ANSYS 12 - Beam

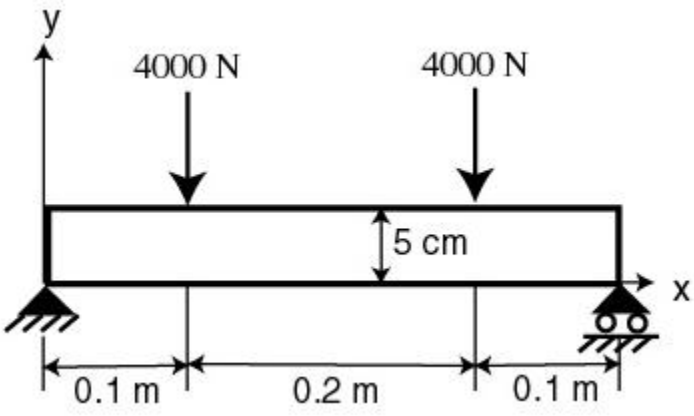
Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Under Construction

The following ANSYS tutorial is under construction.

Problem Specification

Consider the beam in the figure below. There are two point forces acting on the beam in the negative y direction as shown. Note the dimensions of the beam. The Young's modulus of the material is 73 GPa and the Poisson ratio is 0.3. We'll assume that plane stress conditions apply.



Go to Step 1: Pre-Analysis & Start-Up

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

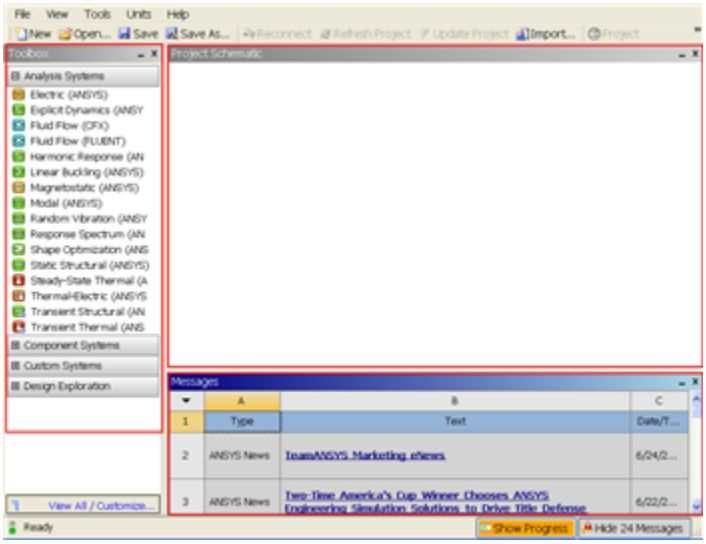
Step 1: Pre-Analysis & Start-Up

Start ANSYS Workbench

We start our simulation by first starting the ANSYS workbench.

Start > All Programs > ANSYS 12.1 > Workbench

Following figure shows the workbench window.



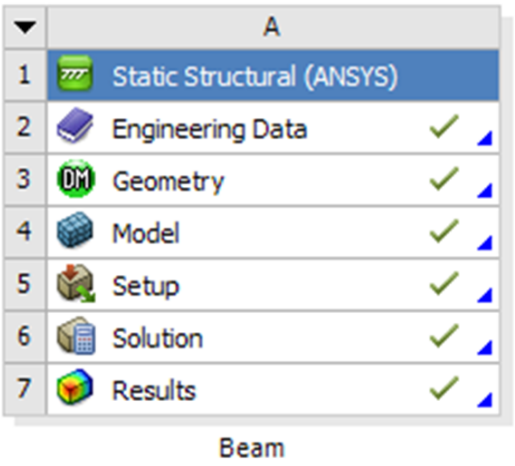
At the left hand side of the workbench window, you will see a toolbox full of various analysis systems. In the middle, you see an empty work space. This is the place where you will organize your project. At the bottom of the window, you see messages from ANSYS.

Select Analysis Systems

Select Analysis System Demo

Since our problem involves static analysis, we will select the ***Static Structural (ANSYS)*** component on the left panel.
Left click (and hold) on ***Static Structural (ANSYS)***, and drag the icon to the empty space in the ***Project Schematic***.

