INDIAN INSTITUTE OF TECHNOLOGY GUWAHATI

INTERNSHIP REPORT

- Drag and Lift analysis on a stationary circular cylinder using openFOAM
- Automation of VFD and Displacement sensors
- Meshing of wind turbine using Gmsh

SAURABH KUMAR
BACHELOR OF TECHNOLOGY IN MECHANICAL ENGINEERING

May 2023 – June 2023

Supervised by : Atul K. Soti

Drag and Lift Analysis on stationary circular cylinder

- **AIM**

Simulate the steady and unsteady flow of water over a cylinder and analyse lift and drag coefficients.

INTRODUCTION

Flow around objects exists from commonly observed movement of animals in nature such as the flight of birds ranging from the smallest in size to the largest to man-made applications viz., in heat exchangers, chimney stacks, and commercial and cargo flight aircrafts respectively and a growing significance to understand the different fluid flow phenomena like lift and drag coefficients, boundary layer separation, vortex shedding, etc., has been extensively exploited in recent years and in many cases to solve existing issues with the previously mentioned engineering applications. One such peculiar interest that gained importance over time was the flow around a circular cylinder and more specifically to comprehend the phenomenon of Karman vortex street which is when the wake behind the circular cylinder becomes unstable beyond a certain value of Reynolds number

In this project, we are going to simulate the flow over a cylinder inorder to capture the vortex shedding that is formed behind the cylinder surface and drag and liftc coefficients. This is done for various cases by varying the velocity of the fluid flow for each case inorder to change the reynolds number and so by taking the user defined fluid with constant density of 1 kg/m^3 and viscosity of 0.05 kg/m-s. This simulation is done with steady state and transient state for Reynolds number is 100 and so as to analyse the vortex formation for both the cases and also to calculate the Strouhal number.

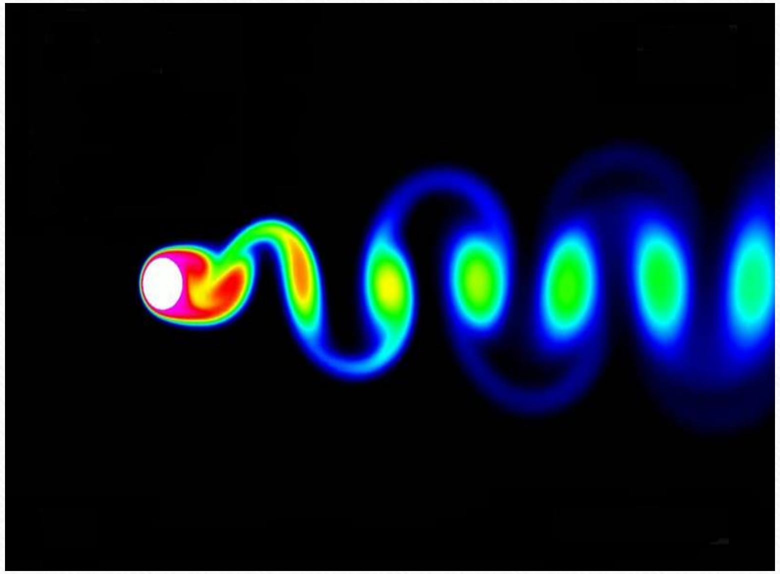
THEORY

Karman vortex

In Fluid Dynamics, a Karman vortex street is a repeating pattern of swirling vortices, caused by a process known as Vortex Shedding, which is responsible for the unsteady separation of Fluid around blunt bodies. It is named after the engineer and fluid dynamicist Theodore Von Karman.

Vortex shedding is a phenomenon when the wind blows across a structural member, vortices are shed alternately from one side to the other, and where alternating low-pressure zones are generated on the downwind side of the structure giving rise to a fluctuating force acting at right angles to the wind direction.

Vortices are a major component of turbulent flow. The distribution of velocity, vorticity (the curl of the flow velocity), as well as the concept of circulation is used to characterize vortices. In most vortices, the fluid flow velocity is greatest next to its axis and decreases in inverse proportion to the distance from the axis.



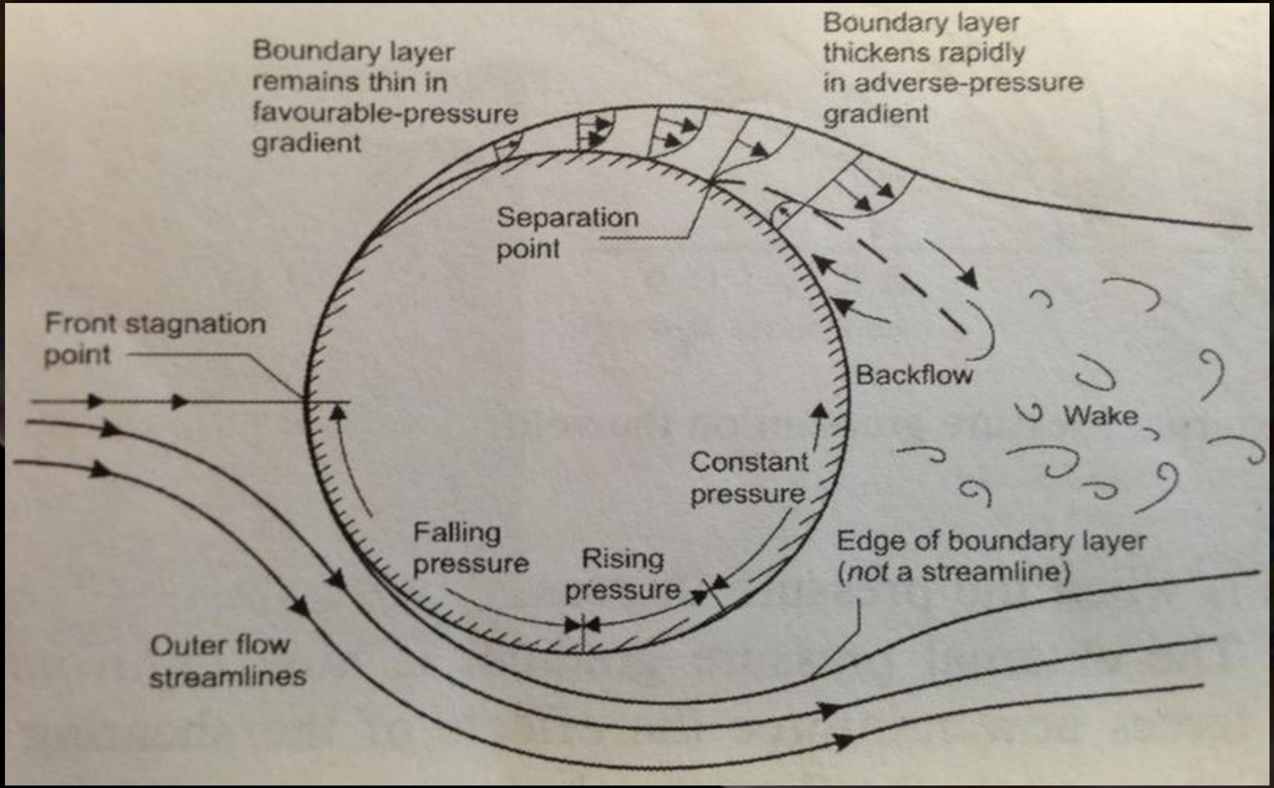
- For common flows (the ones which can usually be considered as incompressible or isothermal), the kinematic viscosity is everywhere uniform over all the flow field and constant in time, so there is no choice on the viscosity parameter, which becomes naturally the kinematic viscosity of the fluid being considered. On the other hand, the reference length is always an arbitrary parameter, so particular attention should be put when comparing flows around different obstacles or in channels of different shapes: the global Reynolds numbers should be referred to the same reference length. This is actually the reason for which most precise sources for air foil and channel flow data specify the reference length at a prefix to the Reynolds number. The reference length can vary depending on the analysis to be performed: for a body with circle sections such as circular cylinders or spheres, one usually chooses the diameter; for an air foil, a generic non-circular cylinder or a bluff body or a revolution body like a fuselage or a submarine, it is usually the profile chord or the profile thickness, or some other given widths that are in fact stable design inputs; for flow channels usually, the hydraulic diameter about which the fluid is flowing.
- When a single vortex is shed, an asymmetrical flow pattern forms around the body and changes the pressure distribution. This means that the alternate shedding of vortices can create periodical lateral (sideways) forces on the body in question, causing it to vibrate. If the vortex shedding frequency is similar to the natural frequency of a body or structure, it causes resonance.

