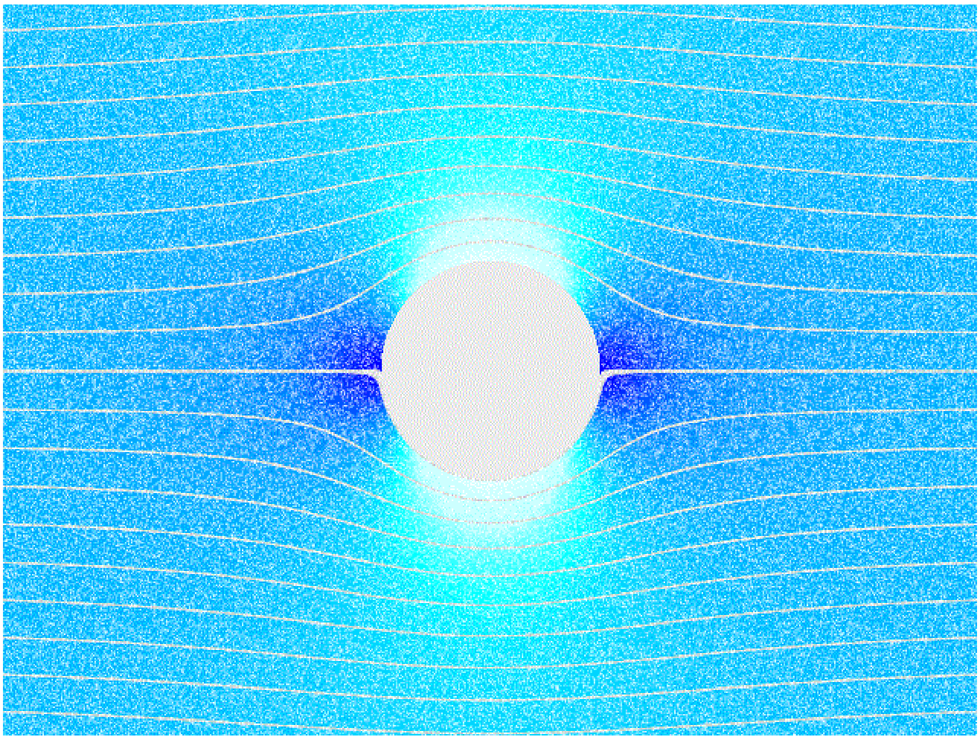
AM5820
Wind Tunnel and Numerical Experiments

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

Numerical Simulation

Pressure distribution 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 pressure distribution over the cylinder for various Reynolds Number

Introduction

For inviscid flow, there is no friction to cause boundary layer separation, vortices, or a subsequent wake. However, inviscid flow over a cylinder will generate areas of different pressure gradients. Potential flow around a cylinder is a classical solution for the flow of an inviscid, incompressible fluid around a cylinder that is transverse to the flow. Far from the cylinder, the flow is unidirectional and uniform. The flow has no vorticity and thus the velocity field is irrotational and can be modeled as a potential flow. The non-lifting flow over the cylinder can be realized by combining a uniform flow and a doublet.

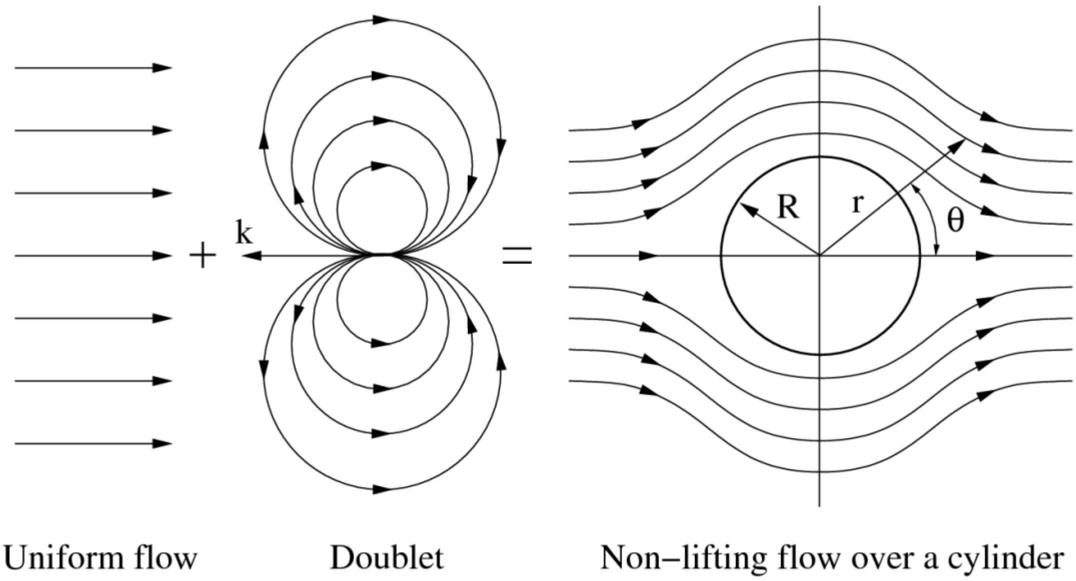


Fig. Flow realization over a circular cylinder

The stream function for the combined flow is

$$\psi = (V_\infty r \sin\theta)(1 - \frac{R^2}{r^2})$$

The velocity field is obtained by differentiating above equation

$$V_r = \frac{1}{r}\frac{\partial\psi}{\partial\theta} = \left(1 - \frac{R^2}{r^2}\right) V_\infty \cos\theta\,, \qquad (a)$$

$$V_\theta = -\frac{\partial \psi}{\partial r} = -\left(1 + \frac{R^2}{r^2}\right) V_\infty \sin\theta\,, \qquad (b)$$

