Cantilever Beam - Numerical Solution (OLD)

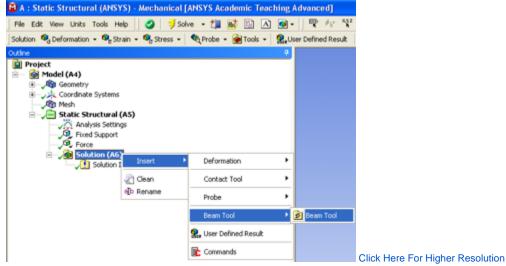
Author: John Singleton, Cornell University Problem Specification 1. Pre-Analysis & Start-Up 2. Geometry 3. Mesh 4. Physics Setup 5. Numerical Solution 6. Numerical Results 7. Verification & Validation Exercises Comments

Numerical Solution

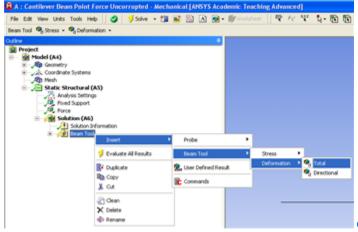
Choosing Results

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.

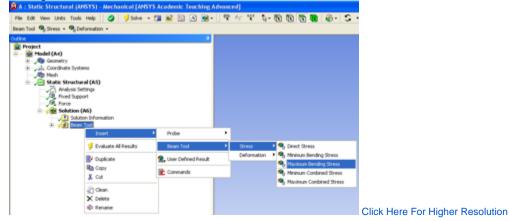


Then, right click on the *Beam Tool* folder that you have just added, then click on *Insert*, then click on *Beam Tool* > *Deformation* > *Total* as displayed below.



Click Here For Higher Resolution

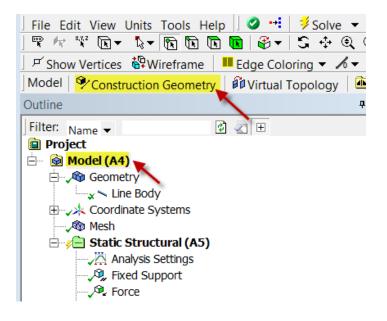
Next, right click on the Beam Tool folder, then click on Insert, then click on Beam Tool > Stress > Maximum Bending Stress as shown below.



Bending Moment along the Beam

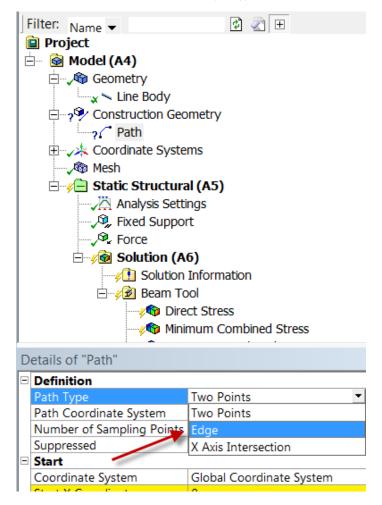
Now, we will set up a result object for the bending moment along the beam. We will do this by setting up a "path" along the line body. To set up a path, click on in the Outline window. This will launch the Model toolbar in the Menu Bar. In the Model toolbar, press

Construction Geometry which will bring up the Construction Geometry Tool bar, then press C Path to create a path.

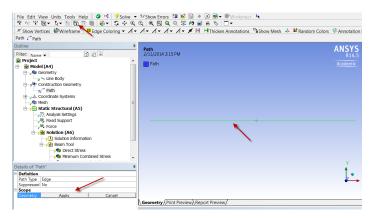


Construction Geometry C Path Surface
Outline Path
Filter: Name 🗸 👔 🕀
💼 Project
🖮 🎯 Model (A4)
🗄 🗸 🖚 Geometry
🛶 🛰 Line Body
🕀 🏑 Koordinate Systems
🗄 🌾 🔁 Static Structural (A5)

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**.

Outline 🕂		
Coordinate Systems Mesh Static Structural (A5) Analysis Settings Fixed Support Fixed Support Settings Fixed Support Settings Fixed Support Fixed		
insert	Deformation	•
Clear Generated Data	Contact Tool	•
allo Rename	Probe	•
Open Solver Files Directory	Coordinate Systems	•
🗤 🖓 Total Deformation	Beam Results	• 🔍 Axial Force
Details of "Solution (A6)" 🛛	Beam Tool	Bending Moment
Adaptive Mesh Refinement	👷 User Defined Result	 Torsional Moment Shear Force
Max Refinement Loops 1.	USER	 Shear-Moment Diagram
Refinement Depth 2.	Commands	T Snear-Woment Diagram

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).

D	Details of "Total Bending Moment"				
-	Scope				
	Scoping Method	Path			
	Path	neutral axis			
	Geometry	All Line Bodies			
Ξ	Definition				
	Туре	Total Bending Moment			
	Ву	Time			
	Display Time	Last	Ξ		
	Calculate Time History	Voc			

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*.

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

Solve

🔣 Solve

In order to solve, click on the solve button, **Solve**, 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