

# Old (4/1/2020) Flat Plate Boundary Layer - Geometry

Author: Rajesh Bhaskaran, Cornell University

[Problem Specification](#)

- [1. Pre-Analysis & Start-Up](#)
- [2. Geometry](#)
- [3. Mesh](#)
- [4. Model Setup](#)
- [5. Numerical Solution](#)
- [6. Post Processing](#)
- [7. Verification & Validation](#)
- [8. Part II: Flat Plate Convection](#)

[Frequently Asked Questions](#)

[Exercises](#)

[Comments](#)

## Geometry

The geometry is a rectangle of length 1 m and height 0.5 m. The following instructions are for creating the geometry in DesignModeler which is now a legacy tool. SpaceClaim is the recommended geometry engine. The steps for creating a rectangle using SpaceClaim are covered in Dr. Bhaskaran's free online course at [edX.org](http://edX.org). Go to section [4.3 Laminar Pipe Flow > Geometry](#). Don't forget to use the dimensions for this problem while creating the rectangle in SpaceClaim.

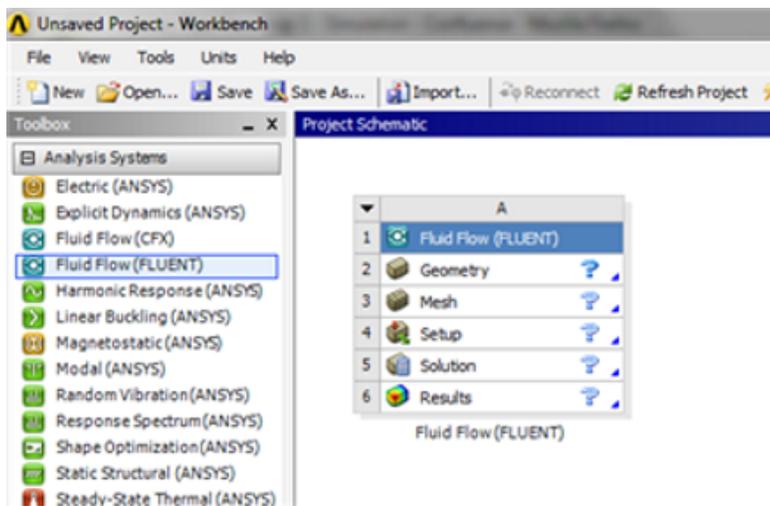
### Saving



It would be of best interest, to save the project at this point. Click on the "Save As.." button, , which is located on the top of the [Workbench Project Page](#). Save the project as "FlatPlateFlow" in your working directory. When you save in ANSYS a file and a folder will be created. For instance if you save as "FlatPlateFlow", a "FlatPlateFlow" file and a folder called "FlatPlateFlow\_files" will appear. In order to reopen the ANSYS files in the future you will need both the ".wbj" file and the folder. If you do not have BOTH, you will NOT be able to access your project.

### Fluid Flow(FLUENT) Project Selection

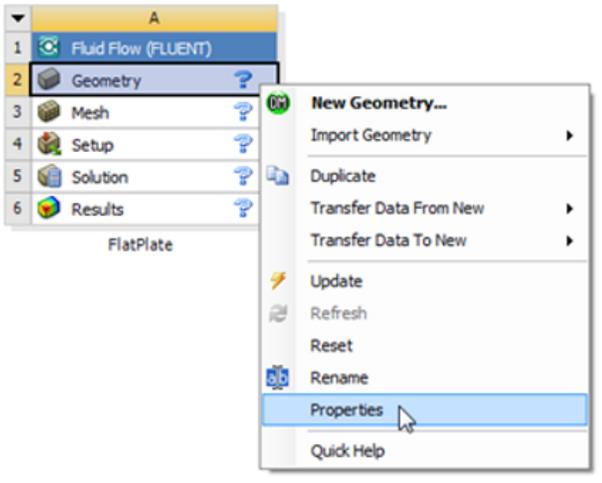
On the left hand side of the workbench window, you will see a toolbox full of various analysis systems. To the right, you see an empty work space. This is the place where you will organize your project. At the bottom of the window, you see messages from ANSYS. Left click (and hold) on [Fluid Flow \(FLUENT\)](#), and drag the icon into the empty space in the [Project Schematic](#). Your ANSYS window should now look comparable to the image below.



Since we selected Fluid Flow(FLUENT), each cell of the system corresponds to a step in the process of performing CFD analysis using FLUENT. Rename the project to "FlatPlate". We will work through each step from top down to obtain the solution to our problem.

### Analysis Type

In the **Project Schematic** of the Workbench window, right click on **Geometry** and select **Properties** , as shown below.



The properties menu will then appear to the right of the Workbench window. Under **Advance Geometry Options** , change the **Analysis Type** to 2D as shown in the image below.