The velocity distribution on the surface of the cylinder is given by equations (a) & (b) with $r = R$, resulting in

$$V_r = 0$$

$$V_\theta = -2V_\infty \sin\theta$$

$$V_c = \sqrt{{V_r}^2 + {V_\theta}^2} = 2V_\infty \sin\theta, \qquad (c)$$

From Bernoulli's equation,

$$P_1 + \frac{1}{2}\rho_1 {V_1}^2 + \rho g h_1 = P_2 + \frac{1}{2}\rho_2 {V_2}^2 + \rho g h_2, \qquad (d)$$

Since there is no change in height,

$$h_1 = h_2$$

We are doing low speed subsonic test. Therefore, flow can be assumed to be incompressible,

$$\rho_1 = \rho_2 = \rho_\infty$$

Now, equation (d) boils down to,

$$P_1 + \frac{1}{2}\rho_\infty {V_1}^2 = P_2 + \frac{1}{2}\rho_\infty {V_2}^2$$

In our experiment, point 1 is free stream & point 2 is the point on the surface of the cylinder. Therefore, we can write

$$P_\infty + \frac{1}{2}\rho_\infty {V_\infty}^2 = P_c + \frac{1}{2}\rho_\infty {V_c}^2, \qquad (e)$$

By definition of coefficient of pressure, we can write,

$$C_P = \frac{P_c - P_\infty}{\frac{1}{2}\rho_\infty {V_\infty}^2}, \qquad (f)$$

At stagnation point

i.e. at $\theta = 0^o$, $\quad V_c = 0$

$$P_\infty + \frac{1}{2}\rho_\infty {V_\infty}^2 = P_c$$

$$(P_c - P_\infty)_{\theta=0^o} = \frac{1}{2}\rho_\infty {V_\infty}^2$$

Substituting above equation in equation (f), we get

$$\boxed{C_{P,Real} = \frac{P_c - P_\infty}{(P_c - P_\infty)_{\theta=0^o}}}$$

Here,

$V_\infty - Free\ stream\ velocity$

$V_c - Velocity\ at\ a\ point\ on\ the\ surface\ of\ the\ cylinder$

$P_\infty - Free\ stream\ pressure$

$P_c - Pressure\ at\ a\ point\ on\ the\ surface\ of\ the\ cylinder$

$\rho_\infty - Free\ stream\ density$

$\theta - Angular\ position\ of\ the\ port\ on\ the\ cylinder$

Experimental Conditions

We have taken a circular cylinder of 1m diameter. Simulation is done by taking air as the working fluid with following properties:

$$\rho = 1.225\, kg/m^3$$

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

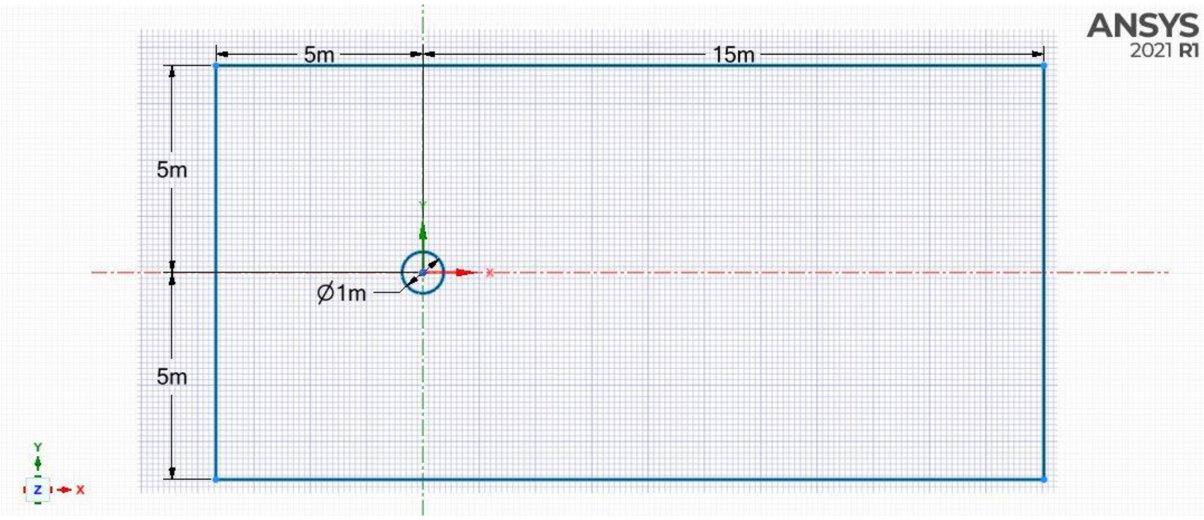
To conduct the experiment at different Reynolds number we have varied the inlet velocity and keeping all other variables constant in the expression of Reynolds number given as follows:

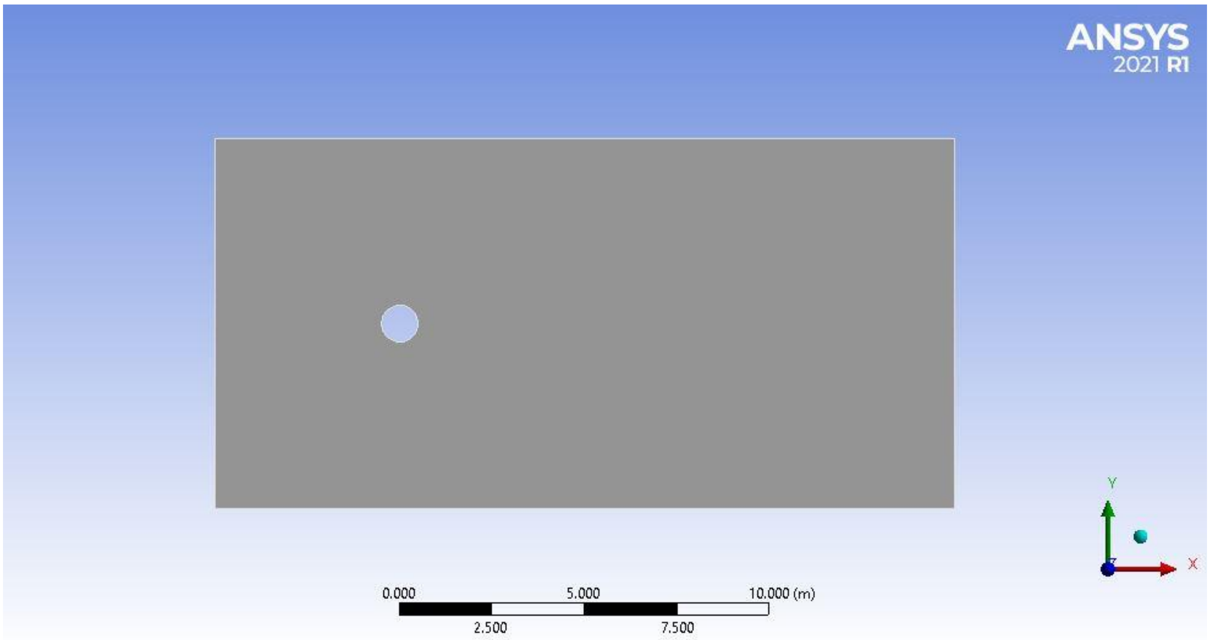
$$Re = \frac{\rho v d}{\mu}$$

For $Re = 100, 10000\ and\ 1000000$ we have taken inlet velocity as $v = 0.00146\frac{m}{s}, 0.146\frac{m}{s}\ and\ 14.6\frac{m}{s}$ respectively.

