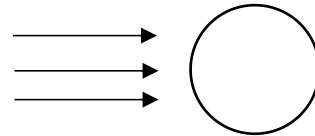


Computation Fluid Dynamics Assignment

Assignment: -

To Run a Simulation to Check Flow Over a Cylinder.

- Fluid Properties: Water Liquid
- Material: Vertical Cylinder
- Diameter Of Material: 1 Metre
- Fluid Reynold's Number: 40
- Mach Number: 0.3



Simulation Result: -

For This Simulation, We Have Used the Software Ansys Workbench in Order to Run This Simulation. Let's First Understand the Theoretical Part.

Theory: -

This is the experiment which is called Von Karmann Vortex Effect Over the Vertical Cylinder. Von Karman Vortex Effect is a recurring pattern of whirling vortices in fluid dynamics that is brought on by the process of vortex shedding, which is what causes the fluid flow to be unsteadily separated about the blunt objects.

In this simulation, fluid is taken as a water liquid, general properties of a water liquid is that the density of water liquid is 1000kg/m^3 at temperature 4°C and viscosity of water-liquid is $10^{-3}\text{ Pa}\cdot\text{s}$. Material taken in this experiment is Cylinder which is withstand vertical circumference or face or base of the cylinder directed perpendicular to the fluid flow. Diameter Of Cylinder is taken as 1 meter. Reynold's Number of fluid flow is taken as 40 which implies that the fluid flow is steady and laminar. Mach Number is 0.3 which implies that the fluid flow is incompressible which implies that the temperature over the fluid flow remains constant. In this simulation, we have to check the unsteadiness of the steady flow of fluid about the blunt object when the steady flow pass over the cylinder and then the unsteady separation of the fluid which will be check by the contour and streamlines.

Software: -

In this assignment, the software used is Ansys Workbench, in Ansys Workbench, Project be opened and Ansys Fluent is opened and now we there are five processes given Geometry, Mesh, Solve, Results. Geometer and Mesh acts as a Pre-processor, Solve is a solver which is used to solve the governing equations by using Finite-Volume Method and Results acts as Post Processor which gives the results of contour and streamlines.

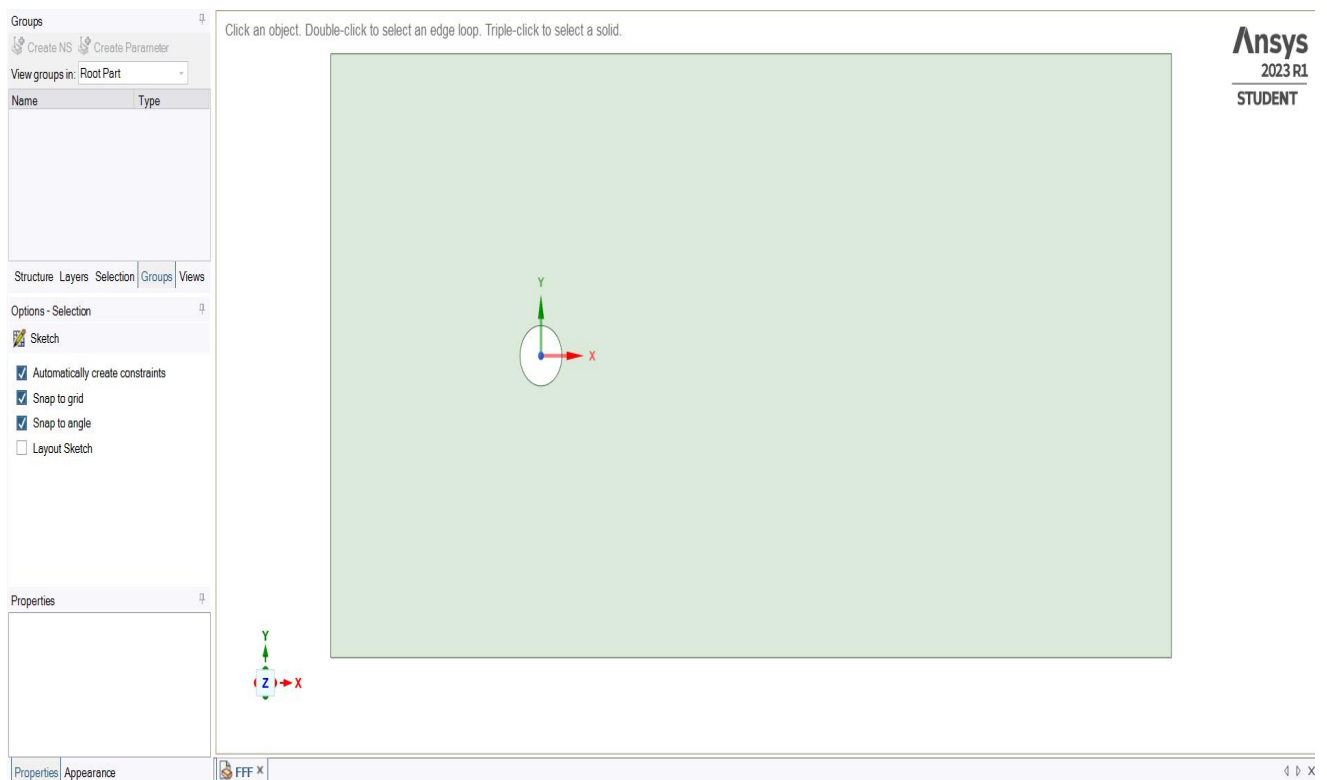
Pre Processor: -

This is the input for the fluid flow to Solver.