Physics behind Vortex formation:

As a Fluid particle flows towards the leading edge of a cylinder, the pressure on the particle rises from the free stream pressure to stagnation pressure. The high fluid pressure near the leading edge impels flow about the cylinder as a boundary layer develops about both sides. The high pressure is not sufficient to force the flow about the back of the cylinder at high Reynolds number. Near the widest section of the cylinder, the boundary layer separates from each side of the cylinder surface and form 2 shear layers. Since the innermost portion of the shear layers, which is in contact with the cylinder, moves much more slowly than the outermost portions of the shear layers which is in contact with the free flow, the shear layers roll near the wake.

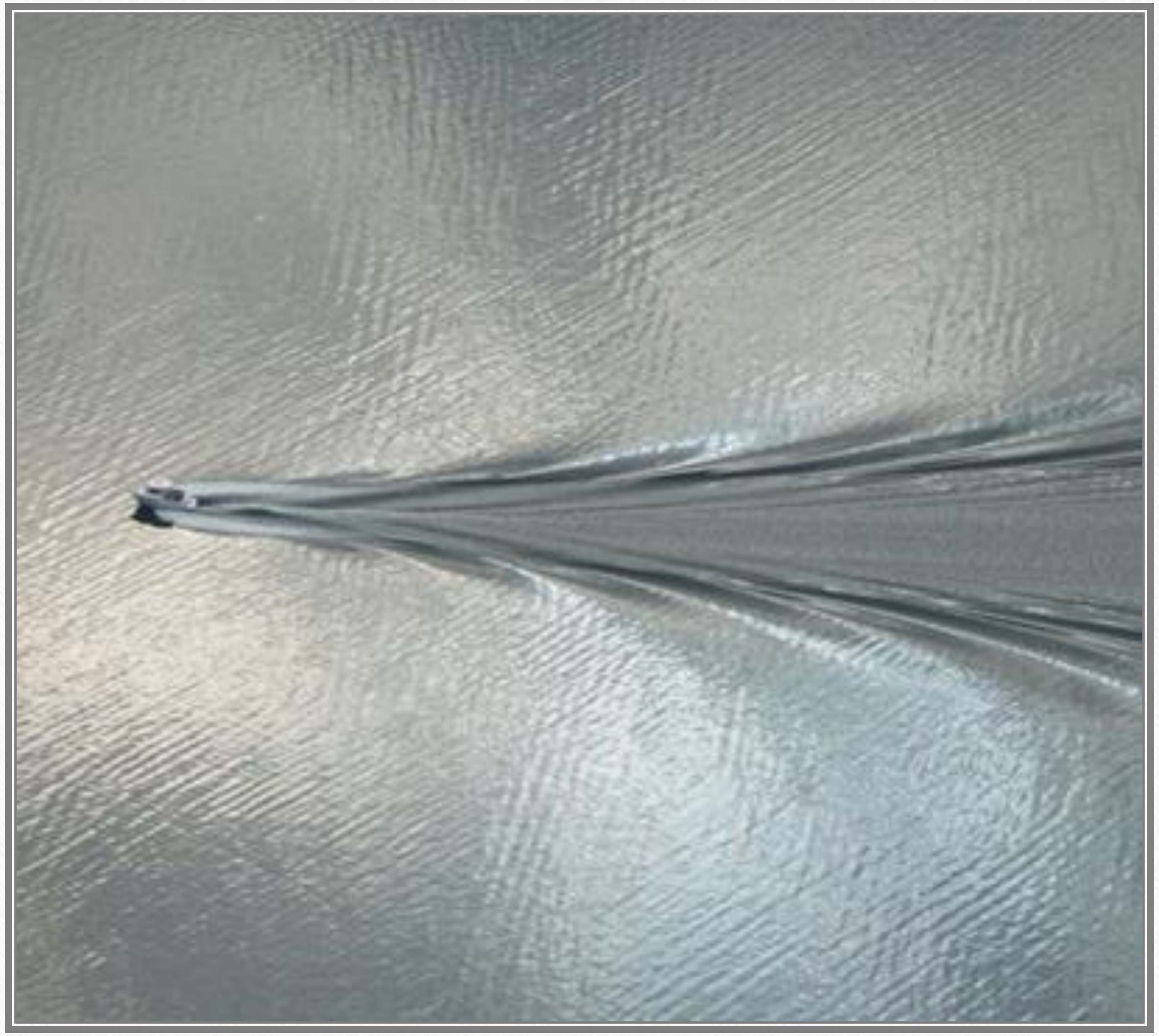
Understanding the physics behind vortex shedding is quite important from an engineering point of view. For example, if the bluff structure is not mounted rigidly and the frequency of vortex shedding matches the resonance frequency of the structure, then the structure can begin to resonate and starts vibrating. These vibrations can cause severe damage to the structure.

- The boundary layer separates from the surface forms a free shear layer and is highly unstable. This shear layer will eventually roll into a discrete vortex and detach from the surface (a phenomenon called vortex shedding). Another type of flow instability emerges as the shear layer vortices shed from both the top and bottom surfaces interact with one another. They shed alternatively from the cylinder and generates a regular vortex pattern (the Karaman vortex street) in the wake. The vortex shedding occurs at a discrete frequency and is a function of the Reynolds number. The dimensionless frequency of the vortex shedding, the shedding Strouhal number, $St = f\ D/V$, is approximately equal to 0.21 when the Reynolds number is greater than 1,000.



Wake

- Wake is the region of recirculating flow immediately behind a moving or stationary blunt body, caused by viscosity, which may be accompanied by flow separation and turbulence. It is basically the region of disturbed flow downstream of a solid body moving through a fluid.



Wake pattern created by small boat

Reynold’s Number

Reynold’s number is a dimensionless quantity that is used to determine the type of flow pattern as laminar or turbulent while flowing through a pipe. Reynolds number is defined by the **ratio of inertial forces to that of viscous forces**.

$$Reynold's\ number = \frac{Inertial\ Force}{Viscous\ Force}$$

The formula is given by,

$$Re = \frac{\rho v D}{\mu}$$

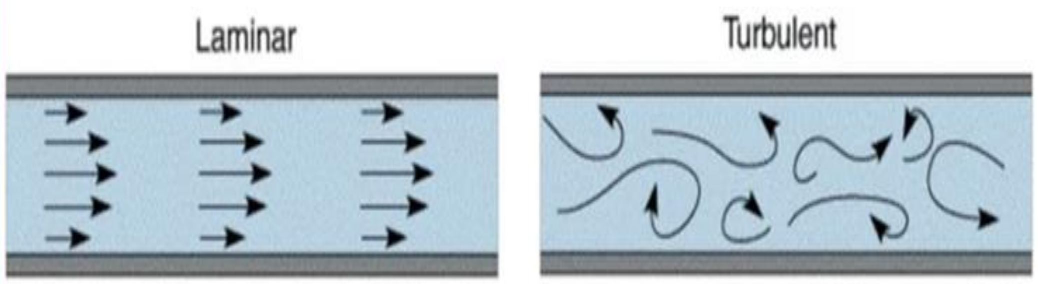
Where,

ρ = Density of fluid Kg/m^3

D = Diameter of Cylinder m

v = Velocity of Fluid m/s

μ= Kinematic Viscosity of Fluid m^2/s



Reynolds Number	Condition of flow
Re < 2000	Laminar
2000 < Re < 4000	Transitional
Re > 4000	Turbulent

Drag Co-efficient

Drag coefficient is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment, such as air or water. A lower drag coefficient indicates the object will have less aerodynamic or hydrodynamic drag. The drag coefficient is always associated with a particular surface area.

$$C_D = \frac{F_D}{\frac{1}{2}.\rho.A.V^2}$$

Where,
F_D = Drag Force
ϱ = Fluid Density
A = Projected Area of Object with respect to which drag force is calculated
v = Flow Velocity.

Lift Co-efficient

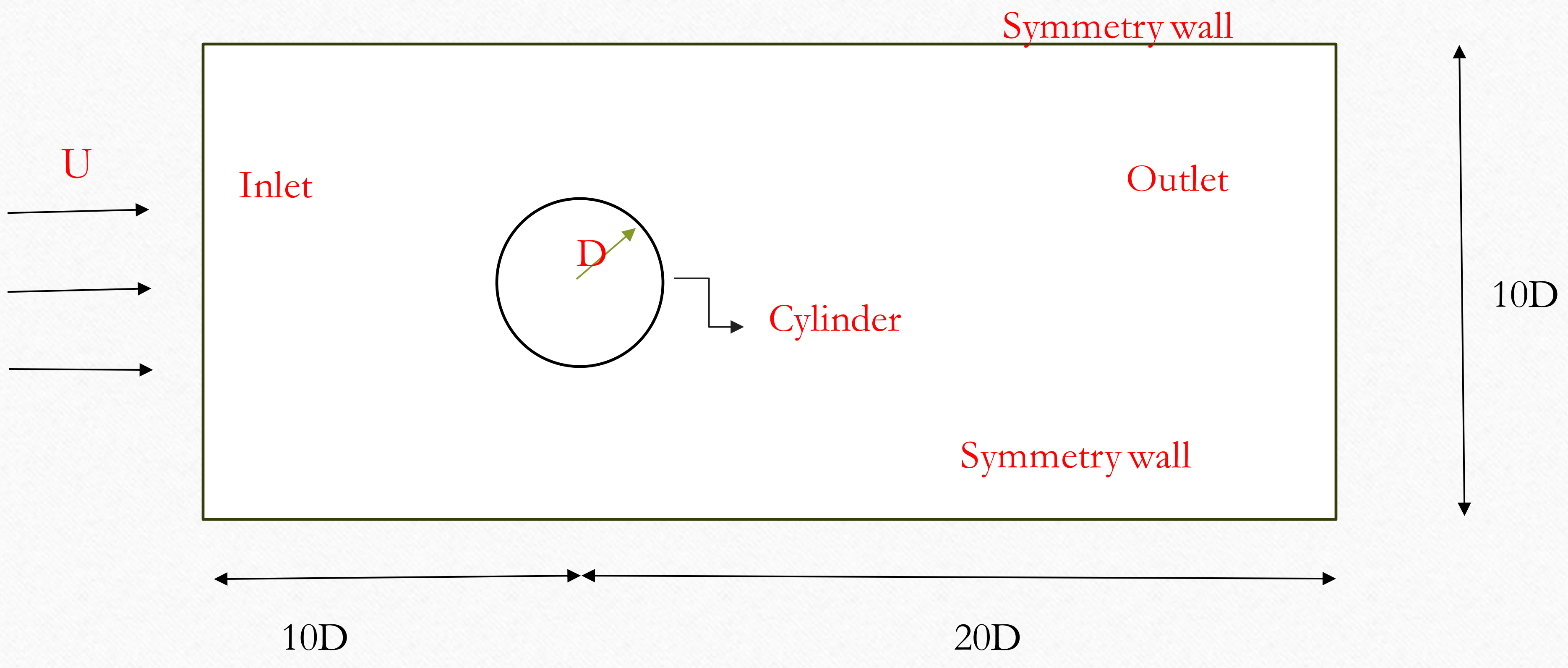
The lift coefficient (CL) is a dimensionless coefficient that relates the lift generated by a lifting body to the fluid density around the body, the fluid velocity and an associated reference area. CLCL is a function of the angle of the body to the flow, its Reynolds number and its Mach number.

$$C_L = \frac{F_L}{\frac{1}{2}.\rho.A.V^2}$$

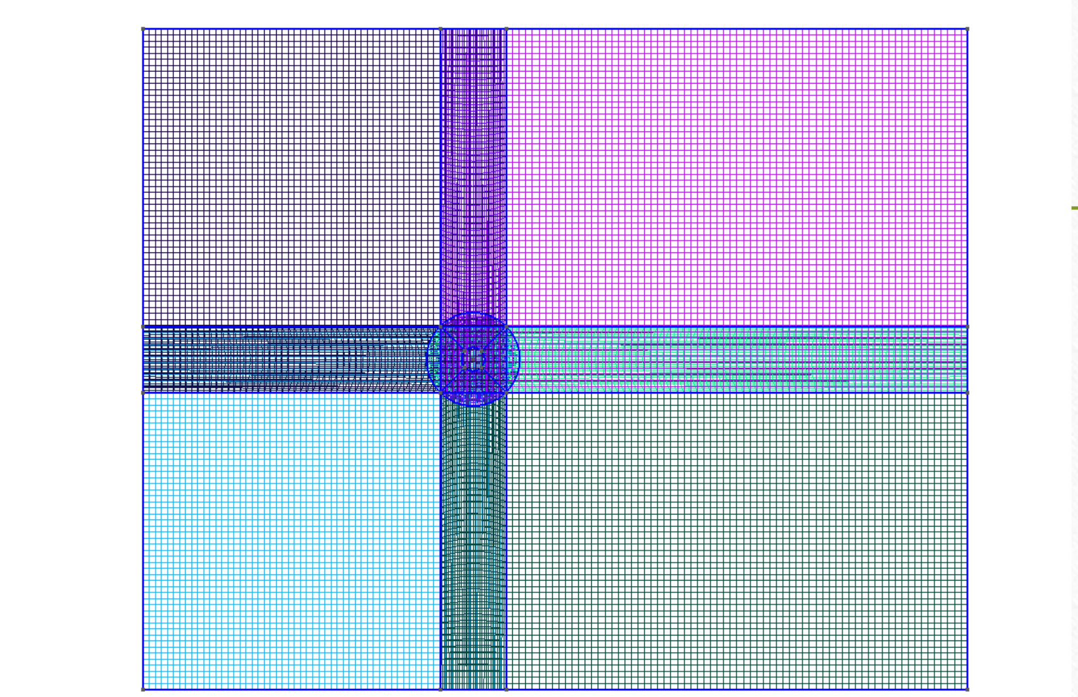
Where,
FL = Lift Force
ρ = Fluid Density
A = Projected Area of Object with respect to which lift force is calculated
v = Flow Velocity

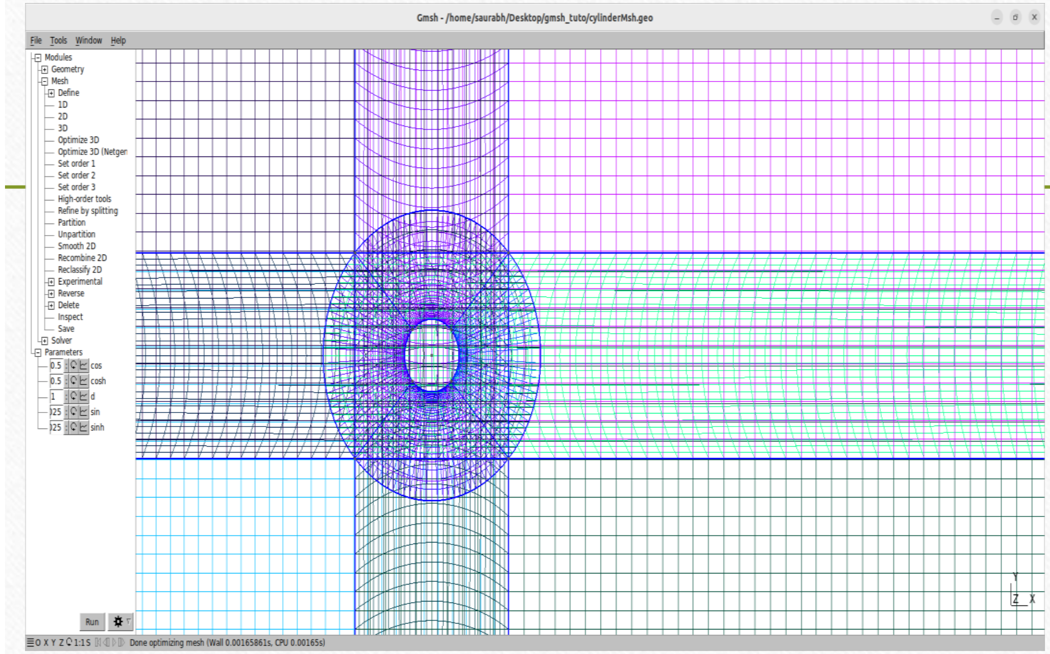
PREPROCESSING AND MODELLING APPROACH

The computational domain for the flow over a cylinder is constructed in space claim



Mesh:





Mesh created using Gmsh software

Link for Gmsh file : Gmsh geo file

Open | *p ~/openFoam_tut/cylinderRef_Flow/0

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  v2212                                 |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 2 -2 0 0 0 0];

internalField   uniform 0;

boundaryField
{
    symmetry
    {
        type            symmetry;
    }

    outlet
    {
        type            fixedValue;
        value           uniform 0;
    }

    inlet
    {
        type            zeroGradient;
    }

    cylinder
    {
        type            zeroGradient;
    }
}
```

Open | U ~/openFoam_tut/cylinderFlow/0 | Save

p | U

```
boundaryField
{
    symmetry
    {
        type            symmetryPlane;
    }

    outlet
    {
        type            zeroGradient;
    }


    inlet
    {
        type            uniformFixedValue;
        uniformValue    constant (1 0 0);
    }

    cylinder
    {
        type            symmetry;
    }

    front
    {
        type            empty;
    }

    back
    {
        type            empty;
    }

    defaultFaces
    {
        type            empty;
    }
}


// ************************************************************************* //
```

Boundary conditions for P and U file from openFOAM

Open

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  v2212                                 |
|   \\  /    A nd           | Website:  www.openfoam.com                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

application     icoFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         300.0;

deltaT          0.01;

writeControl    timeStep;

writeInterval   1000;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

```

Open

```

runTimeModifiable false;


functions
{
forces_object
{
type forces;
functionObjectLibs ("libforces.so");

enabled true;

//writeControl outputTime;
writeControl adjustableRunTime;
writeInterval 0.001;

patches ("cylinder");

pName p;
Uname U;

////Density only for incompressible flows
rho rhoInf;
rhoInf 1;

////Centre of rotation
CofR (0 0 0);
}
/////////////////////////////////////////////////////////////////////////////


/////////////////////////////////////////////////////////////////////////////
forceCoeffs1
{
// Mandatory entries
type forceCoeffs;
libs ("libforces.so");
patches ("cylinder");


// Optional entries

// Field names
```

Open

```
// Optional entries

// Field names
p p;
U U;
rho rhoInf;
rhoInf 1;

// Reference pressure [Pa]
pRef 0;

// Include porosity effects?
porosity no;

// Store and write volume field representations of forces and moments
writeFields no;

// Centre of rotation for moment calculations
CofR (0 0 0);

// Lift direction
liftDir (0 1 0);

// Drag direction
dragDir (1 0 0);

// Pitch axis
pitchAxis (0 0 1);

// Freestream velocity magnitude [m/s]
magUInf 1;

// Reference length [m]
lRef 1;

// Reference area [m2]
Aref 1;


writeControl timeStep;
writeInterval 1;
}
}
// ************************************************************************* //
```

Control Dict. File : openFOAM

RESULTS AND POST PROCESSING:

Reynolds number : 100

Flow : steady

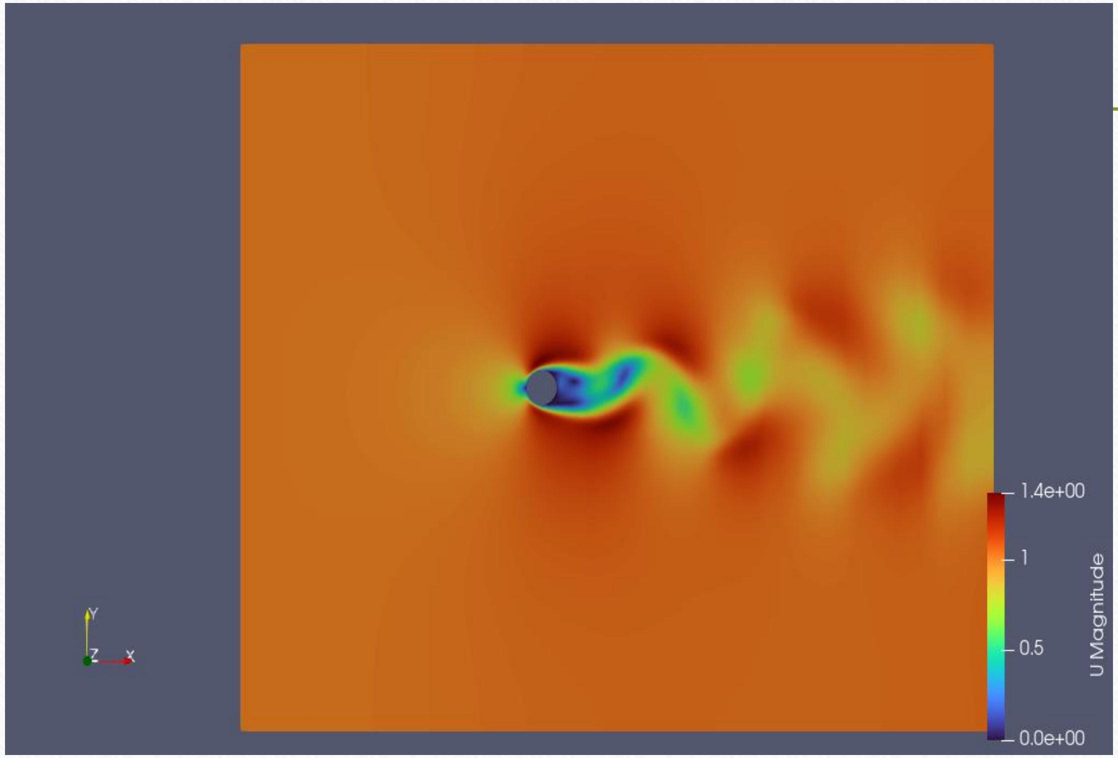


Fig. : Velocity contours as obtained in Paraview

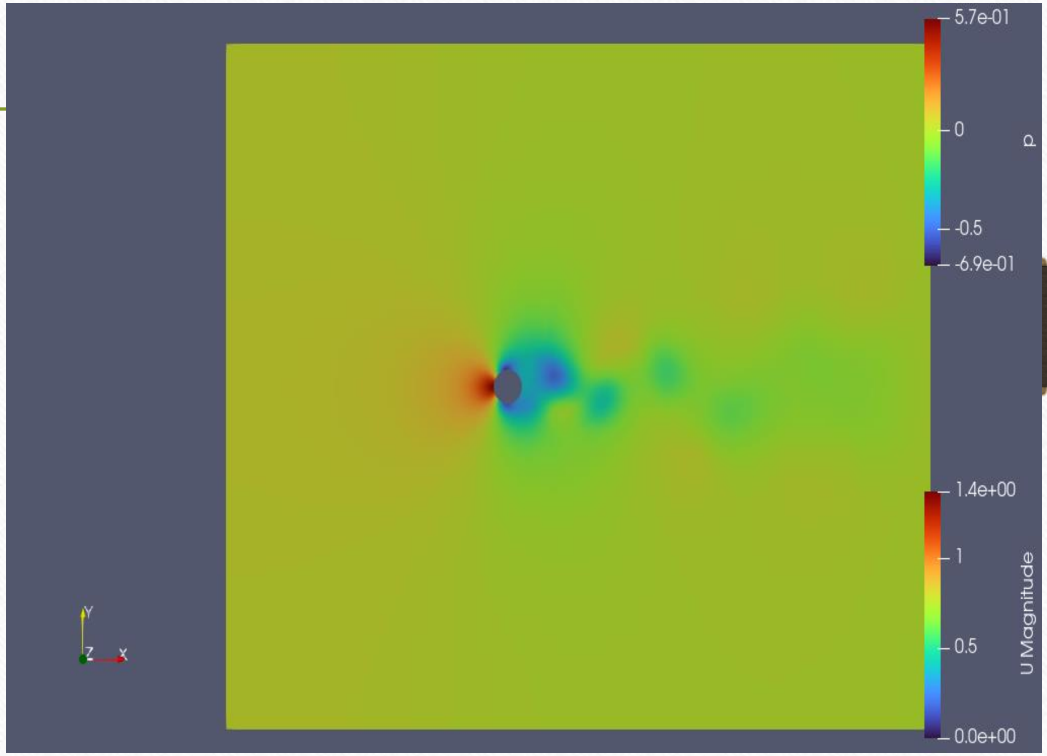
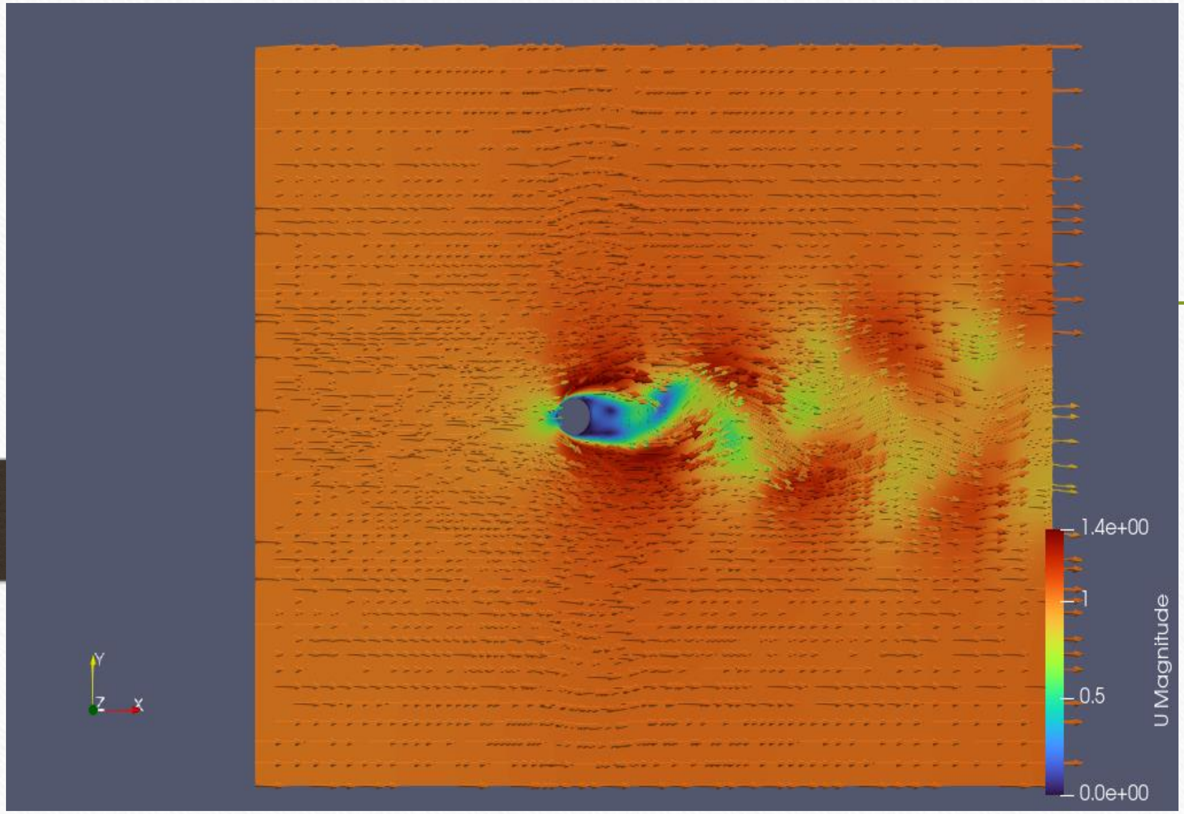
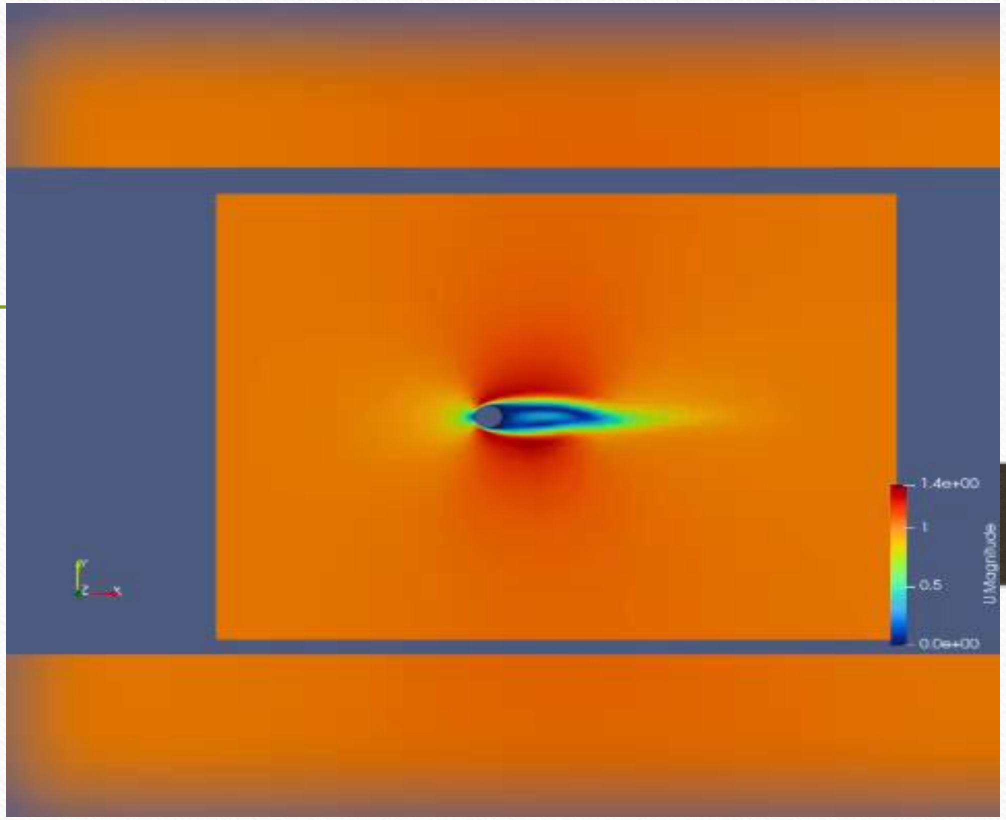


Fig. : Pressure contours as obtained in Paraview



Velocity Vector



Animation

RESULTS:

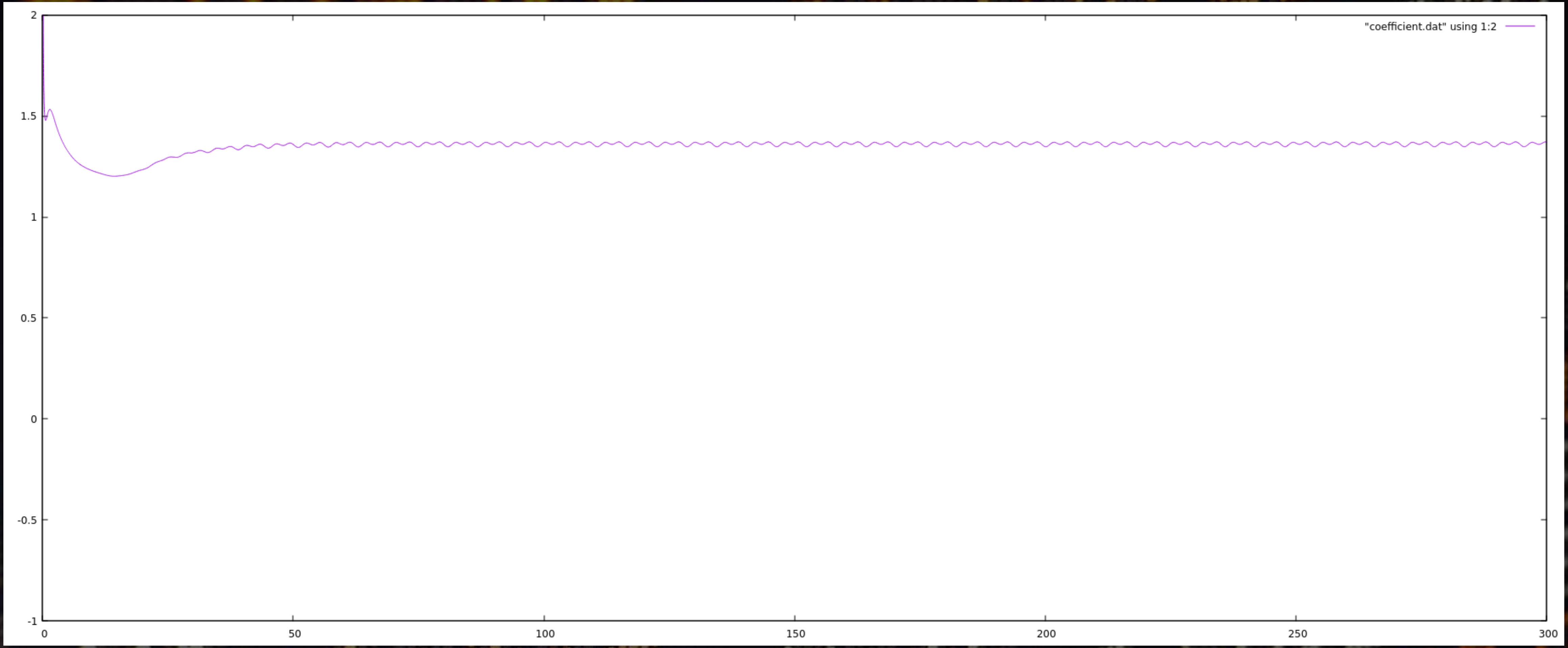


Fig. : Plot of drag with time

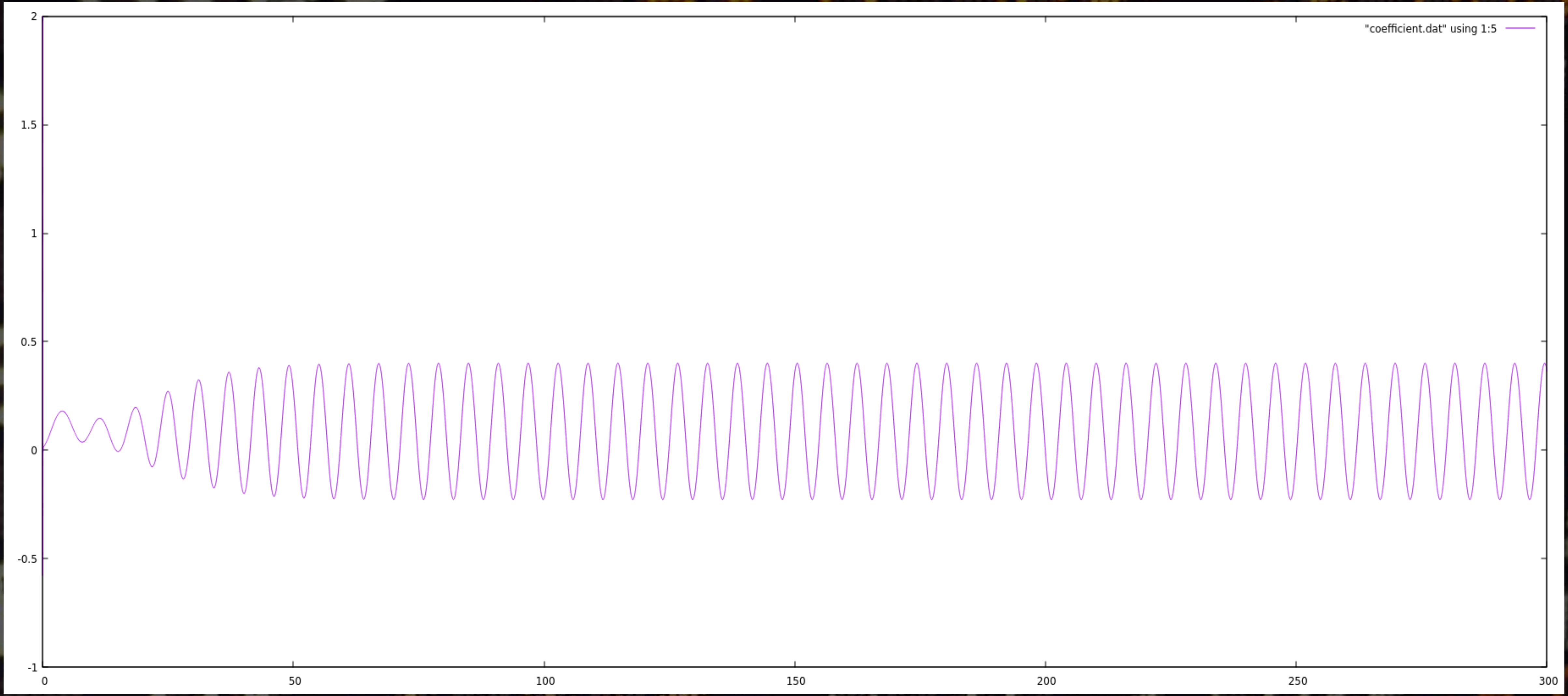
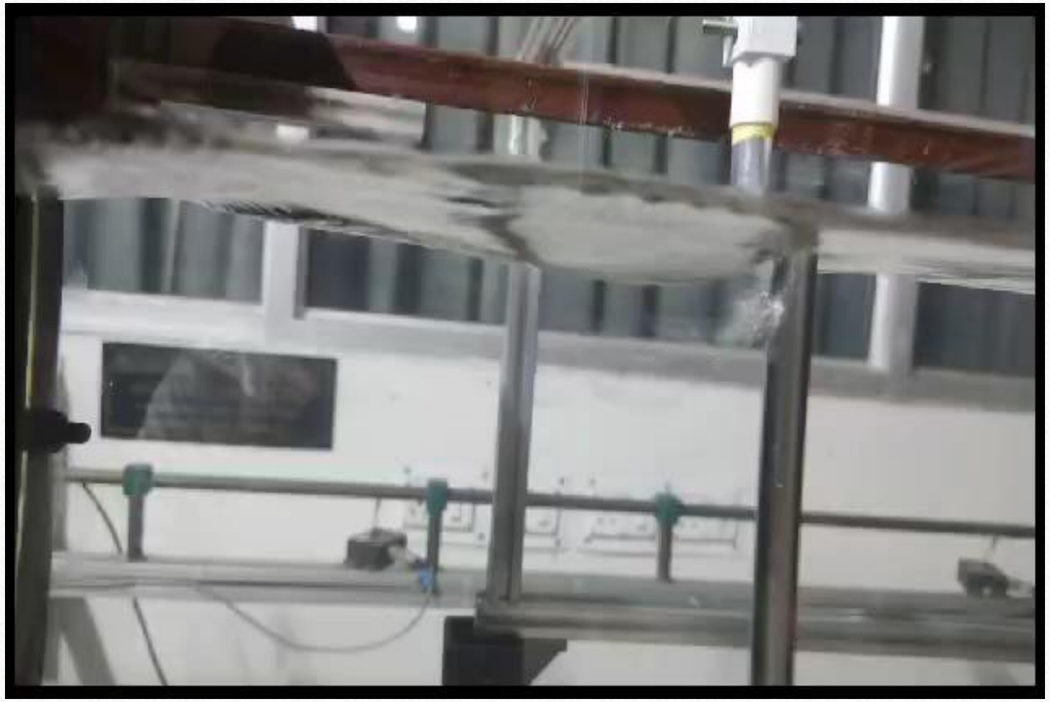
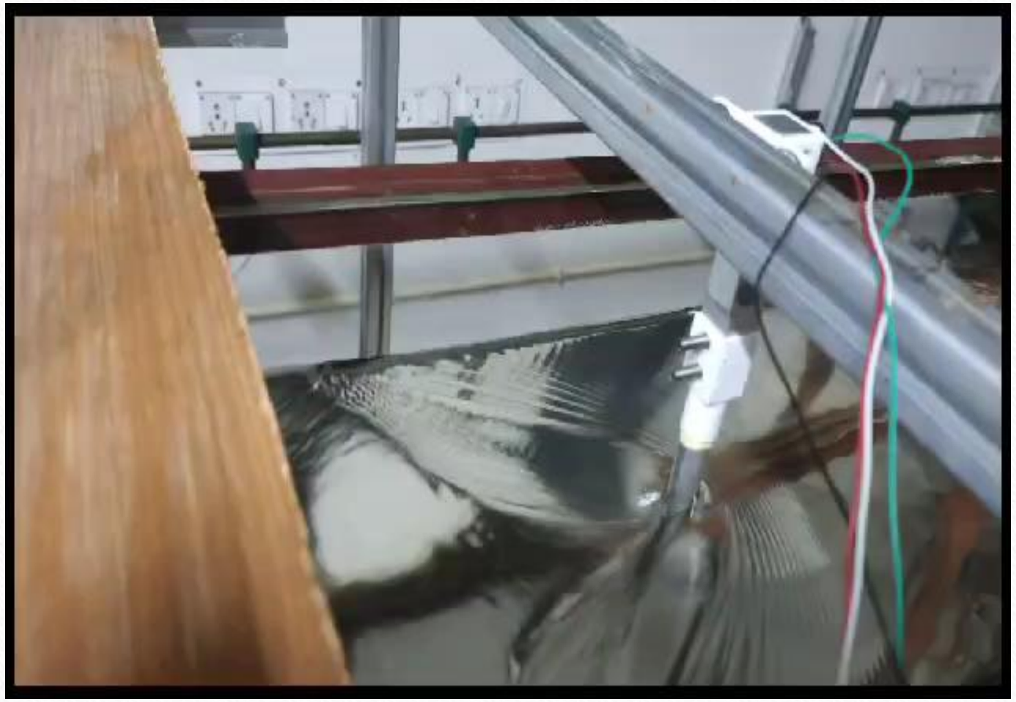


Fig. : Plot of Lift with time

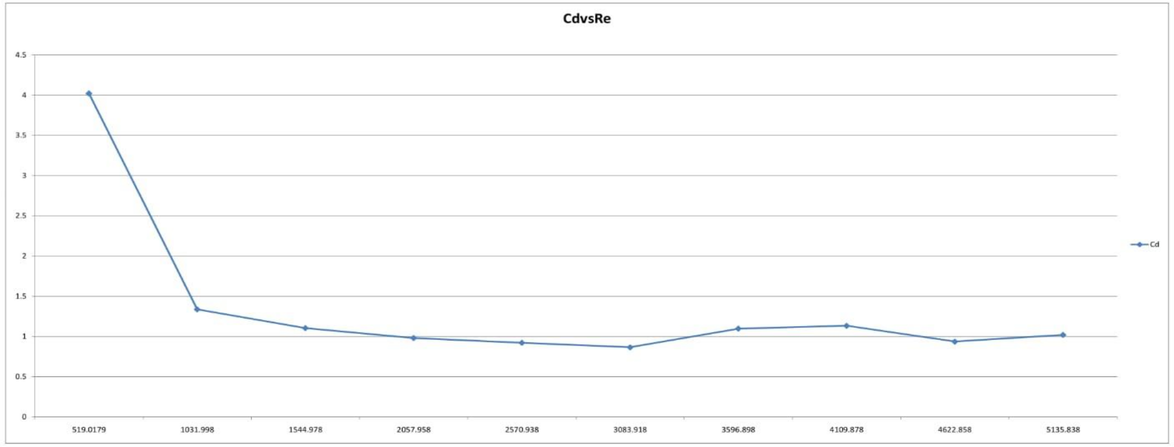
Experimental set-up :





Courtsey : Sanu Sourav (B. Tech, Mech. Engg, IIT Guwahati)

Plot of Cd v/s Re :



Courtsey : Sanu Sourav (B. Tech, Mech. Engg, IIT Guwahati)

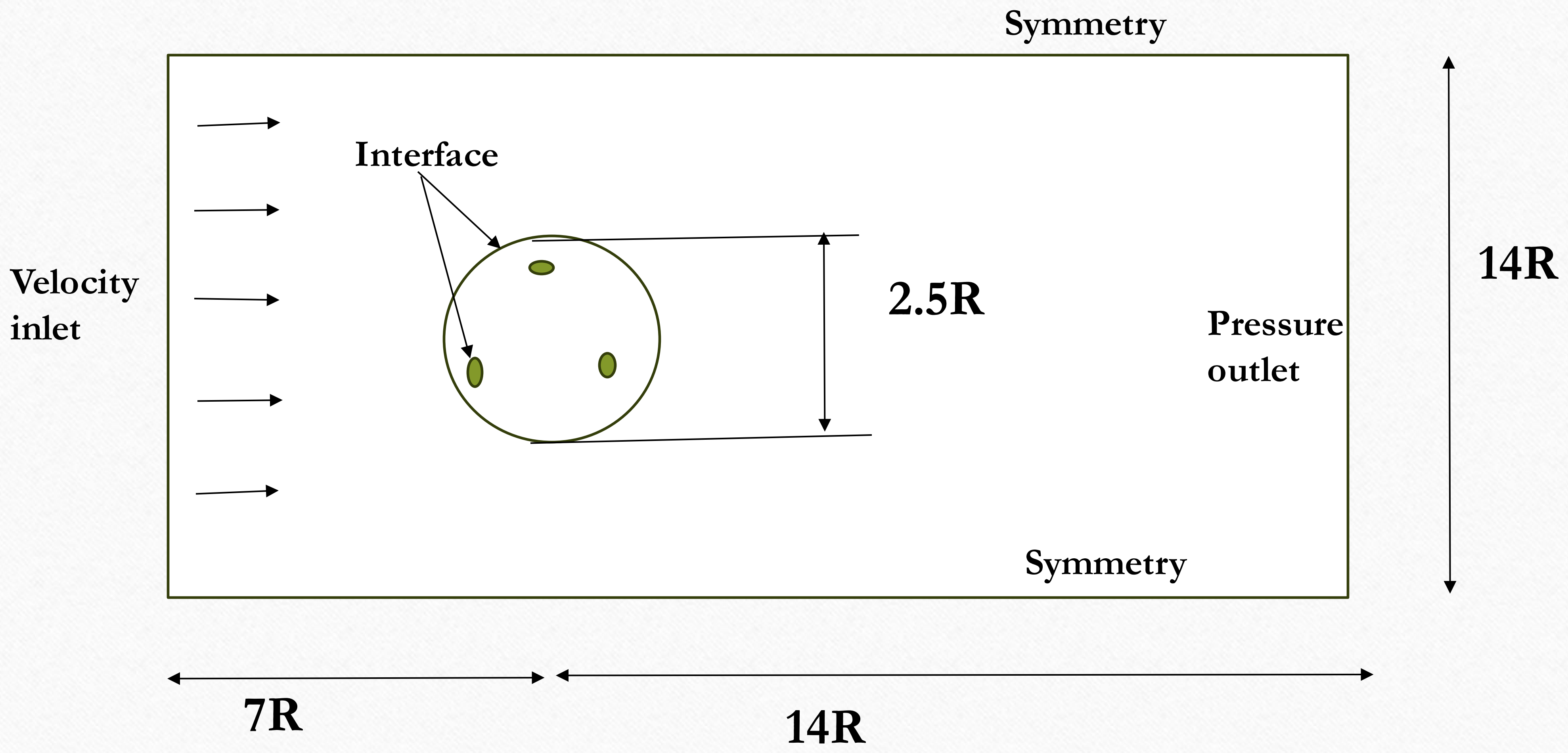
Conclusion:

1. The fluid flow over a cylinder has successfully been simulated for the Reynold's Number 100 and the respective plots and contours have been obtained.
2. Coefficient of drag(Cd) comes out to be close to **1.3 - 1.4.**
3. Coefficient of lift varies sinusoidal type with average zero.
4. Averag**e Lift is Zero.**
5. Vortex shedding is not much significant as we operated on low Reynolds number.
6. Transient state simulation takes a very high simulation time to compute the results and the number of iterations or timesteps it takes to reach a steady state is also very high.

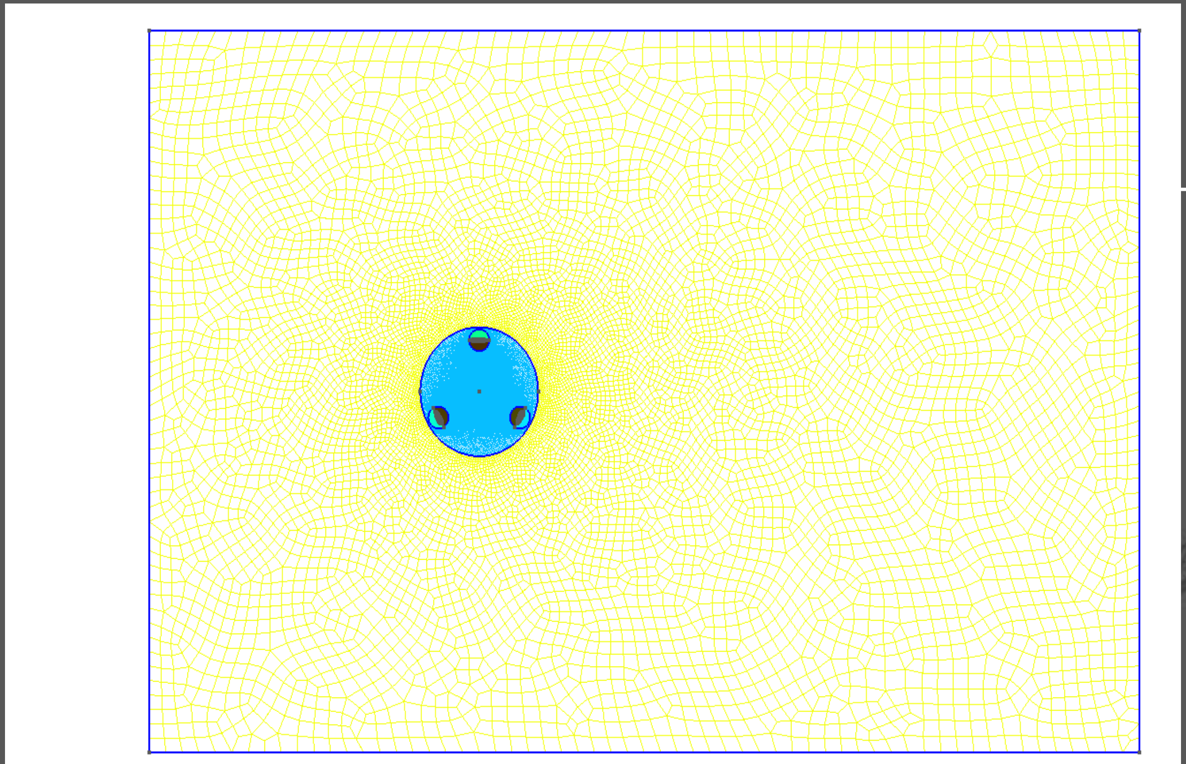
Link : Various coefficients obtained

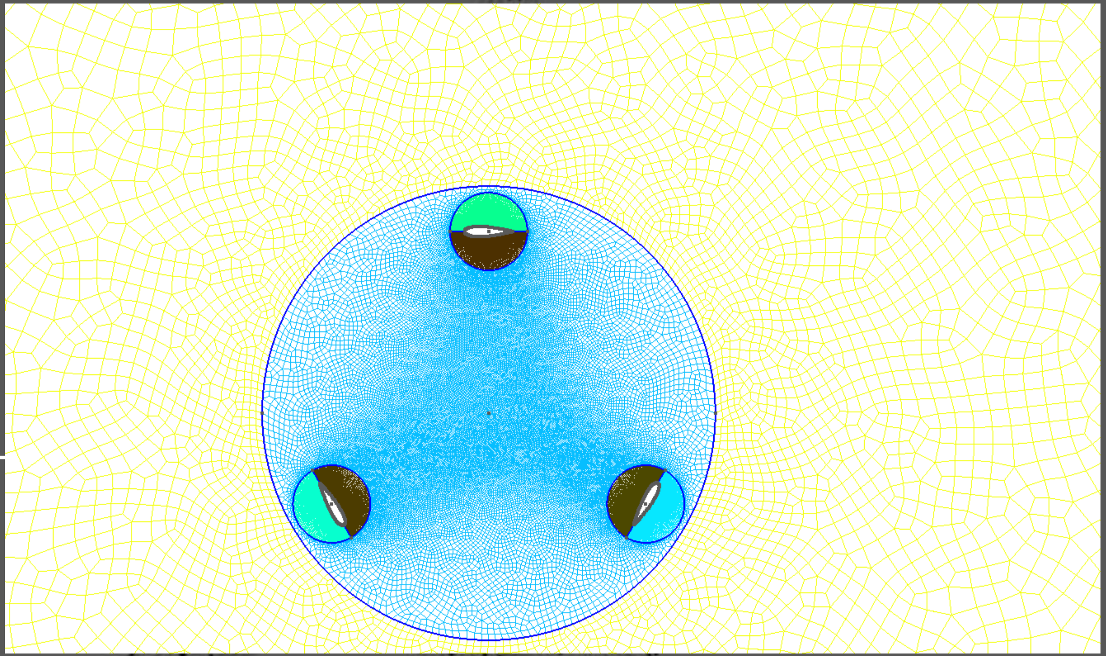
MESHING OF WIND TURBINE:

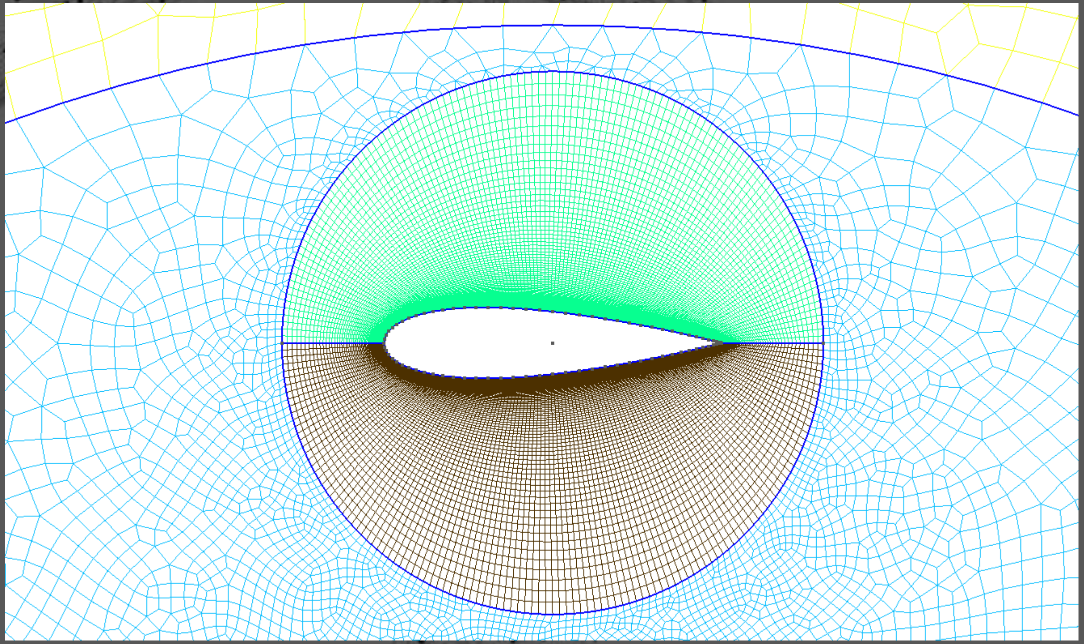
A schematic of the computational domain and the boundary conditions



Meshing of Wind Turbine:







Airfoil : NACA0021

Link : geo file for windTurbine

Link : geo file for airfoil(NACA0021)

Automation of VFD frequency

- VFD : A Variable Frequency Drive (VFD) is a type of motor controller that drives an electric motor by varying the frequency and voltage supplied to the electric motor. Other names for a VFD are **variable speed drive**, **adjustable speed drive**, **adjustable frequency drive**, **AC drive**, **microdrive**, and **inverter**.
- A VFD can be controlled manually or through softwares. Modbus Poll is one of the softwares that is used to control such devices.
- Modbus Poll is a Modbus master simulator designed primarily to help developers of Modbus slave devices or others that want to test and simulate the Modbus protocol.

Link : Code for automation of VFD

Displacement Sensor(ILD 1420)

Link : [Code for ILD1420]

In order to record the data we need to operate the motor at different frequency from time to time, that is being done by VFD.

To record the minute displacements of the object placed in the flow we need high frequency Laser Sensors.

These sensors are being operated by its software ILD 1420 Tool where we can record the data by a single press.

Since recording data is a manual task from software, it is convenient to automate it.

!! THANK YOU !!