Since we selected Static Structural (ANSYS), each cell of the system corresponds to a step in the process of performing the ANSYS Structural analysis. Right click on ***Static Structural ANSYS*** and ***Rename*** the project to ***Beam***.



Now, we just need to work out each step from top down to get to the results for our solution.

- We start by preparing our geometry
- We use geometry to generate a mesh
- We setup the physics of the problem
- We run the problem in the solver to generate a solution
- Finally, we post process the solution to gain insight into the results

Specify Material Properties

We will first specify the material properties of the crank. The material has an Young's modulus E=2.8x10^7 psi and Poisson's ratio =0.3.

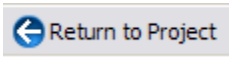
In the Crank cell, double click on ***Engineering Data***. This will bring you to a new page. The default material given is ***Structural Steel***. We will use this material and change the Young's modulus and Poisson's ratio.

Left click on ***Structural Steel*** once and you will see the details of Structural Steel material properties under ***Properties of Outline Row 3: Structural Steel*** . Expand ***Isotropic Elasticity***, change ***Young's Modulus*** and ***Poisson's Ratio*** to E=7.9e10 pa and =0.3. Remember to check that you use the correct unit.

Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Young's Modulus	7.3E+10	Pa		
8	Poisson's Ratio	0.3			
9	Alternating Stress Mean Stress	Tabular			
13	Strain-Life Parameters				
21	Tensile Yield Strength	2.5E+08	Pa		
22	Compressive Yield Strength	2.5E+08	Pa		
23	Tensile Ultimate Strength	4.6E+08	Pa		

Higher Resolution Window

Press the ***Return to Project*** 

to return to Workbench ***Project Schematic*** window.

Go to Step 2: Geometry

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Step 2: Geometry

At Workbench, in the ***Beam*** cell, right click on ***Geometry***, and select ***Properties***. You will see the properties menu on the right of the Workbench window. Under ***Basic Geometry Options***, select ***Line Bodies***. This is because we are going to create a line geometry.

Properties of Schematic A3: Geometry

	A	B
1	Property	Value
2	General	
3	Cell ID	Geometry 1
4	Geometry Source	
5	Geometry File Name	H:\ANSYS Workbench...
6	CAD Plug-In	DesignMod...
7	Basic Geometry Options	
8	Solid Bodies	☑
9	Surface Bodies	☑
10	Line Bodies	☑
11	Attributes	☐
12	Named Selections	☐
13	Material Properties	☐
14	Advanced Geometry Options	
15	Analysis Type	3D
16	Use Associativity	☑
17	Import Coordinate Systems	☐
18	Import Work Points	☐
19	Reader Mode Saves Updated File	☐
20	Import Using Instances	☑
21	Smart CAD Update	☐
22	Enclosure and Symmetry Processing	☑
23	Mixed Import Resolution	None

In the ***Project Schematic***, double left click on ***Geometry*** to start preparing the geometry.

At this point, a new window, ANSYS Design Modeler will be opened. You will be asked to select desired length unit. Use the default ***meter*** unit and click ***OK*** .

Creating a Sketch

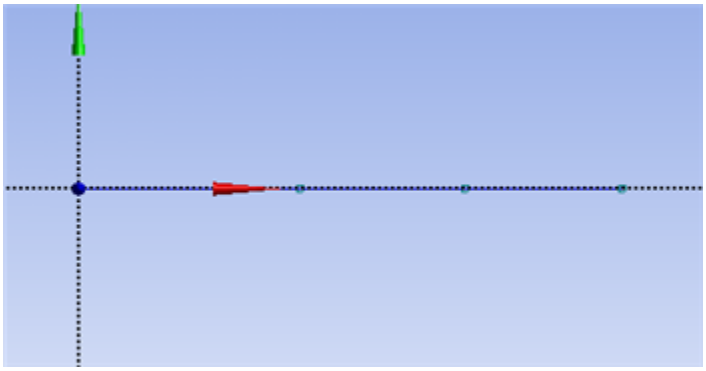
Like any other common CAD modeling practice, we start by creating a sketch.