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 type="checkbox"/>
18	Import Coordinate Systems <input type="checkbox"/>
19	Import WorkPoints <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

### Launch Design Modeler

In the **Project Schematic** , double 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

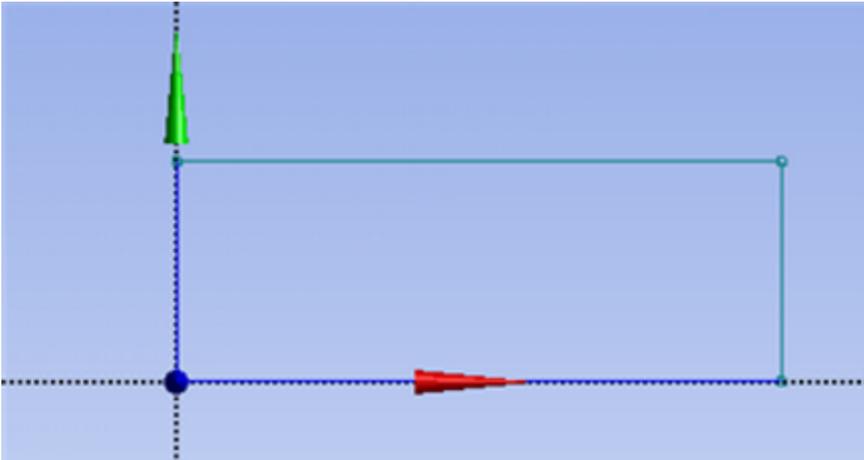
Start by creating a sketch on the **XYPlane** . Under **Tree Outline** , select **XYPlane** , then click on **Sketching** right before **Details View** . This will bring up the **Sketching Toolboxes** .

<https://confluence.cornell.edu/download/attachments/123547957/Select+Sketching+Toolboxes.gif>

Click on the **+Z** axis on the bottom right corner of the **Graphics** window to have a normal look of the XY Plane.

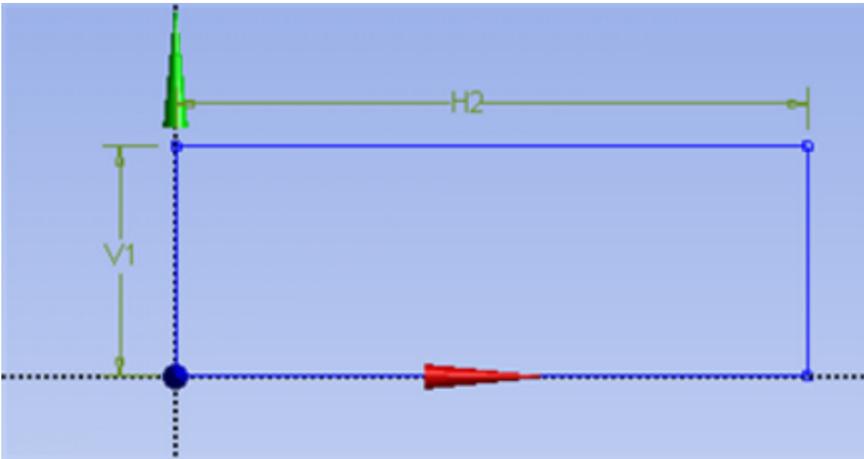
<https://confluence.cornell.edu/download/attachments/111221570/Select+Normal+View.gif>

In the Sketching toolboxes, select **Rectangle**. In the **Graphics** window, create a rough rectangle by clicking once on the origin and then by clicking once somewhere in the positive XY plane. (Make sure that you see a letter P at the origin before you click. The P implies that the cursor is directly over a point of intersection.) At this point you should have something comparable to the image below.



### Dimensions

At this point the rectangle will be properly dimensioned. Under **Sketching Toolboxes**, select **Dimensions** tab, use the default dimensioning tools. Dimension the geometry as shown in the following image.



[https://confluence.cornell.edu/download/attachments/90737913/DimRect\\_Full.png](https://confluence.cornell.edu/download/attachments/90737913/DimRect_Full.png)

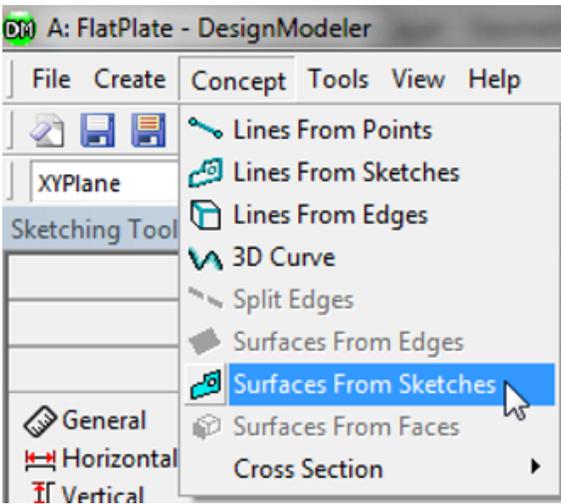
Under the **Details View** table (located in the lower left corner), set  $V1=0.5\text{m}$  and set  $H2=1\text{m}$ , as shown in the image below.

Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 2	
<input type="checkbox"/> H2	1 m
<input type="checkbox"/> V1	0.5 m
Edges: 4	
Line	Ln7
Line	Ln8
Line	Ln9
Line	Ln10

[https://confluence.cornell.edu/download/attachments/90737913/RectDims\\_Full.png](https://confluence.cornell.edu/download/attachments/90737913/RectDims_Full.png)

### Surface Body Creation

In order to create the surface body, first (Click )Concept > Surface From Sketches as shown in the image below.



This will create a new surface SurfaceSK1 . Under Details View , select Sketch1 as Base Objects . Finally, click Generate to generate the surface.

At this point, you can close the Design Modeler and go back to Workbench Project Page . Save your work thus far in the Workbench Project Page.

### Go to Step 3: Mesh

[Go to all FLUENT Learning Modules](#)