

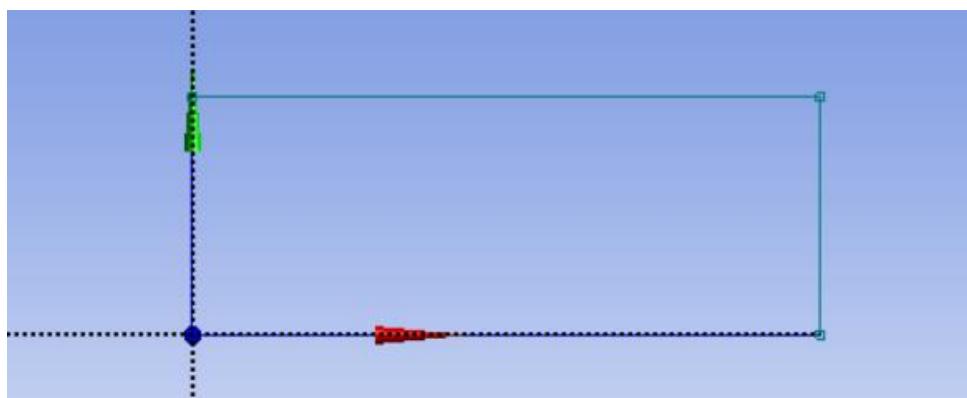
## **EXPERIMENT 1 - FLOW OVER FLAT PLATE**

**Aim:** Obtain the velocity and pressure distribution with boundary layer for laminar flow over a flat plate

### **Procedure/ Steps:**

#### **A. Geometry**

1. Create a sketch on the XY Plane. Under Tree Outline, select XY Plane, then click on Sketching.
2. Click on the +Z axis on the bottom right corner of the Graphics window to have a normal look of the XY Plane.
3. In the Sketching toolboxes, select Rectangle.
4. 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.



5. Under Sketching Toolboxes, select Dimensions tab, use the default dimensioning tools to give dimensions to the sketch.
6. Under the Details View table (located in the lower left corner), set  $V1=0.5\text{m}$  and set  $H2=1\text{m}$



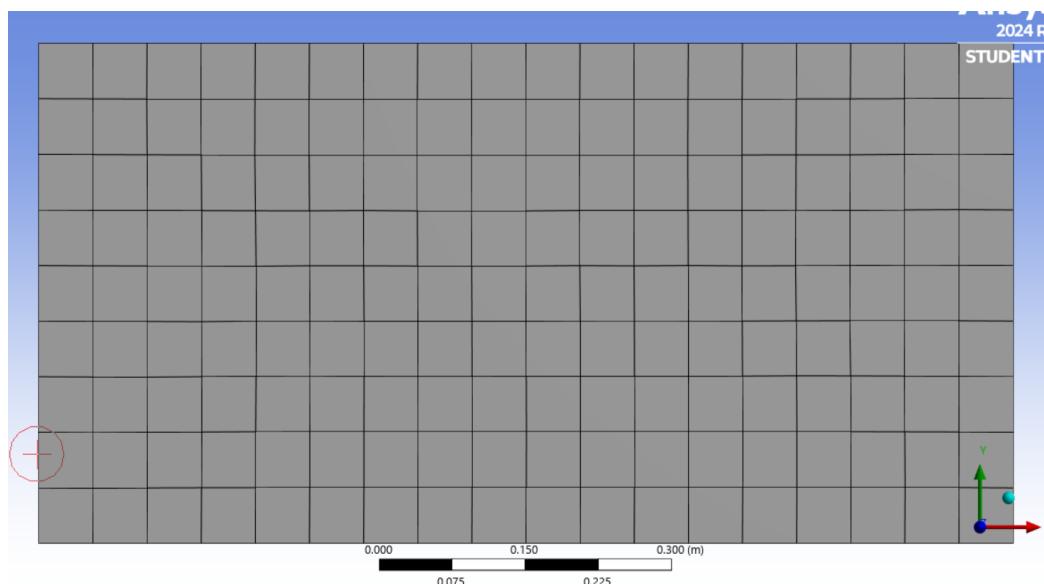
7. In order to create the surface body, first (Click) Concept > Surface from Sketches.
8. This will create a new surface SurfaceSK1. Under Details View, select Sketch1 as Base Objects. Finally, click Generate to generate the surface.



9. Close the Geometry Design Modeler.

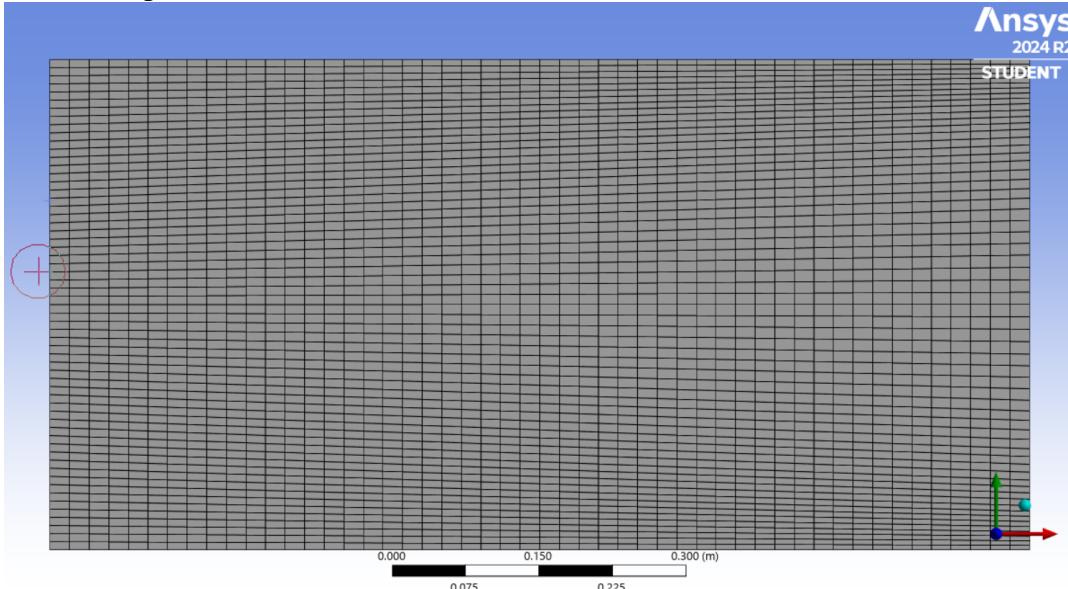
## B. Meshing

1. Double click on meshing to open the meshing.
2. To generate the default mesh first click on Mesh, then click on Update.

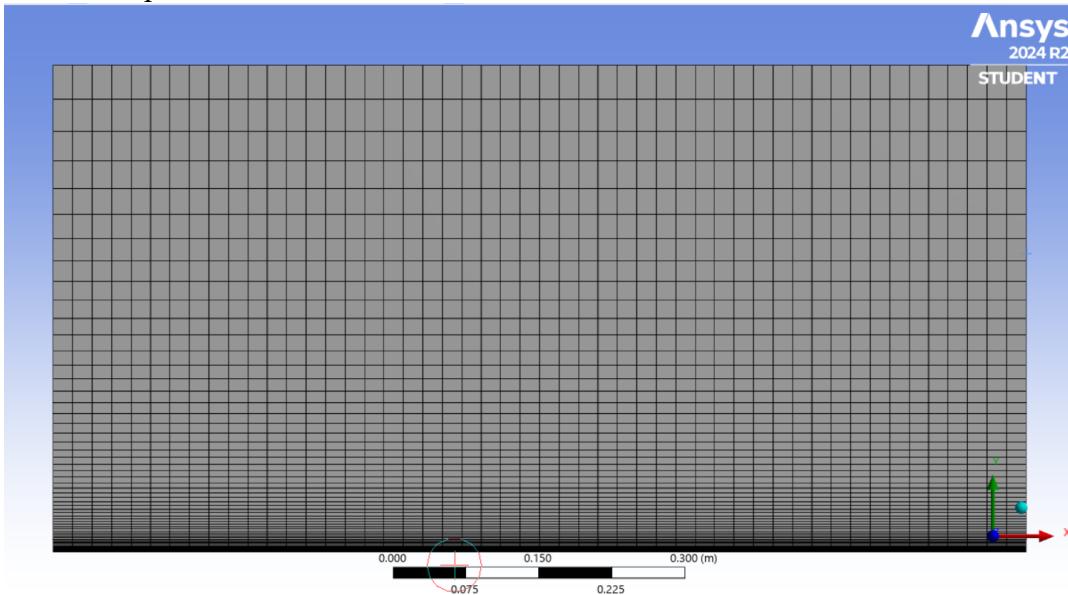


3. Click on Mesh Control > Mapped Face Meshing.
4. In the Details View of Mapped Face meshing Click on geometry, then click on the rectangle and then click on Apply.
5. Click on Generate to Generate the Mesh.
6. Click on Mesh Control > Sizing.
7. In the Details View of Edge Sizing Click on geometry.
8. Use the Edge Selection Tool, hold control and click on the bottom and top edges of the rectangle.
9. Click on Apply
10. Set the Type to Number of Divisions.
11. Set the Number of Divisions to 50.
12. Set the Behaviour to Hard.

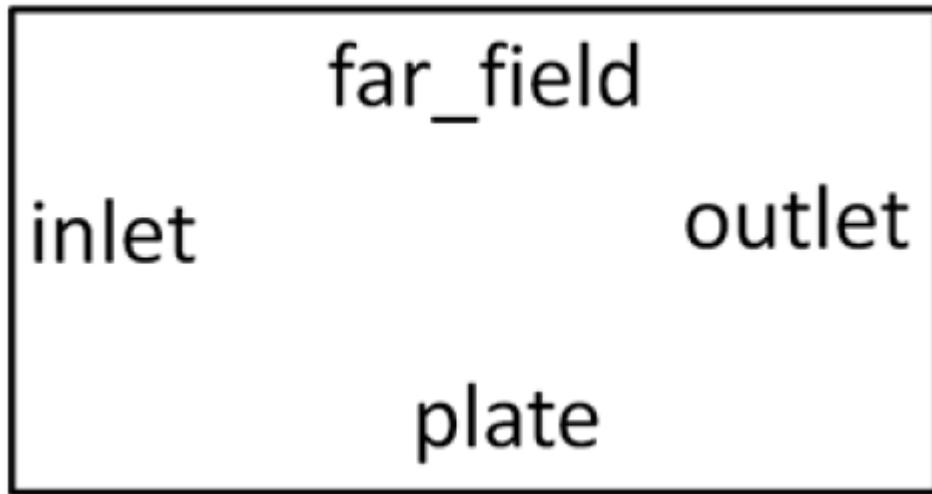
13. Follow the same procedure as for the edge sizing in the x direction, except select only the left side instead of the top and bottom and set the number of divisions to 60.
14. Click on Update to Generate the Mesh.



15. In the Details View expand statistics and the mesh should have 3000 elements.
16. Click on Edge Sizing 2, set the Bias Type to the first Option and give a Bias Factor of 70.
17. Now apply the edge sizing to the right side of the rectangle. Set the number of divisions as 60. Set Behaviour as Hard.
18. Select the Bias type to the Second option and Bias Factor to 70.
19. Click on Update to Generate a new Mesh.



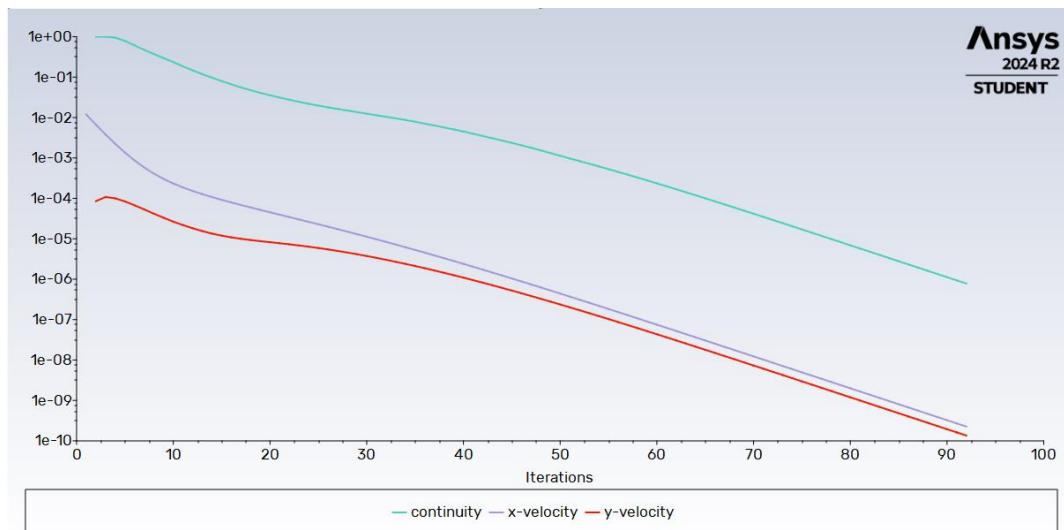
20. Using the Edge Selection Filter, click on each side of the rectangle and then click on create named selection and name the edges as the figure shown below.



21. Close the Mesher Window. Right Click the Mesh and click on Update.

### C. Fluent Operations

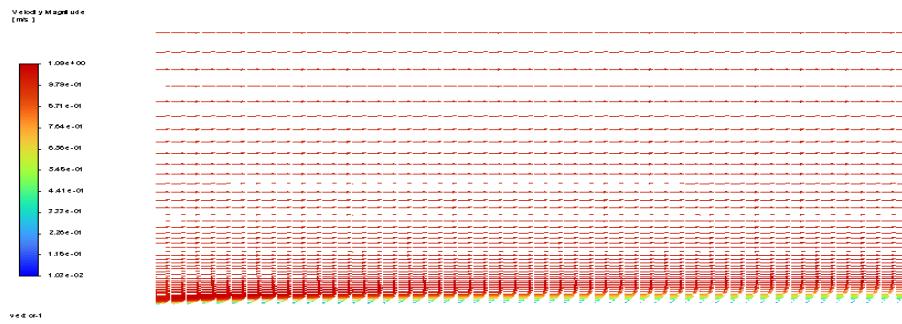
1. Under display select >>option-edges>>edge type –all>> surfaces-select all>>display>>close
2. Select models>>viscous laminar-and all other off
3. Select Materials > Fluid > Create/Edit set the Density to 1kg/m<sup>3</sup> (constant) and set the Viscosity to 1e-4 kg/(ms) (constant)
4. Define boundary conditions
  1. Inlet>>edit>>Velocity Specification Method - Components, >> X-Velocity (m/s) - 1m/s
  2. Outlet >> Edit>>pressure-outlet
    3. Plate>> wall
    4. Far field>>symmetry
  5. Solution method >>momentum-Second Order Upwind>>rest all as it is
  6. Monitors>>Residuals >> Edit>>print and plot>>Convergence Criterion for continuity, xvelocity, and y-velocity, all to 1e-6
  7. Solution Initialization >>Compute from >> inlet>>initialize
  8. Run calculation>>Number of Iterations – 1000>> calculate
  9. The Residuals curve was obtained



## D. Results

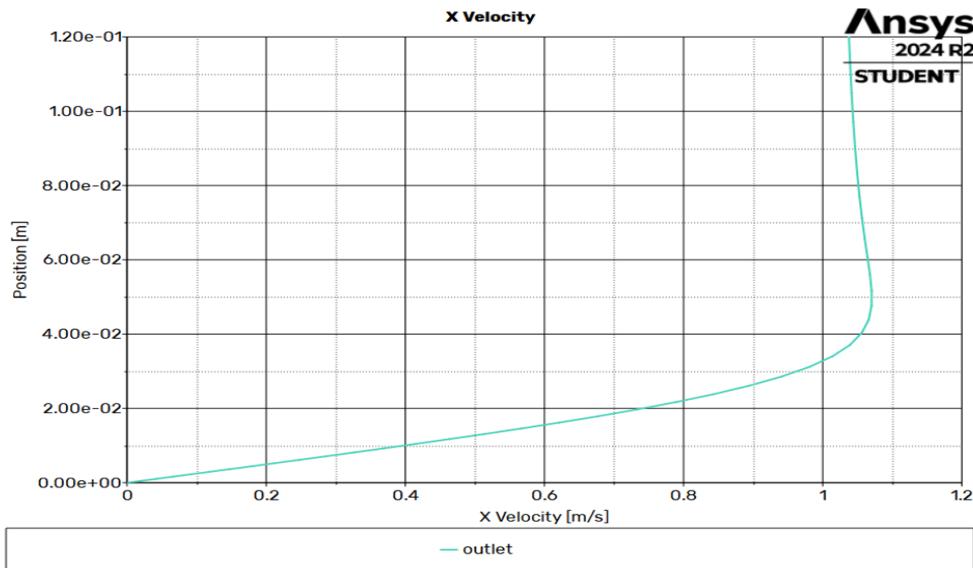
### 1. Velocity vector

Graphics & Animations >> Vectors >> Display  **velocity vectors profile is obtained**



### 2. Outlet velocity profile

Results >> Plots >> XY Plot >> Set Up >> Solution XY Plot > Position on Y -X is set to 0 and Y is set to 1 >> X Axis Function- Velocity >> Y Axis Function-direction vector >> surfaces-outlet >> Plot → **Variation of x velocity vs position is obtained**



### 3. Pressure coefficients

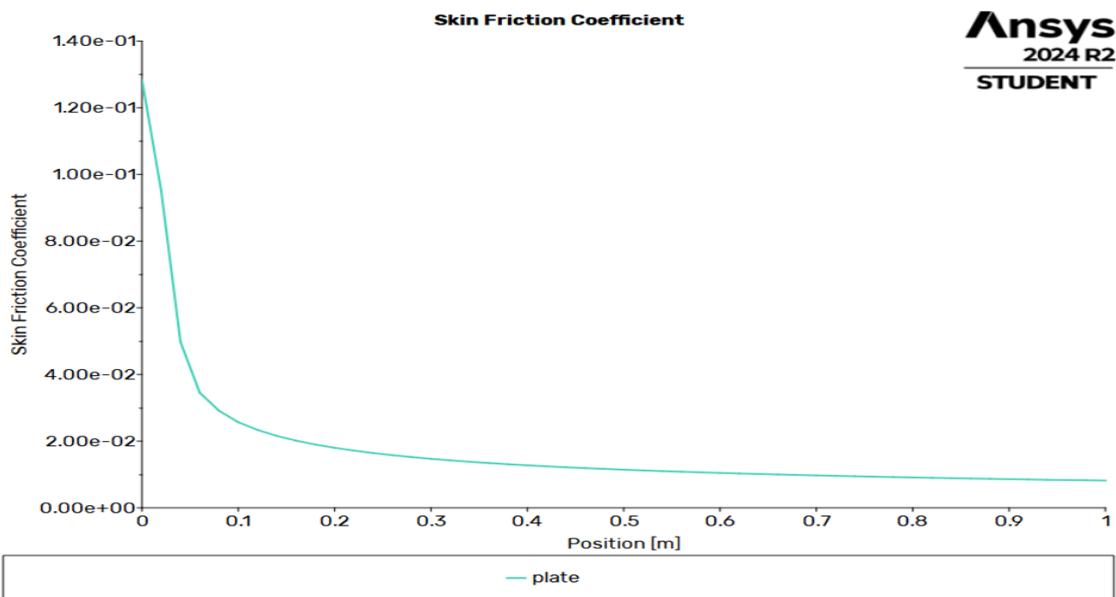
Reference Values >> Compute from >> inlet

Graphics and Animations >> Contours >> Contours of - Pressure... Pressure Coefficient >> options >> Filled ... Levels – 90 → **Pressure contours are obtained**



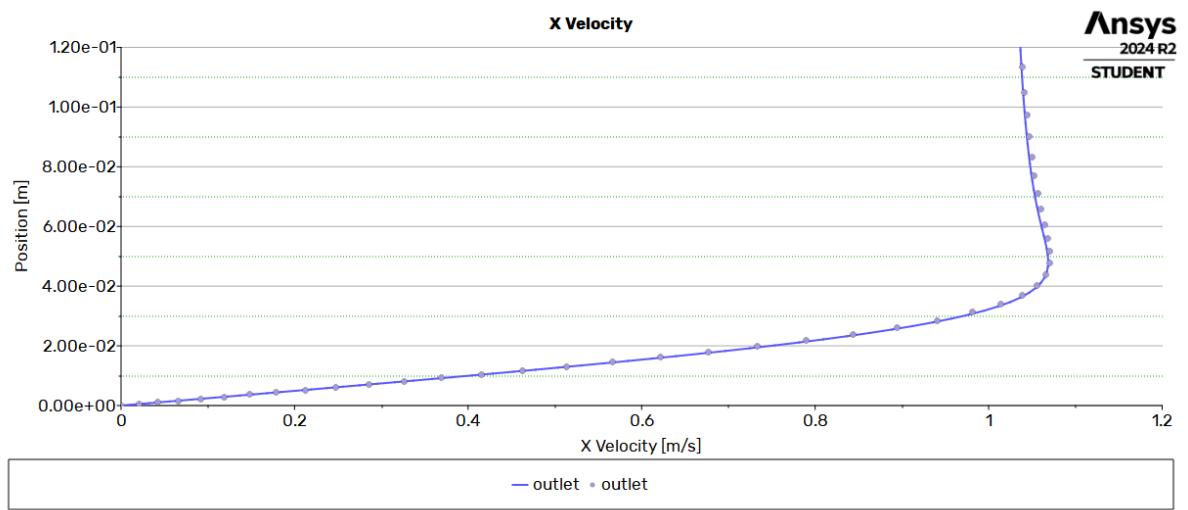
#### 4. Skin friction coefficient

Plots >>XY Plot>> Solution XY Plot >>options-node values..Position on x axis>>Plot direction>>set X to 1 and set Y to 0>> Y Axis Function - Wall Fluxes...Skin Friction Coefficient>> X axis function-direction vector>> surfaces - plate → **Plot of skin friction factor is obtained**