Step 1: Creating A Geometry: -

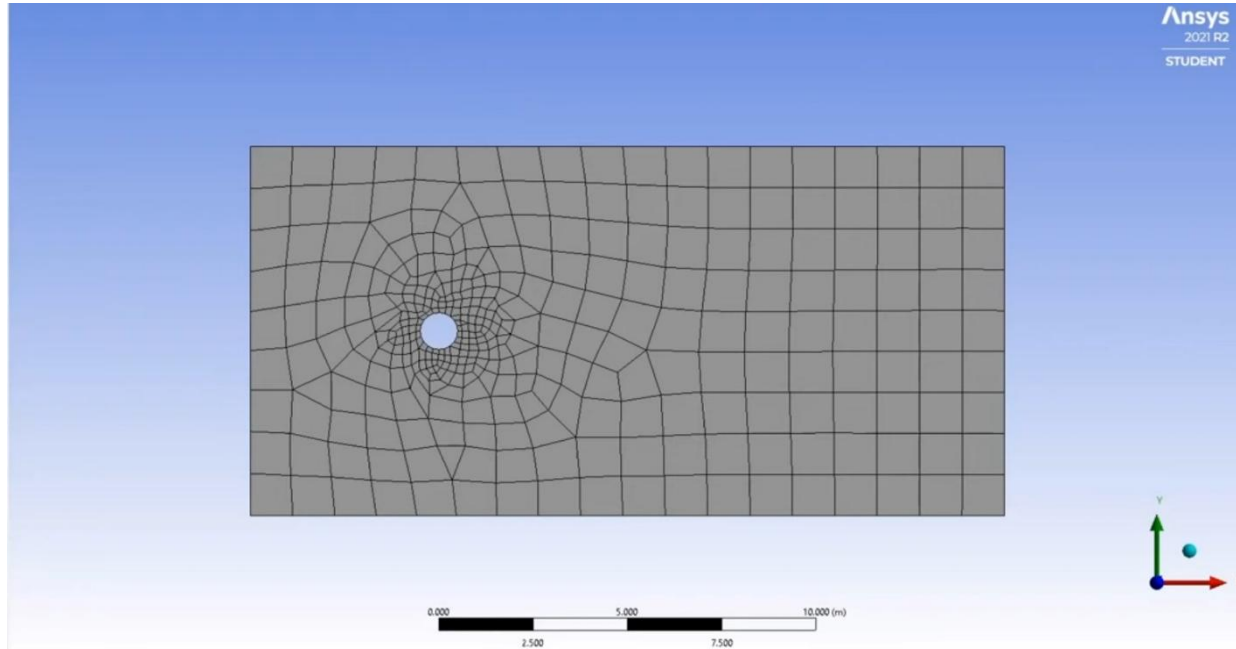
- On Clicking The Geometry, We Have Start Ansys SpaceClaim.
- On Sketch Editing Tool, Drawn $20 \times 10 \text{ m}^2$ Rectangle with centre at (0,0) coordinate.
- On that Rectangle, Drawn A 1m Diameter Circle.
- Put That centre of Circle at (-5,0).
- This how it has given two surfaces one for rectangle and other cylinder.
- Now the cylinder surface is deleted for creating cylinder boundary layer.
- This is how We have Created Two Dimensional Geometry for fluid flow setup.
- Circle Outline is the boundary layer for cylinder.
- Surface inside the rectangle is domain for fluid flow.

SPACECLAIM GEOMETRY:-



Step 2: Generating The Mesh: -

On Starting The Mesh, two dimensional geometry we have created shown on this mesh setup. Now we will click on the Generate Mesh Option to see what mesh it will be generated by itself.



Now To Edit The Meshing, We will Right Click and Select Sizing, Named it as Edge Sizing , Select Circle Edge through edge selecting tool and element size is written as 0.025m and mesh is updated. We will Right Click and Select another Sizing, Named it as Face Sizing , Select Rectangle Face through face selecting tool and element size is written as 0.1m and mesh is updated. Now the mesh is more refined and quite orthogonal in the fluid domain to make calculation simple for the solver.

Now we will Right Click On Mesh, Select Inflation For Geometry We will select the face of the rectangle and for boundary we will select the edge of the rectangle and edge of the cylinder. First Layer Height as 0.025m and selecting maximum layer of 20, growth rate of 1.5 and the mesh is updated.

Edge Sizing: -

Details of "Edge Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	2.5e-002 m
Advanced	
Behavior	Soft
<input type="checkbox"/> Growth Rate	Default (1.2)
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

Face Sizing: -

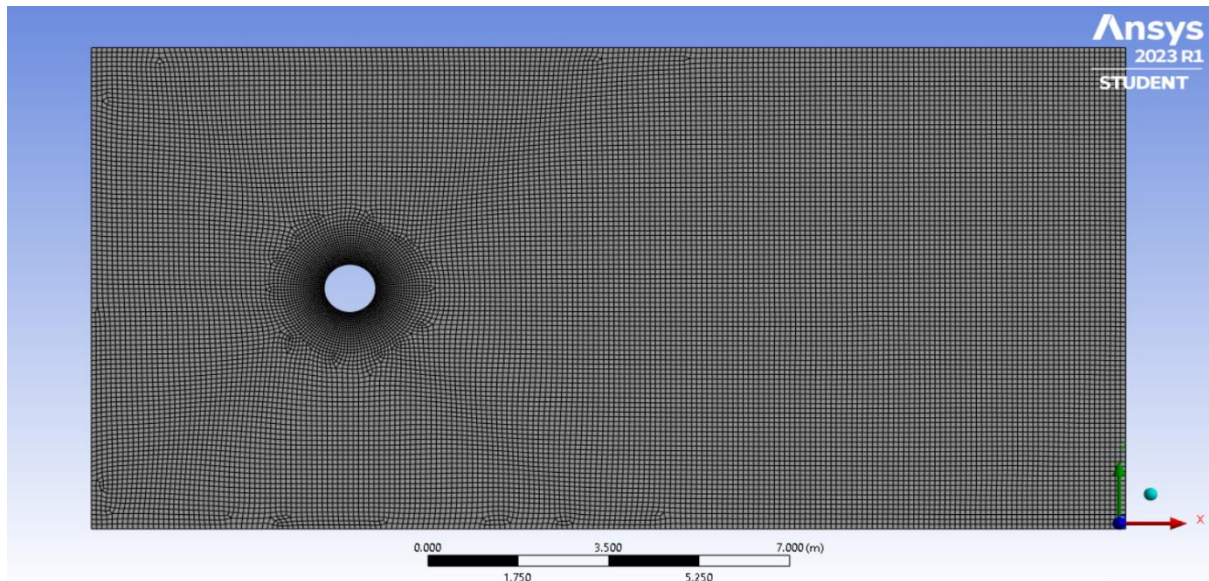
Details of "Edge Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	2.5e-002 m
Advanced	
Behavior	Soft
<input type="checkbox"/> Growth Rate	Default (1.2)
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