Start by creating a sketch on the ***XYPlane***. Under ***Tree Outline***, select ***XYPlane***, then click on ***Sketching*** next to ***Modeling*** tab. This will bring up the ***Sketching Toolboxes***.

Note: In sketching mode, there is ***Undo*** features that you can use if you make any mistake. Select Sketching Toolboxes Demo

On the right, there is a ***Graphic*** window. At the lower right hand corner of the Graphic window, click on the ***+Z*** axis to have a normal look of the ***XY Plane***. Select Normal View Demo

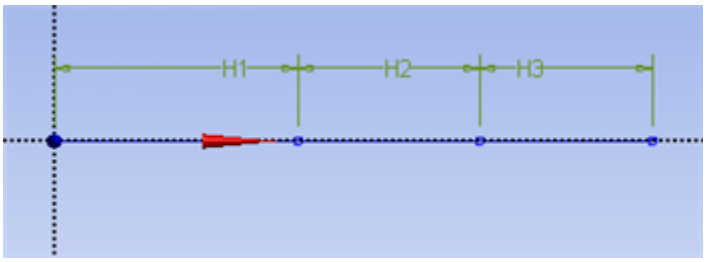
In the ***Sketching Toolboxes***, select ***Line***. In the ***Graphics*** window, create three rough lines from starting from the origin in the positive XY direction (Make sure that you see a letter P at the origin and at each connection between the lines. The letter P the geometry is constrained at the point.)
You should have something like this:



Note: You do not have to worry about dimension for now, we can dimension them properly in the later step.

Dimensions

Under ***Sketching Toolboxes***, select ***Dimensions*** tab, use the default dimensioning tools. Dimension the geometry as shown:



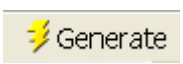
Under ***Details View*** on the lower left corner, input the value for dimension appropriately.
H1: 0.1 m
H2: 0.2 m
H3: 0.1 m

We are done with sketching.

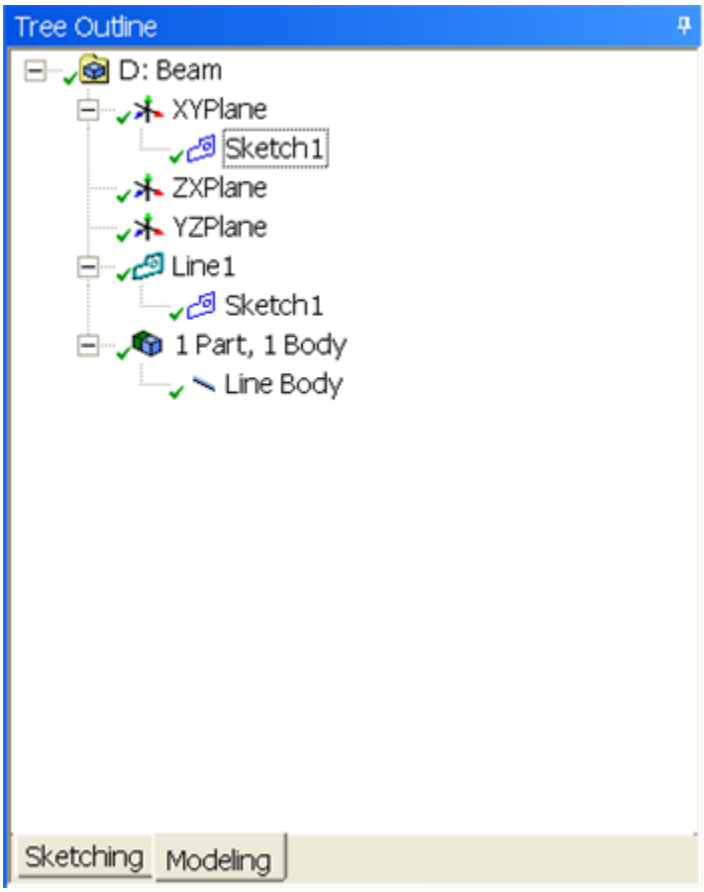
Create Surface

Now that we have the sketch done, we can create a line body for this sketch.