#### 5. Drag

Results >>Report>> Forces>>Print□ Drag coefficient values is obtained as:0.0083



## **EXPERIMENT 2: LAMINAR PIPE FLOW**

**Aim:** Obtain the velocity and pressure distribution with boundary layer for laminar flow in pipe.

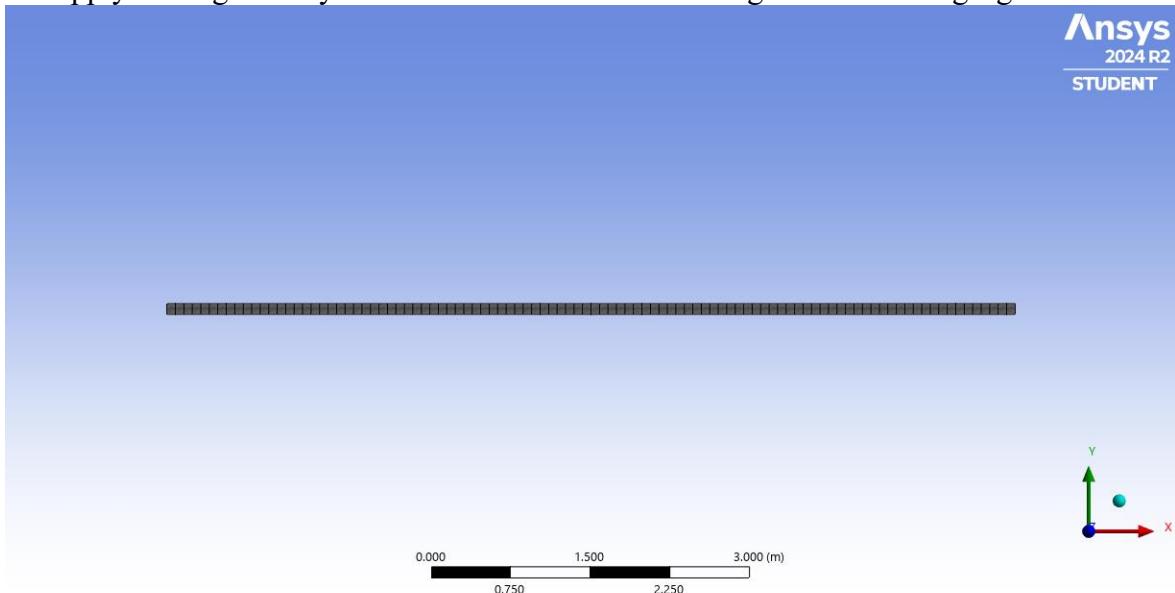
### **Procedure:**

#### **A. Geometry**

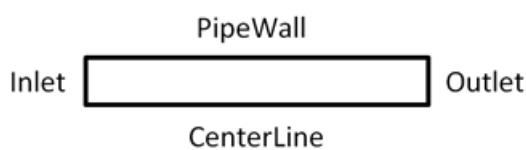
1. Sketch a rectangle of height 8 m and width 0.1 m and created surface from sketch

#### **B. Meshing**

1. Mesh the geometry with meshed with 500 elements. That is, the field will be divided into 100 elements in the x direction and 5 elements in the y direction. Use mapped face meshing and edge sizing for the respective faces.
2. Apply biasing to the y face with a bias factor of 70 to get the following figure



3. Provide name selection as below



#### **C. Fluent operations**

1. Under display select >>option-edges>>edge type -all>> surfaces-selectall>>display>>close
2. Select models>>viscous laminar-and all other off
3. Select Materials > Fluid > Create/Edit set the Density to 1kg/m^3 (constant) and set the Viscosity to 2e-3 kg/(ms) (constant)
4. Define boundary conditions
  1. Inlet>>edit>>Velocity Specification Method - Components, >> X-Velocity (m/s) - 1m/s
  2. Outlet >> Edit>>pressure-outlet
  3. Centreline>>Type>>Axis

4. Pipe wall>> wall Far field>>asymmetric
5. Solution method >>momentum-Second Order Upwind>>rest all as it is
6. Monitors>>Residuals >> Edit>>print and plot>>Convergence Criterion for continuity, xvelocity, and y-velocity, all to 1e-6
7. Solution Initialization >>Compute from >> inlet>>initialize
8. Run calculation>>Number of Iterations – 1000>> calculate

## D. Results

- a) Velocity vector

Graphics & Animations >> Vectors >> Display  **velocity vectors profile is obtained [Figure 1]**

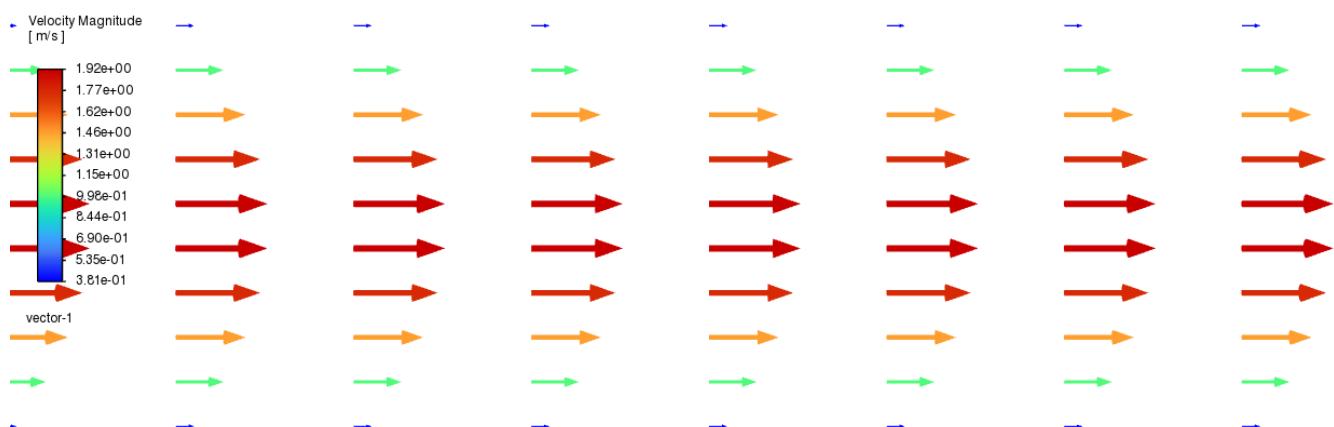
- b) Outlet velocity profile

Results >> Plots >> XY Plot >> Set Up>>Solution XY Plot > Position on Y -X is set to 0 and Y is set to 1>> X Axis Function- Velocity>> Y Axis Function-direction vector>>surfaces-outlet>> Plot → **Variation of x velocity vs position is obtained[Figure 2]**

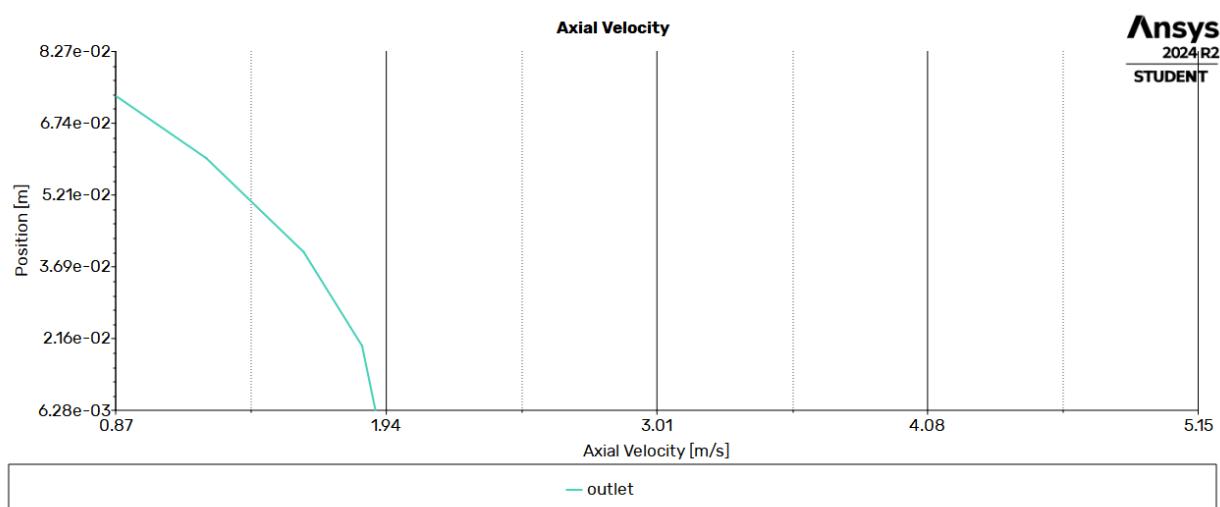
- c) Skin friction coefficient

Plots >>XY Plot>> Solution XY Plot >>options-node values..Position on x axis>>Plot direction>>set X to 1 and set Y to 0>> Y Axis Function - Wall Fluxes...Skin Friction Coefficient>> X axis function-direction vector>> surfaces - plate → **Plot of skin friction factor is obtained [Figure 4]**

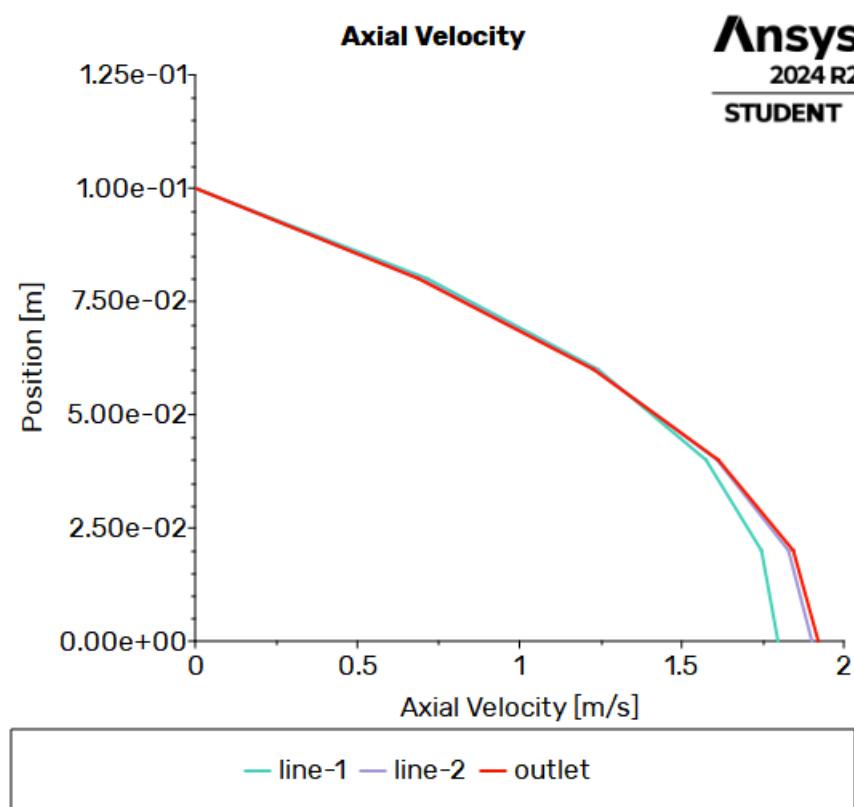
**Ansys**  
2024 R2  
STUDENT

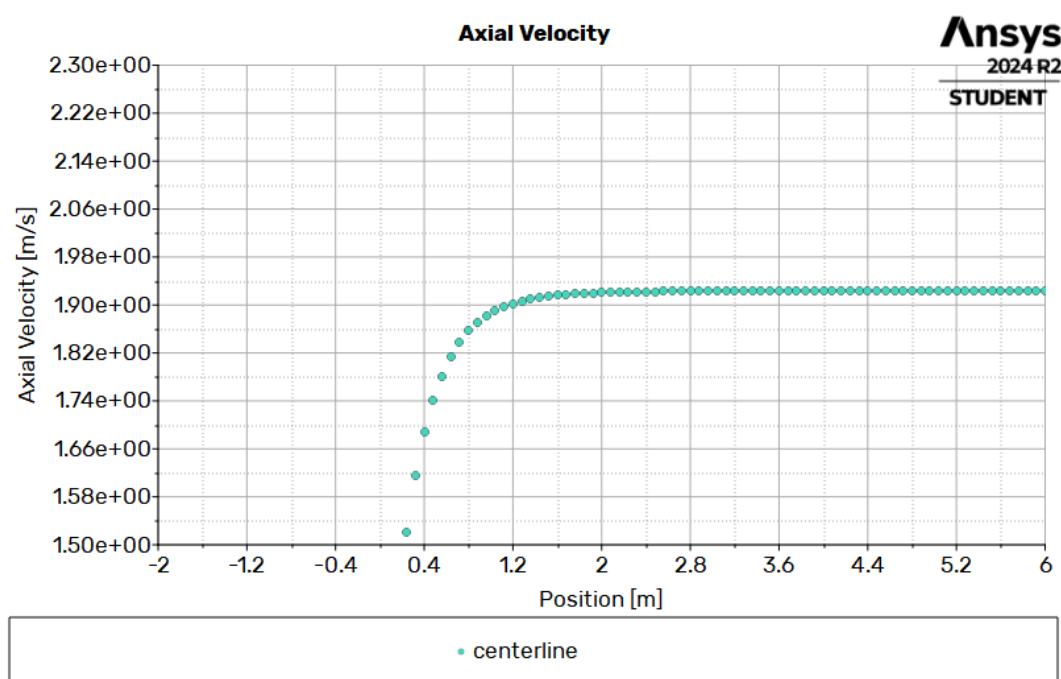
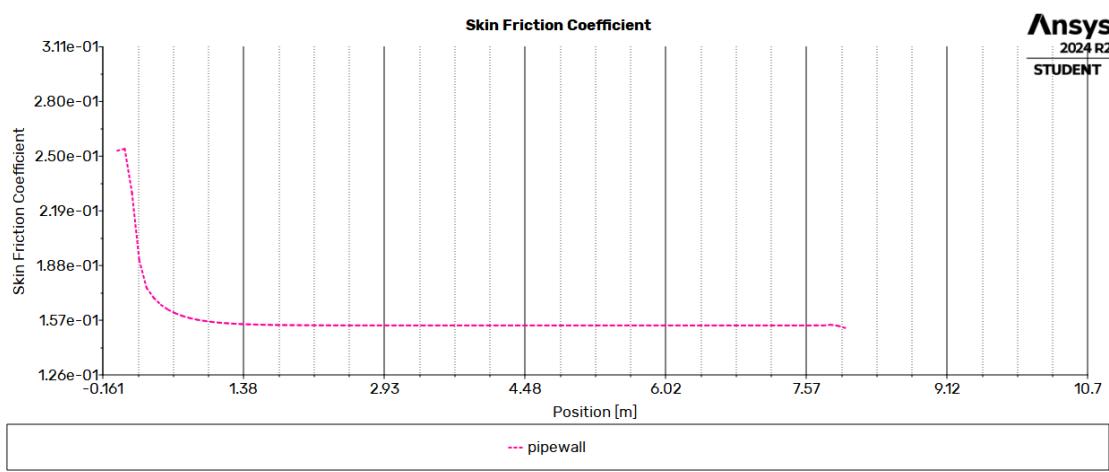


**Figure 1: Velocity vector field**



**Figure 2: Variation of x velocity vs position**





**Figure 3: Plot for skin friction**

## **EXPERIMENT3: TURBULENT PIPE EFLOW**

**Aim:** Obtain the velocity and pressure distribution with boundary layer for turbulent flow in pipe.

### **A. Geometry**

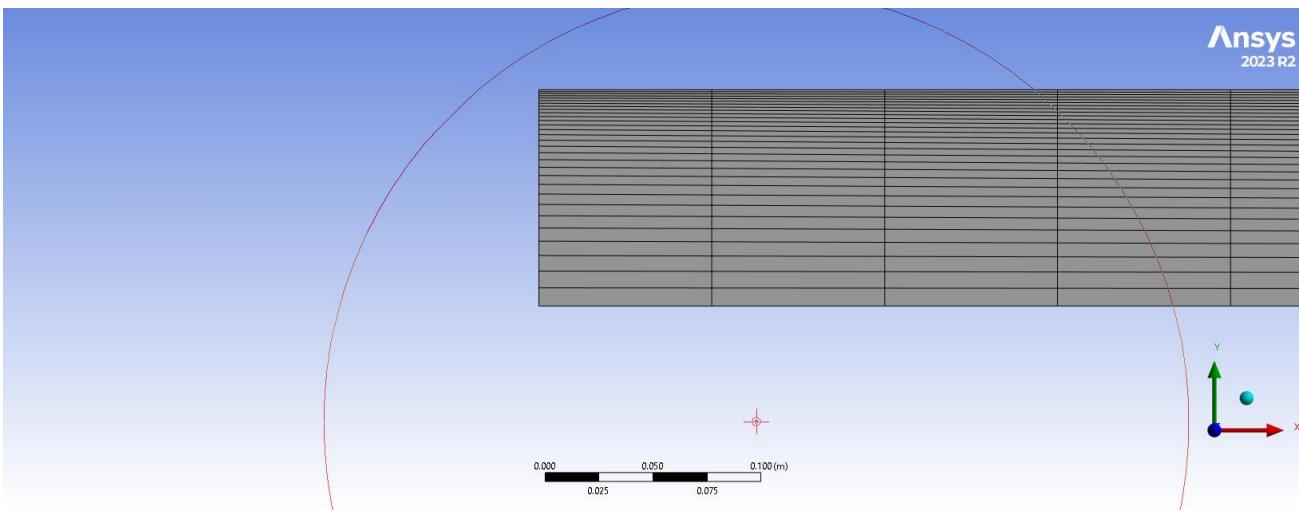
1. Sketch a rectangle of height 8 m and width 0.1 m and created surface from sketch.