Inflation: -

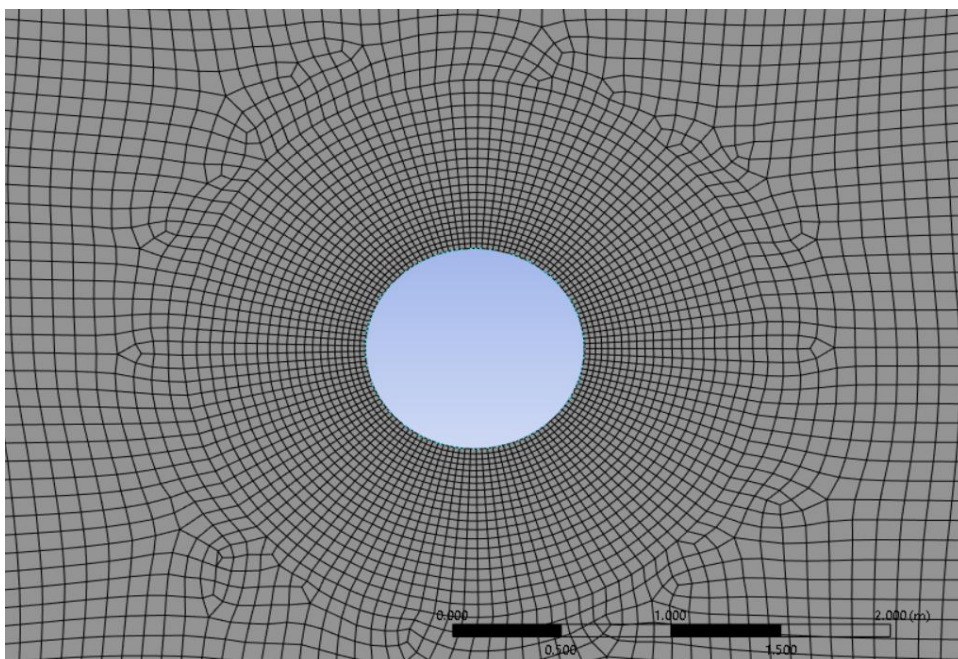
Details of "Inflation" - Inflation	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	1 Edge
Inflation Option	First Layer Thickness
First Layer Height	2.5e-002 m
<input type="checkbox"/> Maximum Layers	20
<input checked="" type="checkbox"/> Growth Rate	1.5
Inflation Algorithm	Pre

Now, We will name the boundary layer of the rectangle and cylinder. On clicking the left side of the rectangle, we will named it as **inlet** using named selection, on clicking the right side of the rectangle, we will named it as **outlet** and then on clicking both the top and bottom side of the rectangle, we will named it as **wall** and then on clicking the edge of the cylinder, we will named it as the **cylinder** and thus we have named each edges using named selection.

On Generating The Mesh :-



Mesh Of Vertical Cylinder:-



Step 3: Solving The Equations:-

On Opening the setup selecting 2-D Simulation with double precision, for solver type we are going to want to keep this to **pressure based absolute and velocity formulation absolute and 2-D space as Planar** is fine here going to want to change the time to **transient** because we want to see the **von Karman effect** in the simulation. For the model type, we are going to select the **viscous laminar**. Now on the material side, select the fluid database as **water-liquid** as the database is pre-defined for the properties of the water- fluid is already given such as **density** and **viscosity** so we will select the model flow as water-liquid.

Now Selecting the cell zone condition, select the fluid and then select the **fff_surface(fluid id=2)** and here select the material name as **water-liquid** and then apply it. Now Selecting the boundary condition, all the named selection is given in the boundary condition. Checking each boundary condition **cylinder, wall as wall** and **inlet as velocity inlet** and **outlet as pressure outlet**. On selecting the inlet and then setting velocity magnitude as **0.00004m/s** the reason for this because we have to set the **Reynold's Number of 40**.

Selecting The Reference Value, Selecting Compute from **inlet**, the pre-defined reference values given in that setup, the **density as 998.2 kg/m³**, **viscosity as 0.001003 Pa-s**, **velocity as 0.00004m/s** and **temperature as 298K** constant, the reason for this to make a constant temperature at 298 K, for making density constant to make it incompressible flow of **Mach Number=0.3**.

