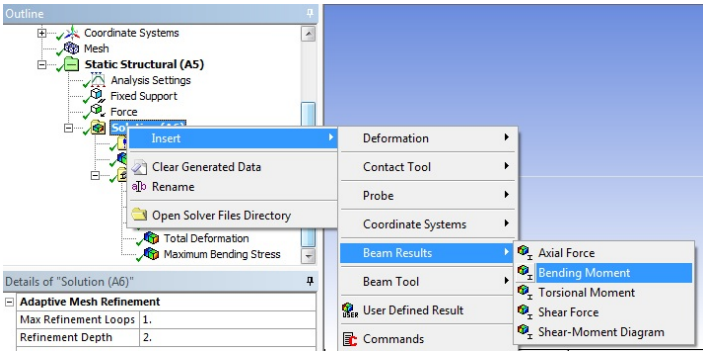


Next, using the ***Edge Selection filter***  select the line body in the *Graphics* window. Back in the *Details of "Path"* window, select ***Geometry > Apply***. Rename it "neutral axis".



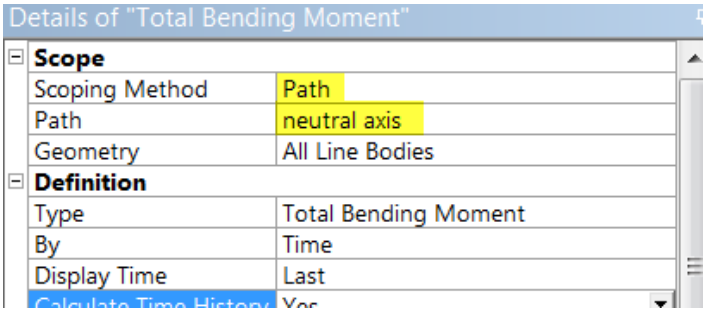
Now that we have created the path, we need to create a solution object that gives the bending moment along the path. Click on 

in the *Outli ne* window to bring up the solution menu, then select **Beam Results > Bending Moment**.



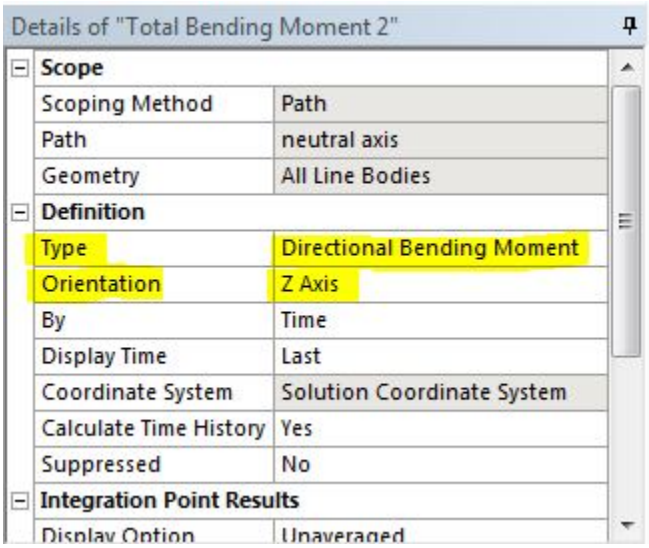
click here for full view

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).



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***.



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.

Go to Step 6: Numerical Results

Go to all ANSYS Learning Modules