### **B. Mesh**

1. Apply Mapped Face Meshing to the body.
2. First, (Click) Mesh Control > Sizing as shown below. Now, the geometry and the number of divisions need to be specified. First (Click) Edge Selection Filter. Then hold down the "Control" button and then click the bottom and top edge of the rectangle. Both sides should highlight green. Next, hit Apply under the Details of Sizing table. Now change Type to Number of Divisions and set Number of Divisions to 100.
3. Click Mesh Control > Sizing. Using the edge selection tool, highlight the inlet (left end) of the pipe and click Apply next to Geometry. As in the Laminar Pipe Flow tutorial, change Type to Number of Divisions, and enter 30. Change Behavior to Hard. Set Capture Capture to No. Under Bias Type, select the second option, - — -----. Enter a Bias Factor of 30.
4. Apply an edge sizing to the outlet, the right end of the pipe. We'll use 30 divisions, with a bias factor of 10 and with the smaller divisions at the top, near the wall. This time, when selecting Bias Type, choose the first option, ---- — -- -. Set Capture Capture to No. Change Behavior to Hard.
5. Next, create the following named selections:

Edge Position	Name	Type
Left	inlet	VELOCITY_INLET
Right	outlet	PRESSURE_OUTLET

Top	wall	WALL
Bottom	centerline	AXIS

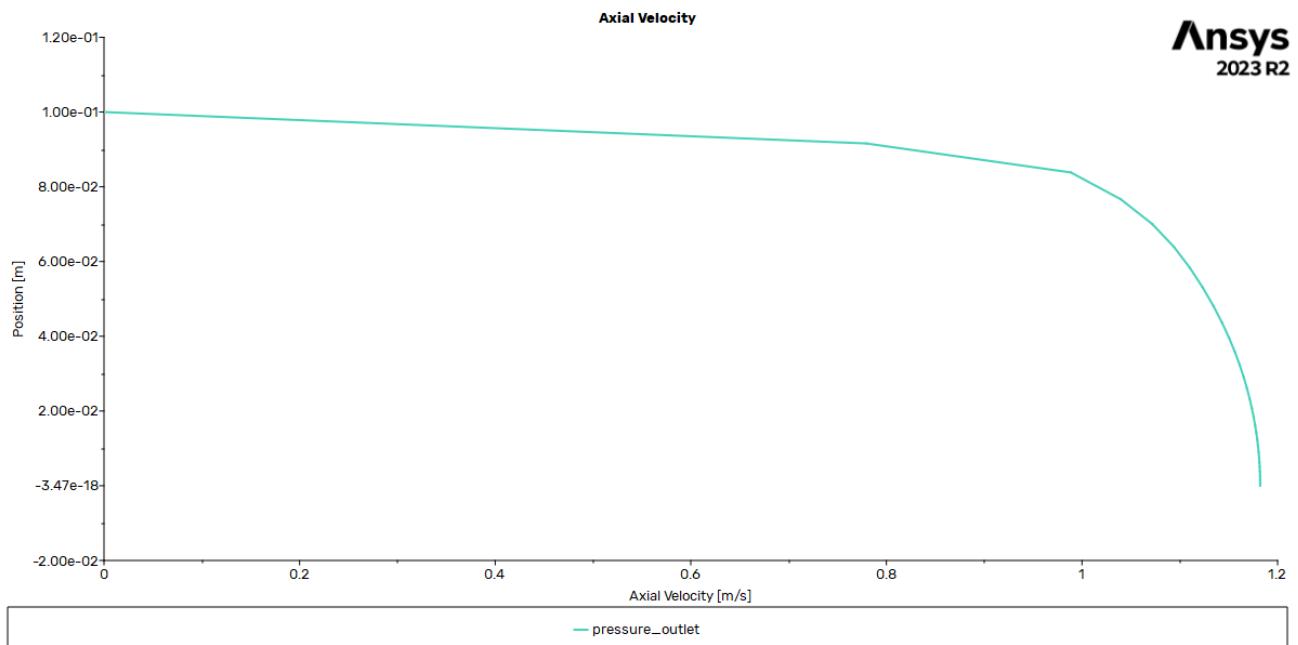


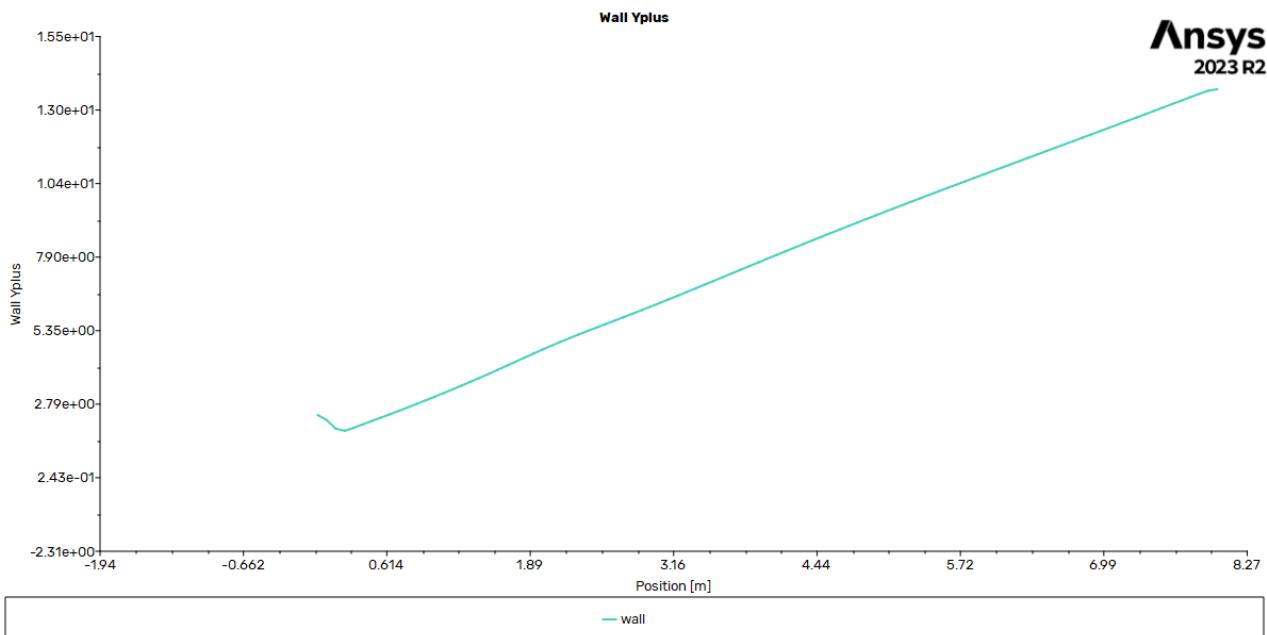
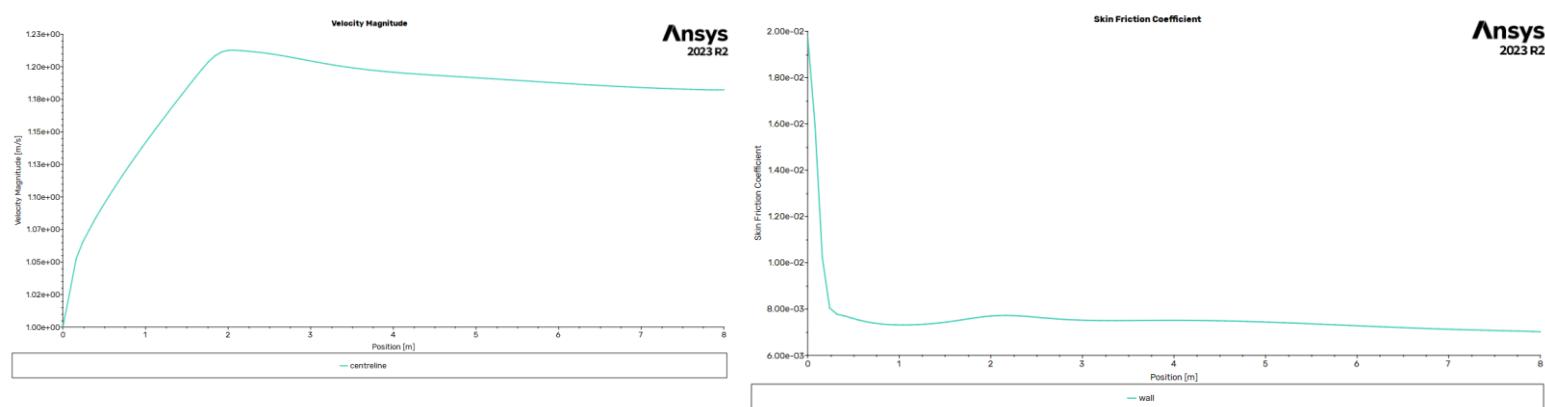
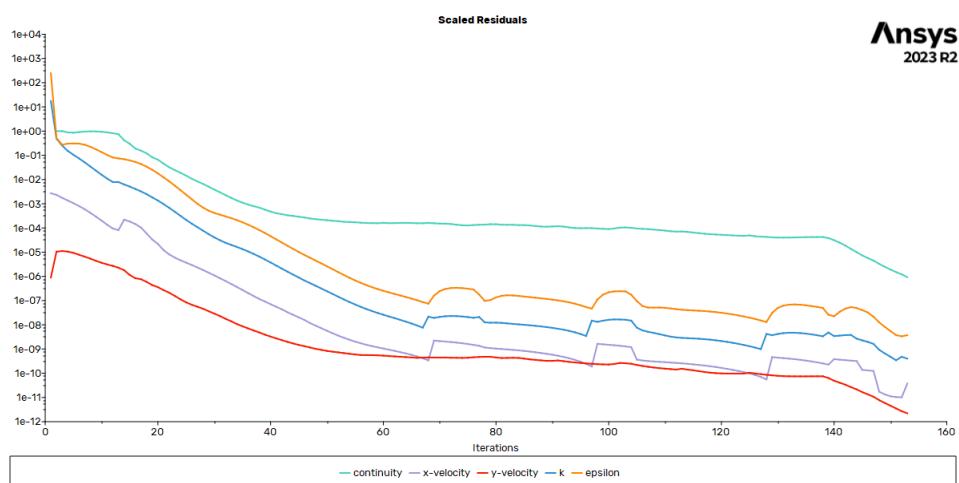
## C. Fluent Operations

1. Make sure the **Double Precision** option is selected, and if using a computer with multiple cores, you can select the **Parallel** option, and choose the number of processors to use. Once Fluent has opened, select Problem Setup > General.
2. Problem Setup > General > Solver. Choose **Axisymmetric** under **2D Space**. We'll use the defaults of Pressure-Based Type, Steady flow and Absolute Velocity Formulation. Problem Setup > Models > Energy... The energy equation can be turned off since this is an incompressible flow and we are not interested in the temperature. Make sure **Energy - Off** appears. Problem Setup > Models > Viscous – Laminar. Click **Edit...** and choose **k-epsilon (2eqn)**. Notice that the window expands and additional options are displayed on choosing the k-epsilon turbulence model. Under **Near-Wall Treatment**, pick **Enhanced Wall Treatment** so that we may get a more accurate result. Click **OK**.
3. Double click on **air** and change **Density** to  $1.0 \text{ kg/m}^3$  and **Viscosity** to  $2e-5 \text{ kg/(m*s)}$ . These are the values in the Problem Specification. We'll take both as constant. Click **Change/Create** and close the window.
4. Problem Setup > Boundary conditions > Operating Conditions...Recall that for all flows, FLUENT uses the gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the gauge pressure. We'll use the default value of 1 atm (101,325 Pa) as the **Operating Pressure**. Click **Cancel** to leave the default in place.
5. Problem Setup > Boundary conditions. We don't need to set any parameters for the **wall** zone. FLUENT will automatically detect that this location should be set as a wall based on its name. Verify this by selecting that zone and looking at its type in the drop down menu. Next, let's look at the centerline. Since we are solving an axisymmetric problem, we will set the centerline as the axis. Set **centerline** to axis boundary type, using the drop down menu.
6. Choose inlet and click on **Edit....** Change the **Velocity Specification Method** to **Magnitude, Normal to Boundary**. Enter  $1 \text{ m/s}$  for **Velocity Magnitude**. This indicates that the fluid is coming in normal to the inlet at the rate of 1 meter per second. Select **Intensity** and **Hydraulic Diameter** next to the **Turbulence Specification Method**. Then enter 1% for **Turbulence Intensity** and 0.2m for **Hydraulic Diameter**. Click **OK** to set the boundary conditions for the inlet.
7. Problem Setup > Reference Values. Select **Compute from > inlet**.

8. Solution > Solution Methods Change the Discretization for Momentum, Turbulence Kinetic Energy and Turbulence Dissipation Rate equations to Second Order Upwind.
9. Solution > Monitors > Residuals, Statistic and Force Monitors. Double click on Residuals. Notice that Convergence Criterion has to be set for the k and epsilon equations in addition to the three equations in the last tutorial. Set the Convergence Criterion to be 1e-06 for all five equations being solved.
10. Solution > Solution Initialization. In the Solution Initialization menu that comes up, choose inlet under Compute From. The Axial Velocity for all cells will be set to 1 m/s, the Radial Velocity to 0 m/s and the Gauge Pressure to 0 Pa. The Turbulence Kinetic Energy and Dissipation Rate (scroll down to see it) values are set from the prescribed values for the Turbulence Intensity and Hydraulic Diameter at the inlet.
11. Solution > Run Calculation > Data File Quantities. Under Additional Quantities, select Skin Friction Coefficient.
12. Solution > Run Calculation In the Iterate menu that comes up, change the Number of Iterations to 700. Click Calculate. The solution converges in a total of about 220 iterations. Plot the necessary contours, charts and vectors.

## SCREENSHOTS FROM FILE



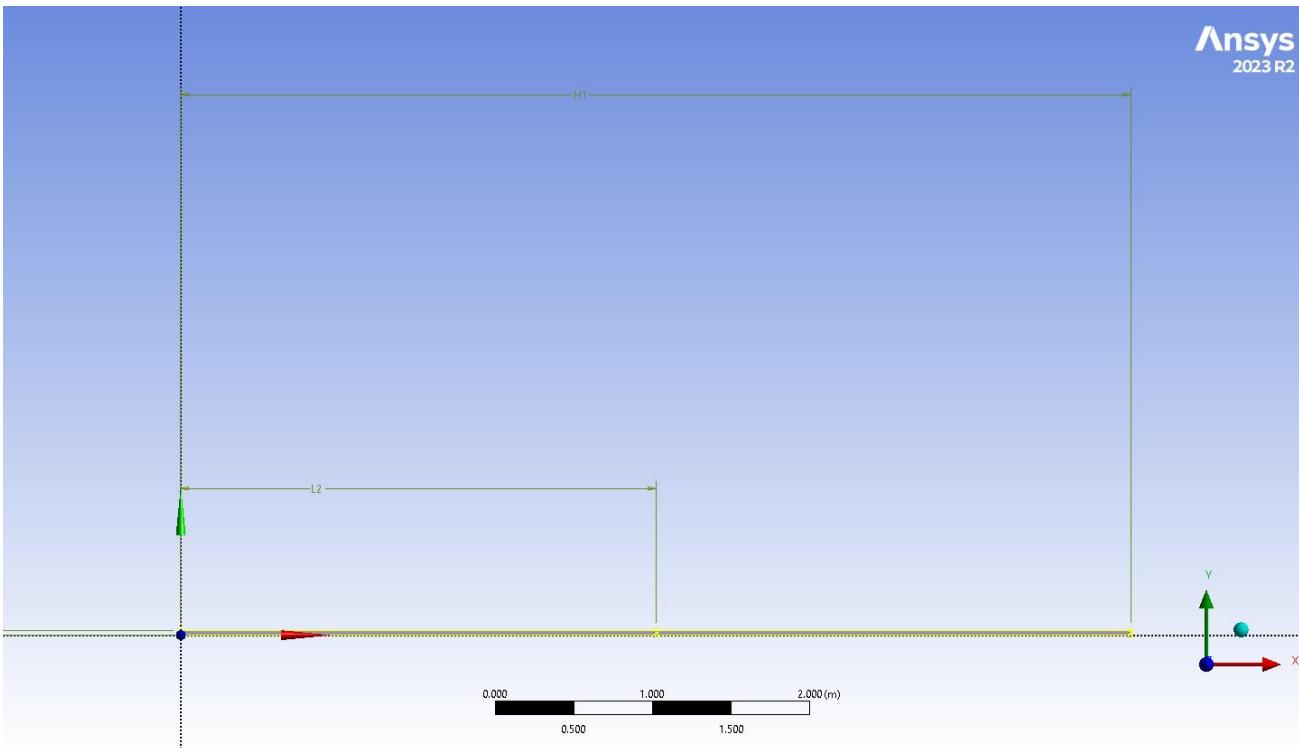


## **EXPERIMENT 4: LAMINAR FORCED CONVECTION**

**AIM:** To determine temperature in pipe with heated sections with laminar flow.

### **A. Geometry**

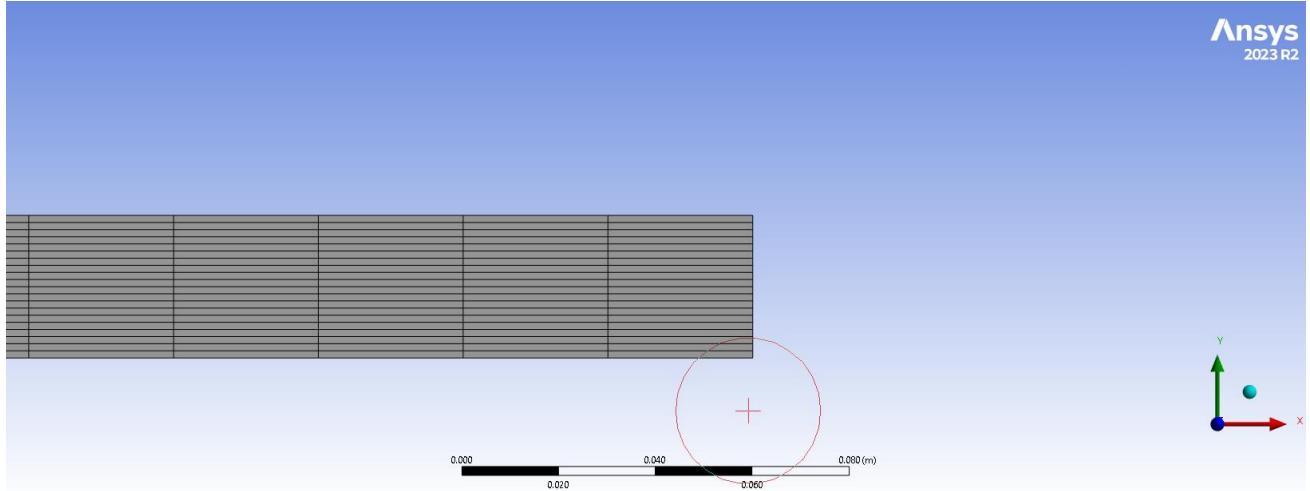
1. First, we need to specify that the geometry is 2-dimensional. Right click the Geometry box and select **Properties**. This will open the Properties of Schematic A2: Geometry Window. Under Advance Geometry Options change **Analysis Type** from **3D** to **2D**.
2. After the analysis type has been set, we are ready to launch Design Modeler, the design tool in ANSYS. Open Design Modeler by double clicking the geometry box. After launching the Design Modeler, you will be prompted to choose a standard unit of measurement. Select **Meter** as the standard unit, and click **OK**.
3. The convection pipe flow can be simplified to a 2D axis symmetric problem and solved in FLUENT. The following list shows the key features in creating the simplified model for the 2D steady convection problem:
  - Sketch
  - Rectangle
  - Split Edge
  - Equal Length
  - Dimensions
  - Create Surface
4. To begin sketching, click on the **Sketching** tab in the Tree Outline window. To draw our domain, we will use the Rectangle tool. Click on in the Sketching Toolboxes window. In the graphics window, draw the rectangle by first clicking on the origin (make sure the P icon is showing, meaning you are in fact selecting the point), then select a point in the 1st quadrant. Because there are two sections to our domain, a heated section and an isothermal section, we will need to split the upper surface of our domain. To split the line, select the **Modify** tab in the Sketching Toolboxes window, and select . Next, click any point along the upper surface of the rectangle. This will split the line into 2 segments. Do this as well for the bottom line.
5. Next, go to **Constraints** tab and select "Equal Length." Click on both the top and bottom right partitions of the rectangle to make the splits equidistant.
6. The next step in creating the domain will be adding dimensions. In the Sketching Toolboxes window, select **Dimensions > General**. First, click the left segment of the upper surface of the rectangle, then click off of the geometry to place the dimension. Repeat this process for the right segment of the upper surface of the rectangle, then the left surface of the rectangle. We can set the dimension in the Details View window. In the Details View window, change **H1** to 5.76, **H2** to 2.88, and **V3** to .06.
7. To finish the domain, we must create a surface. To accomplish this, click **Concept > Surfaces From Sketches** in the menu bar. Next, click any edge on the sketch in the Graphics window. In the Details View window, select **Base Objects > Apply**. Next, click **Generate** to create the surface.



## B. Mesh

1. The mesher in workbench provides various meshing tools to create appropriate finite volumes for a given geometry. In this tutorial, we will utilize the following basic meshing tools:
  - Default Mesh
  - Sizing
  - Hard/Soft Behavior
  - Mapped Face
  - Advanced Size Functions
  - Named Selections
2. First we will apply a mapped face meshing. First, in the Outline window, click Mesh to show the Mesh menu in the menu bar. In the Meshing Menu, select Mesh Control > Mapped Face Meshing. In the Graphics window, click on the domain face to select it, then in the Details window, click **Geometry > Apply**.
3. To create the edge sizings, go to Mesh Control > Sizing. Next, we need to select the edge selection filter. Click and drag over the bottom, and top two surfaces (remember, the top surface has been split into two surfaces). The surfaces will highlight green when they've been selected. In the details window, select **Geometry > Apply**. Ensure that **Type** is set to **Element Size**, then change the **Size** from **Default** to **0.05**.
4. Now, we will create an edge sizing for the vertical edges of the geometry. Create another edge sizing, and this time, choose the left and right vertical edges of the geometry by holding Ctrl and selecting each of them, and go to **Geometry > Apply**. Change the **Type** to **Number of Divisions**. Next, specify **Number of Divisions** to **30**.
5. In the Mesh menu, Mesh > Generate Mesh.
6. Select the left, vertical edge of the geometry with a left mouse click - it should highlight green. Right click, and select **Create Named Selection**. Name the selection **Inlet**. Click OK once finished.
7. Next, we will specify the axis/centerline of the pipe. Select the bottom surface of the pipe, and create a named selection. Name it **Centerline**.

8. We will now specify the portion of the wall that is isothermal. Select the left portion of the upper surface of the pipe. Create a named selection and call it Isothermal Wall.
9. We will now specify the portion of the wall that is heated. Select the right portion of the upper surface of the pipe. Create a named selection and call it Heated Wall.
10. Finally, we will specify the outlet of the pipe. Select the right, vertical edge of the pipe. Create a named selection, and call it Outlet.