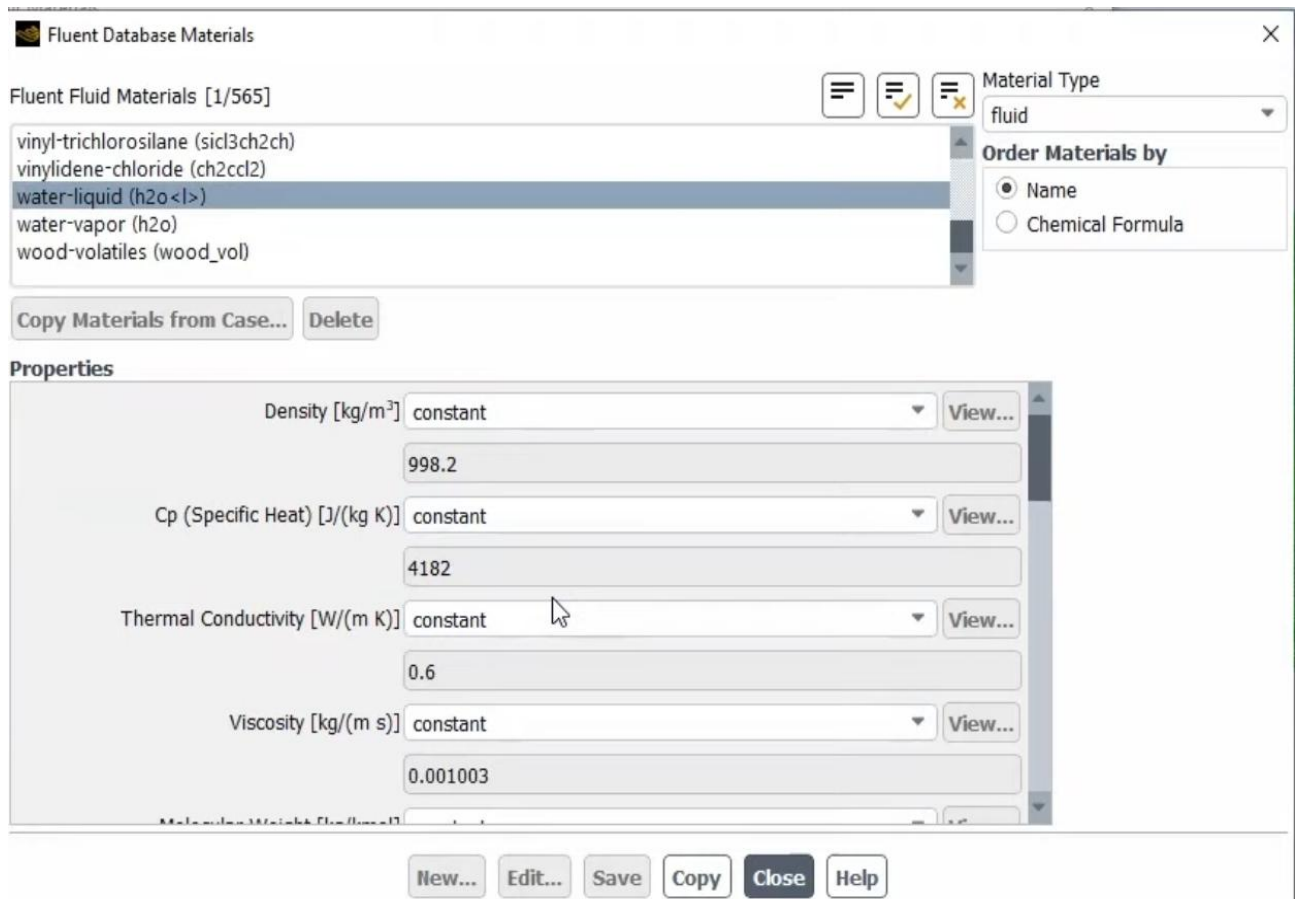
Reference Values

Compute from
inlet

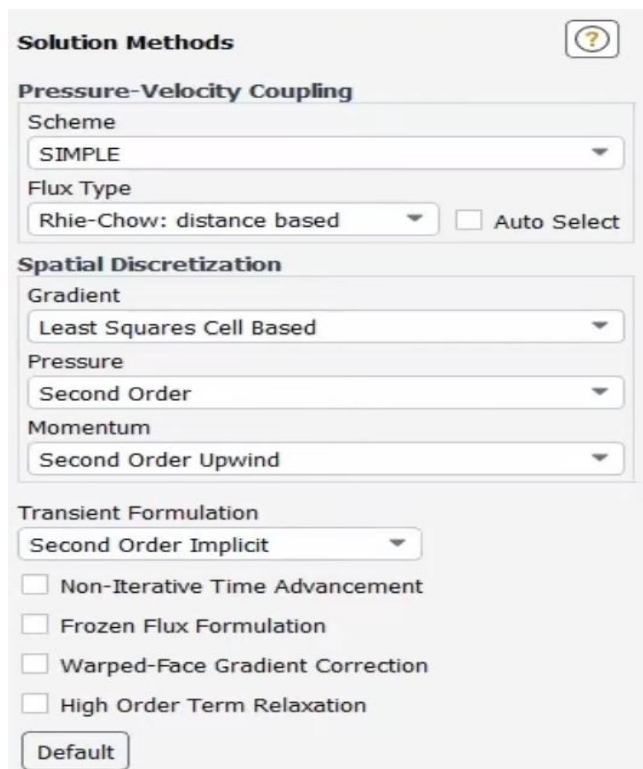
Reference Values

Area [m ²]	1
Density [kg/m ³]	998.2
Depth [m]	1
Enthalpy [J/kg]	0
Length [m]	1
Pressure [Pa]	0
Temperature [K]	288.16
Velocity [m/s]	0.0001
Viscosity [kg/(m s)]	0.001003
Ratio of Specific Heats	1.4
Yplus for Heat Tran. Coef.	300

Reference Zone



For Solution Methods, Clicking On Methods making a Scheme of **SIMPLE** Algorithm, Gradient of **Least Squares Cell Based**, Pressure as **Second Order** and Momentum as **Second Order Upwind** and Transient Formulation as **Second Order Implicit**.