Concept > Lines From Sketches

This will create a new line ***Line1***. Under ***Details View***, select ***Sketch1*** as ***Base Objects*** and click ***Apply***. Finally click ***Generate*** 

to generate the surface. This is what you should see under your ***Tree Outline***.



Create Cross Section

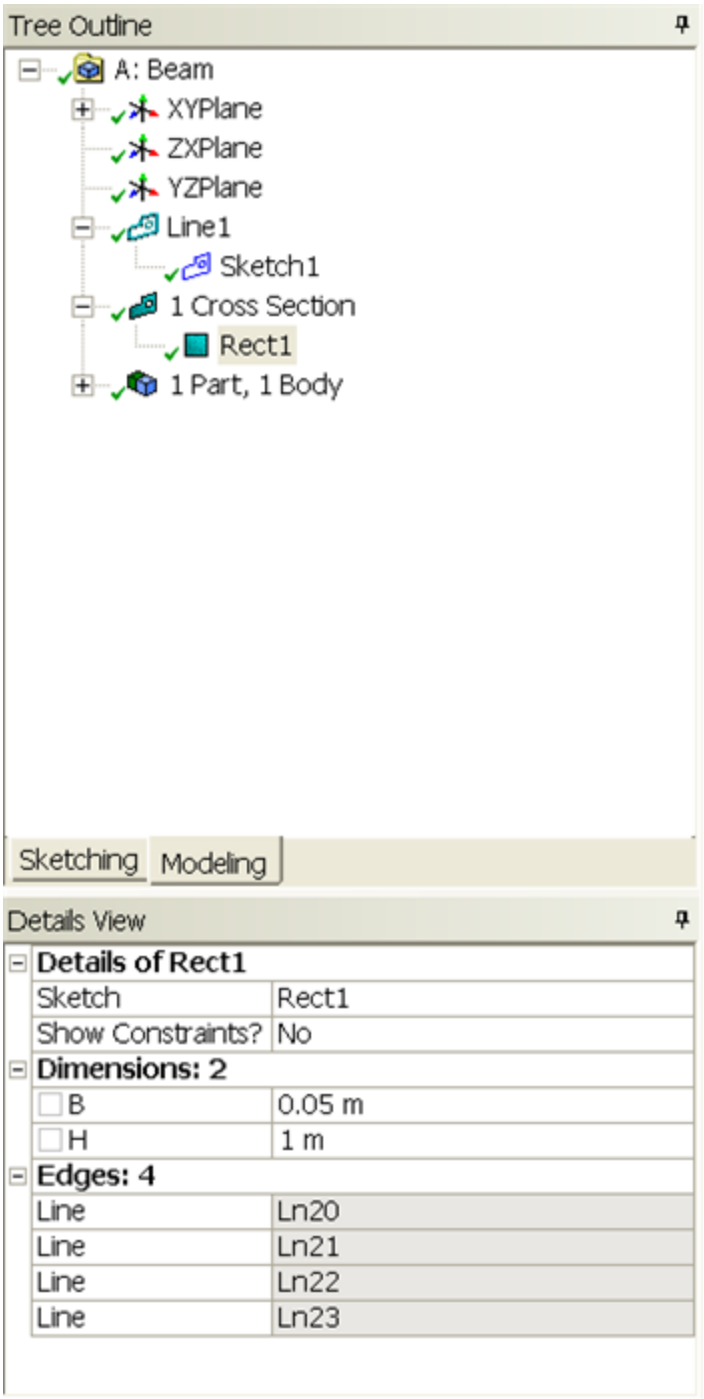
We will now add a cross section to the line body.