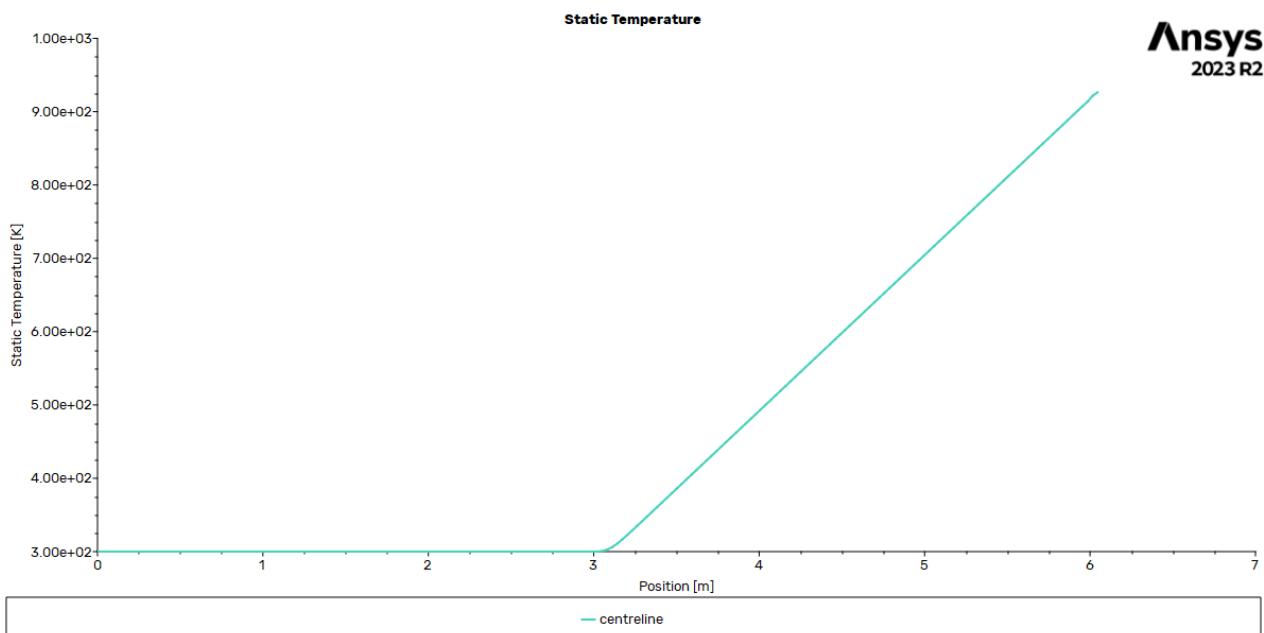


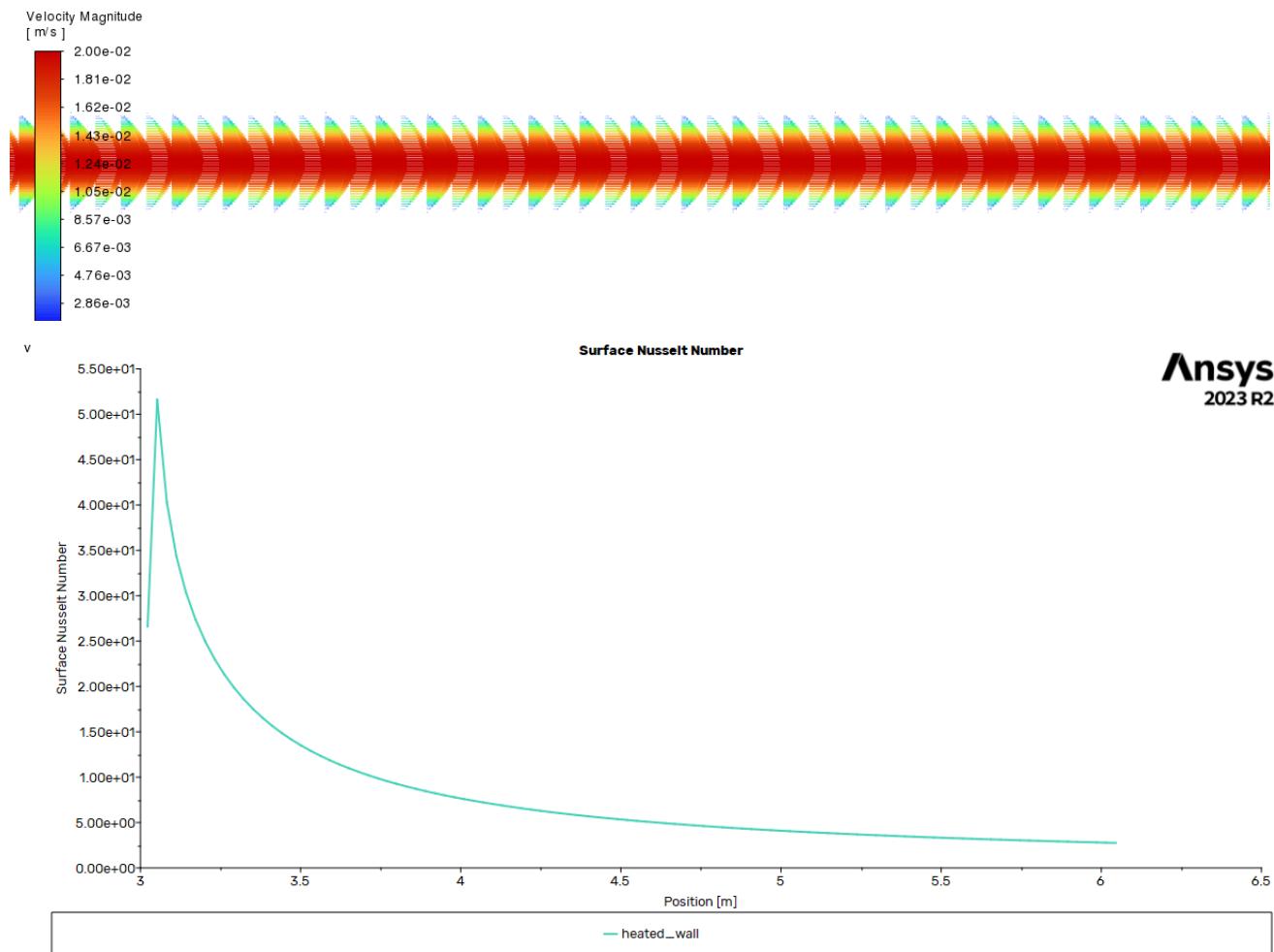
## C. Fluent

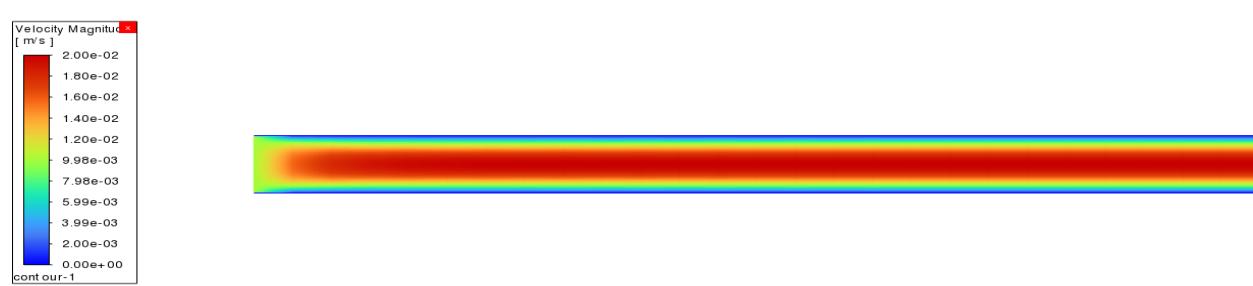
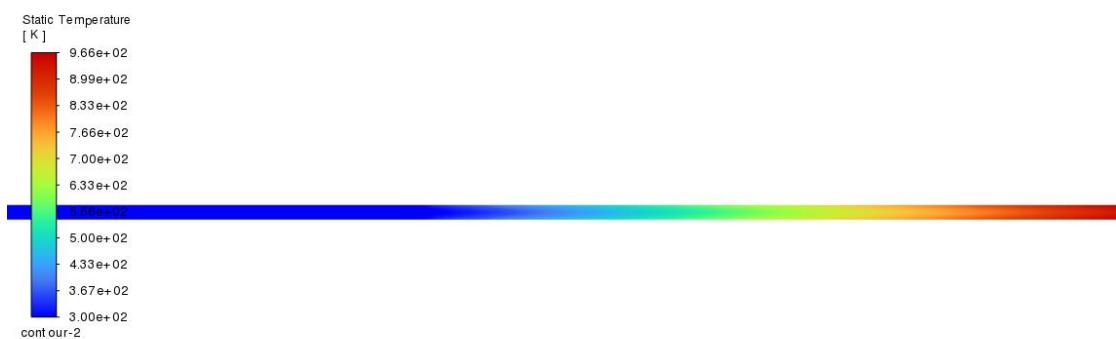
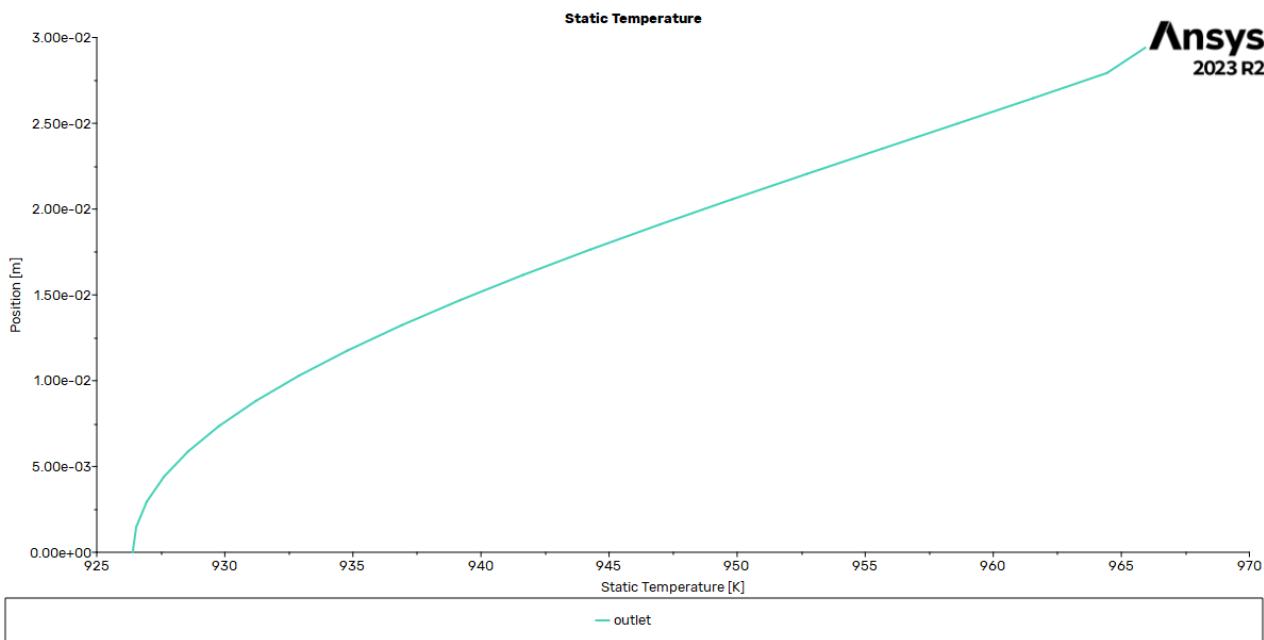
1. Before FLUENT launches, we will be prompted to set some options. In Options check the box next to **Double Precision**. Once the options are set, click OK.
2. Now, FLUENT should open. We will begin setting up some options for the solver. In the left hand window (in what I will call the Outline window), under **Problem Setup**, select **General**. The only option we need to change here will address the fact that pipe domain we created is axisymmetric. Under 2D Space, click the radio box next to **Axisymmetric**.
3. In the outline window, click **Models**. We will need to utilize the energy equation in order to solve for the temperature. Under Models highlight **Energy - Off** and click **Edit....** Now, the Energy window will launch. Check the box next to **Energy Equation** and hit OK.
4. In the Outline window, highlight **Materials**. In the Materials window, highlight **Fluid**, and click **Create/Edit....** this will launch the Create/Edit Materials window; here we can specify the properties of the fluid. Set the **Density** to 1.2, the **Specific Heat** to 1000, the **Thermal Conductivity** to .02, and the **Viscosity** to 1.8e-5. Once finished, click **Change/Create**, then **Close**.
5. Now we will specify the boundary conditions governing the problem. In the Outline window, highlight **Boundary Conditions**.
6. The default operating pressure in FLUENT is 1 atm, which is 101325 Pa. We can equate the operating pressure to the absolute pressure by setting the **gauge pressure** to zero.
7. Under Zone, highlight **Centerline**. Change the **Type** to **axis**. Confirm you are changing the selection, then leave the name as the default centerline.
8. Under Zone, highlight **heated\_wall**. The **Type** should have defaulted to **wall**. Next, click **Edit....** Click the **Thermal** tab, and select the **Heat Flux** radio button. Change the **Heat Flux (w/m<sup>2</sup>)** to 37.5. Click **OK**.
9. Under Zone, highlight **inlet**. The **Type** should have defaulted to **velocity-inlet**. Next, click **Edit....** In the Momentum tab, change the **Velocity Specification Method** to **Components**, and specify the **Axial Velocity** to 0.1. Click **OK**.

10. Under Zone, highlight **isothermal\_wall**. The **Type** should have defaulted to **wall**. Next, click **Edit....** Click the **Thermal** tab, and select the **Temperature** radio button. Change the **Temperature (k)** to 300. Click **OK**.
11. Under Zone, highlight **outlet**. The **Type** should have defaulted to **pressure-outlet**. Next, click **Edit....** In the **Momentum** tab, ensure the **Gauge Pressure** is 0. Click **OK**.
12. In the Outline window, select **Reference Values**. Under **Compute From**, select **Inlet**. Ensure that the values displayed are the values we specified. We are now ready to setup the solution.
13. In the Outline window, select **Solution Methods** to open the Solution Methods window. Under **Spatial Discretization**, change the option under **Momentum** from **First Order Upwind** to **Second Order Upwind**. Under **Energy**, also change the option to **Second Order Upwind**.
14. In the Outline window, click **Monitors** to open the Monitors window. In the Monitors window, select **Residuals - Print,Plot** and press **Edit....** This will open the Residual Monitors window. We want to change the convergence criteria for our solution. Under **Equation** and to the right of **Continuity**, change the **Absolute Criteria** to 1e-6. Repeat for **x-velocity**, **y-velocity**, and **energy**, then press **OK**.
15. In the Outline window, select **Solution Initialization**. We need to make an "Initial Guess" to the solution so FLUENT can iterate to find the final solution. In the Solution Initialization window, under **Compute from**, select **Inlet** from the drop down box. Check to see that the values that generate match our inputted values, then press **Initialize**.
16. In the Outline window, select **Run Calculation**. Change the **Number of Iterations** to 200. Double click **Calculate** to run the calculation. After the calculation is complete, save the project.
17. Generate the required plots, vectors and contours.

## SCREENSHOTS FROM FILE







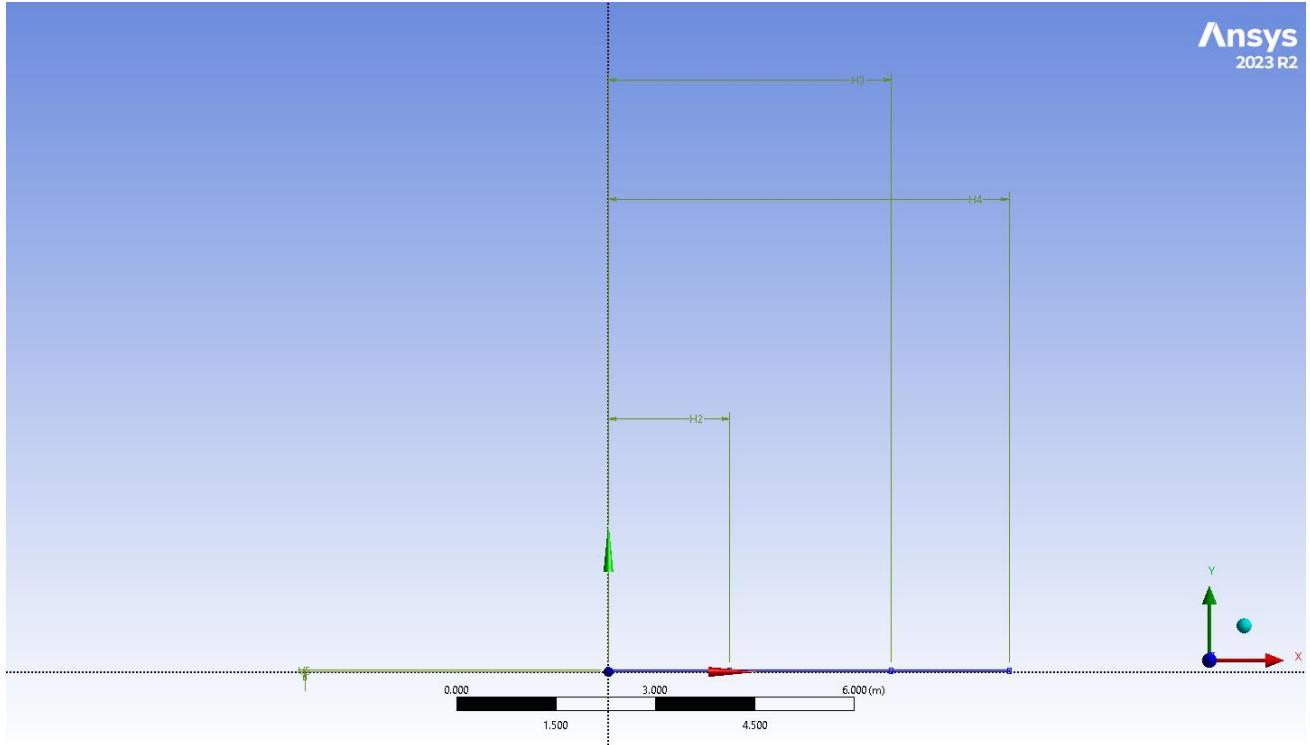
## **EXPERIMENT 5: TURBULENT FORCED CONVECTION**

**AIM: To determine temperature in pipe with heated sections with turbulent flow.**

### **A. Geometry**

1. First, we need to specify that the geometry is 2-dimensional. Right click the Geometry box and select **Properties**. This will open the Properties of Schematic A2: Geometry Window. Under Advance Geometry Options change **Analysis Type** from **3D** to **2D**. After the analysis type has been set, we are ready to launch Design Modeler, the design tool in ANSYS. Open Design Modeler by double clicking the geometry box. After launching the Design Modeler, you will be prompted to choose a standard unit of measurement. Select **Meter** as the standard unit, and click **OK**.
2. The convection pipe flow can be simplified to a 2D axis symmetric problem and solved in FLUENT. The following list shows the key features in creating the simplified model for the 2D steady convection problem:
  - Sketch**
  - Rectangle**
  - Split Edge**
  - Equal Length**
  - Dimensions**
  - Create Surface**
3. 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 on the +Z axis on the bottom right corner of the Graphics window to have a normal look of the XY Plane. 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.
4. 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
5. 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.
6. 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

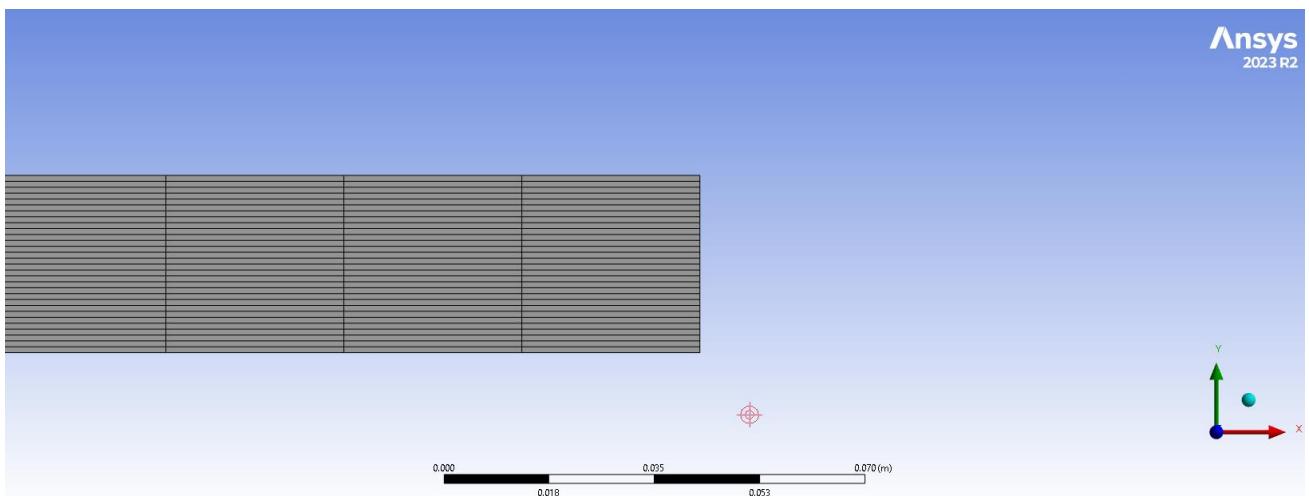
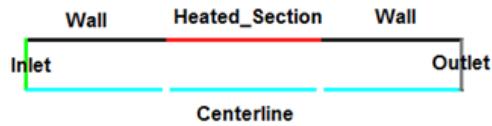
- Now that we have the sketch done, we can create a surface for this sketch. Then click on the **Concept** tab in the **DesignModeler** window, then click on **Surface from sketches**. This will create a new surface **SurfaceSK1**. Under the **Tree Outline**, click on the **X-Y Plane** and select **Sketch1** as **Base Objects** and under **Details View**, click **Apply**. Finally click **Generate** to generate the surface.



## B. Mesh

- We would also like to create a structured mesh where the opposite edges correspond with each other. Let's insert a Mapped Face mesh. Under Outline, right click on Mesh, move cursor to Insert, and select Mapped Face Meshing. Finally select the pipe surface body in the Graphics window and click Apply next to Geometry.
- Now let us move on to specify the element sizing along the pipe radial direction.  
Outline > Mesh > Insert > Sizing  
In the Graphics window, select both the left and right edge of the geometry (click on the Edge tab on the Fluid flow Fluent - Mesh window and then press Ctrl + mouse click to multiple select). Under Details of "Edge Sizing", click Apply next to Geometry. Change the edge sizing definition Type to Number of Divisions. Enter 30 for Number of Divisions. Next to Behaviour, change Soft to Hard.
- Now continue with the sizing in the axial direction. Outline>Mesh>Insert>Sizing  
In the Graphics window, select all the top and bottom edge of the geometry (press Ctrl + mouse click to multiple select). Under **Details of "Edge Sizing"**, click **Apply** next to **Geometry**. Enter 0.03 for **Element Size** (this will give us roughly 200 divisions). Next to **Behaviour**, change **Soft** to **Hard**. Update the Mesh.
- Click on Mesh and look under Details of "Mesh", next to Statistics, you should see that we have approximately 6000 Elements for our mesh.

5. Next, we will name the edges accordingly so that we can specify the appropriate boundary conditions in the later step. We know the bottom edges of the geometry are the centerline of the pipe, the left edge is the inlet of the pipe, the right edge is the outlet of the pipe, top side edges are the unheated wall sections and the top middle edge is the heated wall section.



## C. Fluent

1. We ask FLUENT to solve the axisymmetric form of the governing equations. When you do this, the solver switches to cylindrical polar coordinates. So from here on, you should interpret the horizontal coordinate as axial and the vertical coordinate as radial. General > Solver > 2D Space > Axisymmetric
2. The energy equation is turned off by default. Turn on the energy equation. Note that in most cases, you'll have to double-click on an item to select it. Models>Energy-Off>Edit... Turn on the Energy Equation and click OK.
3. Models > Viscous - Laminar > Edit... Under **Model**, select **k-epsilon (2 eqn)**. Since we'll use the default settings for the k-epsilon turbulence model, click **OK**.
4. Materials > Fluid air > Create/Edit... Since variations in absolute pressure are small in our pipe, we'll use a constant absolute pressure in the ideal gas law as discussed in the Pre-Analysis step. This is called the "Incompressible ideal gas" model in FLUENT (it's non-standard nomenclature). Change the **Density (kg/m3)** from **constant** to **incompressible-ideal-gas**. The constant absolute pressure to be used in the ideal gas equation is specified later as Operating Pressure.

5. Enter the following constant values:

Cp (Specific Heat) (j/kg-k): 1005

Thermal Conductivity (w/m-k): 0.0266

Viscosity (kg/m-s): 1.787e-5

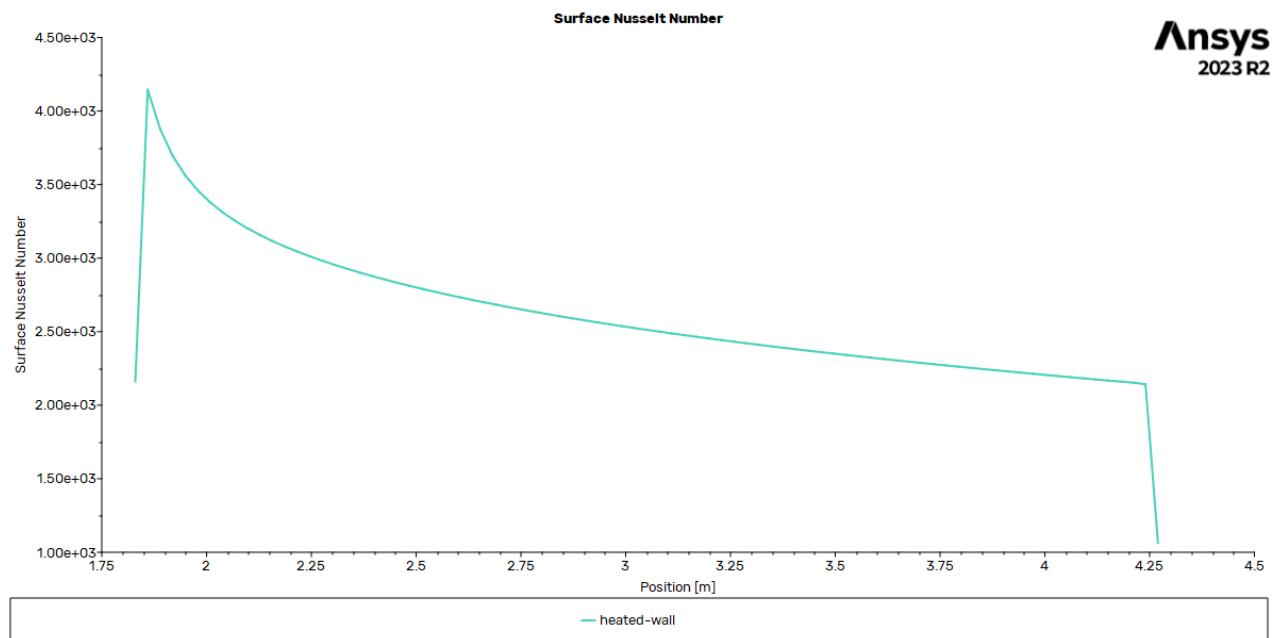
Molecular Weight (kg/kmol): 28.97

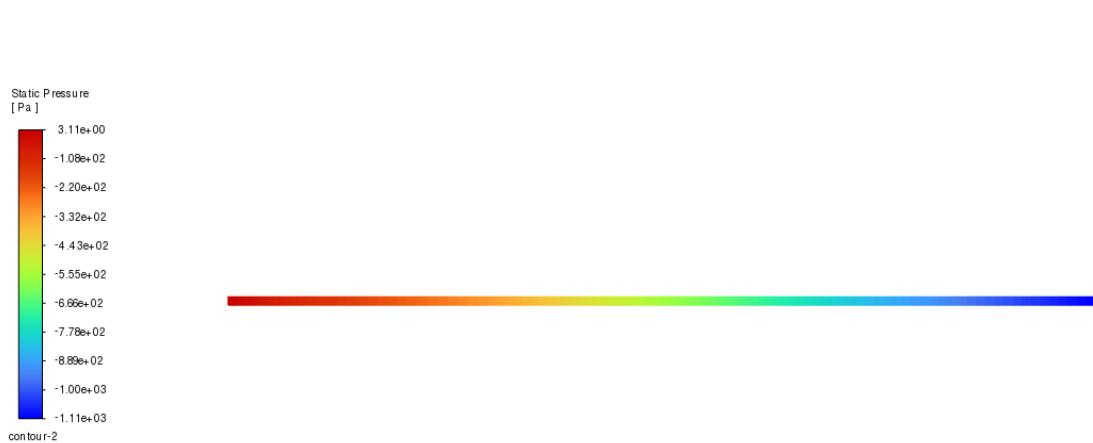
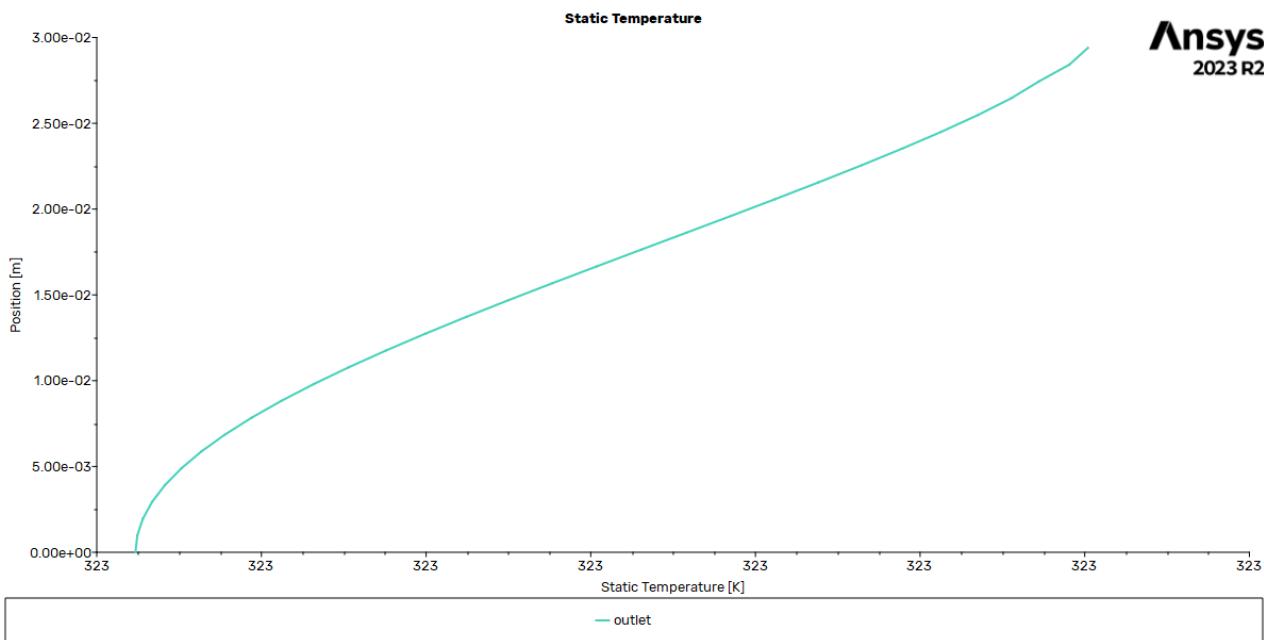
6. Click Change/Create and Close the Create/Edit Materials window.
7. (*double-click*) **Boundary Conditions**>**Operating Conditions...** Enter 98338.2 under Operating Pressure and click **OK**.
8. Next we will specify the boundary condition for the centerline. Boundary Conditions > axis. Change the **Type** to axis and click **OK**. FLUENT will set all radial derivatives at this boundary to zero in accordance with the axisymmetric assumption.
9. Now let's specify the boundary condition at the walls. By default, FLUENT correctly picks the Wall boundary type for these boundaries. It will impose the no-slip condition for velocity at these boundaries. Additionally, for the heated wall section, we need to specify the heat flux into the flow. Boundary Conditions > heated\_section > Edit... A new Wall window will open. Click on **Thermal** tab and enter 5210.85 next to **Heat Flux (w/m<sup>2</sup>)** and click **OK**.
10. **Boundary Conditions > inlet.** Note that the boundary Type is automatically set to velocity-inlet. FLUENT has an automatic mechanism to pick a boundary type according to the name you give and settings that you have selected previously (this can be dangerous if FLUENT selects the wrong boundary type and a lackadaisical user doesn't change it). In this case, it gets it right. Click **Edit...** to set up the correct inlet parameters. The Velocity Inlet window pops up. Enter 30.06 next to Velocity Magnitude (m/s). Under Turbulence, select the specification method to be Intensity and Viscosity Ratio. Use the default values for Turbulent Intensity (5%) and Turbulent Viscosity Ratio (10). These are plausible guess values for the turbulence level at the inlet. FLUENT will calculate k and epsilon at the inlet from these values and use them as boundary conditions for the k and epsilon equations. The results should not be sensitive to these inputs since most of the turbulence is generated in the boundary layers (ideally, you should check the sensitivity of your calculation to this setting).
11. Now click on **Thermal** tab and enter 298.15K for Temperature. Click **OK** to close the window.
12. **Boundary Conditions > Outlet.** FLUENT selects the pressure-outlet boundary type and its guess turns out to be right. Click **Edit...** to specify the gauge pressure at the outlet. Enter -1112.3 for Gauge Pressure and click **OK**. (From experiment, measured outlet pressure is 97225.9 Pa. Corresponding gauge pressure = 97225.9 Pa - operating pressure = -1112.3 Pa.) The negative sign indicates that the pressure at the outlet is lower than the ambient value.
13. Solution > Methods. The FLUENT solver converts our BVP to a set of algebraic equations through a process called discretization. We'll use second-order discretization for which the error is of the order of the square of the mesh spacing. This is more accurate (albeit less stable) than first-order discretization where the error is of the order of the mesh spacing. Choose Second-

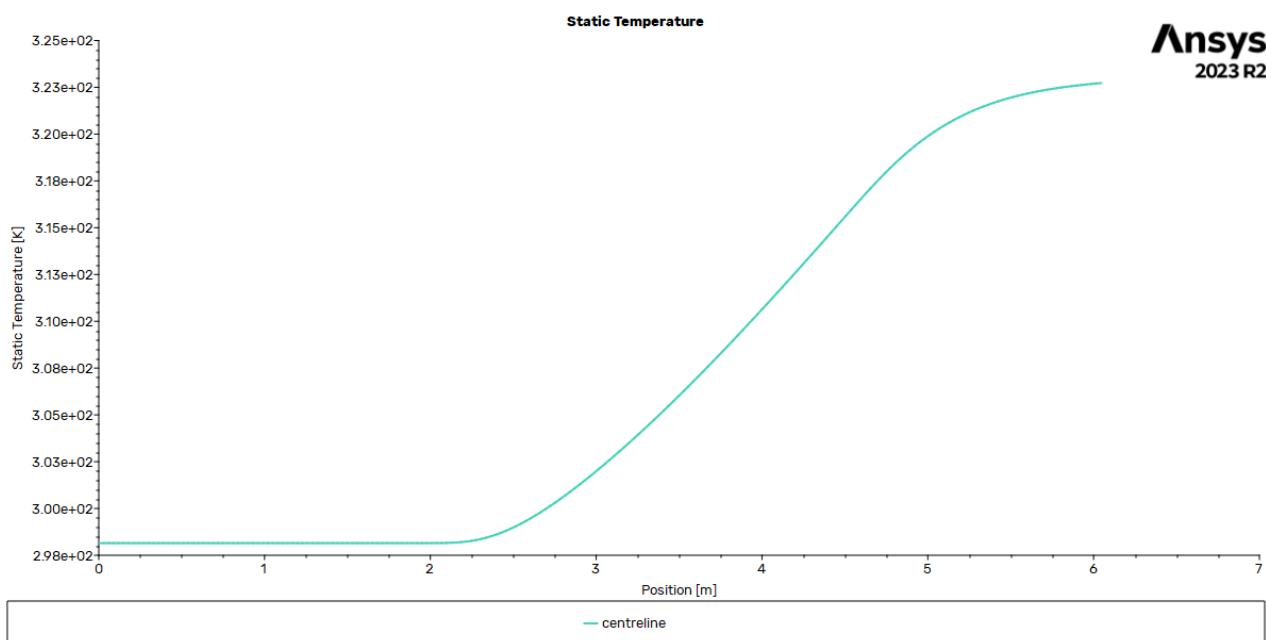
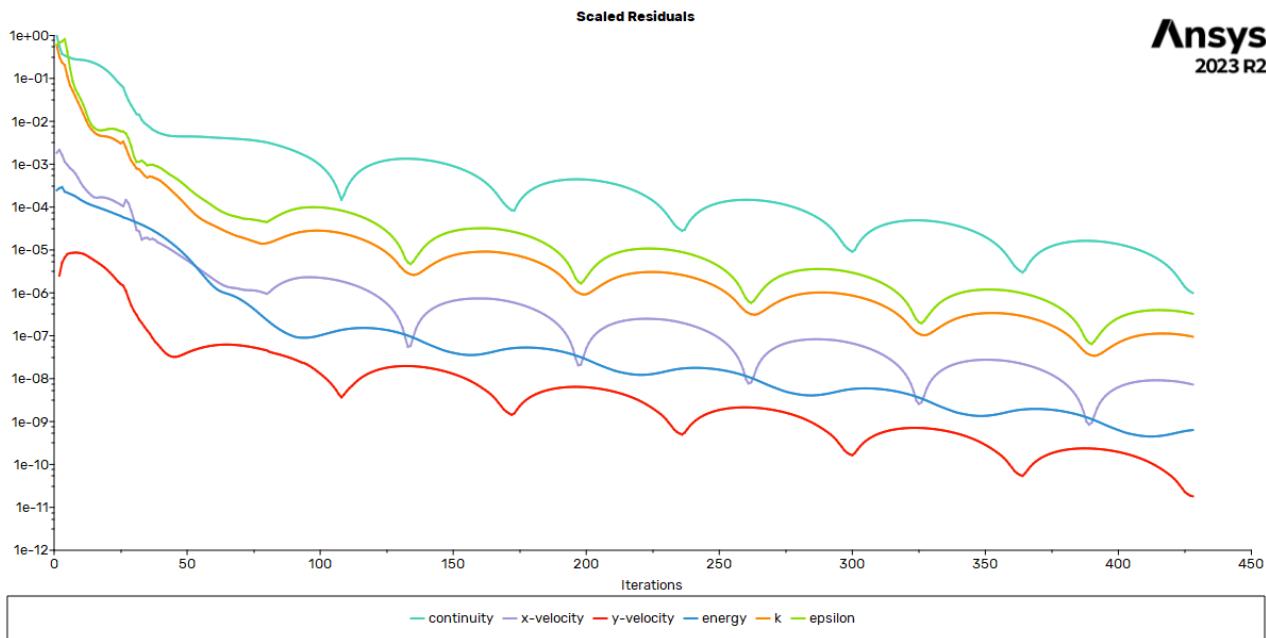
Order Upwind for all equations as shown below. Set Pressure-Velocity Coupling to SIMPLE if it is not by default.

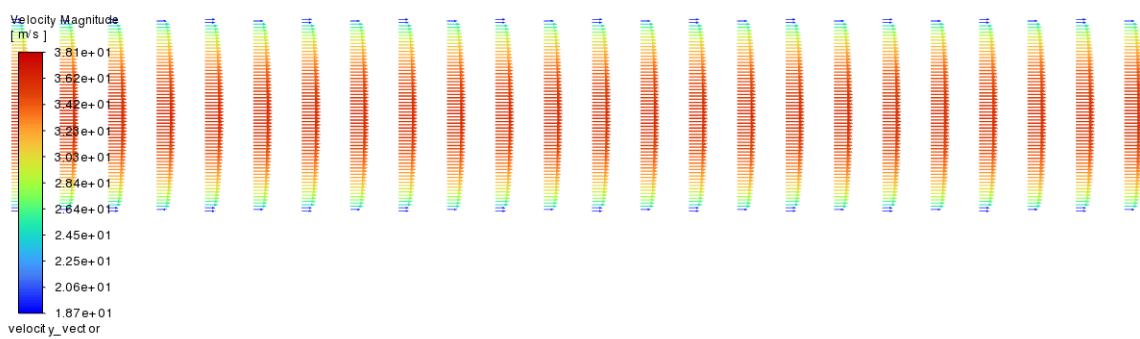
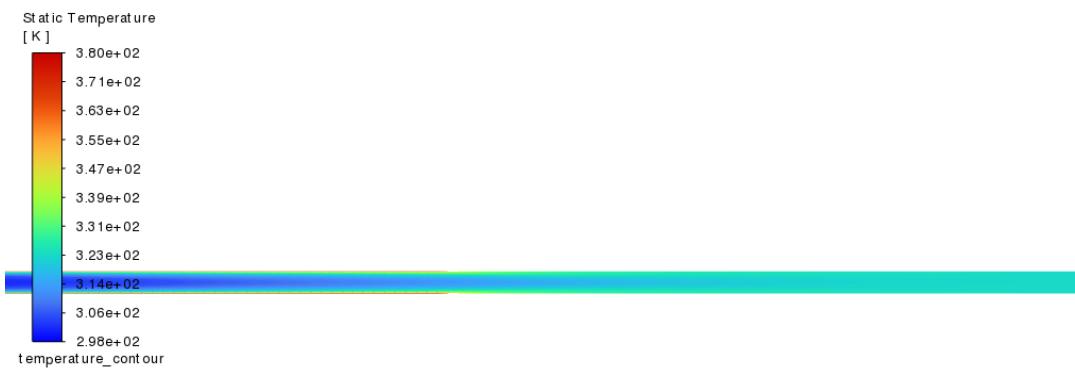
14. Solution > Monitors > Residual. We see that we need to provide a convergence criterion for each PDE that is being solved. The solver will stop iterating when mass, momentum, energy, k and epsilon imbalances (called residuals) fall below the convergence tolerance. We'll use a residual tolerance of  $10^{-6}$  for all six PDE's being solved.
15. Solution > Initialization. We need to provide FLUENT with an initial guess for the flow variables (velocity, pressure etc.) to start the iterations. For this example, we know the conditions at the inlet of the pipe (except for pressure which is set to zero gauge by default). Initialize the entire flowfield to the specified values at the inlet: First, select **Standard Initialization**, then under **Compute from**, select **Inlet** and click **Initialize**.
16. Solution > Run Calculation. Enter 500 for Number of Iterations and click Calculate.
17. Plot the necessary numerical charts, contours and vectors.

## SCREENSHOTS FROM FILE









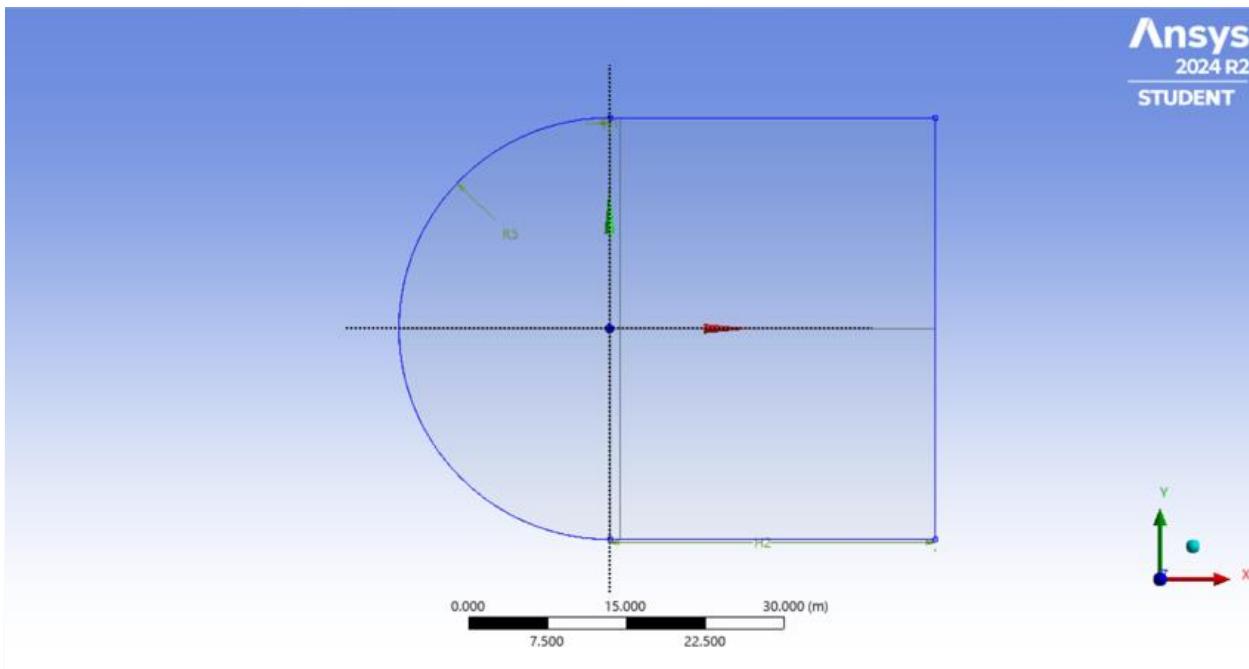
## **EXPERIMENT 6: FLOW OVER AN AIRFOIL**

**Aim:** Obtain the pressure and streamline with boundary layer for Viscous (K- $\Omega$ ) SST over an airfoil.

### **Procedure:**

#### **I. Design Modular (Geometry) of airfoil in ansys:**

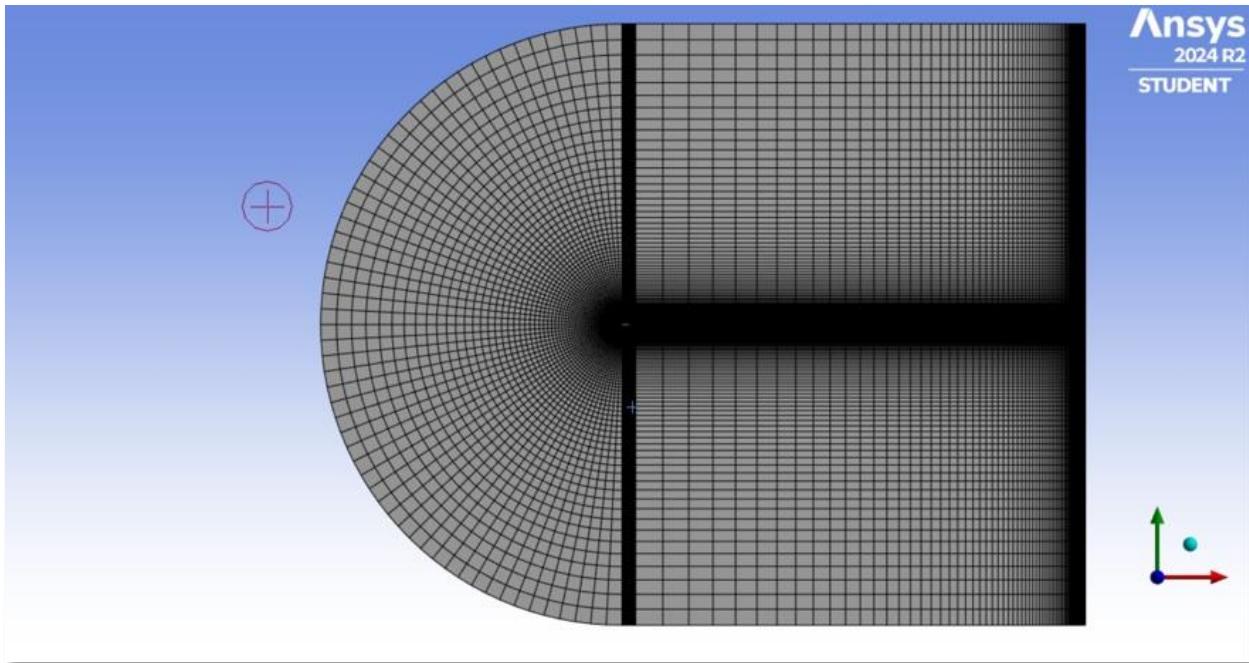
1. Open workbench ansys.
2. Drag into project schematic (fluid flow(fluent))
3. Right click on geometry and select properties. Select analysis type>2D.
4. Right click on geometry new modular design.
5. Concept >> 3D Curve >> Detail View >> Coordinate File >> Open Text File (naca0012).
6. Concept >> Surface from edges >>select geometry >>apply >>generate.
7. Choose a new sketch in XY plane. Sketching a rectangle cutting the airfoil.
8. Choosing dimension as horizontal select length H1 = 0.1m and H2 = 31m and also vertical change last two vertical column to each 20m.
9. Draw >> Arc by Center (Draw an arc using the common point of airfoil and rectangle) >> Trim (Extra part above and below airfoil) >> Dimension Radius =20m.
10. Concept >>Surfaces from sketches >>Select all geometry face and apply >>Change operation >>Frozen >>generate.
11. Create >>Boolean >>Subtract >>airfoil geometry from large geometry.
12. Tools >> face split >> By plane>> Target face and ZX plane>> generate.
13. Select new plane >>type- from coordinates.
  - (i) Point X - 1m
  - (ii) Point Y - 0m
  - (iii) Point Z - 0m
  - (iv) Normal X - 1
  - (v) Normal Y - 0
  - (vi) Normal Z - 0
14. Repeat step – 12 and select geometry, target Both faces and new plane.
15. Select new plane >>type- from coordinates.
  - (vii) Point X – 0.1m
  - (viii) Point Y – 0m
  - (ix) Point Z – 0m
  - (x) Normal X – 1
  - (xi) Normal Y – 0
  - (xii) Normal Z – 0
16. Tools >> face split >> By plane>> Target both front face and new plane>> generate.
17. Change surface body >> fluid and suppress >> line body.
18. Save the geometry.



**Fig1.** Geometry of an airfoil and cover

## II. MESHING:

1. Open mesh by double clicking on it.
2. Update and see the mesh, not in proper form.
3. Click edge selection filter (Ctrl+E) and select all 3 horizontal line >>insert sizing >> element size – No. of division >>100 >> Behavior >> hard >>bias >> 1<sup>st</sup> option >> Bias factor >>500 >> reverse bias(any line is not proper).
4. Click edge selection filter (Ctrl+E) and select all 6 vertical line + 1 horizontal line in front of airfoil >>insert sizing >> element size – No. of division >>200 >> Behavior >> hard >>bias >> 2<sup>nd</sup> option >> Bias factor >>50000 >> reverse bias(any line is not proper).
5. Click edge selection filter (Ctrl+E) and select small horizontal line + airfoil up and below (back) >>insert sizing >> element size – No. of division >>80 >> Behavior >> hard.
6. Click edge selection filter (Ctrl+E) and select both front curve + airfoil up and below (front) >>insert sizing >> element size – No. of division >>30 >> Behavior >> hard.
7. Click on face selection filter (Ctrl+F) and select all faces >> Insert>> Face Meshing.
8. Change in Sizing option >> De-filtering >>No.
9. Update and generate mesh.
10. Click of edge selection select the front >> press N>> Inlet.
11. Click of edge selection select the last two vertical >> press N>> Outlet.
12. Click of edge selection select airfoil in the center >> press N>> Aerofoil-wall.
13. Click on Update and Save the mesh and close.
14. Update the project. This will load the mesh into FLUENT.



**Fig2.** Meshing of the geometry

### III.SETUP:

1. Double click **Setup**. The *Fluent Launcher* Window should open. Check the box marked **Double Precision**. Start
2. **Problem Setup > Models > Viscous-Laminar**. Then press **Edit...** This will open the *Viscous Model* Menu Window. Select **(K- $\Omega$ ) SST** and press **OK**.
3. To define the density, click **Problem Setup > Materials > (double click) Air**. This will launch the *Create/Edit Materials* window.
4. Under *Properties*, ensure that density is set to **Constant** and enter  $1 \text{ kg/m}^3$  as the density. Click **Change/Create** to set the density.
5. Bring up the boundary conditions menu by selecting **Problem Setup > Boundary Conditions**.
6. Select **Inlet** to see the details of the boundary condition. The boundary condition type should have defaulted to **velocity-inlet**: If it didn't, select it. Now, click **Edit** to bring up the *Velocity-Inlet* Window.
7. Select **Velocity Specification Method > Components**. Specify **X-Velocity** as  $0.9945 \text{ m/s}$  and **Y-Velocity** as  $0.1045 \text{ m/s}$ . press **OK**.
8. In the *Boundary Conditions* window, look under *Zones*. Select **Outlet** to see the details of the boundary condition. The boundary condition type should have defaulted to **pressure-outlet**: if it didn't, select it. Click **Edit**, and ensure that the **Gauge Pressure** is defaulted to  $0$ .
9. In the *Boundary Conditions* window, look under *Zones* and select **airfoil**. Select **Type > Wall** if it hasn't been defaulted.
10. Go to **Problem Setup > Reference Values**. In the *Reference Values* Window, select **Compute From > Inlet**.

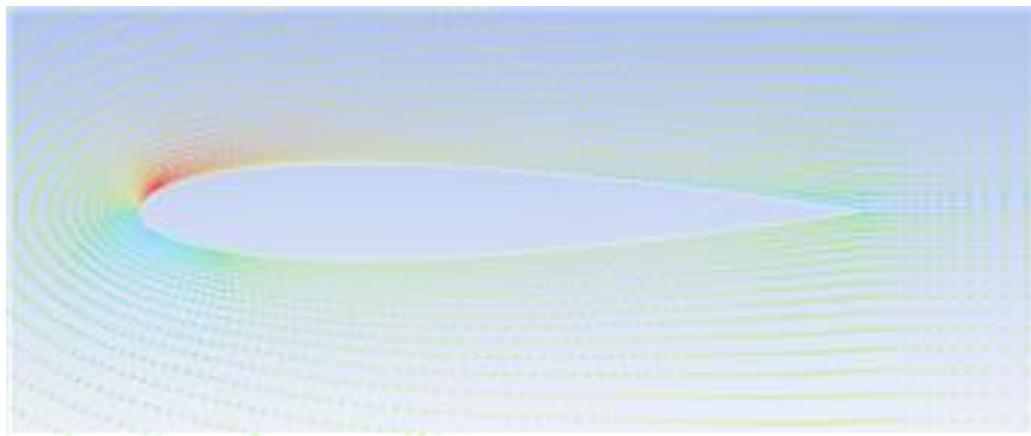
### IV.SOLUTION:

1. **Solution > Solution Methods**. Everything in this section should have defaulted to what we want, but let's make sure that under **Flow** the selection is **Second Order Upwind**.

2. **Solution > Monitors.** In the *Monitors* Window, look under **Residuals, Statistic, and Force Monitors**. Select **Residuals - Print,Plot** and press **Edit**. In the *Residual Monitors* Window, we want to change all of the **Absolute Criteria** to  $1e-6$ .
3. Go to **Solution > Solution Initialization**. In the *Solution Initialization* Window, select **Compute From > Inlet**.
4. Go to **Solution > Run Calculation**. Change **Number of Iterations** to 3000, then double click **Calculate**.

## V.RESULTS:

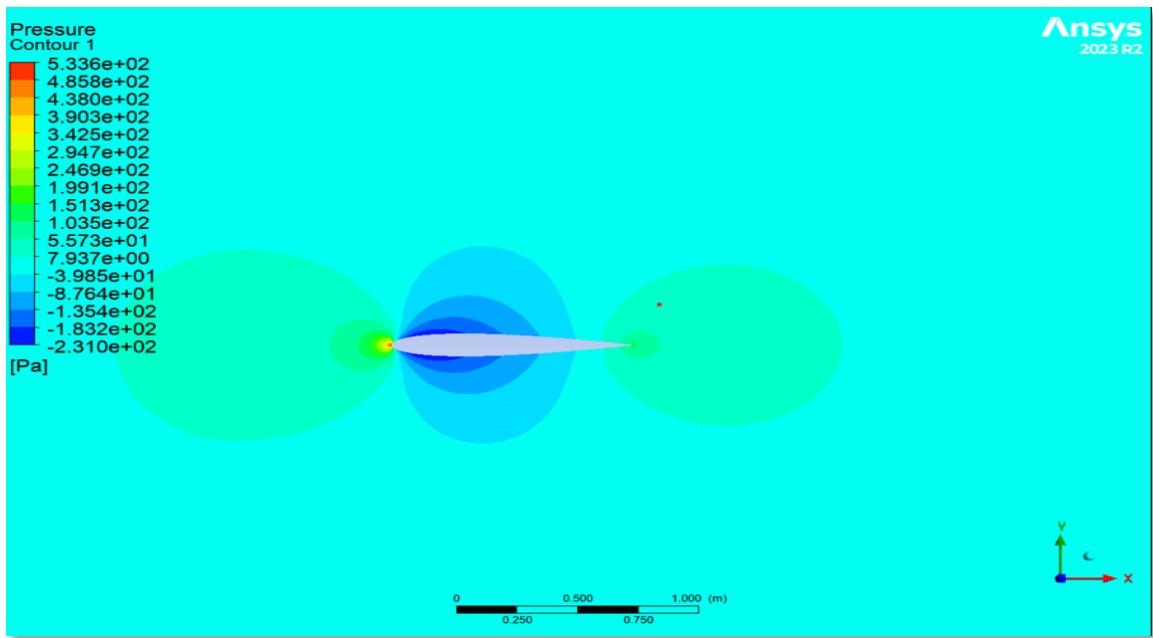
1. To plot the velocity vectors, go to **Results > Graphics and Animations**. In the *Graphics and Animations* Window, select **Vectors** and click **Set Up...**. This will bring up the *Vectors* Menu.
2. **Vectors of > Velocity, Color by > Velocity**, and set the second box as **Velocity Magnitude**. To see the velocity vectors, press **Display**.



**Fig3.** Vector Profile

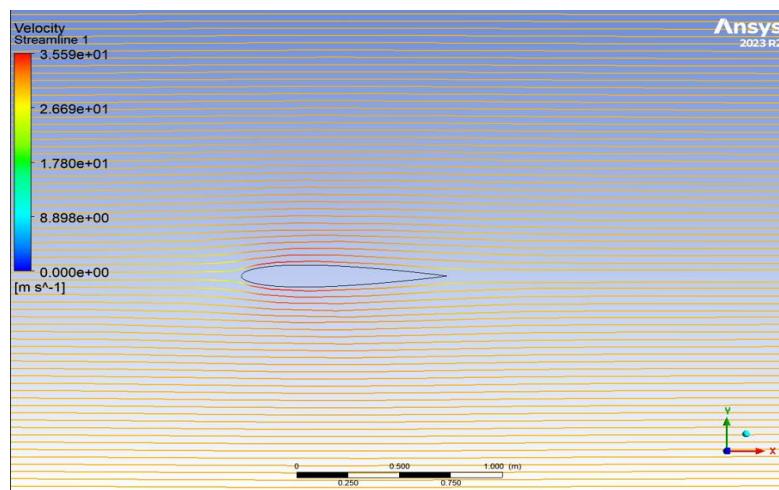
3. To view the pressure contours over the entire mesh, go to **Results > Graphics and Animations** again, and in the *Graphics and Animations*. Select **Contours**. Click **Set Up...** to bring up the *Contours* Menu. Check the box next to **Filled**. Under *Contours Of*, ensure that the two boxes that are selected are **Pressure...** and **Static Pressure**.

Once these parameters are set, press **Display** to see the pressure contours.



**Fig4.** Pressure Contour Profile

To view the streamlines, keep the *Contours* window open, and change the **Contours Of** box to **Velocity**, and the box below to **Stream Function**. Change **Levels** to 100. Also, uncheck the box marked **Auto Range**, and set **Min(kg/s)** to 13.11, and **Max(kg/s)** to 14.16 . Display.



**Fig5.** Streamline Profile

## **EXPERIMENT 7: STEADY FLOW PAST A CYLINDER**

### **AIM:**

1. To analyze the steady-state fluid flow past a cylindrical object using Computational Fluid Dynamics (CFD) techniques
2. to study the velocity distribution, streamline patterns, vorticity, and drag coefficient around the cylinder when subjected to a uniform flow.

### **PROCEDURE:**

#### **STEP 1. Pre-Analysis & Start-Up**

- **Solution Domain:**
  - Use a circular domain with an outer boundary 64 times the diameter of the cylinder.
- **Boundary Conditions:**
  - **Velocity Inlet:** Set the left half of the outer boundary with a velocity of 1 m/s in the x-direction.
  - **Pressure Outlet:** Set the right half of the outer boundary with a gauge pressure of 0 Pa.
  - **No Slip Boundary:** Apply this to the cylinder wall.

#### **STEP 2. Geometry Creation**

- **Fluid Flow(FLUENT) Project Selection:**
  - Drag Fluid Flow(FLUENT) into the Project Schematic window.
- **Analysis Type:**
  - Right-click Geometry > Properties > Set Analysis Type to 2D.
- **Launch Design Modeler:**
  - Double-click **Geometry**.
- **Create Inner Circle:**
  - Create a circle centered at the origin in the xy-plane with a diameter of 1m.
- **Create Outer Circle:**
  - Create another circle with a diameter of 64m.
- **Surface Body Creation:**
  - Create surface bodies for both circles. The outer circle should be frozen to avoid merging with the inner circle.

- Use Concept > Surfaces From Sketches to create surfaces.
- **Boolean Operation (Subtraction):**
  - Use Create > Boolean to subtract the inner circle from the outer circle.
- **Create Bisecting Line:**
  - Draw a line along the y-axis that bisects both circles and project it onto the geometry

## STEP 3. Meshing

- **Launch Mesher:**
  - Double-click Mesh.
- **Mapped Face Meshing:**
  - Right-click Mesh > Insert > Mapped Face Meshing and apply it to both portions of the surface body.
- **Edge Sizing:**
  - **Circumferential Edge Sizing:** Apply 96 divisions to the circumference.
  - **Radial Edge Sizing:**
    - Top Half: Set 96 divisions with a bias factor of 460.
    - Bottom Half: Set 96 divisions with an opposite bias factor.
- **Verify Mesh Size:**
  - Check that there are 18,624 nodes and 18,432 elements.

## STEP 4. Setup (Physics)

- **Launch Fluent:**
  - Double-click Setup.
- **Double Precision & Parallel Processing:**
  - Enable Double Precision.
  - Set Processing Options to Parallel if multiple cores are available.
- **Check Mesh:**
  - Verify that the mesh has 18,432 cells and check for errors.
- **Specify Material Properties:**
  - Set the fluid density to  $1 \text{ kg/m}^3$  and viscosity to  $0.05 \text{ kg/m}\cdot\text{s}$ .
- **Boundary Conditions:**
  - **farfield1:** Set as velocity-inlet with X-Velocity = 1 m/s.
  - **farfield2:** Set as pressure-outlet.

- **cylinderwall**: Set as a wall.
- **Reference Values**:
  - Set the density to  $1 \text{ kg/m}^3$ .

## STEP 5. Solution

- **Second Order Upwind Momentum Scheme**:
  - Set Momentum to Second Order Upwind in Solution Methods.
- **Convergence Criterion**:
  - Set the absolute criteria for residuals to  $1e-6$ .
- **Initial Guess**:
  - Set Compute From to farfield1 or X-Velocity to  $1 \text{ m/s}$  and initialize.
- **Iterate Until Convergence**:
  - Set the number of iterations to 2000 and click Calculate.

## 6. Results

- **Velocity Vectors**:
  - Display velocity vectors under Graphics and Animations.
- **Stream Lines**:
  - Generate streamlines by setting contours of velocity.
- **Vorticity**:
  - Display vorticity magnitude in the contour plots.
- **Drag Coefficient**:
  - Calculate the drag coefficient under Result Reports.

### RESULT:

The simulation successfully modeled the steady-state flow past a cylinder.

The velocity vector field and streamlines were visualized, showing the characteristic flow separation and wake formation behind the cylinder.

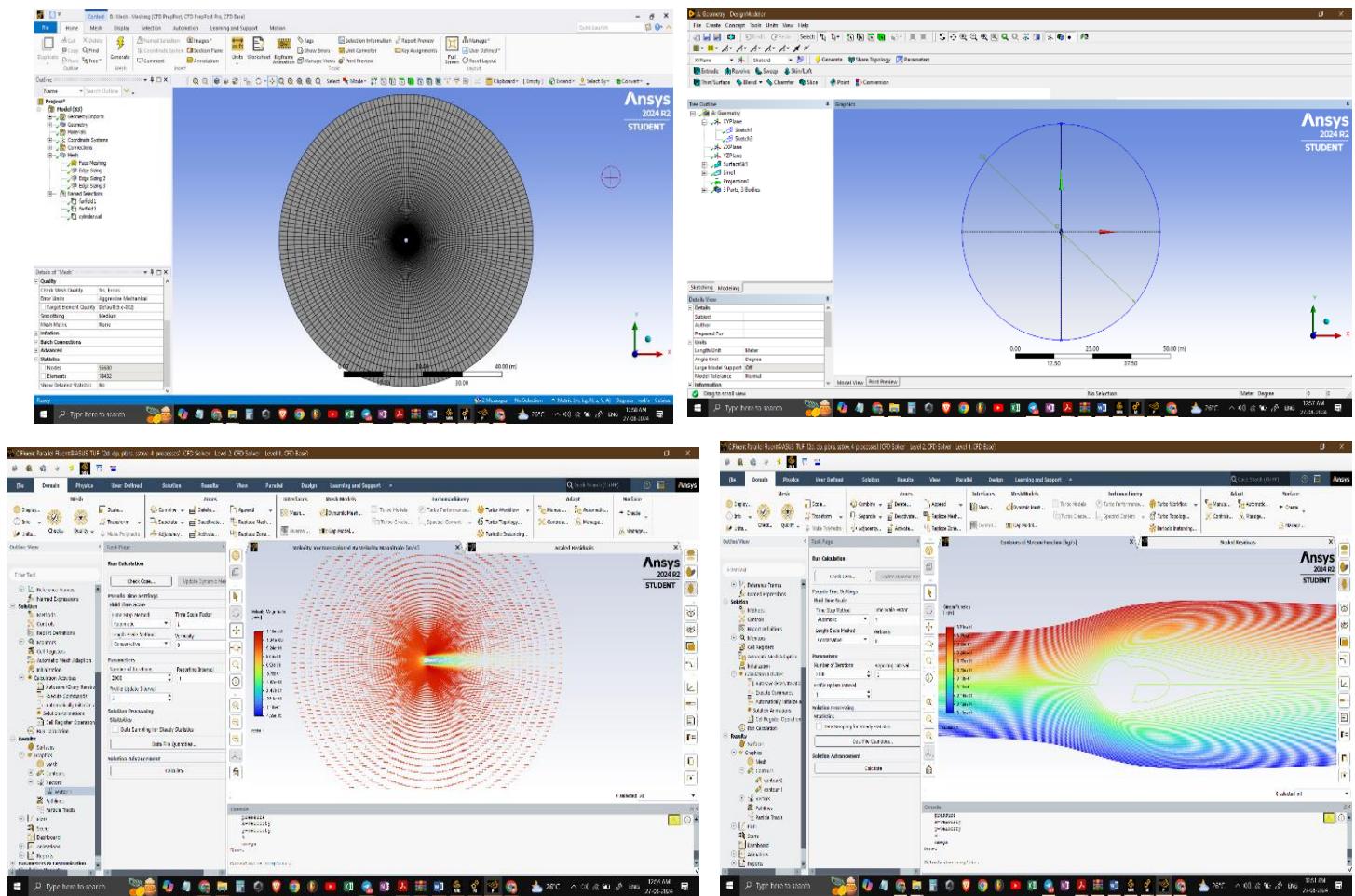
The vorticity contours highlighted regions of high rotational flow near the cylinder surface. The drag coefficient was calculated to be approximately 2.06, which is consistent with typical values for flow

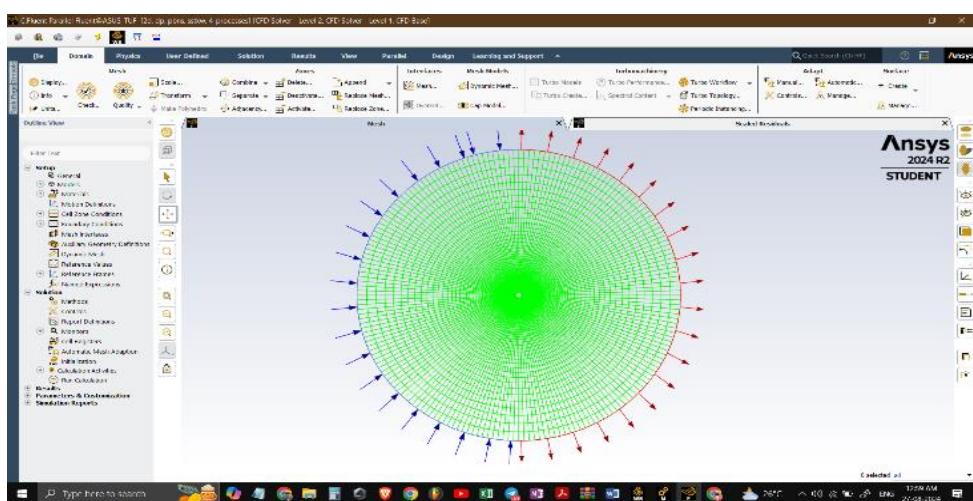
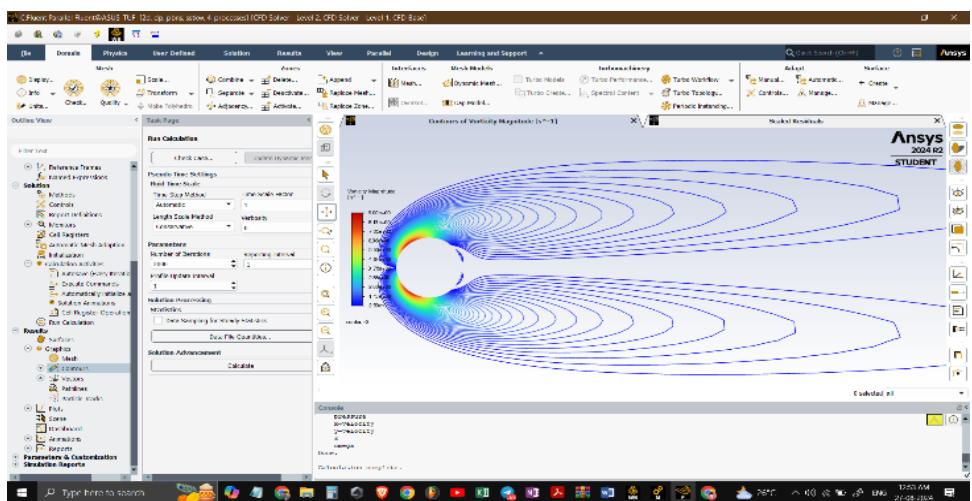
at similar Reynolds numbers. These results demonstrate the capability of CFD to accurately predict flow behavior around bluff bodies like a cylinder.

## CONCLUSION:

The CFD simulation accurately modeled steady-state flow past a cylinder, capturing key flow features like wake formation and flow separation. The calculated drag coefficient of approximately 2.06 aligns with expected values, validating the simulation's accuracy. This confirms the effectiveness of CFD in predicting flow behavior around bluff bodies.

## SCREENSHOTS FROM THE PROJECT FILE:





## **EXPERIMENT 8: STEADY FLOW IN PIPE ELBOW**

### **AIM:**

To analyze the steady-state fluid flow in a pipe elbow using Computational Fluid Dynamics (CFD) techniques.

### **1. Geometry**

1. In Ansys Geometry, select “Properties” and set Dimensions to 2D.
2. Open Ansys Geometry using Design Modeler.
3. Create “New Sketch”.
4. Draw a straight line offset from the x-axis, representing the upper edge. Use Spline to obtain the curved bend of the elbow.
5. Similarly, draw the intermediate edge of the elbow, and the bottom edge of the elbow.  
Alternatively, one can use arcs instead of splines.
6. Use “Create Surfaces from Sketches” to complete the Sketch.

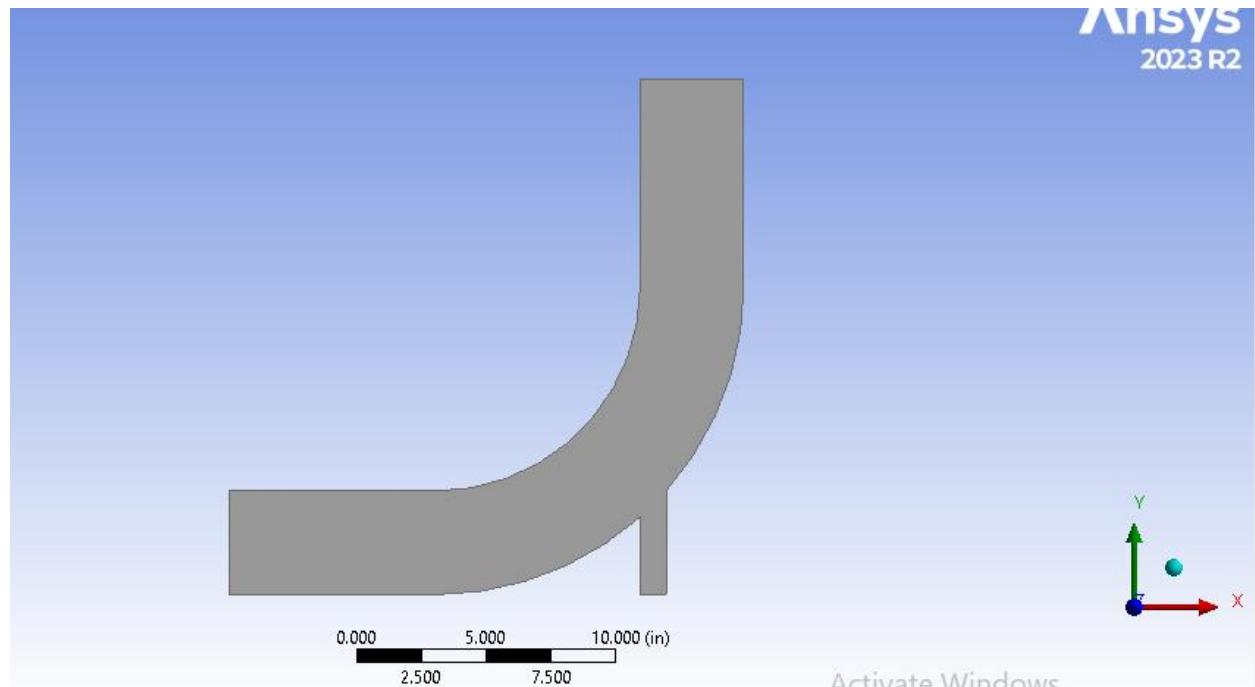


Fig. 1 Geometry

### **2. Mesh**

1. Open Mesh.
2. Rename the left hand side inlet as “inlet-1”, and the bottom inlet as “inlet-2” using Named Selections.
3. For the outer edges, create a common Named Selection and name it as “wall”.
4. Create the “outlet” Named Selection for the Outlet.

5. Change Meshing setting to CFD from Mechanical.
6. Apply Face Meshing on the body.
7. Overall, change Mesh setting to “Element Size” and set Element Size to 0.01 m.
8. Generate the Mesh.

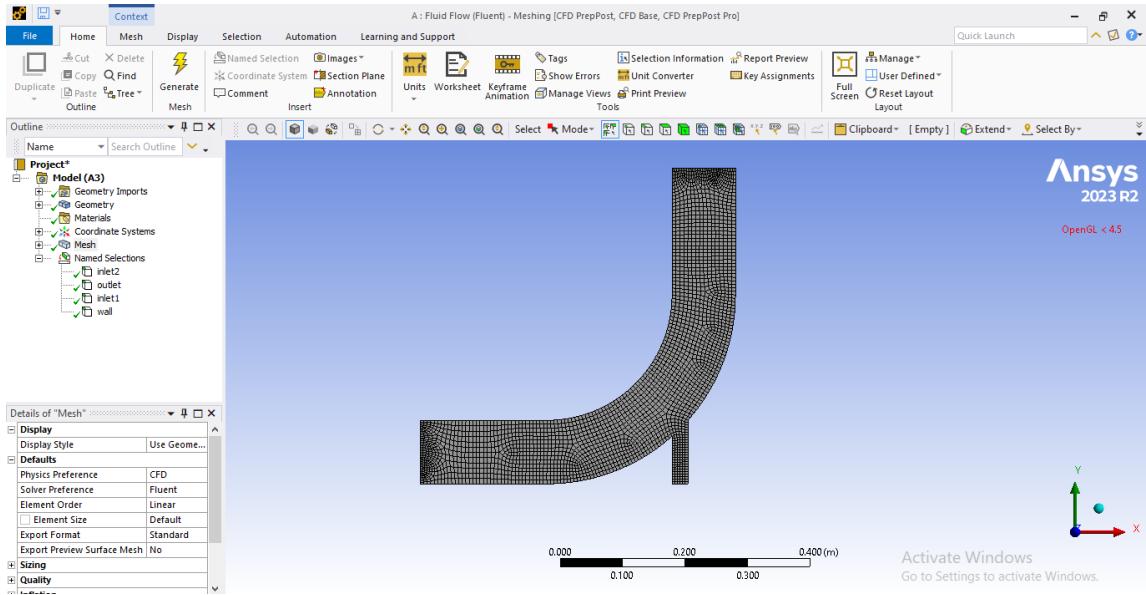


Fig 2. Mesh

### 3. Fluent

1. Open Ansys Fluent. Set Processes to Parallel and Number of Processors to 4.
2. Under Models, set the flow to k-epsilon (2 eqn).
3. Turn on Energy equation.
4. Enter the specified values of the materials under the Materials option.
5. Under Boundary conditions, set “inlet-1” as “velocity-inlet”. Set the inlet velocity and temperature. Repeat the same procedure for “inlet-2”.
6. For the outlet, keep the Gauge pressure as 0 (or atmospheric).
7. Under Monitors, set all the six residual values as 1e-06.
8. Initialise the solution using “Standard Initialisation.”
9. Set Number of Iterations to 1000 and press Calculate.
10. The Temperature and Velocity contours and vectors are then determined and calculated from the Results Menu.

## Screenshots from the File:

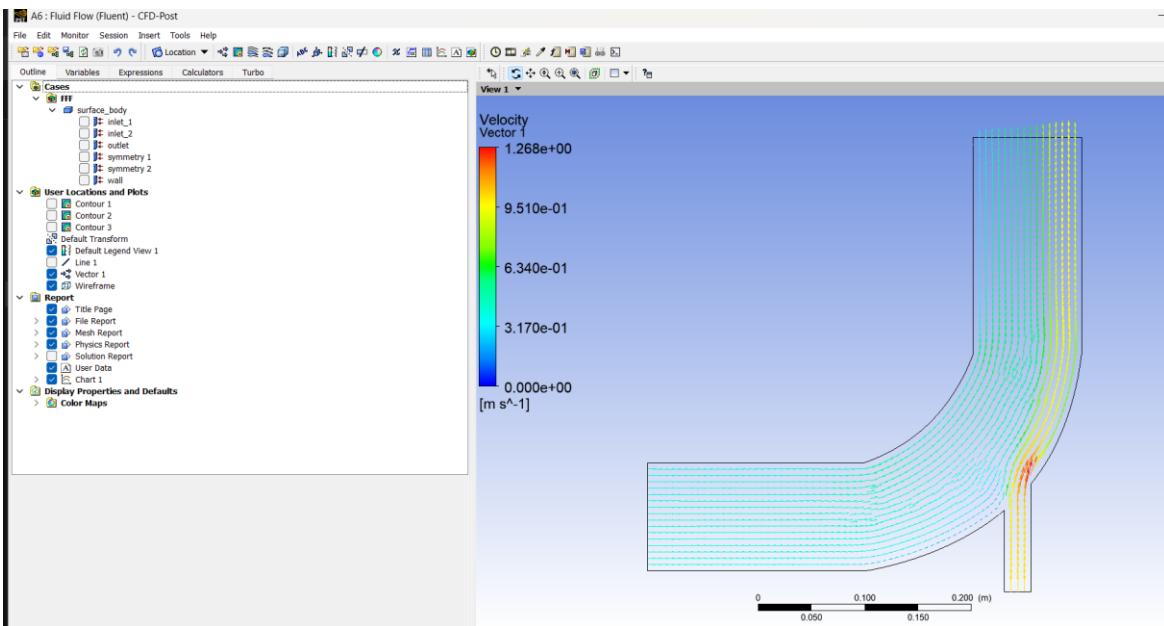


Fig 3. Velocity Vector

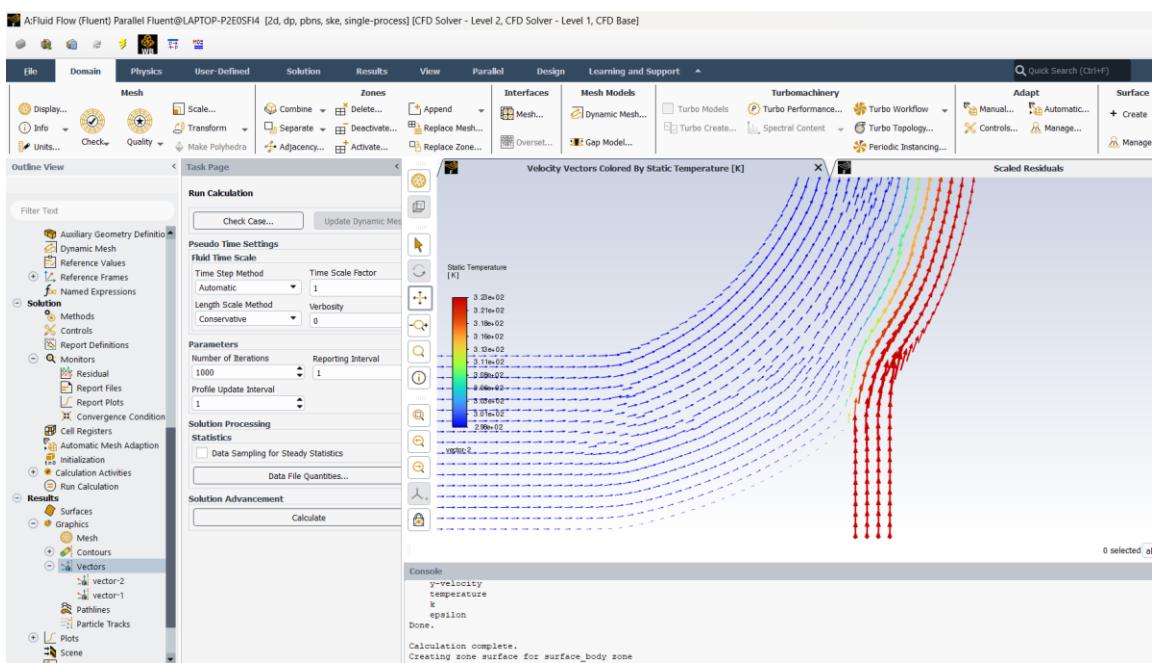


Fig 4. Temperature Vector

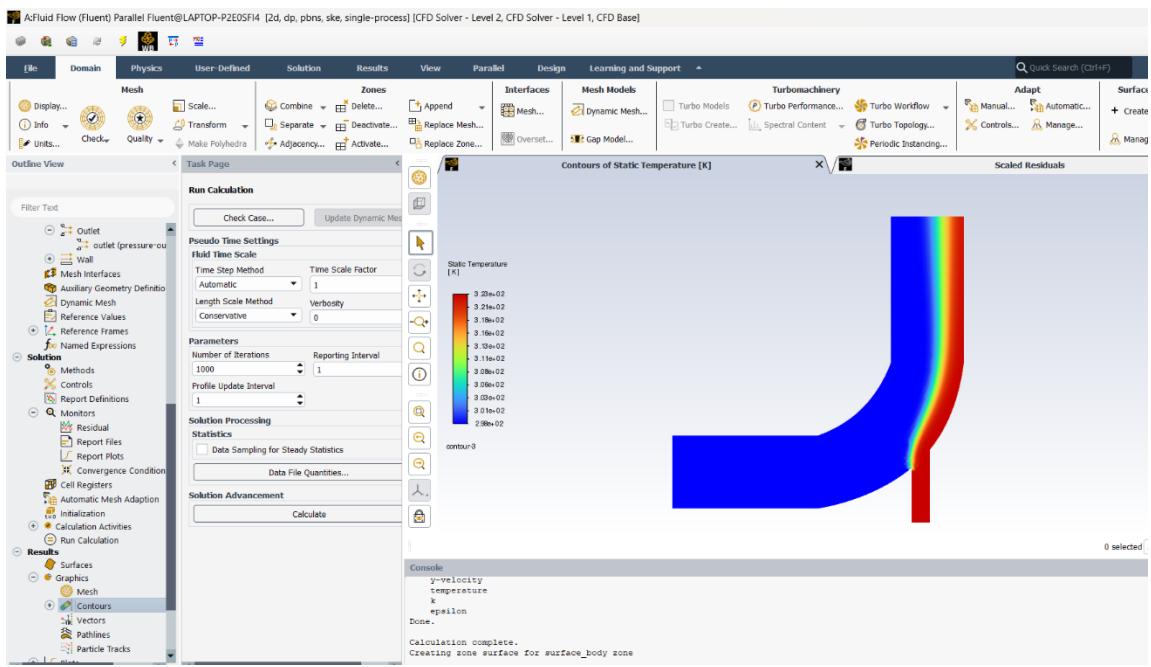


Fig 5. Temperature Contour

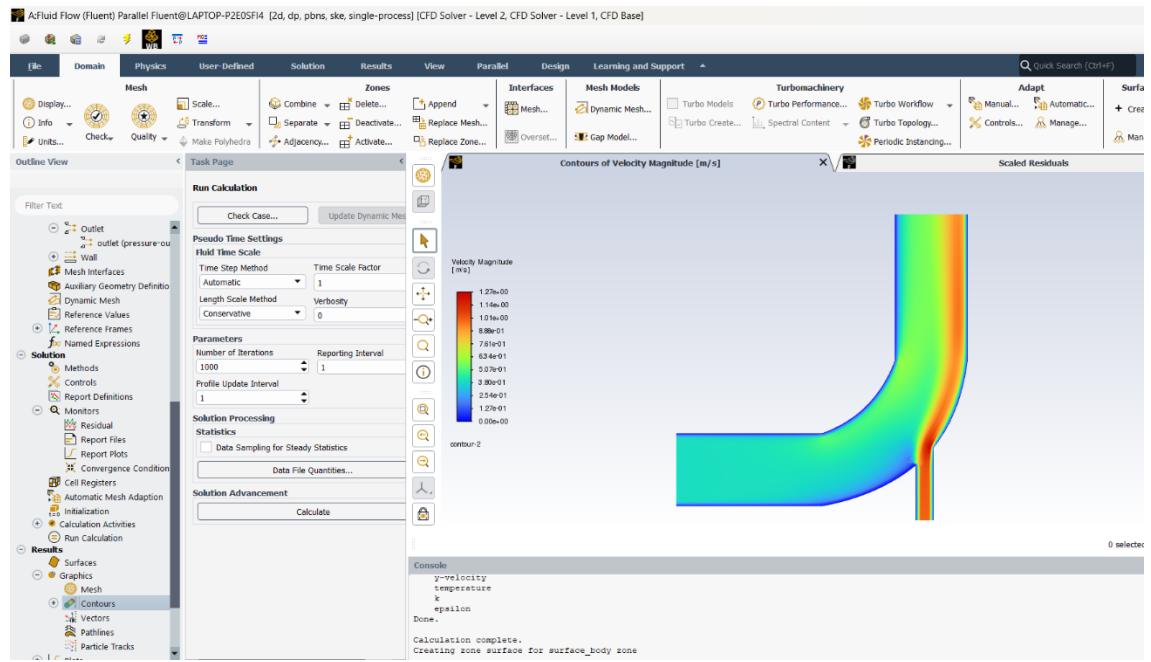


Fig 6. Velocity Contour

## **EXPERIMENT 9: STEADY FLOW IN NOZZLE**

### **AIM:**

To analyze the steady-state fluid flow in a nozzle using Computational Fluid Dynamics (CFD) techniques.

### **1. Geometry**

1. Left click Geometry, and select “Properties”. Set Dimensions to 2D.
2. Open Ansys Design Modeler.
3. Create “New Sketch”.
4. Make a rectangle of the specified dimensions.
5. Split it at 3 points in order to make the nozzle arc.
6. Now, use ‘Arc by Center’ to make the nozzle arc, and Trim the lines outside the arc on the rectangle.
7. Use ‘Create Surface from Sketches to complete the Sketch.

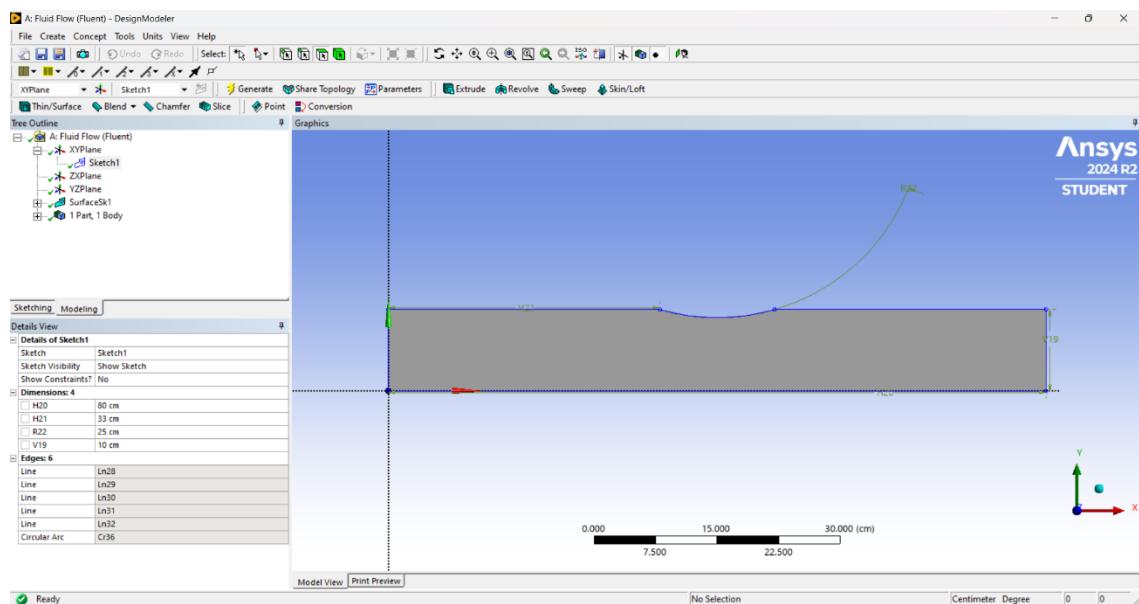


Fig 1. Geometry

### **2. Mesh**

1. Change Mesh setting from Mechanical to CFD.
2. Apply Face Meshing on the Body.
3. Apply Edge sizing to inlet edge, walls and outlet edge.
4. Create named selections of “inlet”, “outlet” and wall.
5. Generate Mesh.

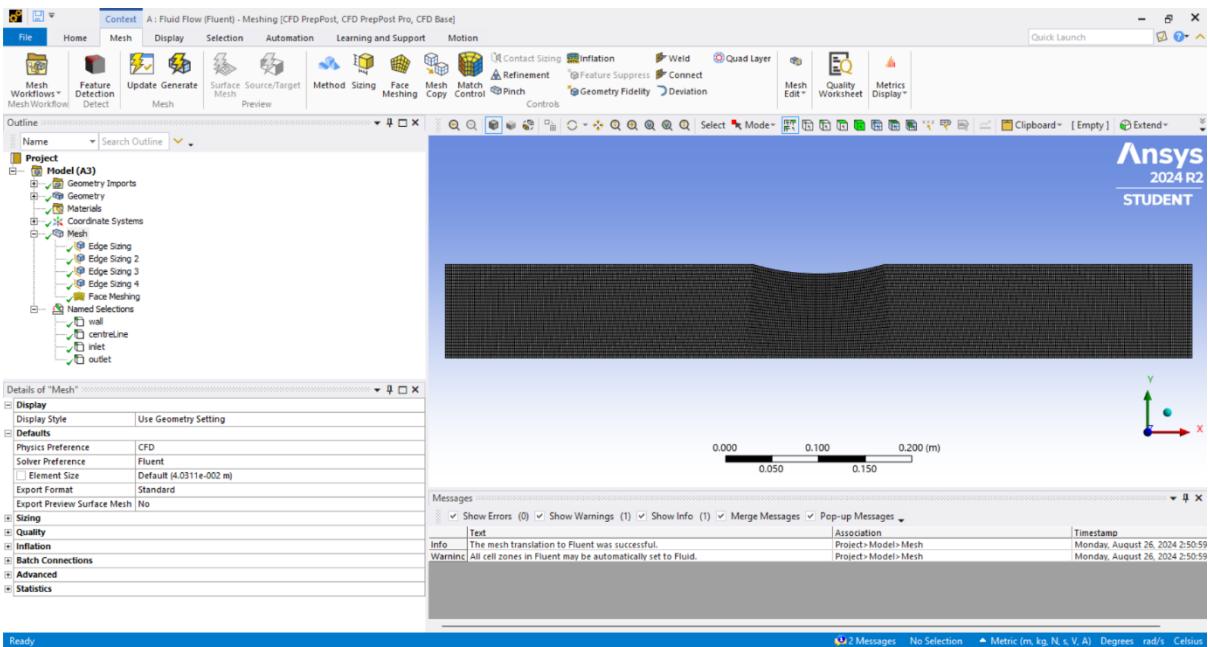


Fig 2. Mesh

### 3. Ansys Fluent

1. Open Ansys Fluent. Set Processes to Parallel and Number of Processors to 4.
2. Set Model to “Laminar”.
3. Select the “Axisymmetric” option.
4. Change the values of the materials to their appropriate values.
5. Set the inlet as “velocity inlet” and specify the velocity.
6. Set the outlet as “pressure outlet” and specify gauge pressure as 0.
7. Initialise the solution.
8. Under Monitors, set all the residual values to 1e-06.
9. Set number of iterations as 500. Run the calculations.
10. Calculate the relevant contours and vectors.

#### Screenshots from the File:

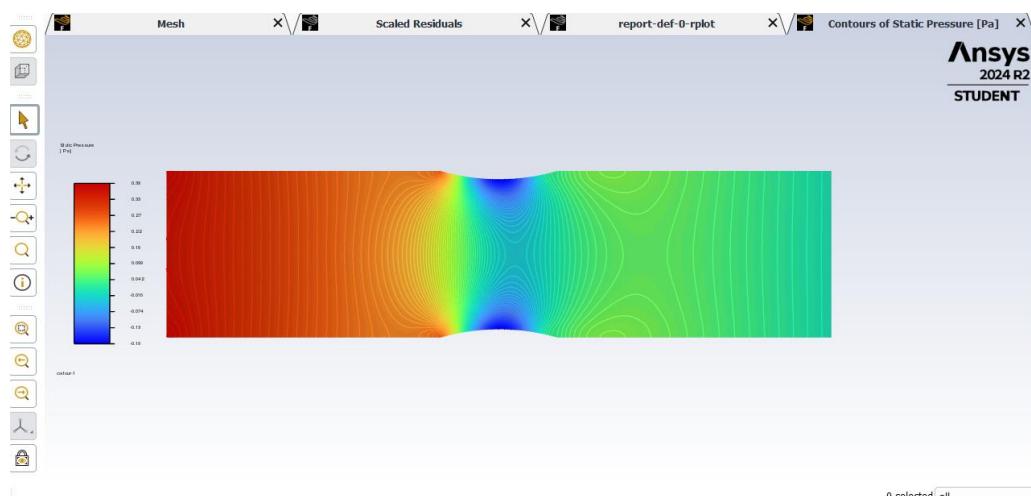


Fig 3. Pressure Contour

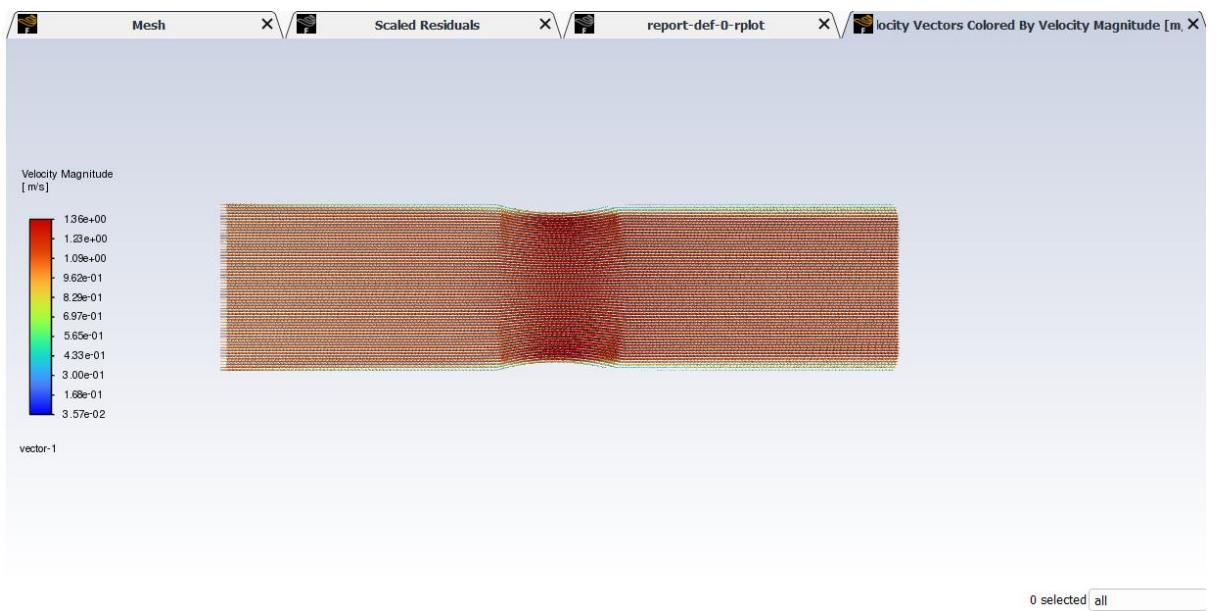


Fig 4. Velocity Vector

## **EXPERIMENT 10: STEADY FLOW IN TUBE BANKS**

### **AIM:**

To analyze the steady-state fluid flow in a nozzle using Computational Fluid Dynamics (CFD) techniques.

### **1. Geometry**

1. Left click Geometry and click on “Properties”. Set Dimensions to 2D.
2. Open Ansys Design Modeler.
3. Create a New Sketch.
4. Draw a rectangle of specified dimensions. Draw two circles with their centres on the rectangle’s top and bottom edges, respectively.
5. Trim the arcs of the circle outside the rectangle and the radii of the circles.
6. Dimension the circles such that the centre of the bottom circle is 1 cm displaced from the right edge, and the centre of the upper circle is 1 cm displaced from the left edge. Use “Surfaces from Sketches” to create the Surface.

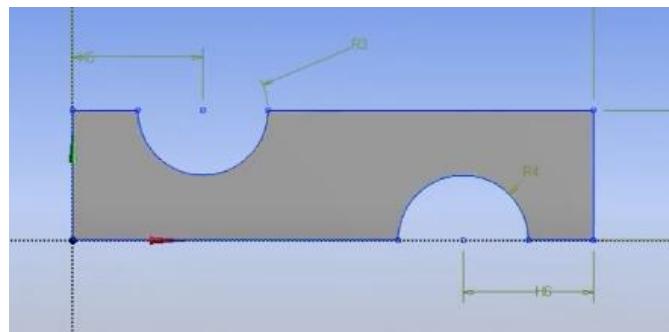


Fig 1. Geometry

### **2. Mesh**

1. Open Mesh and change setting to “CFD” from “Mechanical”.
2. Change Meshing method to “triangles” from “Quadrilaterals”.
3. In Sizing, change the setting from “Coarse” to “Fine.”
4. Apply “Inflation” to the tube walls. Change Inflation setting to “First Layer Thickness”, and set the First Layer Height as 0.01 mm. Set the number of maximum layers as 20, and keep the Growth Rate as 1.2.
5. Create the named selections of “wall”, “tubewall”, “inlet” and “outlet”.

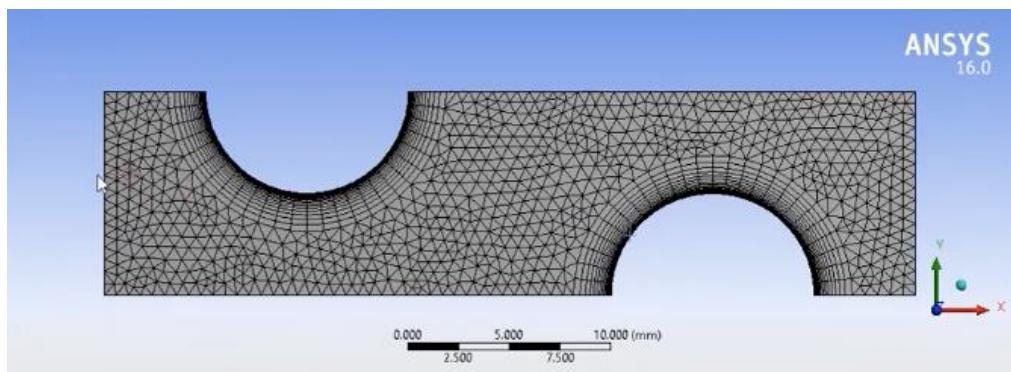


Fig 2. Mesh

### 3. Ansys Fluent

1. Open Ansys Fluent. Set Processes to Parallel and Number of Processors to 4.
2. In the Ansys Fluent Terminal, press ‘Enter’, followed by ‘mesh’, ‘modify-zones’, ‘make-periodic’, ‘inlet’, ‘outlet’, ‘no’, and ‘yes’. This will create the inlet as the periodic zone and the outlet as the shadow zone.
3. In Models set the model as Laminar. Turn on the Energy equation.
4. Create the appropriate material properties.
5. Under Boundary Conditions, set the ‘inlet’ periodic conditions (mass flow rate 0.05 kg/h and temperature 300 K).
6. Set the tubewall temperatures to 400 K.
7. Under Monitors, set the Residuals to 1e-06.
8. Initialise the solution at the inlet.
9. Set Number of Iterations at 500 and run the Calculations.
10. Obtain the required contours and graphs.

#### Screenshots from the File:

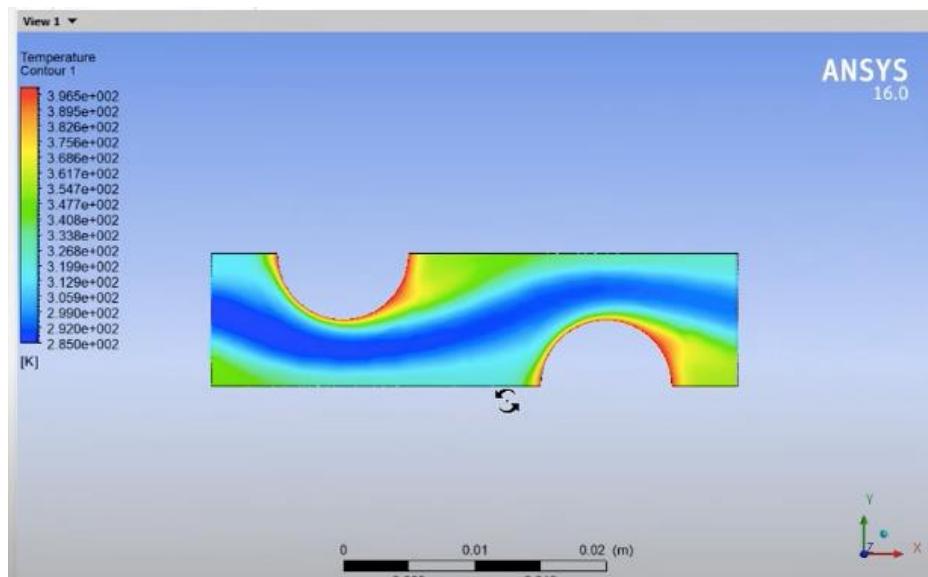


Fig 3. Temperature Contour

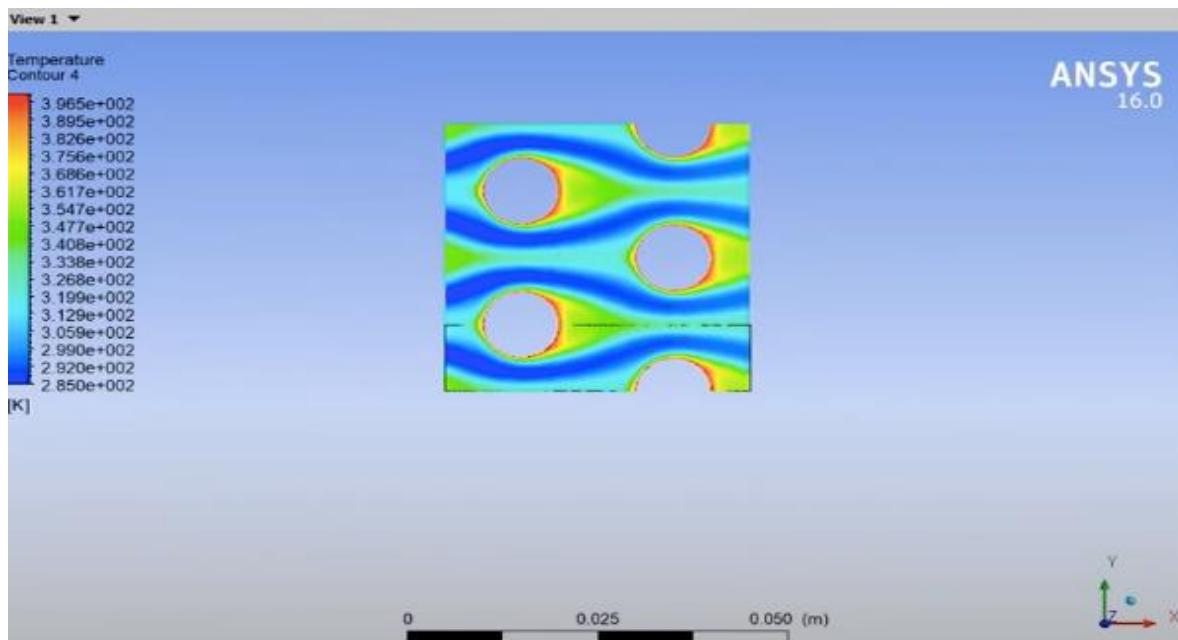


Fig 4. Temperature Contour with Periodicity