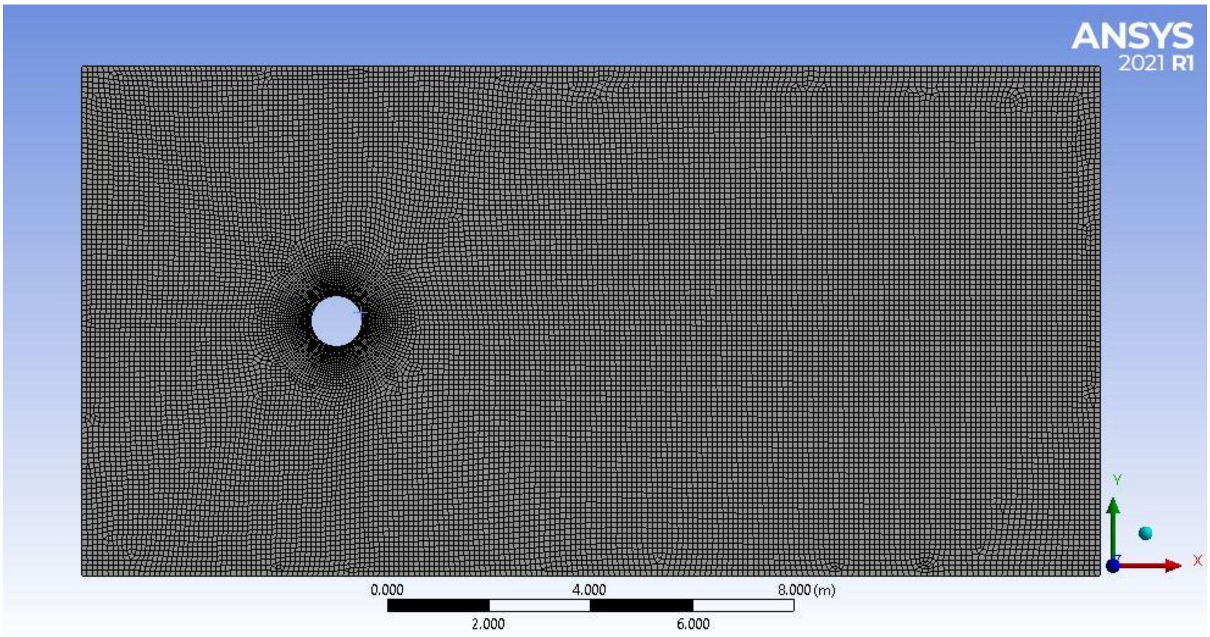
CFD Simulation Process

<u>Geometry</u>





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

Total number of elements = 22373

Total number of nodes = 22719

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