Concept > Cross Section > Rectangular

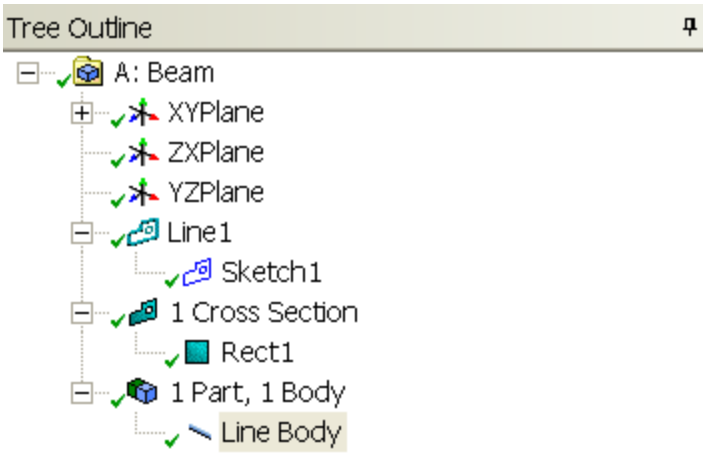
Under Details View, input value as follow:

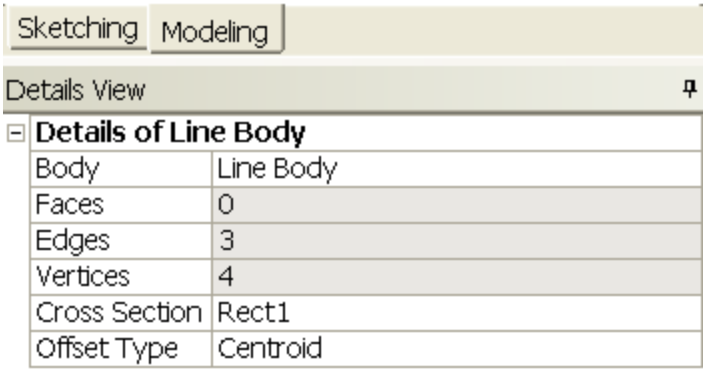
B - 0.05m

H - 1m



Finally, under expand the Line Body
Outline > 1 Part, 1 Body > Line Body
And attach ***Rect1*** to ***Cross Section*** under ***Details View***.





We are done with geometry. You can close the ***Design Modeler*** and go back to ***Workbench*** (Don't worry, it will auto save).

Go to Step 3: Mesh

See and rate the complete Learning Module

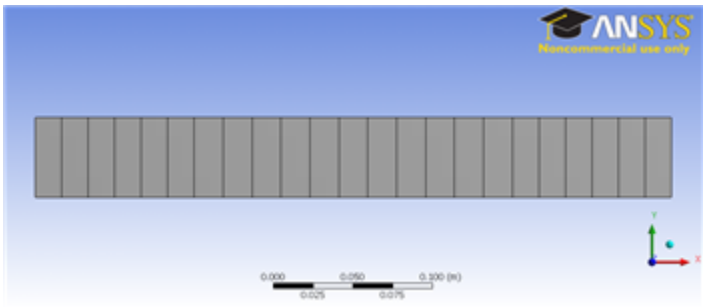
Go to all ANSYS Learning Modules

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Step 3: Mesh

Save your work in ***Workbench*** window. In the ***Workbench*** window, right click on ***Mesh***, and click ***Edit***. A new ***ANSYS Mesher*** window will open.

Use the default mesh. Under ***Outline***, right click on ***Mesh*** and click ***Generate Mesh***. This should be the mesh appear in the Graphics window.



Go to Step 4: Setup (Physics)

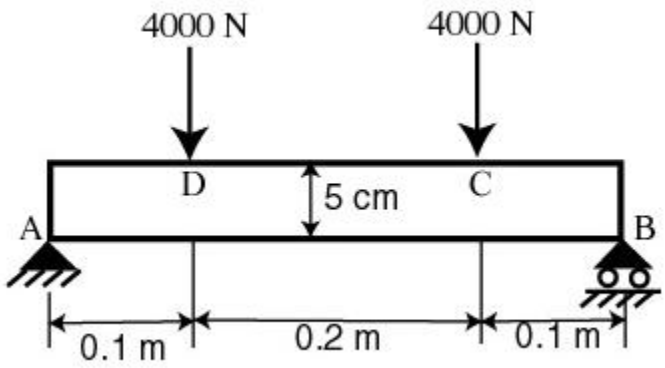
See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Step 4: Setup (Physics)

We need to specify point BC's at *A*, *B*, *C* and *D*.



Let's start with setting up boundary condition at A.
Outline > Static Structural (A5) > Insert > Remote Displacement
Select point A in the ***Graphics*** window and click ***Apply*** next to Geometry under ***Details of "Remote Displacement"***. Enter 0 for all UX, UY, UZ, ROTX and ROTY except for ROTZ. Let ROTZ to be free.

Details of "Remote Displacement"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Coordinate System	Global Coordinate System
X Coordinate	0. m
Y Coordinate	0. m
Z Coordinate	0. m
Location	Click to Change
Definition	
Type	Remote Displacement
X Component	0. m (ramped)
Y Component	0. m (ramped)
Z Component	0. m (ramped)
Rotation X	0. ° (ramped)
Rotation Y	0. ° (ramped)
Rotation Z	Free
Suppressed	No

Let's move on to setting up boundary condition B.
Outline > Static Structural (A5) > Insert > Remote Displacement
Select point B in the ***Graphics*** window and click ***Apply*** next to Geometry under ***Details of "Displacement 2"***. Enter 0 for all UY, UZ, ROTX and ROTY except for ROTZ. Let UX and ROTZ to be free.

Details of "Remote Displacement 2"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Coordinate System	Global Coordinate System
X Coordinate	0.4 m
Y Coordinate	0. m
Z Coordinate	0. m
Location	Click to Change
Definition	
Type	Remote Displacement
X Component	Free
Y Component	0. m (ramped)
Z Component	0. m (ramped)
Rotation X	0. ° (ramped)
Rotation Y	0. ° (ramped)
Rotation Z	Free
Suppressed	No

We can move on to setting up point force at point C and D.

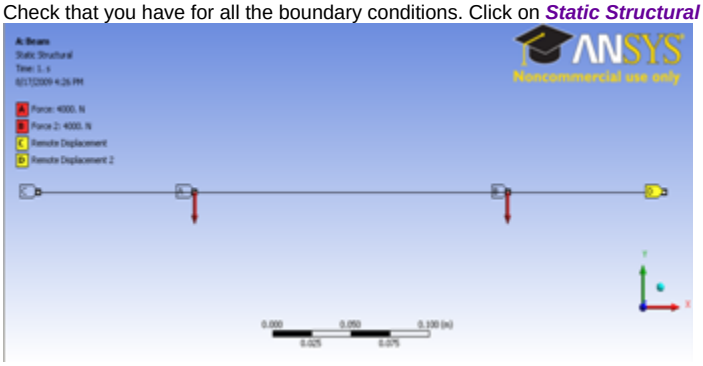
Outline > Static Structural (A5) > Insert > Force

Select point C in the ***Graphics*** window and click ***Apply*** next to Geometry under ***Details of "Force"***. Next to ***Define By***, change ***Vector*** to ***Components***. Enter -4000 for ***Y Component***.

Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. N (ramped)
Y Component	-4000. N (ramped)
Suppressed	No

Do the same for point D.

Check that you have for all the boundary conditions. Click on ***Static Structural (A5)*** to view this in Graphics window.



Higher Resolution Image

Go to Step 5: Solution

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Step 5: Solution

Now that we have set up the boundary conditions, we can actually solve for a solution. Before we do that, let's take a minute to think about what is the post-processing that we are interested in. We are interested in the deflection and bending stress on the beam. We would also like to look at the force and moment reaction at our support A and B. Let's set up those post-processing parameters before we click solve button.

Let's start with inserting Total Deformation.