For Simulation Clicking on Report Files, Add **Drag Report** to calculate the drag coefficient over the cylinder and check the print as console down and then apply it and **Lift Report** and lift coefficient over the cylinder and check the print as console down and then apply it

Selecting the initialization as **hybrid initialization** and then initialize it and then at run calculation select the number of time steps as **4000** and time step size as **200s** and max iteration as **40** and the click on **run calculation** to make the solver to calculate for what we have input in pre processor.

Run Calculation:-

Run Calculation

Check Case... Preview Mesh Motion...

Time Advancement

Type: Fixed Method: User-Specified

Parameters

Number of Time Steps: 4000 Time Step Size [s]: 250

Max Iterations/Time Step: 40 Reporting Interval: 1

Profile Update Interval: 1

Options

☐ Extrapolate Variables

☐ Report Simulation Status

Solution Processing

Statistics

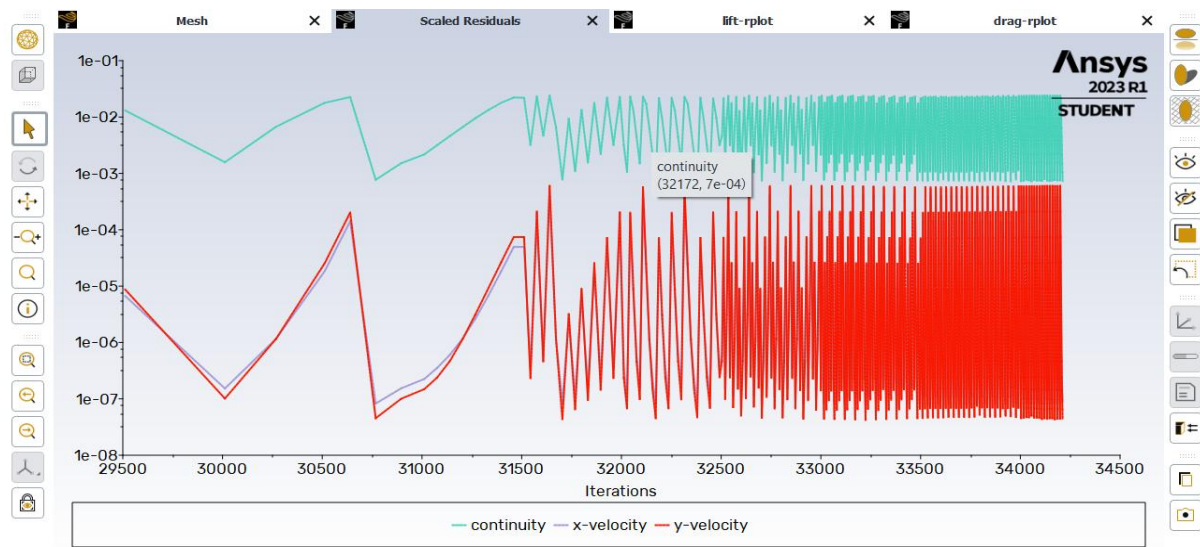
☐ Data Sampling for Time Statistics

Data File Quantities...

