

# ANSYS 12 - Beam (2D Element) - Step 2

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.

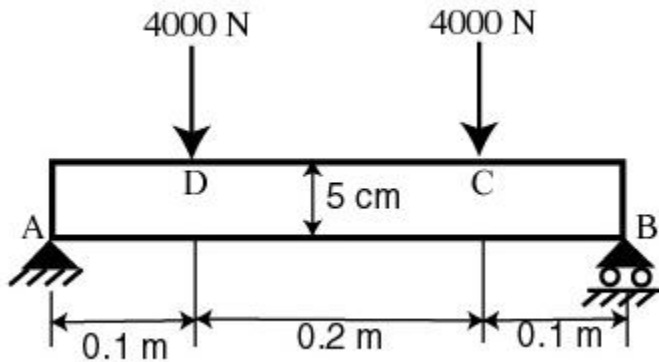
Properties of Schematic A2: Geometry		
	A	B
1	Property	Value
2	General	
3	Cell ID	Geometry
4	Geometry Source	
5	Geometry File Name	
6	Basic Geometry Options	
7	Solid Bodies	<input checked="" type="checkbox"/>
8	Surface Bodies	<input checked="" type="checkbox"/>
9	Line Bodies	<input type="checkbox"/>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

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

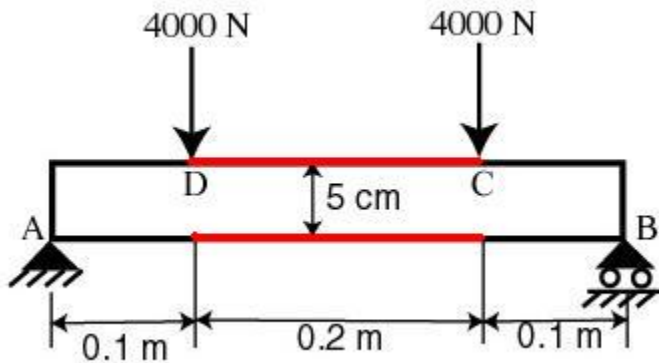
At this point, a new window, ANSYS Design Modeler will be opened. You will be asked to select desired length unit. Use the default **meter** unit and click **OK**.

### 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.

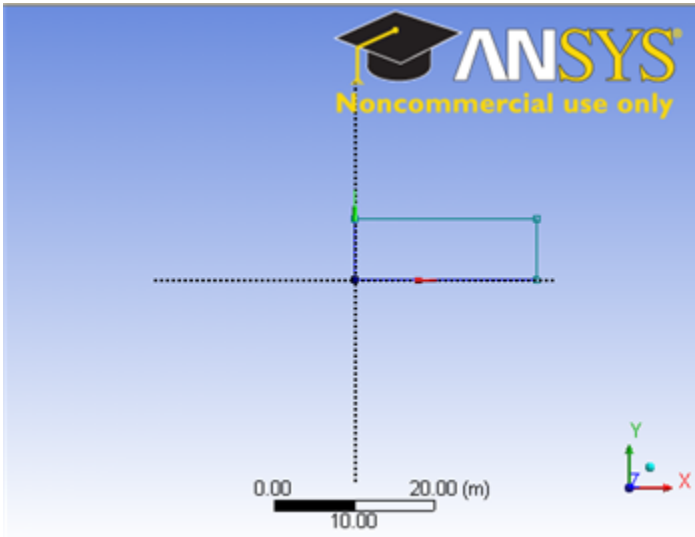
Start by creating a sketch on the *XYPlane*. Under *Tree Outline*, select *XYPlane*, then click on *Sketching* next to *Modeling* tab. This will bring up the *Sketching Toolboxes*.

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.)

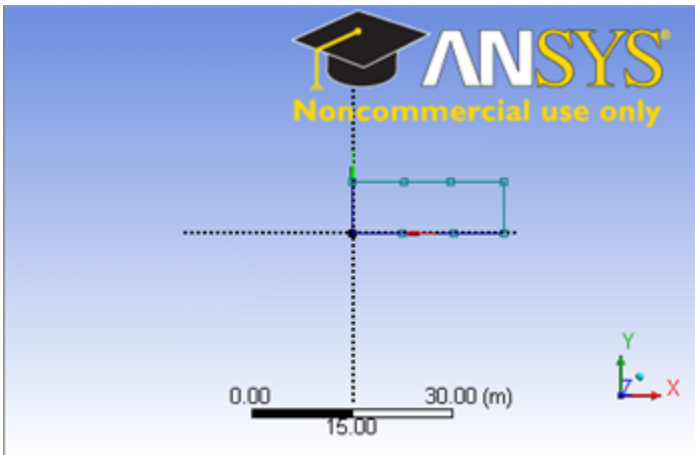
You should have something like this:



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

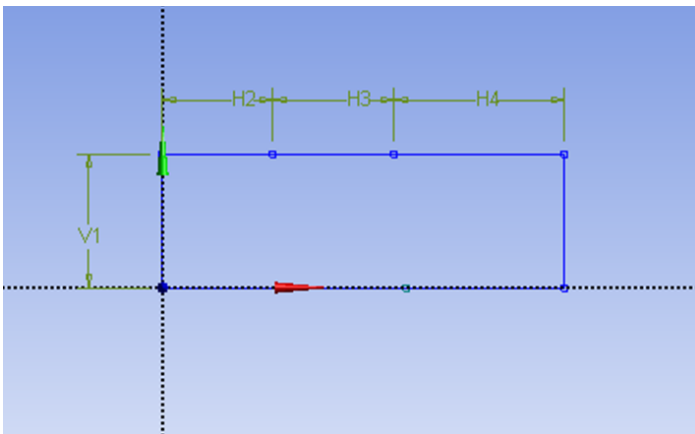
### Modify the Sketch

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.

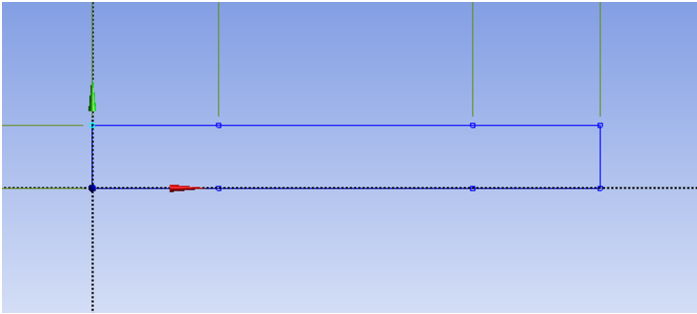
V1: 0.05 m

H2: 0.1 m

H3: 0.2 m

H4: 0.1 m

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

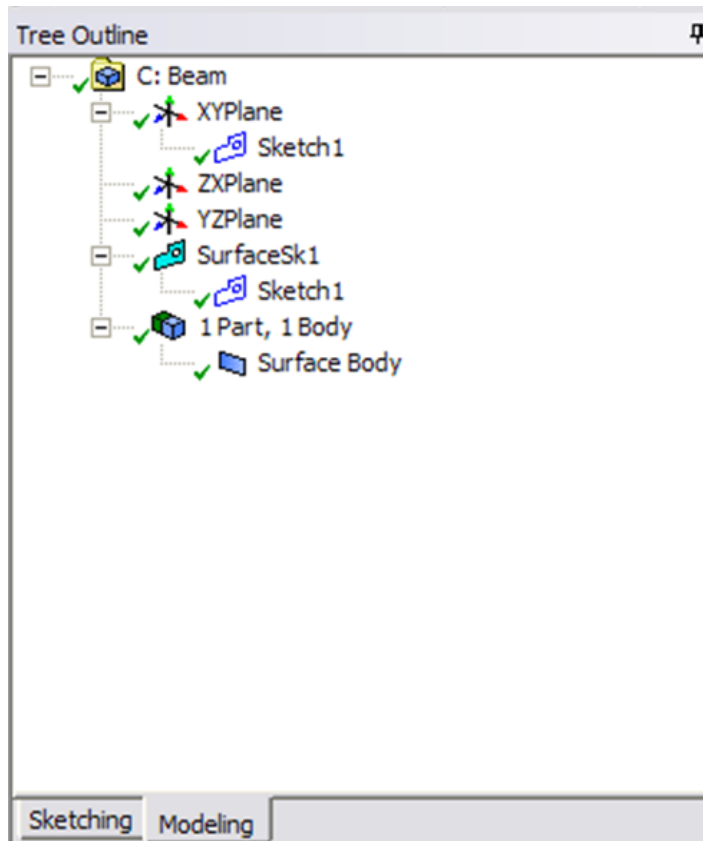
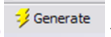


## Create Surface

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](#)