

Forced Convection - Geometry

Author: Rajesh Bhaskaran & Yong Sheng Khoo, Cornell University

Problem Specification

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

Exercises

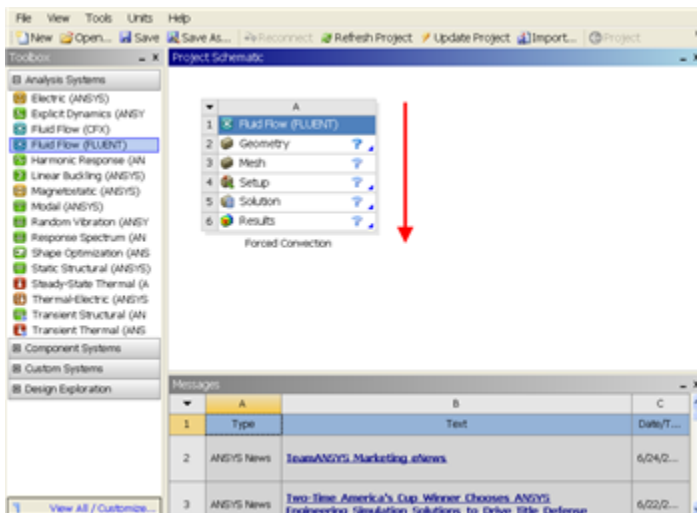
Comments

Geometry

Note to Cornell students enrolled in MAE 4272: You can skip the geometry and mesh steps. Download the mesh [here](#). Unzip the downloaded file. After unzipping, you should see a file called *pipe_flow.wbpj* and a folder called *pipe_flow_files*. Double click on *pipe_flow.wbpj* and then skip to **Step 4: Setup (Physics)**.

Since our problem involves fluid flow, we will select the FLUENT component on the left panel.

Left click (and hold) on **Fluid Flow (FLUENT)**, and drag the icon to the empty space in the **Project Schematic**. Here's what you get:



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 Forced Convection.

We will work through each step from top down to obtain the solution to our problem.

In the **Project Schematic** of the Workbench window, right click on **Geometry** and select **Properties**. You will see the properties menu on the right of the Workbench window. Under **Advanced Geometry Options**, change the **Analysis Type** to 2D.

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

In the **Project Schematic** , right click on **Geometry** and select *Create New DesignModeler Geometry* to start preparing the geometry. To make best use of screen real estate, move the windows around and resize them so that you approximate

[this screen arrangement](#)

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

The geometry is a rectangle of length $L = 6.045m$ and height $R = 2.94 \times 10^{-2} m$.

Creating a Sketch



For users of ANSYS 15.0, please check [this link](#) for procedures for turning on the Auto Constraint feature before creating sketches in **DesignModeler**.

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** .

[Click Here for Select Sketching Toolboxes Demo](#)

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

[Click Here for Select Normal View Demo](#)

In the Sketching toolboxes, select **Rectangle** . Hover the cursor near the origin until you see a letter **P** . This means the cursor is coincident with a point, in this case the **origin** . Then drag the cursor to draw a rectangle.

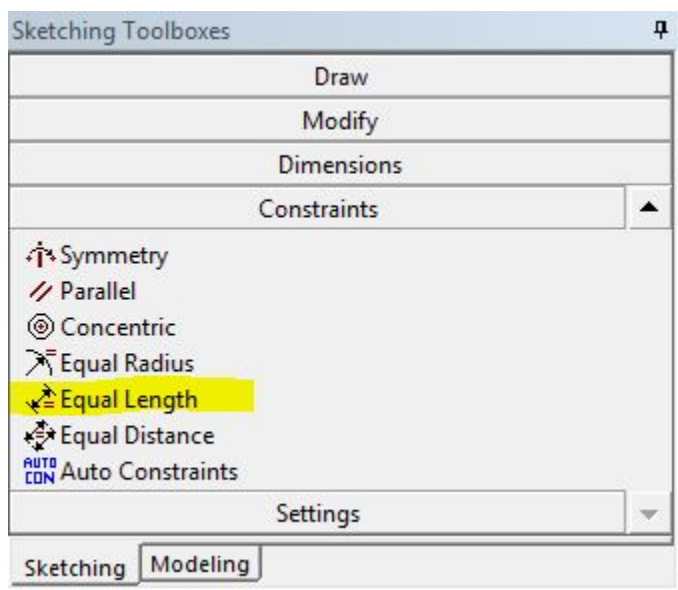
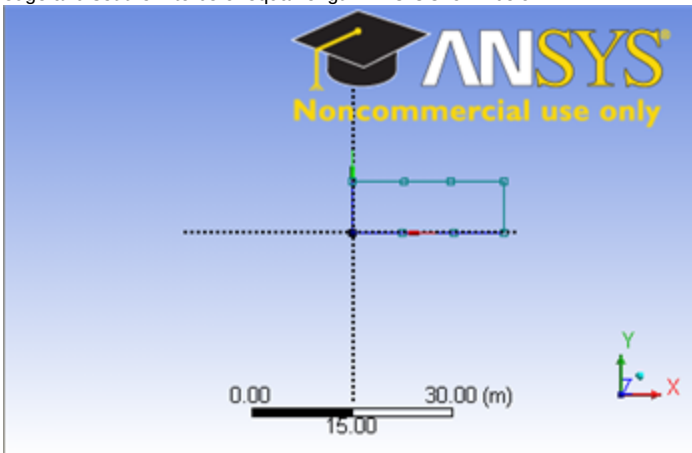