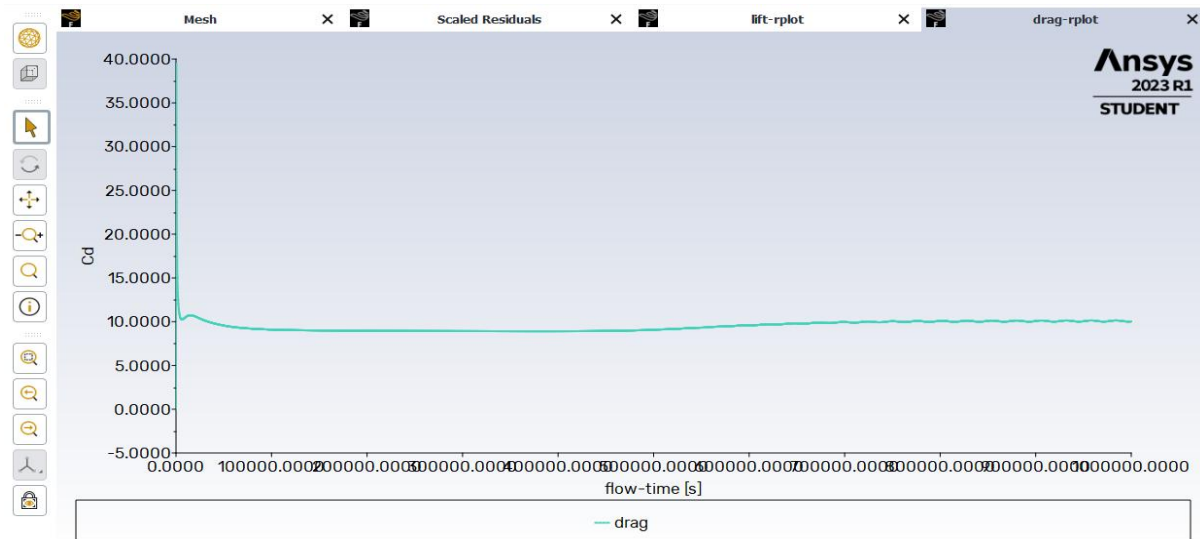
Solution Advancement

Calculate

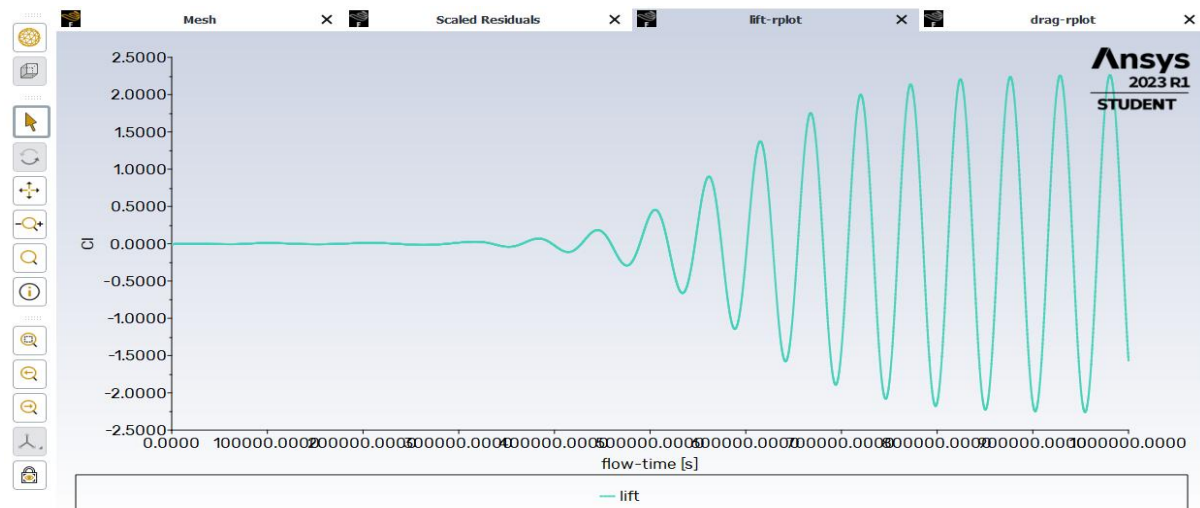
Scaled Residuals:



Drag Reports:



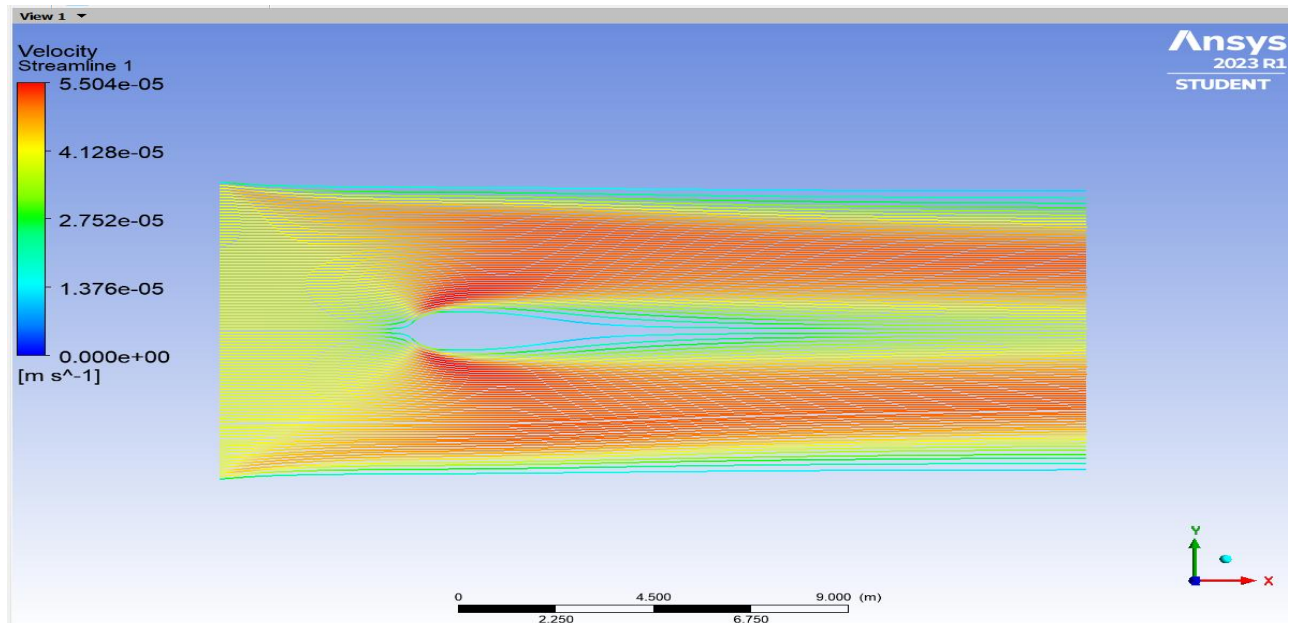
Lift Reports:



Step 4: Post Processor:-

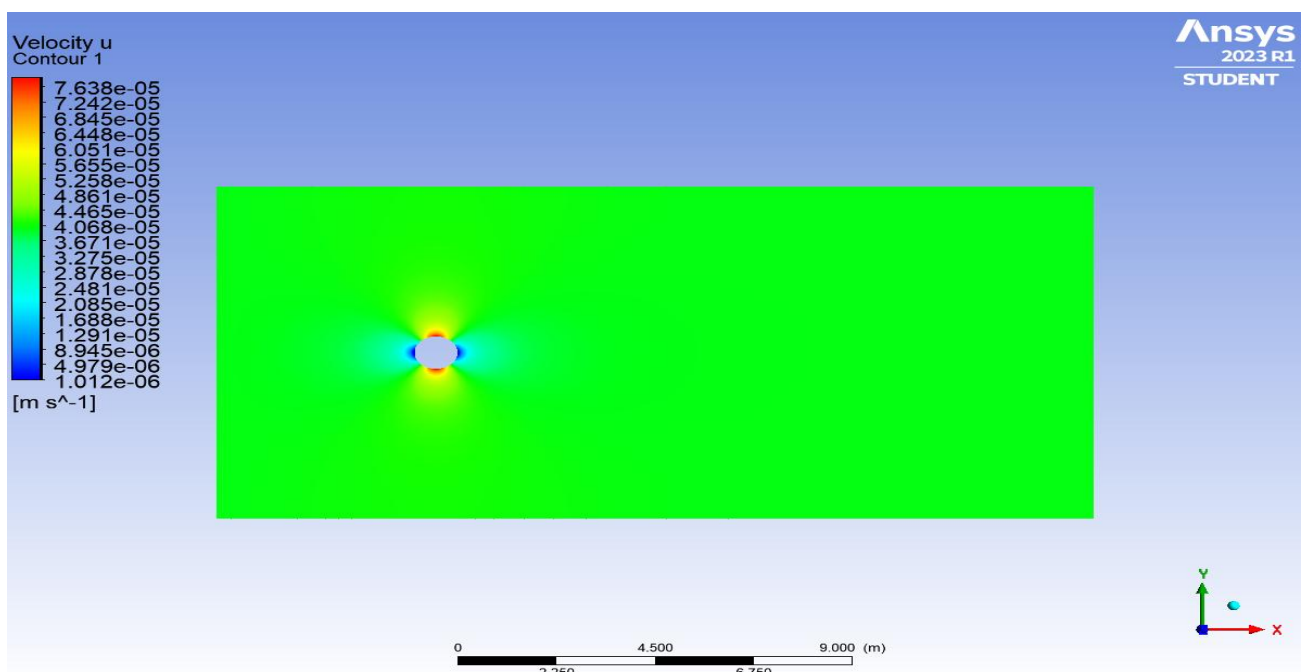
Opening the CFD-Post, Selecting the contour and streamlines on the option given in the ribbon. Selecting domains as **fff_surface**, Locations as **Symmetry 1**, Variable as **Velocity u** and Range as **Global** and Number Of Contour as **100**. Now then at different timesteps check the different contour and streamlines in the CFD Post and thus simulation has don on von Karman effect on vertical cylinder. Timesteps varies from 0 to 4000.

Streamline:-

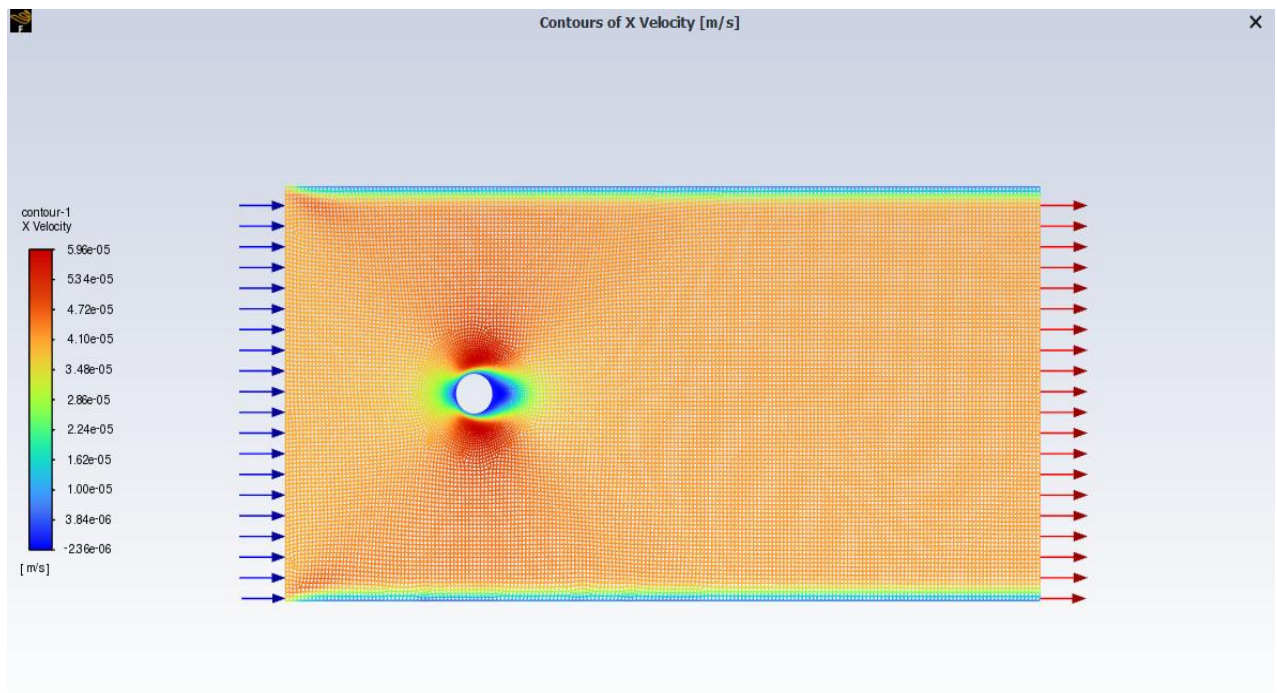


Contour:-

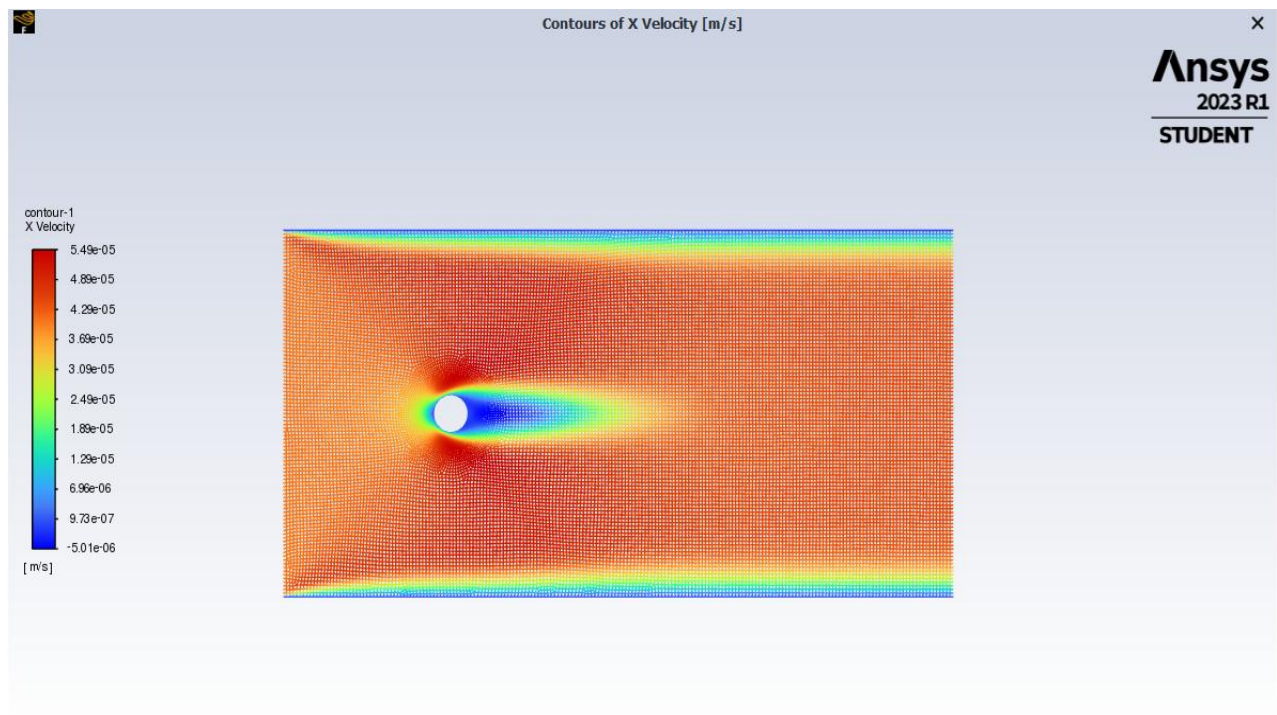
At Timesteps 10: -



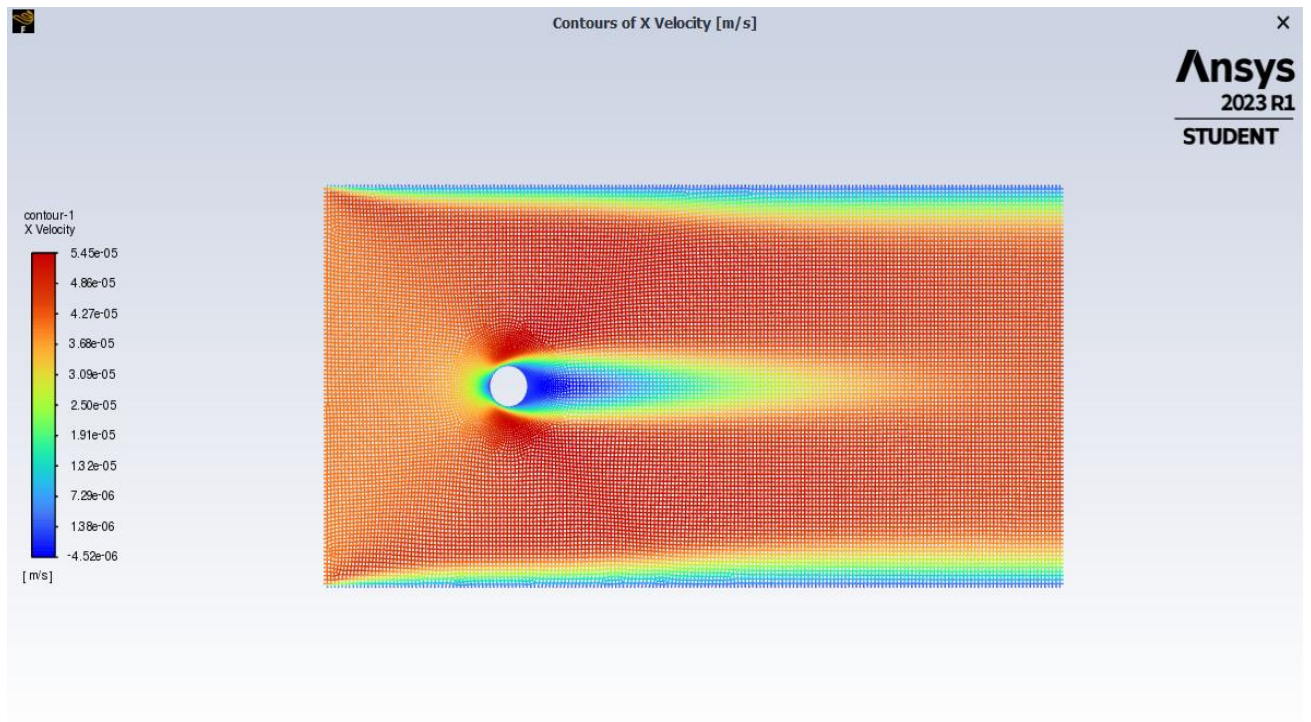
At Timesteps 100: -



At Timesteps 1500: -



At Timesteps 2800:



At Timesteps 4000:

