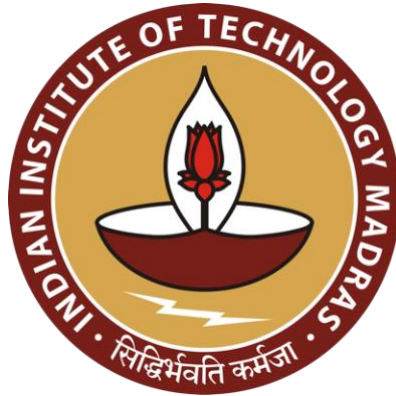


AM5820

Wind Tunnel and Numerical Experiments



Department of Applied Mechanics
Indian Institute of Technology Madras
Chennai – 600036

Numerical Simulation

Wake velocity measurement for flow over a circular
cylinder



Submitted by:
Nitin Yadav
AM20M004

Objective

To perform CFD simulation of air flow over a circular cylinder inside a wind tunnel using ANSYS Fluent to measure flow velocity in the wake region behind the cylinder for different L/D ratios.

Introduction

External flows past objects have been studied extensively because of their many practical applications. Flow past a blunt body, such as a circular cylinder, usually experiences boundary layer separation and very strong flow oscillations in the wake region behind the body. In certain Reynolds number range, a periodic flow motion will develop in the wake as a result of boundary layer vortices being shed alternatively from either side of the cylinder. This regular pattern of vortices in the wake is called a Karman vortex street. It creates an oscillating flow at a discrete frequency that is correlated to the Reynolds number of the flow. The periodic nature of the vortex shedding phenomenon can sometimes lead to unwanted structural vibrations, especially when the shedding frequency matches one of the resonant frequencies of the structure. In this experiment, we are going to investigate the flow past a circular cylinder and measure the velocity of flow field in the wake of the cylinder.

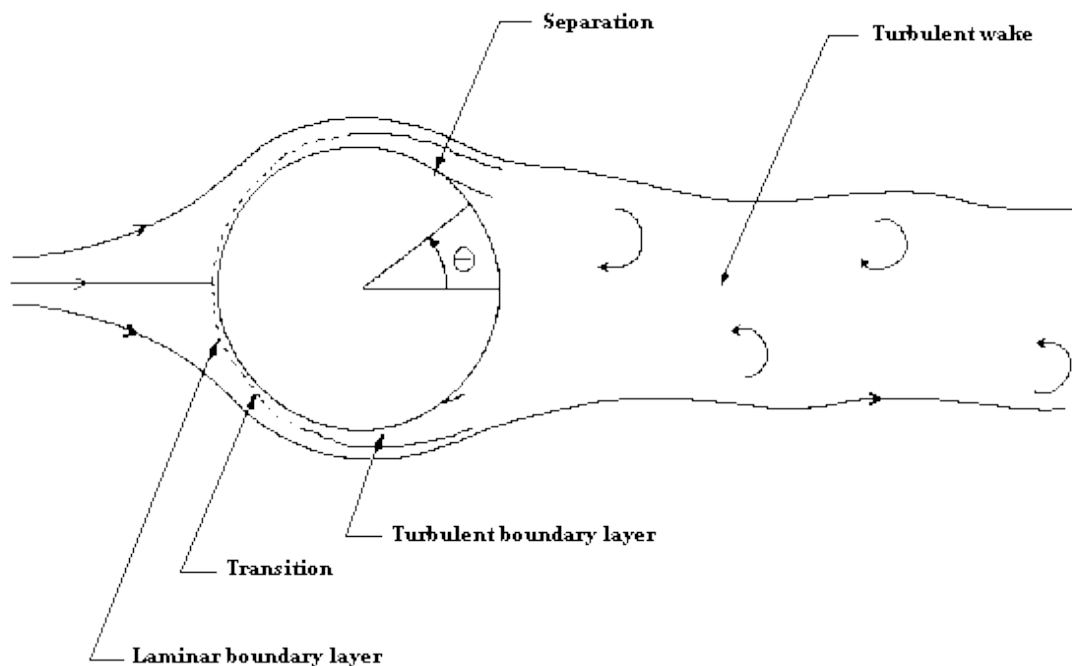


Fig. Wake behind circular cylinder

Experimental Conditions

We have taken a circular cylinder. Simulation is done by taking air as the working fluid with following properties:

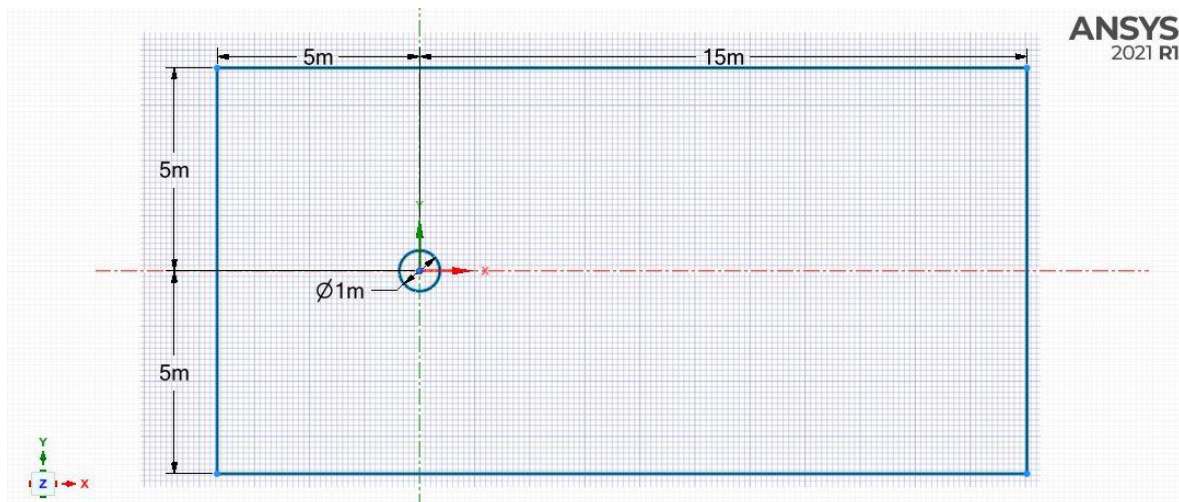
$$\rho = 1.225 \text{ kg/m}^3$$

$$\mu = 1.7894e - 05 \text{ kg/(m-s)}$$

To conduct the experiment at different L/D ratios we have varied the length (L) of cylinder and kept length (L) constant equal to 1m. For $\frac{L}{D} = 1, 0.5, 0.33$ we have taken cylinder diameter as $D = 1 \text{ m}, 2 \text{ m}, 3 \text{ m}$ respectively. Also, inlet flow velocity is kept 21 m/s for all the three cases.

CFD Simulation Process

Geometry



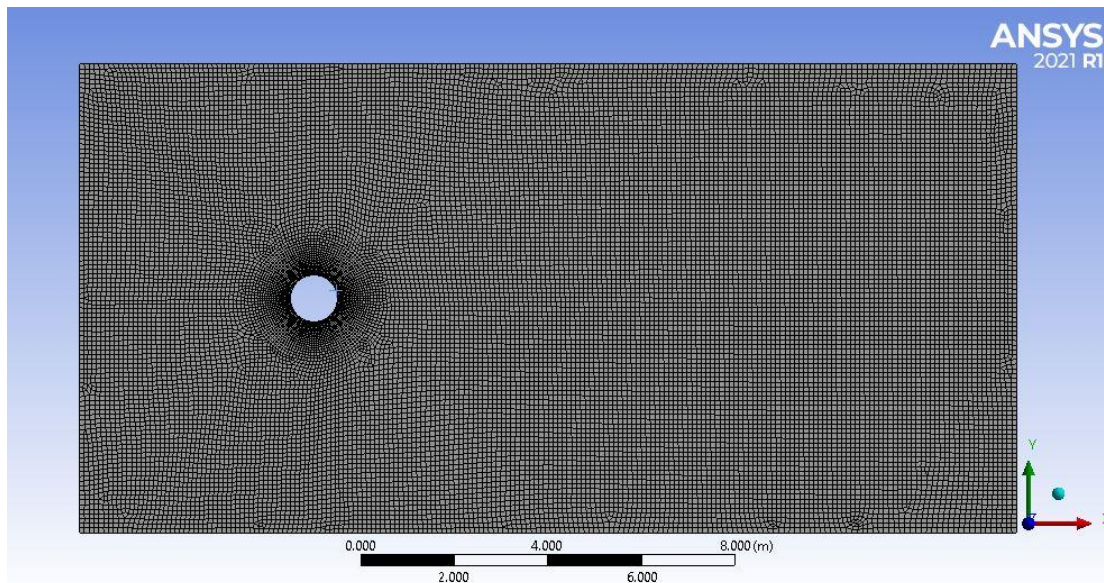
Select and drag a face to offset it. Select and drag an edge to round it.