Note: You do not have to worry about dimensions for now, we will specify them properly in the later step.

Modify the Sketch

Since we have a heated section in the middle of the pipe, we need to split the geometry appropriately. Click **Modify** tab and select **Split**. Select two points at the top of the rectangle, where there will be a heated section. Then select two points at the bottom of the rectangle.

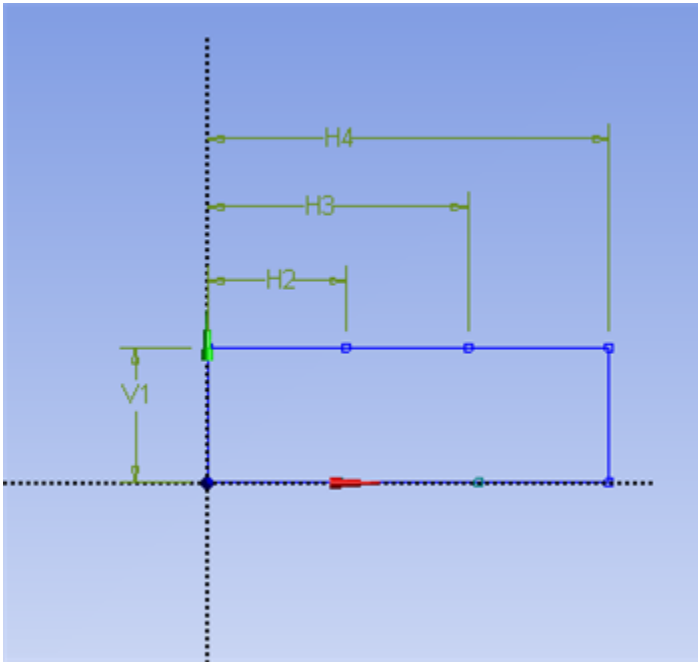
Now we can constraint the lower rectangle with the top of the rectangle. Click **Constraints** tab, select **Equal Length**. Click the appropriate top and bottom edge and set them to be of equal length. This is shown below:



Dimensions

Under **Sketching Toolboxes** , select **Dimensions** tab, use the default dimensioning tools. Then click on the lines and drag upwards or sideways as the case may be to place the dimensions (V1, H2, H3, H4). **Note:** For horizontal dimensioning (shown in H2, H3 and H4), click first on the horizontal dimension tab under the dimensions tab and then click (turns yellow) on the end points of the split section lines (H2, H3 and H4). Then click on any point on the y-axis and drag up. For the vertical dimensioning (V1), click on the vertical dimension tab under the dimensions tab. Then click on the any point on the x-axis then click on V1 (turns yellow). Then drag V1 to the left side.

Dimensioning of the geometry is shown below:



Under **Details View** in the lower left corner, input the values for the appropriate dimensions. Then hit enter each time each dimension is entered.

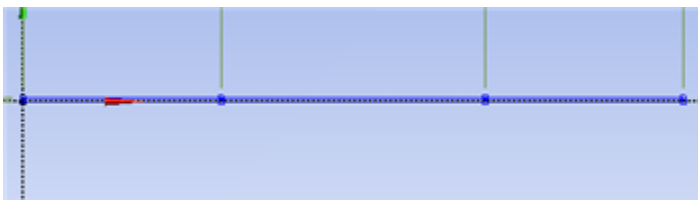
V1: 0.0294 m

H2: 1.83 m

H3: 4.27 m

H4: 6.045 m

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



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

Then click on the **Concept** tab in the **DesignModeler** window, then click on **Surface from sketches** .

This will create a new surface **SurfaceSK1** . Under the **Tree Outline** , click on the **X-Y Plane** and select **Sketch1** as **Base Objects** and under **Details View** , click **Apply** . Finally click **Generate** to generate the surface.

[Click Here for Create Surface Demo](#)

Remember to save the project: File > Save Project

You can close the **Design Modeler** and go back to **Workbench** .

Go to Step 3: Mesh

[Go to all FLUENT Learning Modules](#)