

ANSYS 12 - Beam - 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

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	<input checked="" type="checkbox"/>
9	Surface Bodies	<input checked="" type="checkbox"/>
10	Line Bodies	<input checked="" type="checkbox"/>
11	Attributes	<input type="checkbox"/>
12	Named Selections	<input type="checkbox"/>
13	Material Properties	<input type="checkbox"/>
14	Advanced Geometry Options	
15	Analysis Type	3D ▼
16	Use Associativity	<input checked="" type="checkbox"/>
17	Import Coordinate Systems	<input type="checkbox"/>
18	Import Work Points	<input type="checkbox"/>
19	Reader Mode Saves Updated File	<input type="checkbox"/>
20	Import Using Instances	<input checked="" type="checkbox"/>
21	Smart CAD Update	<input type="checkbox"/>
22	Enclosure and Symmetry Processing	<input checked="" type="checkbox"/>
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

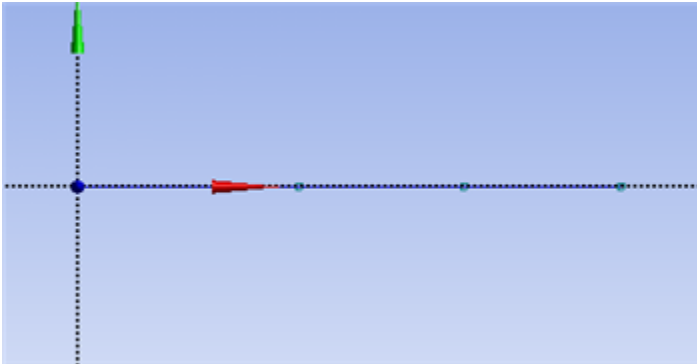
Like any other common CAD modeling practice, we start by creating a sketch.

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](#)

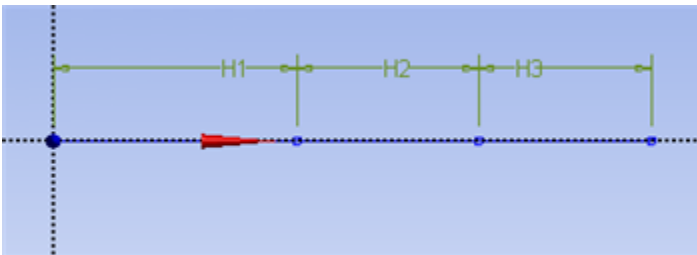
In the **Sketching Toolboxes**, select **Line**. In the **Graphics** window, create three rough lines from starting from the origin in the positive XY direction (Make sure that you see a letter P at the origin and at each connection between the lines. The letter P the geometry is constrained at the point.)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

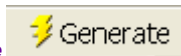
We are done with sketching.

Create Surface

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

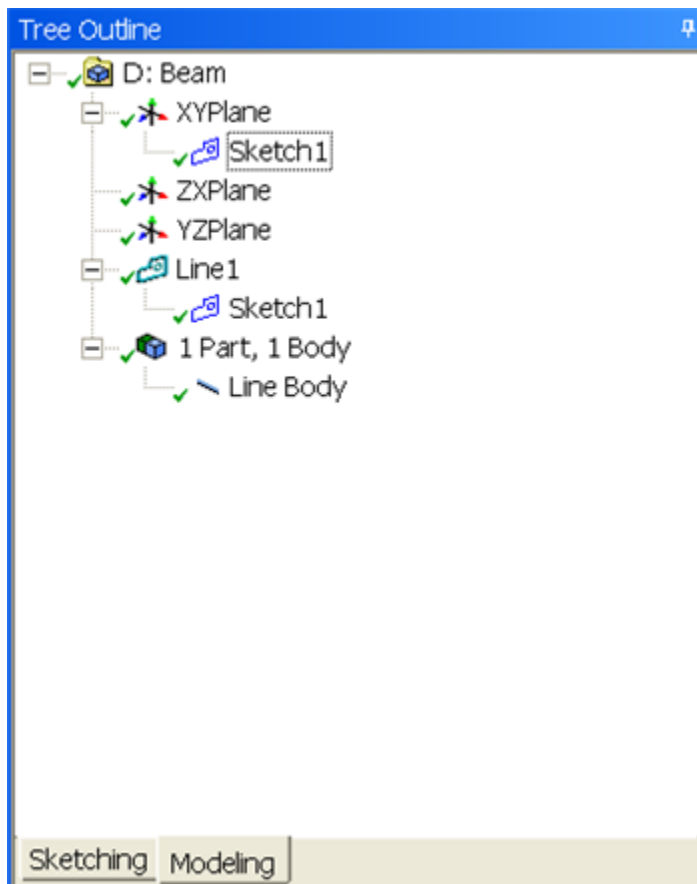
Concept > Lines From Sketches

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

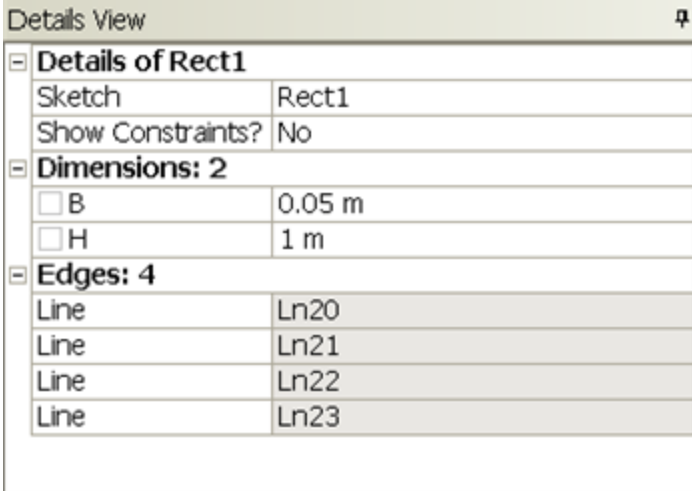
We will now add a cross section to the line body.

Concept > Cross Section > Rectangular

Under Details View, input value as follow:

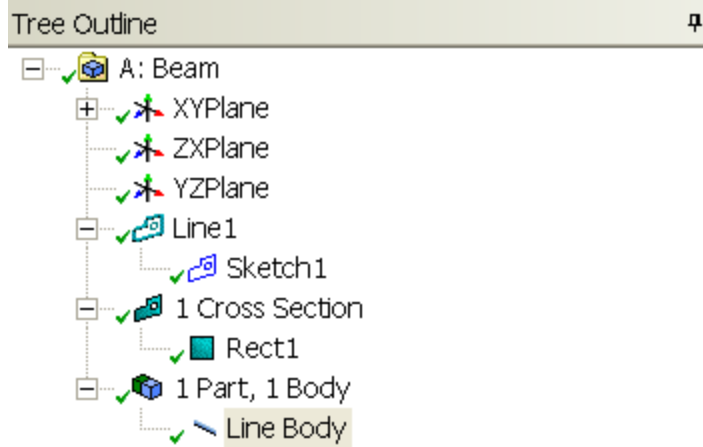
B - 0.05m

H - 1m



Outline > 1 Part, 1 Body > Line Body

And attach **Rect1** to **Cross Section** under **Details View**.



Sketching Modeling

Details View

Details of Line Body	
Body	Line Body
Faces	0
Edges	3
Vertices	4
Cross Section	Rect1
Offset Type	Centroid

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](#)