ANSYS
2021 R1



Meshing



To make fine mesh with uniform shape, we have inserted “inflation” & “sizing” in mesh design as follows:

- Sizing for cylinder edge is of element size 0.025m and growth rate 1.2
- Sizing for face of fluid domain is of element size 0.1m and growth rate 1.2
- Inflation of face of fluid domain to boundary of cylinder is of first layer thickness 0.025m, maximum layers 20 & growth rate 1.5

For $\frac{L}{D} = 1$

Total number of elements = 22373

Total number of nodes = 22719

For $\frac{L}{D} = 0.5$

Total number of elements = 28736

Total number of nodes = 28934

For $\frac{L}{D} = 0.33$

Total number of elements = 37693

Total number of nodes = 37989

Boundary Conditions

To set the boundary conditions as well as making sure that the flow will run along x-axis, we have named every side of geometry as follows:

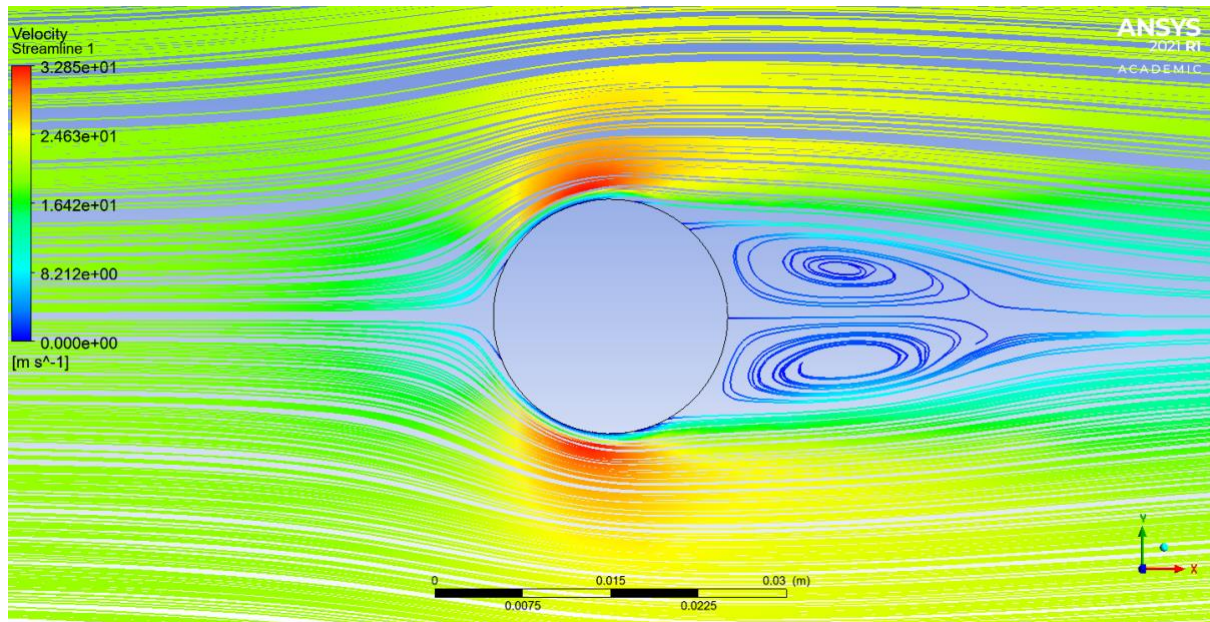
- Inlet for the left end surface

- Outlet for the right end surface
- Wall for upper and lower sides surfaces
- Cylinder for cylinder surface

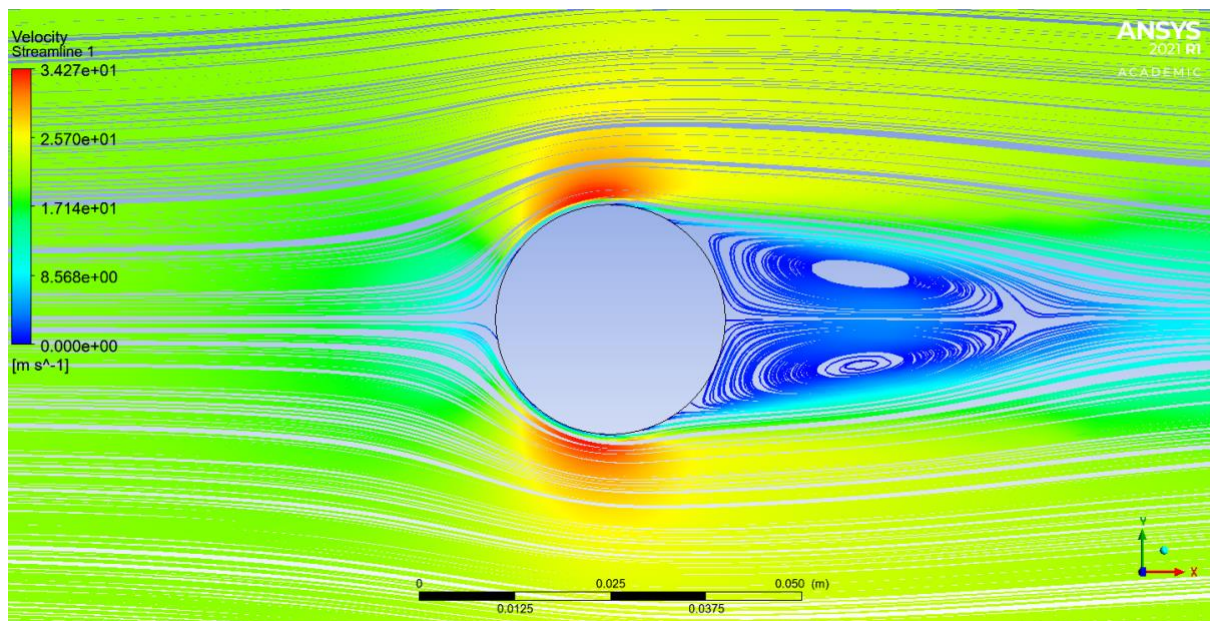
Simulation Results & Analysis

Velocity streamlines

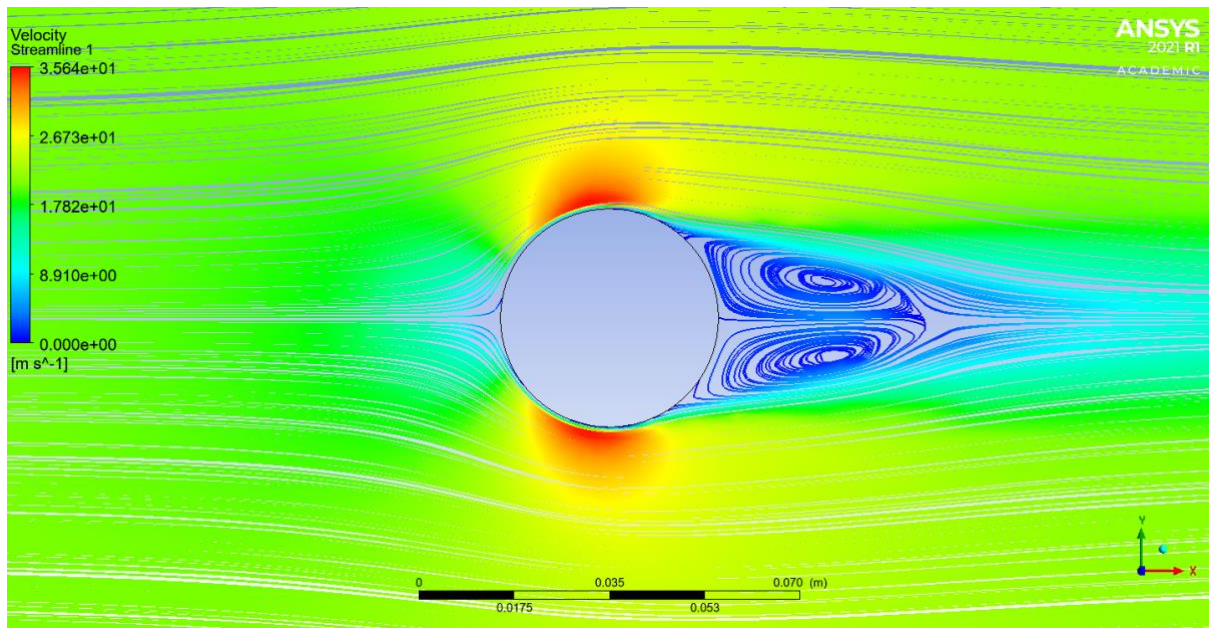
For $\frac{L}{D} = 1$



For $\frac{L}{D} = 0.5$

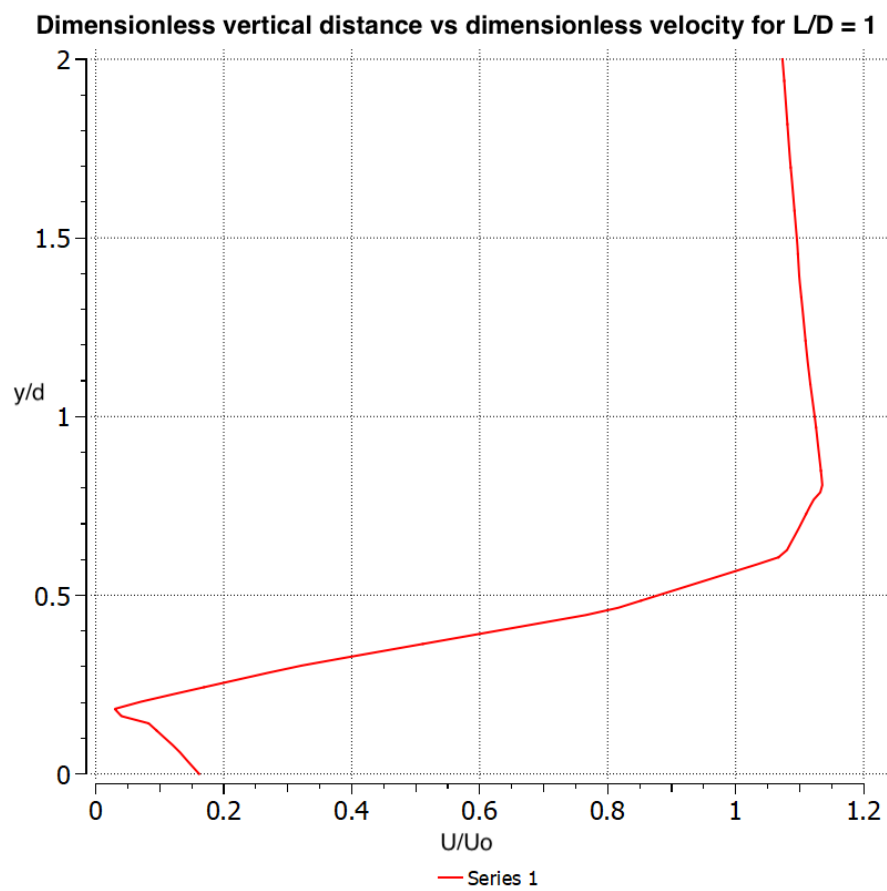


For $\frac{L}{D} = 0.33$



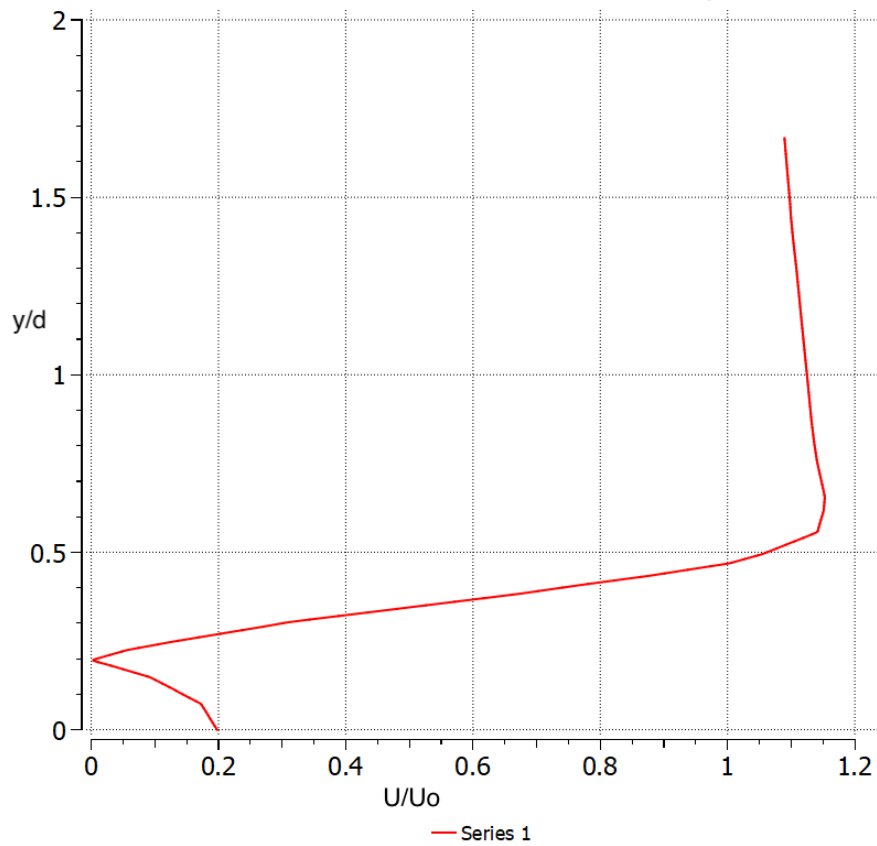
Plot of y/d vs U/U_m

For $\frac{L}{D} = 1$



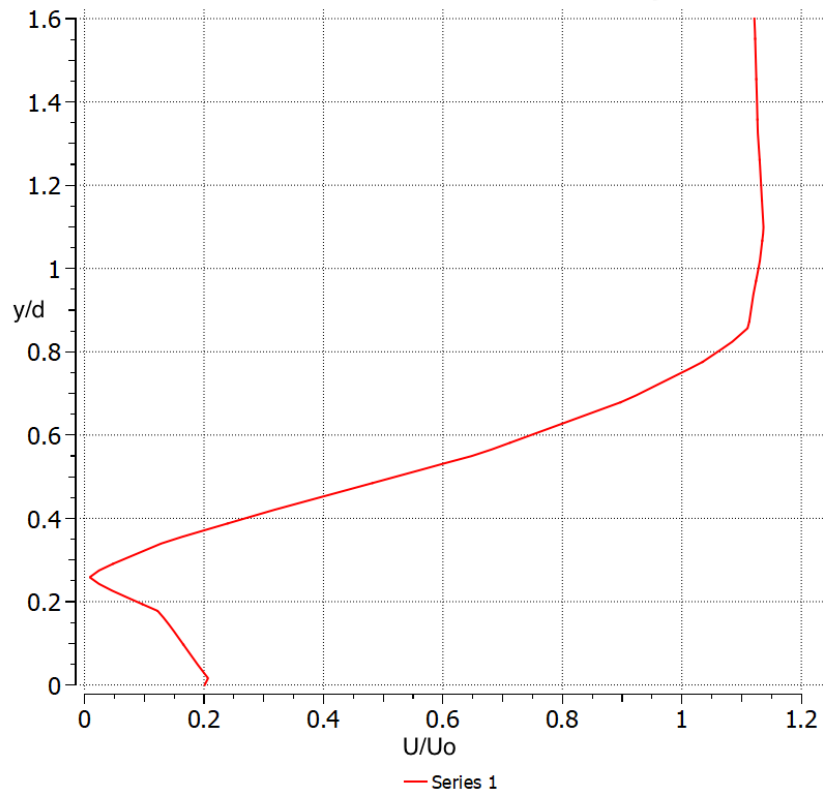
For $\frac{L}{D} = 0.5$

Dimensionless vertical distance vs dimensionless velocity for $L/D = 0.5$



For $\frac{L}{D} = 0.33$

Dimensionless vertical distance vs dimensionless velocity for $L/D = 0.33$



Comparison of Numerical & Experimental Results