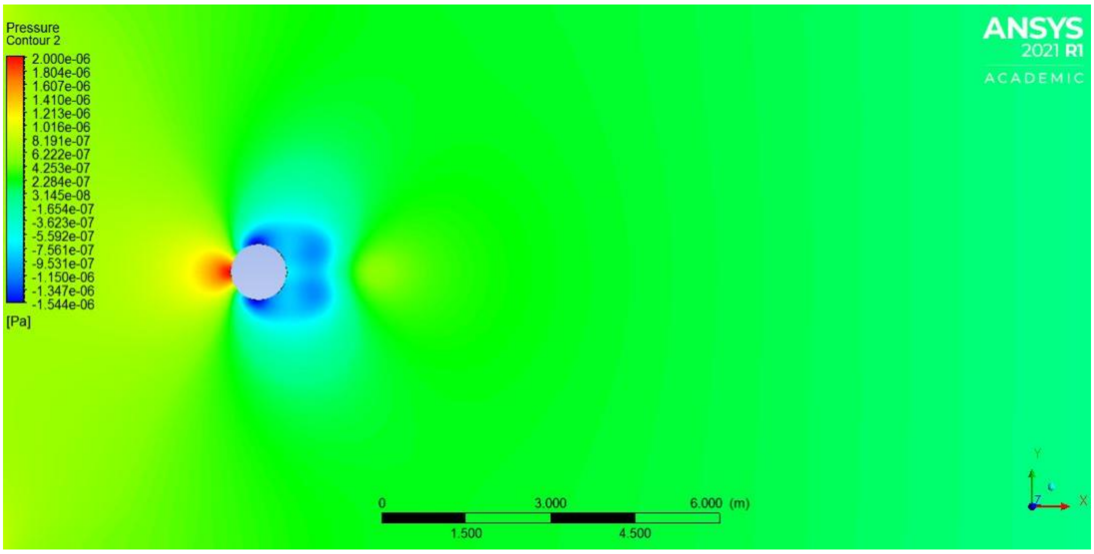
Model

- Viscous Laminar

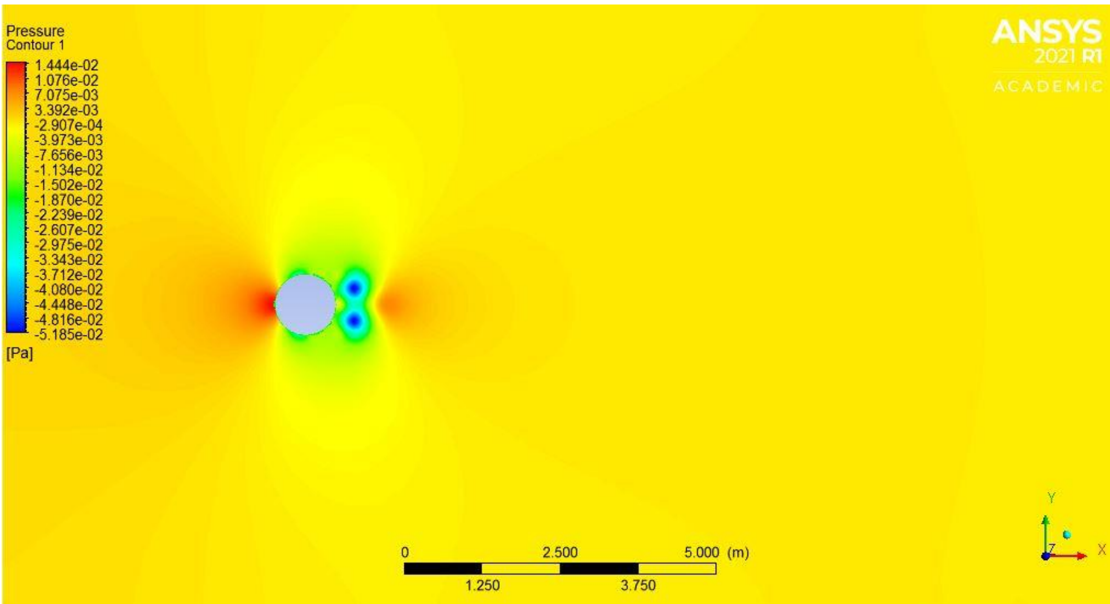
Simulation Results & Analysis

Pressure Contours

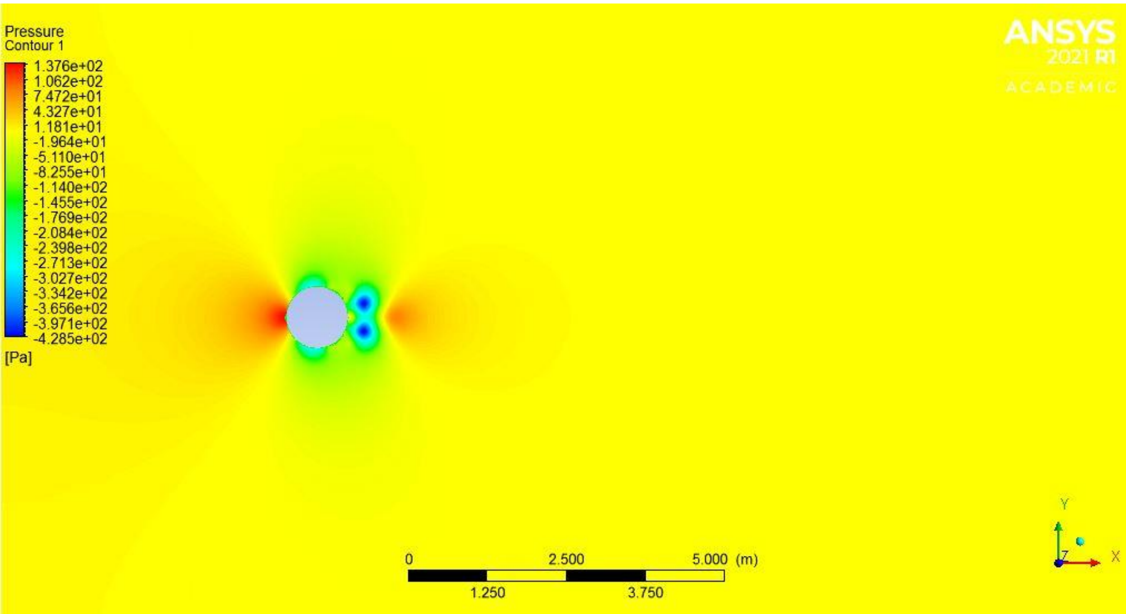
For Re = 100



For Re = 10000

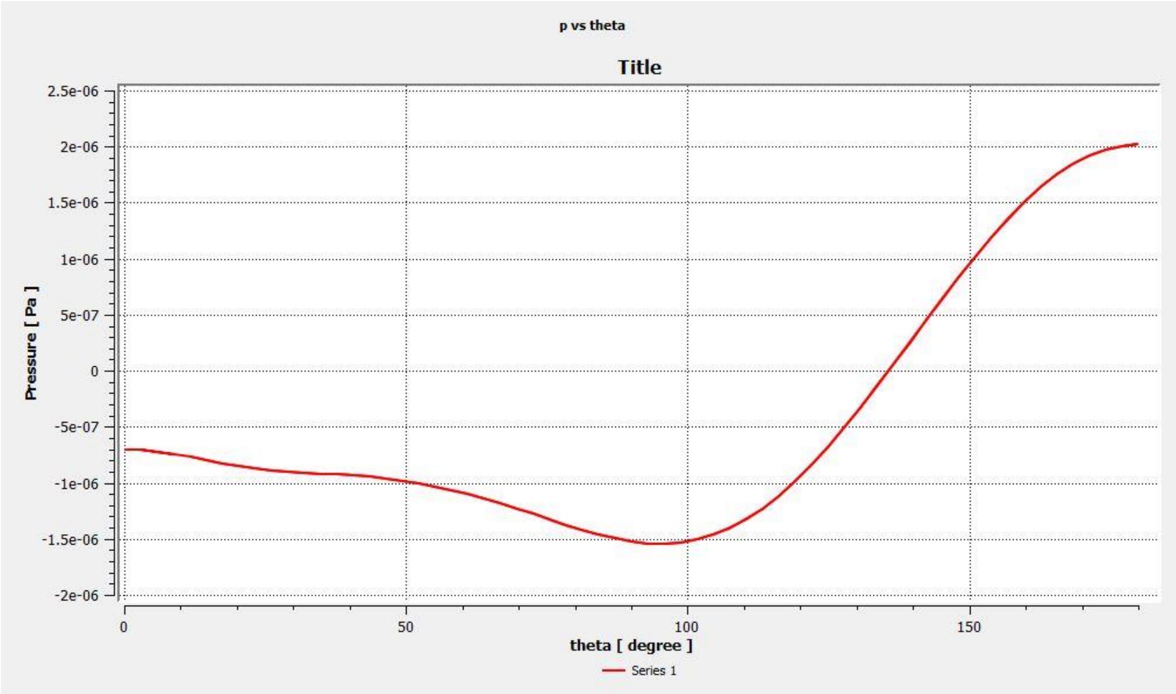


For Re = 1000000

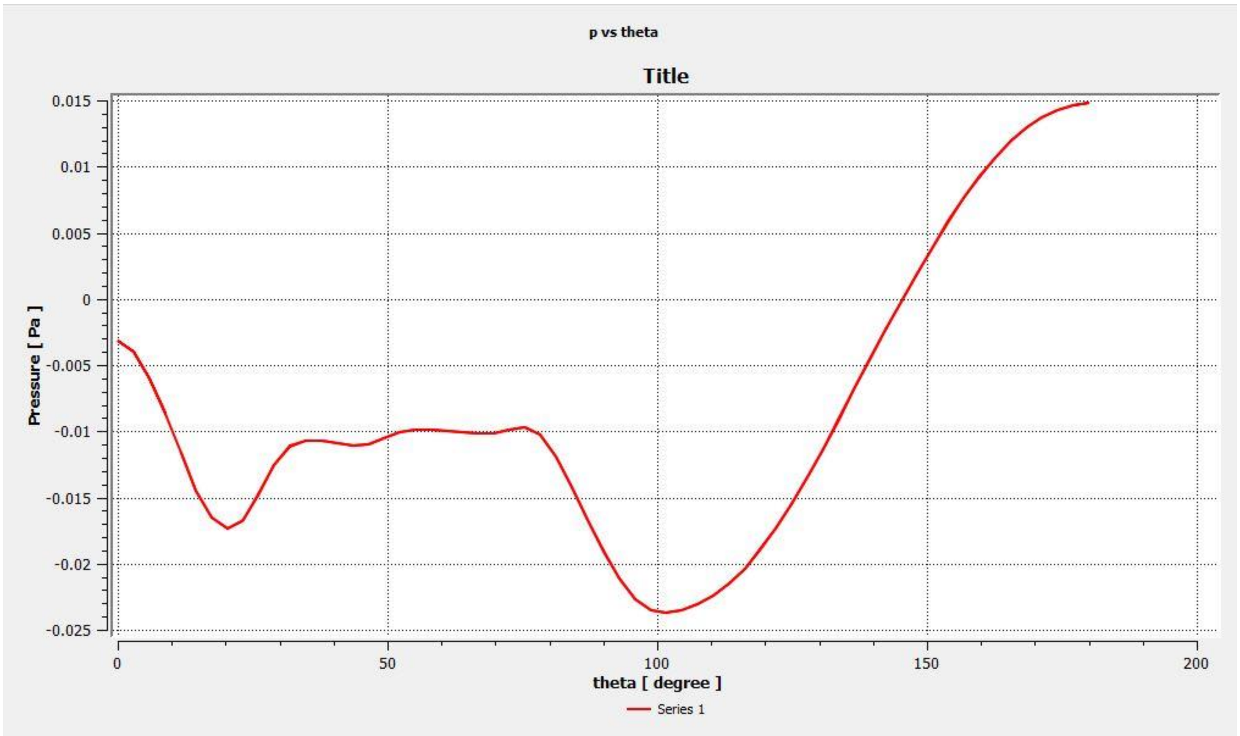


<u>Pressure Plots</u>

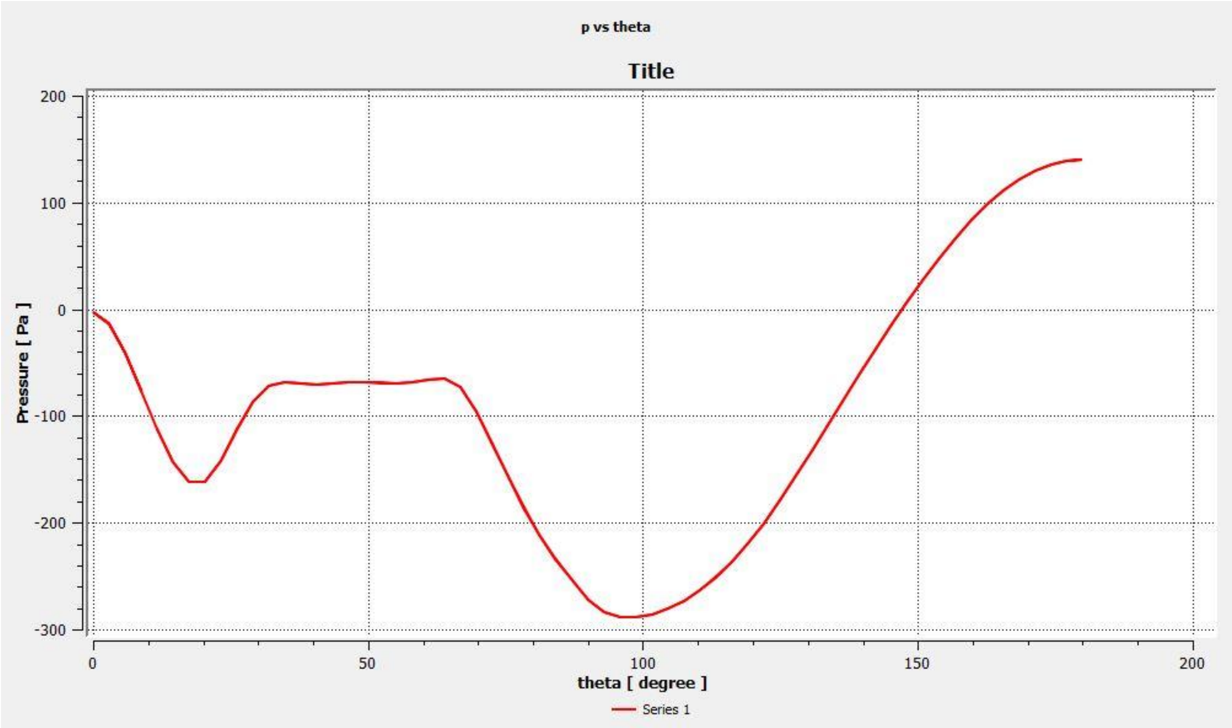
For Re = 100



For Re = 10000

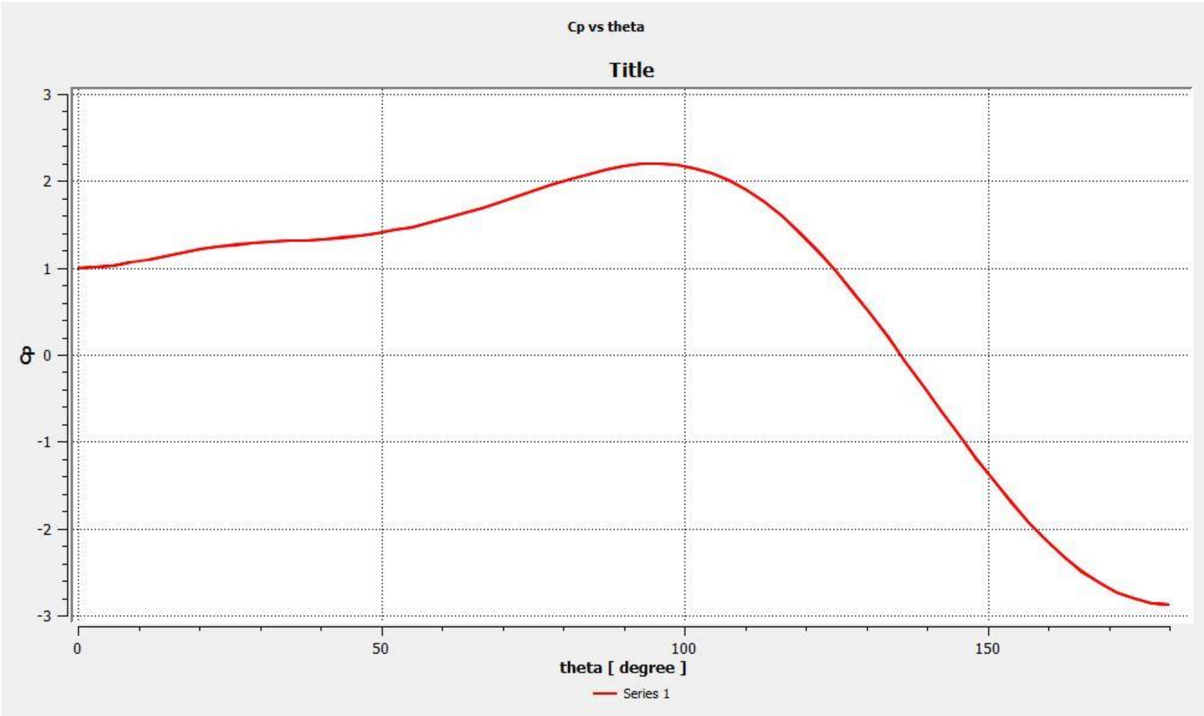


For Re = 1000000

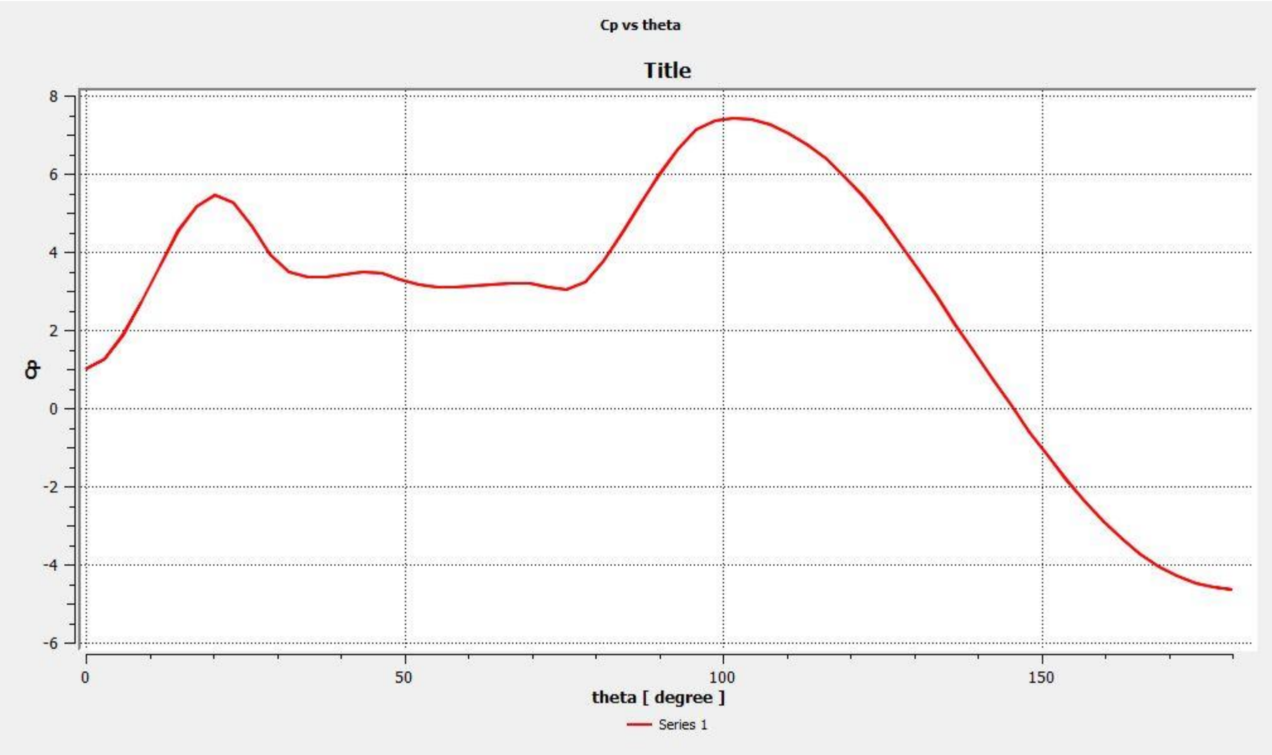


Pressure coefficient Plots

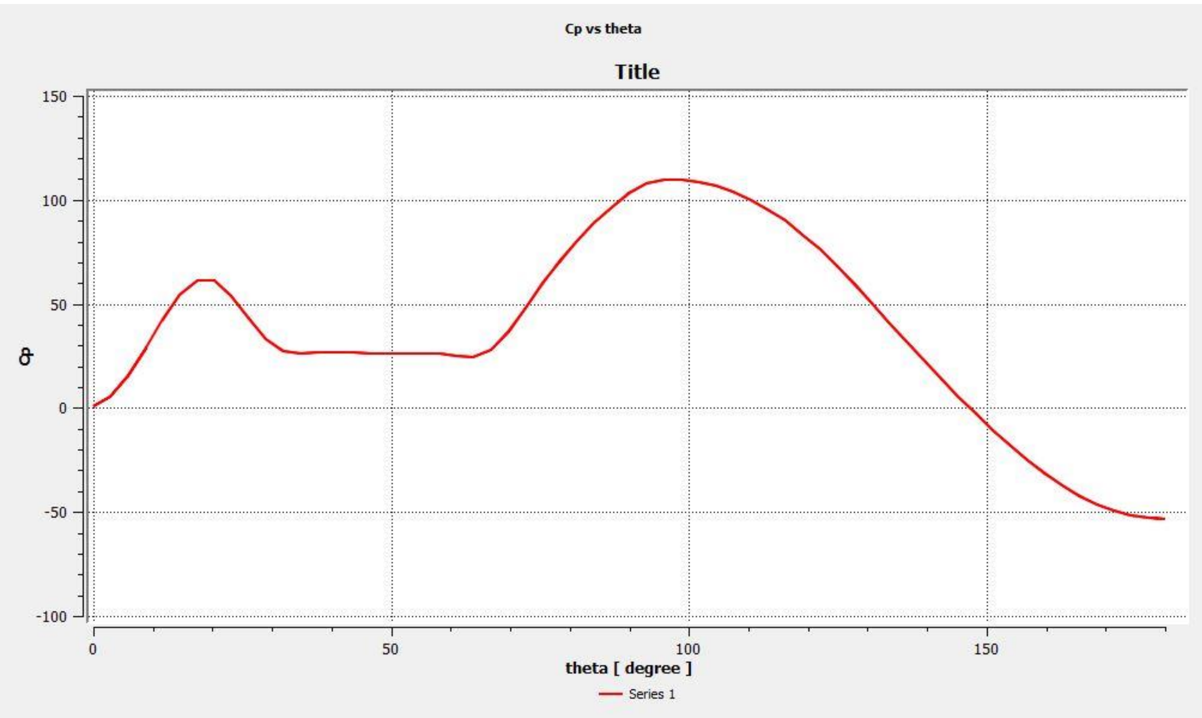
For Re = 100



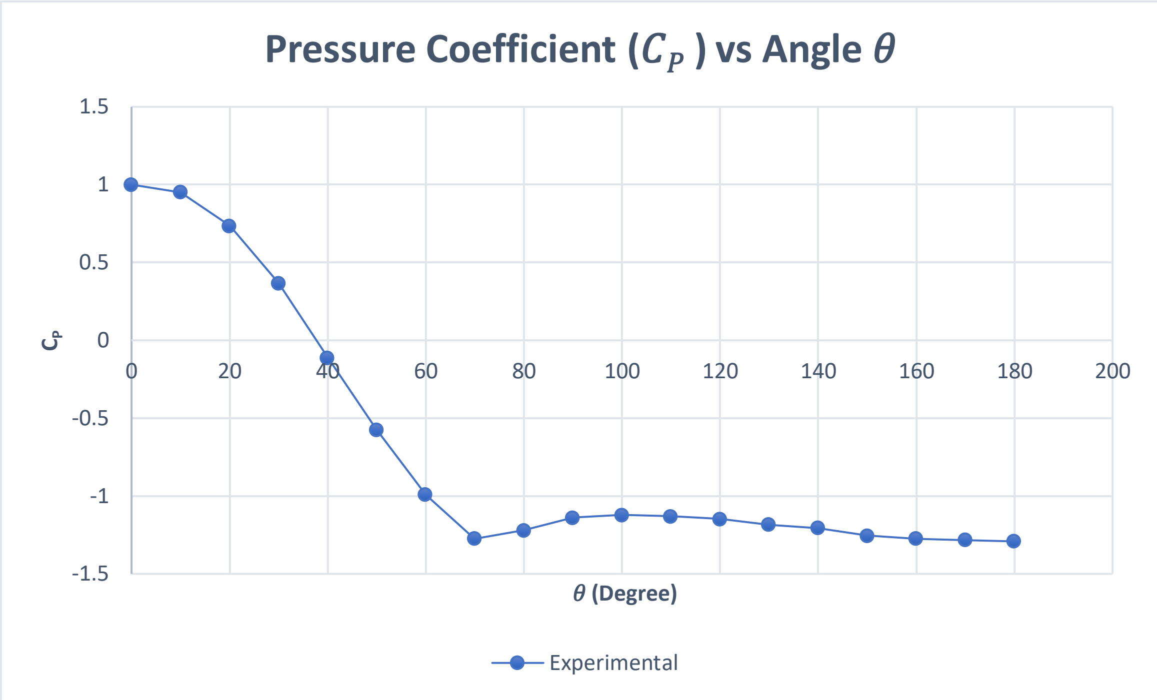
For Re = 10000



For Re = 1000000



Comparison with Experimental Results



Conclusion

The experimental and simulation plots of pressure coefficient are different. In experiment, the measured values of pressure are all negative which is not so in simulation as can be seen from pressure plots. It could be due to the difference in Reynolds number due to different cylinder size.