Outline > Solution (A6) > Insert > Total Deformation

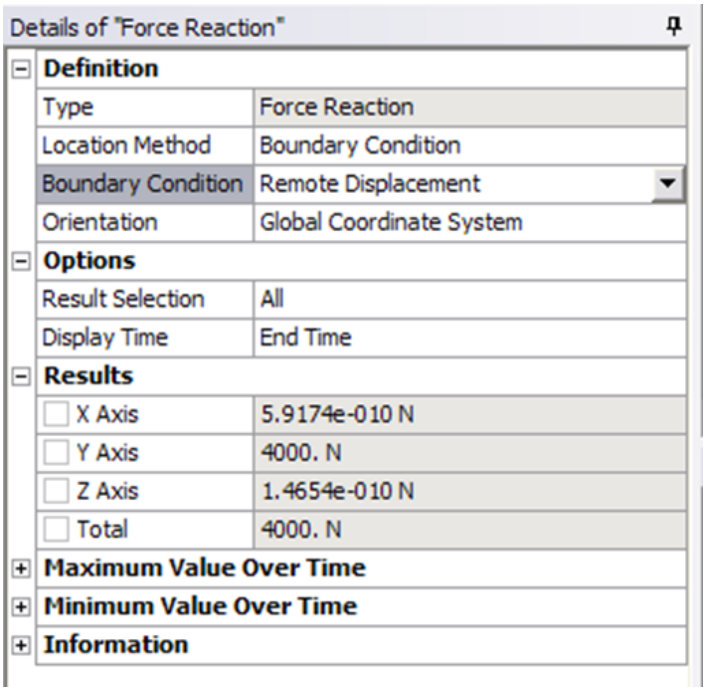
Next let's insert beam tool that will enable us to look at the stresses on the beam.

Outline > Solution (A6) > Insert > Beam Tool > Beam Tool

We would also like to look at the Force Reaction at point A and B.

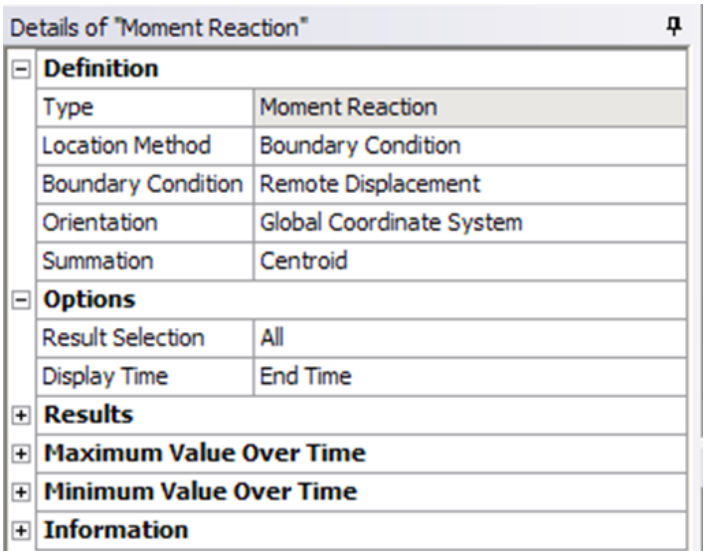
Outline > Solution (A6) > Insert > Probe > Force Reaction

Select ***Remote Displacement*** (which is point A) next to ***Boundary Condition*** under ***Details of "Force Reaction"***.



Details of "Force Reaction"	
Definition	
Type	Force Reaction
Location Method	Boundary Condition
Boundary Condition	Remote Displacement
Orientation	Global Coordinate System
Options	
Result Selection	All
Display Time	End Time
Results	
X Axis	5.9174e-010 N
Y Axis	4000. N
Z Axis	1.4654e-010 N
Total	4000. N
Maximum Value Over Time	
Minimum Value Over Time	
Information	

Do the same step for Remote Displacement 2 (point B).
Next we will like to check and see that the moment at point A and B is zero.
Outline > Solution (A6) > Insert > Probe > Moment Reaction
Select ***Remote Displacement*** (which is point A) next to ***Boundary Condition*** under ***Details of "Moment Reaction"***.



Details of "Moment Reaction"	
Definition	
Type	Moment Reaction
Location Method	Boundary Condition
Boundary Condition	Remote Displacement
Orientation	Global Coordinate System
Summation	Centroid
Options	
Result Selection	All
Display Time	End Time
Results	
Maximum Value Over Time	
Minimum Value Over Time	
Information	

Do the same step for Remote Displacement 2 (point B).
We are done setting up all the results. Click ***Solve*** at the top menu to obtain a solution. Wait for a minute for the solution.

Go to Step 6: Results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules