

ANSYS 12 - Beam (2D Element)

- [Problem Specification](#)

Step 1: Pre-Analysis & Start-Up

- [Start ANSYS Workbench](#)
- [Select Analysis Systems](#)
- [Specify Material Properties](#)

Step 2: Geometry

- [Strategy for Geometry Creation](#)
- [Creating a Sketch](#)
- [Modify the Sketch](#)
- [Dimensions and Constraints](#)
- [Create Surface](#)

Step 3: Mesh

Step 4: Setup (Physics)

Step 5: Solution

Step 6: Results

- [Total Deformation](#)
- [Bending Stress](#)

Step 7: Verification & Validation

- [Simple Checks](#)
- [Refine Mesh](#)

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Problem Specification

1. [Pre-Analysis & Start-Up](#)
2. [Geometry](#)
3. [Mesh](#)
4. [Setup \(Physics\)](#)
5. [Solution](#)
6. [Results](#)
7. [Verification & Validation](#)

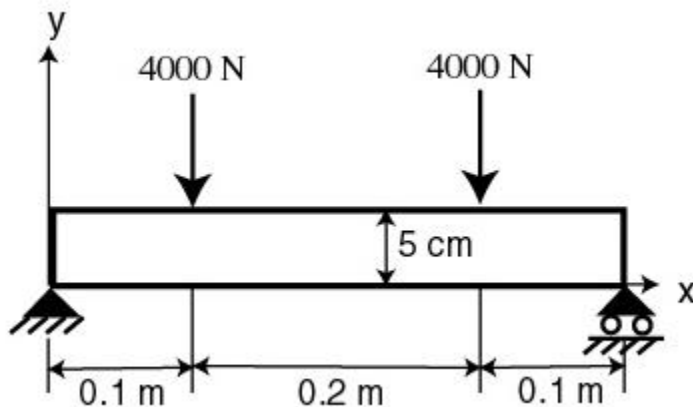


Under Construction

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

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solution
6. Results
7. Verification & Validation

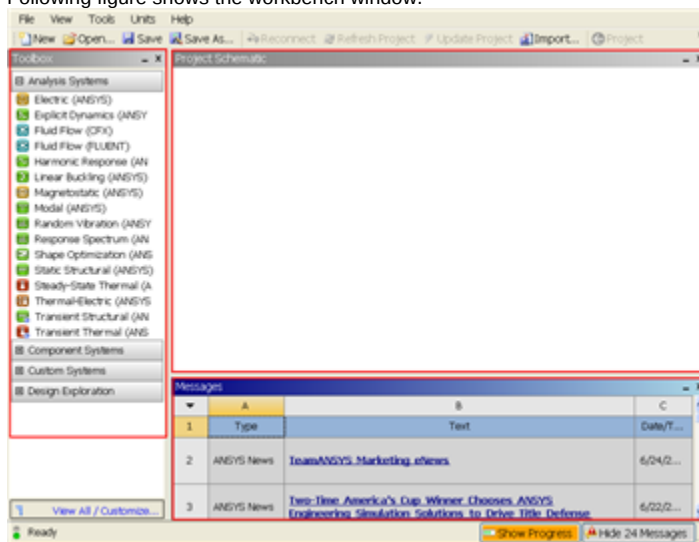
Step 1: Pre-Analysis & Start-Up

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.



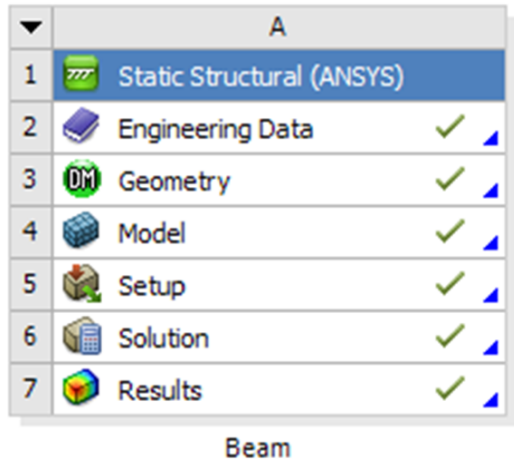
At the left hand side of the workbench window, you will see a toolbox full of various analysis systems. In the middle, you see an empty work space. This is the place where you will organize your project. At the bottom of the window, you see messages from ANSYS.

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 $E=2.8 \times 10^7$ psi and Poisson's ratio $\nu=0.3$.

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 $E=7.9 \times 10^{10}$ pa and $\nu=0.3$. 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	
24	Compressive Ultimate Strength	0	Pa	

[Higher Resolution Window](#)

Press the **Return to Project** button 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

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solution
6. Results
7. Verification & Validation

Step 2: Geometry

Our geometry is 2D. 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 **Advance Geometry Options**, change the **Analysis Type** to 2D.

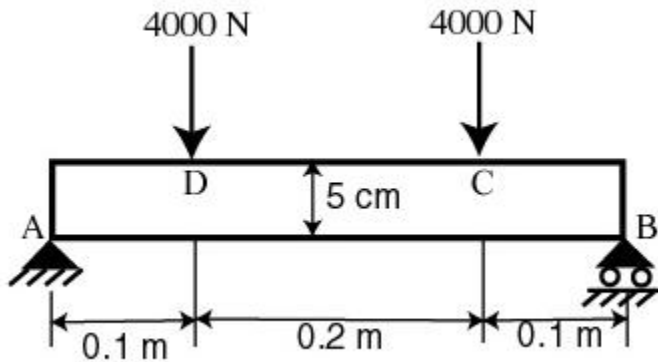
Properties of Schematic A2: Geometry		
	A	B
1	Property	Value
2	General	
3	Cell ID	Geometry
4	Geometry Source	
5	Geometry File Name	
6	Basic Geometry Options	
7	Solid Bodies	☒
8	Surface Bodies	☒
9	Line Bodies	☐
10	Parameters	☒
11	Parameter Key	DS
12	Attributes	☐
13	Named Selections	☐
14	Material Properties	☐
15	Advanced Geometry Options	
16	Analysis Type	2D ▼
17	Use Associativity	☒
18	Import Coordinate Systems	☐
19	Import Work Points	☐
20	Reader Mode Saves Updated File	☐
21	Import Using Instances	☒
22	Smart CAD Update	☐
23	Enclosure and Symmetry Processing	☒
24	Mixed Import Resolution	None ▼

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

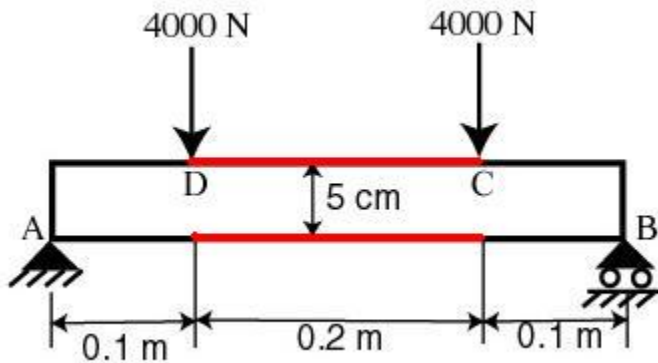
At this point, a new window, ANSYS Design Modeler will be opened. You will be asked to select desired length unit. Use the default **meter** unit and click **OK**.

Strategy for Geometry Creation

We need to apply point boundary conditions at four points *A*, *B*, *C* and *D* shown in the figure below. In ANSYS, point boundary conditions are applied to nodes. When we mesh the rectangle using the ANSYS Mesher, nodes will automatically be created at *A* and *B* since these are corner points; the corresponding displacement BC's can be applied to these corner nodes. However, there is no guarantee that there will be nodes exactly at *C* and *D* since these are not corners. In this case, ANSYS will apply the forces at the nodes that are closest to *C* and *D*. This is possibly acceptable if the mesh is sufficiently fine.



An alternative scenario is that we get clever and force *C* and *D* to be exact points. This can be done by splitting the top and bottom edges into three sections as shown below. This will force nodes to be created at points *C* and *D*. Then, the point forces can be applied to these nodes. This is the strategy we'll use in creating the geometry.



Creating a Sketch

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

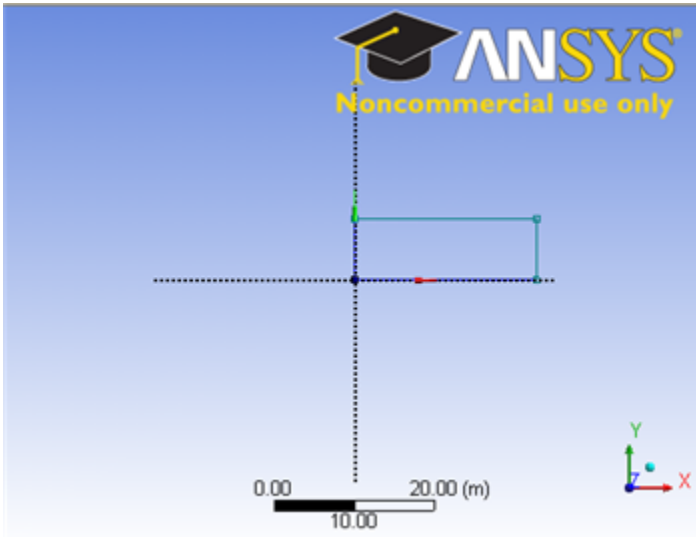
Start by creating a sketch on the *XYPlane*. Under *Tree Outline*, select *XYPlane*, then click on *Sketching* next to *Modeling* tab. This will bring up the *Sketching Toolboxes*.

Note: In sketching mode, there is *Undo* features that you can use if you make any mistake. [Select Sketching Toolbox 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 *Rectangle*. In the *Graphics* window, create a rough Rectangle from starting from the origin in the positive XY direction (Make sure that you see a letter P at the origin before you start dragging the rectangle. The letter P at the origin means the geometry is constrained at the origin.)

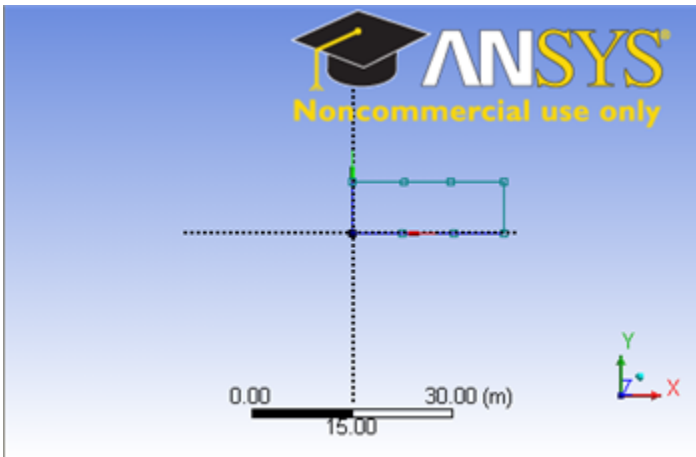
You should have something like this:



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

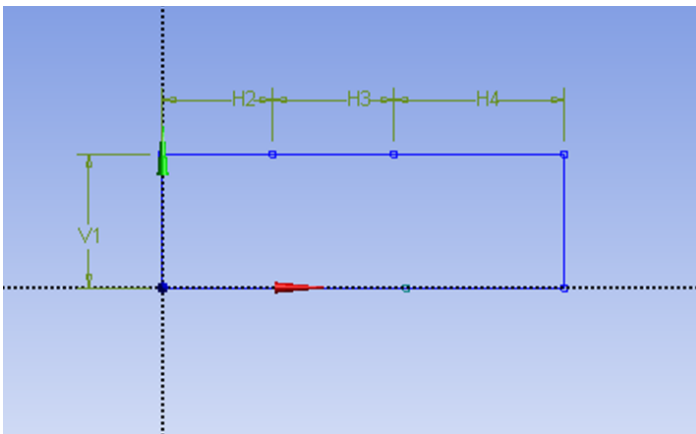
Modify the Sketch

We would like to split the top and bottom with three edges. Click **Modify** tab and select **Split**. Roughly select four points on the top and bottom of the rectangle.



Dimensions and Constraints

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



Now we need to constraint the lower rectangle with the top of the rectangle which has been properly dimensioned. Click **Constraints** tab, select **Equal Length**. Click the appropriate top and bottom edge and set them to be of equal length.

Under **Details View** on the lower left corner, input the value for dimension appropriately.

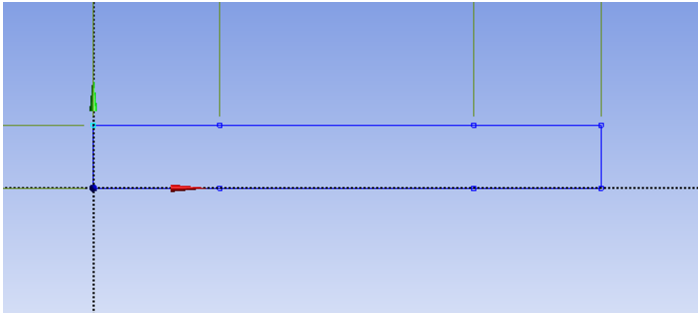
V1: 0.05 m

H2: 0.1 m

H3: 0.2 m

H4: 0.1 m

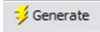
At this point, you should see something like this for your sketch:

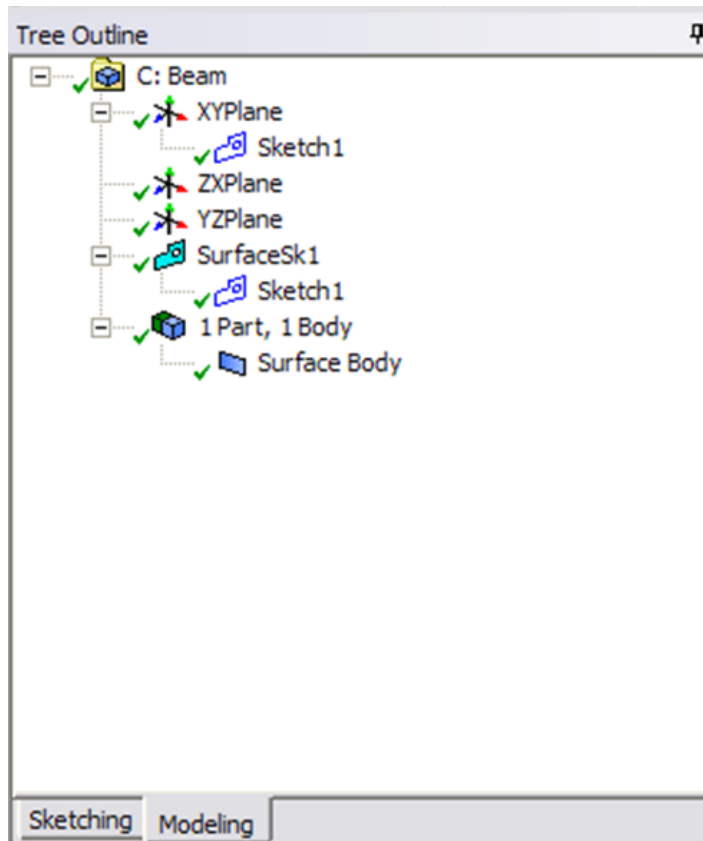


Create Surface

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

Concept > Surfaces From Sketches

This will create a new surface **SurfaceSK1**. 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**.



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

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solution
6. Results
7. Verification & Validation

Step 3: Mesh

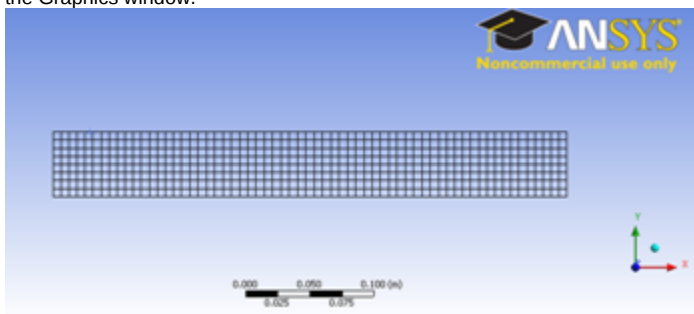
Save your work in **Workbench** window. In the **Workbench** window, right click on **Mesh**, and click **Edit**. A new **ANSYS Mesher** window will open.

We would like to create a structured mesh where the opposite edges correspond with each other. Let's insert a Mapped Face mesh.

Outline > Mesh > Insert > Mapped Face Meshing

Under **Outline**, right click on **Mesh**, move cursor to **Insert**, and select **Mapped Face Meshing**. Finally select the beam surface body in the **Graphics** window and click **Apply** next to **Geometry**.

We can now generate the mesh using the default setting. Under **Outline**, right click on **Mesh** and click **Generate Mesh**. This should be the mesh appear in the Graphics window.



[Higher Resolution Image](#)

Under **Details of "Mesh"**, you should see that we have 504 elements when you expand the **Statistics** tree.

Details of "Mesh"	
[-] Defaults	
Physics Preference	Mechanical
Relevance	0
[+] Sizing	
[+] Inflation	
[+] Advanced	
[+] Pinch	
[-] Statistics	
Nodes	1655
Elements	504
Mesh Metric	None

[Go to Step 4: Setup \(Physics\)](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)

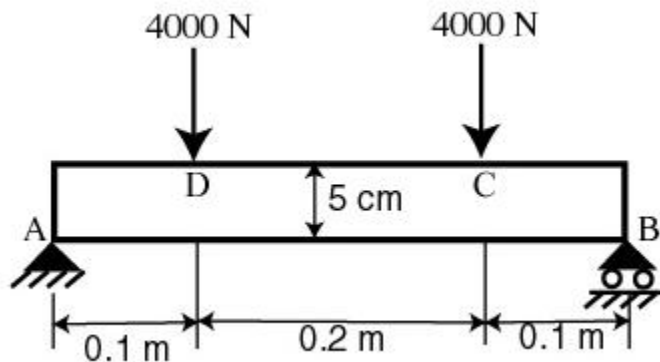
Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solution
6. Results
7. Verification & Validation

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 > Displacement

Select point A in the **Graphics** window and click **Apply** next to Geometry under **Details of "Displacement"**. Enter 0 for both **X Component** and **Y Component**.

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

Let's move on to setting up boundary condition B.

Outline > Static Structural (A5) > Insert > Displacement

Select point B in the **Graphics** window and click **Apply** next to Geometry under **Details of "Displacement 2"**. Enter 0 for **Y Component** and leave **X Component** to be free.

Details of "Displacement 2"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Vertex
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
☐ Y Component	0. m (ramped)
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.

[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

[Problem Specification](#)

- [1. Pre-Analysis & Start-Up](#)
- [2. Geometry](#)
- [3. Mesh](#)
- [4. Setup \(Physics\)](#)
- [5. Solution](#)
- [6. Results](#)
- [7. Verification & Validation](#)

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. Let's set up those post-processing parameters before we click solve button.

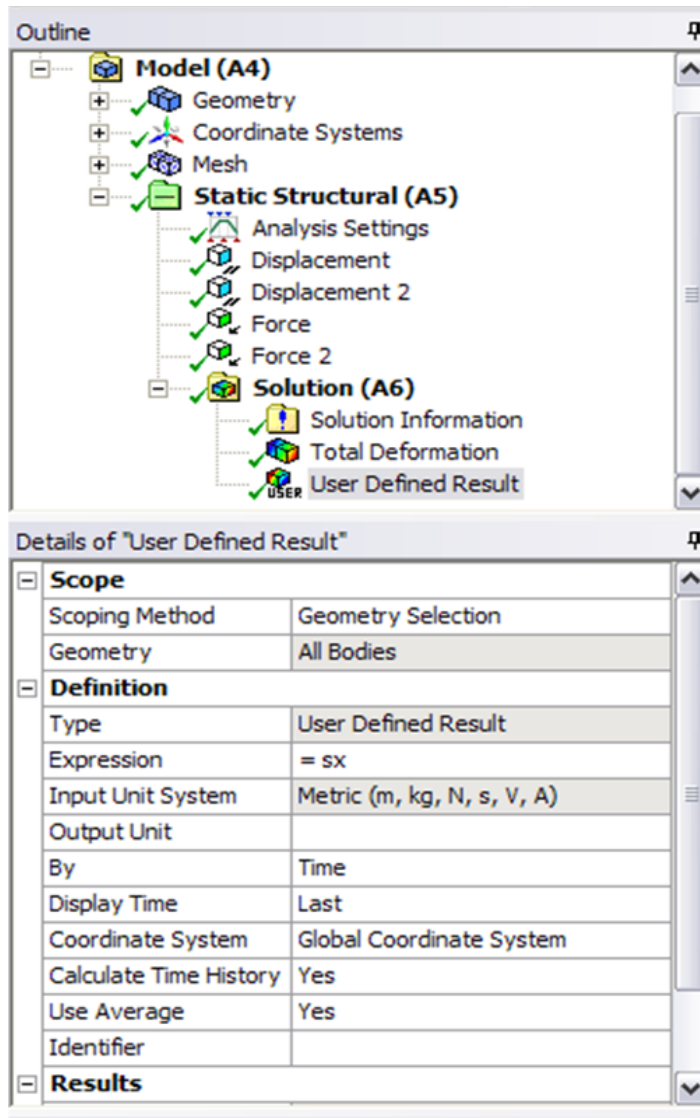
Let's start with inserting Total Deformation.

Outline > Solution (A6) > Insert > Total Deformation

Next let's insert bending moment. This is the stress in the x direction. Unfortunately, this value is not readily available in ANSYS. Let's defined our own variable.

Outline > Solution (A6) > Insert > User Defined Result

Under *Details of "User Defined Result"*, enter SX for *Expression*. Finally click *Solve* at the top menu.



[Go to Step 6: Results](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

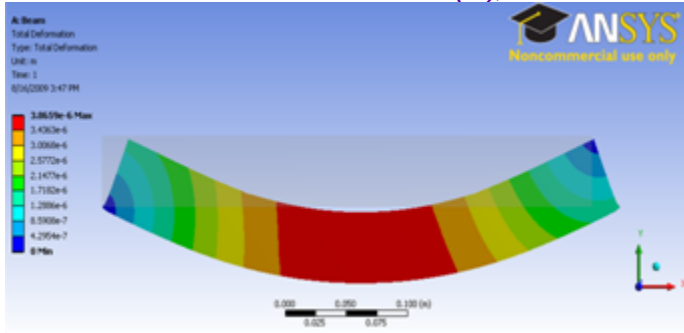
[Problem Specification](#)

- [1. Pre-Analysis & Start-Up](#)
- [2. Geometry](#)
- [3. Mesh](#)
- [4. Setup \(Physics\)](#)
- [5. Solution](#)
- [6. Results](#)
- [7. Verification & Validation](#)

Step 6: Results

Total Deformation

Let first look at Total Deformation. Under **Solution (A6)**, click on **Total Deformation**. The Total Deformation plot is then shown in the Graphics window.



[Higher Resolution Image](#)

Note: To show the original undeformed beam, go to third menu click



and click on



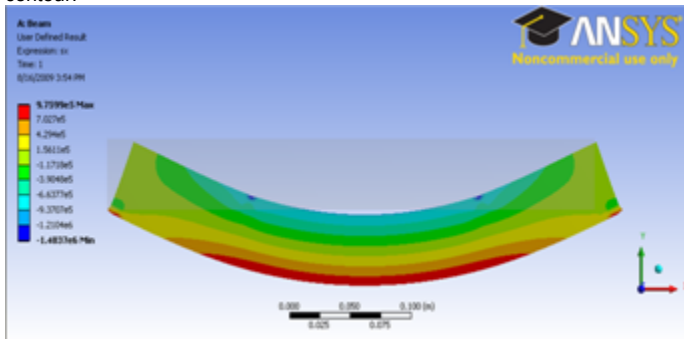
Show Undeformed Model

Notice the deformation is exaggerated, revealing that deformation is caused by bending.

You can also animate the deformation by clicking play button right under **Graphics** window.

Bending Stress

Now let's look at the stress on the beam. Left clicking on **User Defined Result** under **Solution (A6)**. In the Graphics window show the crank stress contour.



[Higher Resolution Image](#)

We expect a pure bending stress in the central region between the two applied forces. The stress is tensile on the bottom surface and compressive on the top surface as expected. Elementary beam theory predicts the bending stress as $\sigma_{xx} = My/I$. Here

$$M = 4000 \cdot 0.1 = 400 \text{ N m}$$

$$I = bh^3/12 = (1) \cdot (0.05)^3/12 = 1.04 \cdot 10^{-5} \text{ m}^4 \text{ (assuming unit thickness in the z direction)}$$

For this geometry, we expect the neutral axis to be at $y = h/2 = 0.025 \text{ m}$. So the max value of $\sigma_{xx} = M \cdot (h/2)/I = 9.6 \cdot 10^5 \text{ Pa}$. This is reasonably close to both the maximum value of tensile and compressive stresses from the computational solution. We have a relatively stubby beam; the agreement with beam theory should improve as the length/height ratio of the beam is increased. Also, the FEA solution perhaps can be made more accurate by refining the mesh. This is left as an exercise to the reader.

Note: We see stress concentration appears around the region of the point force and point displacement. A force and displacement applied to a vertex is not realistic and leads to singular stresses (that is, stresses that approach infinity near the loaded vertex). We should disregard stress values in the vicinity of the loaded points.

[Go to Step 7: Verification & Validation](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Setup (Physics)
5. Solution
6. Results
7. Verification & Validation

Step 7: Verification & Validation

It is **very important** that you take the time to check the validity of your solution. This section leads you through some of the steps you can take to validate your solution.

Simple Checks

Does the bending stress agree with the theoretical value? We checked this in step 6.

Refine Mesh

Let's repeat the solution on a finer mesh with smaller element size. Repeat the mesh steps, but this time we have one additional step after inserting the mapped face mesh. Under **Details of "Mesh"**, expand **Defaults** and enter value of 100 for **Relevance**. Click **Solve** to obtain a new solution.

Details of "Mesh"	
[-] Defaults	
Physics Preference	Mechanical
Relevance	100
[+] Sizing	
[+] Inflation	
[+] Advanced	
[+] Pinch	
[-] Statistics	
Nodes	6433
Elements	2048
Mesh Metric	None

How does the refined mesh compare with the original mesh?

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)