FOSSEE, IIT Bombay
OpenFOAM Case Study Project
February 11, 2020

NUMERICAL SIMULATION & EXPERIMENTAL VALIDATION OF VORTEX SHEDDING SUPPRESSION FROM A CIRCULAR CYLINDER USING RIGID SPLITTER PLATE

VINOTH KUMAR D

Mtech – Aerospace Engineering (Aerodynamics)
Department of Aerospace Engineering, IIT Bombay
Email – vinoth1593@gmail.com

Abstract

Flow seperation control technique is an important research field of aerodynamics. Flow Seperation control implies a small change of a configuration serving an ideally large engineering benefit .There are two broad classifications of flow control seperation – Active & Passive Methods. The aim of this study is to implement numerically one of the passive flow seperation control technique using rigid splitter plate. It is noted that active flow seperation control technology has obvious advantages over passive control but passive control is easy to implement and practically used.
We use opensource CFD tool called openFoam-4.x version inorder to first get numerical results for only flow over cylinder then implement the rigid splitter plate & study the phenomenon of vortex shedding suppression due to the splitter plate. The opensource mesh program Gmsh is used for pre-processing & a python's script inorder to compute FFT, Cd, Cl plots with time.
This report also presents a review of experimental data on the flow over cylinder with and without splitter plates & we try to understand the deviations in the measured & numerical results especially on strouhal number. In future several interesting study related to the same method but with different configurations can be tried in numerical approach to arrive at the best configuration to achieve effective votex shedding separation.

Keywords : *Flow Seperation Control, Active & Passive Methods, Rigid Splitter Plate, Numerical Approach, Opensource Tool, FFT.*

TABLE OF CONTENTS

ACKNOWLEDGEMENTS

I take the opportunity to express my sincere gratitude and indebtedness to **Prof. Kowsik Bodi,** Department of Aerospace Engineering, Indian Institute of Technology Bombay, for his guidance and oppurtunity to work on openFOAM platform without which it would not have been possible to finish the tasks mentioned in the report. I would also like to thank **Prof.Sharma.S.D & Mr.Abhishek (Lab. Incharge)** for providing us oppurtunity to conduct experiments in water tunnel in our department. Also thanks to all my friends *(Anant, Ved, Mrinal, and Naveen)* for providing me valuable inputs and support during this case study.

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS & SYMBOLS USED

St = Strouhal Number.

Re =Reynolds Number.

fs=Frequency of vortex shedding In Hz.

U∞ =Fluid Velocity In m/s.

MaxCo = Maximum CFL number limit in the computations.

Cd = Drag Co-Efficient.

CHAPTER-1: INTRODUCTION

1.1. Objectives Of The Project:

This study investigates the fluid forces acting on a circular cylinder in a laminar flow regime while using a passive control strategy of attaching a rigid splitter plate behind the cylinder at 0°. Initially, the numerical analysis for Re=100 in water is done using pisoFoam solver(Augmented with adjustable timestep) of openFOAM-4.x version for flow over cylinder. FFT of the vortex shedding is done & the Cd & St are validated with literature.
In the second approach, The experimental validation of the same is carried out in the water tunnel facility & St number obtained numerically is validated.

1.2. Basic Understanding (THEORY):

Consider fluid flow over a cylinder. Right at the front of the cylinder fluid particles must come to rest. This point is aptly called the stagnation point and is the point of maximum pressure (to conserve energy the pressure needs to fall as fluid velocity increases, and vice versa). Further downstream, the curvature of the cylinder causes the flow lines to curve, and in order to equilibrate the centripetal forces, the flow accelerates and the fluid pressure drops. Hence, an area of accelerating flow and falling pressure occurs between the stagnation point and the poles of the cylinder. Once the flow passes the poles, the curvature of the cylinder is less effective at directing the flow in curved streamlines due to all the open space downstream of the cylinder. Hence, the curvature in the flow reduces and the flow slows down, turning the previously favourable pressure gradient into an adverse pressure gradient of rising pressure. To understand boundary layer separation we need to understand how these favourable and adverse pressure gradients influence the shape of the boundary layer.

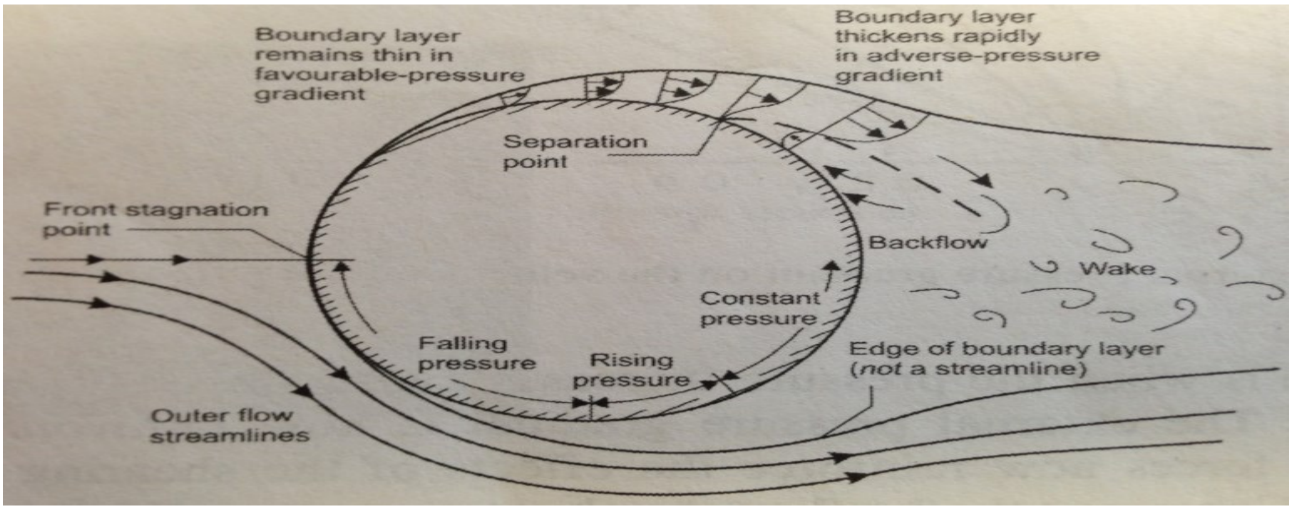

Figure 1.1: Boundary layer separation over a cylinder (axis out out the page). Image Courtesy[2].

From our discussion, we know that the fluid travels slower the closer we are to the surface due to the retarding action of the no-slip condition at the wall. In a favourable pressure gradient, the falling pressure along the streamlines helps to urge the fluid along, thereby overcoming some of the decelerating effects of the fluid's viscosity. As a result, the fluid is not decelerated as much close to the wall leading to a fuller U-shaped velocity profile, and the boundary layer grows more slowly. By analogy, the opposite occurs for an adverse pressure gradient, i.e. the mainstream pressure increases in the flow direction retarding the flow in the boundary layer. So in the case of an adverse pressure gradient the pressure forces reinforce the retarding viscous friction forces close to the surface. As a result, the difference between the flow velocity close to the wall and the mainstream is more pronounced and the boundary layer grows more quickly. If the adverse pressure gradient acts over a sufficiently extended distance, the deceleration in the flow will be sufficient to reverse the direction of flow in the boundary layer. Hence the boundary layer develops a point of inflection, known as the point of boundary layer separation, beyond which a circular flow pattern is established.

1.3. Literature Study:

After the seperation from the cylinder walls, the fluid rolls into vortices & starts shedding from the cylinder surface & this phenomenon is called as vortex shedding. As flow passes over a bluff body, two recirculating zones, known as vortices, are formed downstream of the bluff body. When Reynolds number (Re), of the flow exceeds a threshold number, the vortices start to peel off periodically from the cylinder. When the vortex shedding occurs behind a cylinder, drag on the cylinder increases. Moreover, the periodic forces that were applied on the cylinder in the cross flow direction cause undesirable vibrations in the cylinder **(Kwon and Choi, 1996).** Many practical applications involve vortex-induced vibrations, including heat exchanger tube bundles, marine structures, bridges, power transmission lines etc. Therefore, controlling the vortex shedding is very important in engineering applications to prevent the possible structural damages **(Sudhakar and Vengadesan, 2012).** Several active & passive methods were employed by researchers to control the seperation & also to reduce the vibrations that are induced due to vortex shedding. One such well experimented passive method of vortex sheddign suppression from cylinder is employing rigid & flexible splitter plates in different configurations (say in ahead & behind the cylinder) & also in different configurations 0°,15°,45°,90° etc..

CHAPTER-2: NUMERICAL APPROACH

2.1. OpenFOAM Case Setup:

As discussed earlier the aim of this study is to understand the effect of rigid splitter plates attached to the cylinder & understand their effects on vortex shedding. Initially, we tried to solve the problem of flow over cylinder with & without splitter plates numerically using opensource CFD platform openFOAM-4.x using the pisoFoam solver with slight modifications that will be explained in the forthcoming sections of this report.

PisoFoam is transient solver for incompressible flow as we are within the sub-critical zone we opt for laminar flow type & we used Gmsh-3.0.1 which is an open-source tool for mesh generation.

We employed the computational domain from well-established work of (Rezvan Abdi, Niki Rezazadeh) as shown below inorder to validate the results of our computations with the literature.

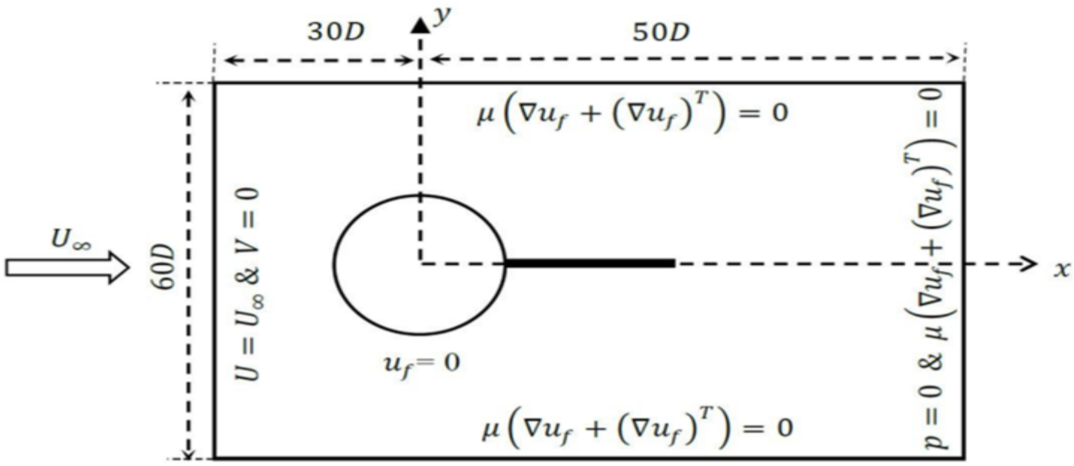

Figure 2.1: The computational domain and boundary conditions. Image Courtesy[4].

The flow geometry and coordinate system as well as boundary conditions are shown in figure-2.1. The computational domain was defined within $-30D \le x \le 50D$ and $-30D \le y \le 30D$,where the centre of the cylinder was located at the origin (x = 0, y = 0). The effect of the outlet boundary and the blockage effect were negligible in this problem as the size of the domain was large enough in both flow and cross-flow directions, respectively. The inlet velocity U was uniform, aligned with the x-direction and was chosen such that $Re = \rho U_\infty D/\mu = 100$. Pressure outlet boundary condition was applied on the far downstream outlet. Assuming the environment had the atmosphere conditions, a pressure value of zero was used on the outlet boundary. The open boundary conditions with no viscous stress were imposed to the lateral surfaces. A no-slip boundary condition specified as uinf = 0 was applied on the cylinder wall and on the splitter plate, where u f is the fluid velocity. Based on the Re=100 selected by us, We estimated the necessary boundary conditions for the problem and are tabulated below,

S.No	Patch Field	Velocity in m/s	Pressure in Pa
1	Inlet	Fixed Value=0.0178,0,0	zeroGradient
2	Outlet	zeroGradient	Fixed Value=0
3	Front	empty	empty
4	Back	empty	empty
5	Top	slip	zeroGradient
6	Bottom	slip	zeroGradient
7	Cylinderwalls	Fixed Value=0,0,0	zeroGradient
8	CylinderandPlateWalls	Fixed Value=0,0,0	zeroGradient

Table 2.1: The boundary conditions defined for both with & without plate

The computations are then carried out with mpirun parallel processing with 4 cores for 5500 seconds for only cylinder case for time of 400 seconds. For cylinder with splitter plate the case was run for 6.5hrs in 4 cores.

2.2. Pre-Processing – Mesh Generation For With & Without Splitter Plate:

As discussed earlier the mesh generation is done using Gmesh-3.0.1 & the mesh topology is as shown below,

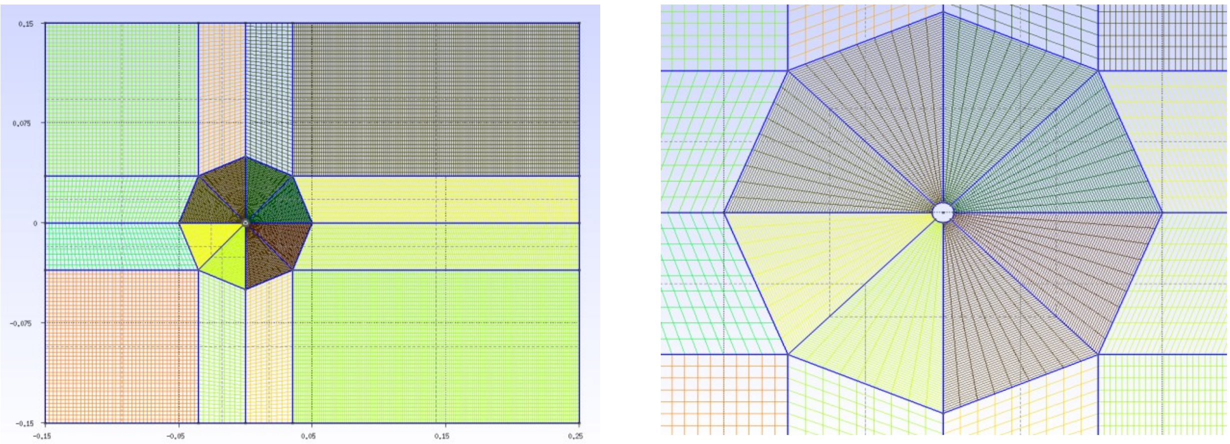

Figure 2.2: Mesh Topology Adopted (For Only Flow Over Cylinder)

Mesh Details checked using checkMesh(Only Cylinder Case):

No. of hexahedra cells: 26100.

Max skewness : 1.03444 OK.

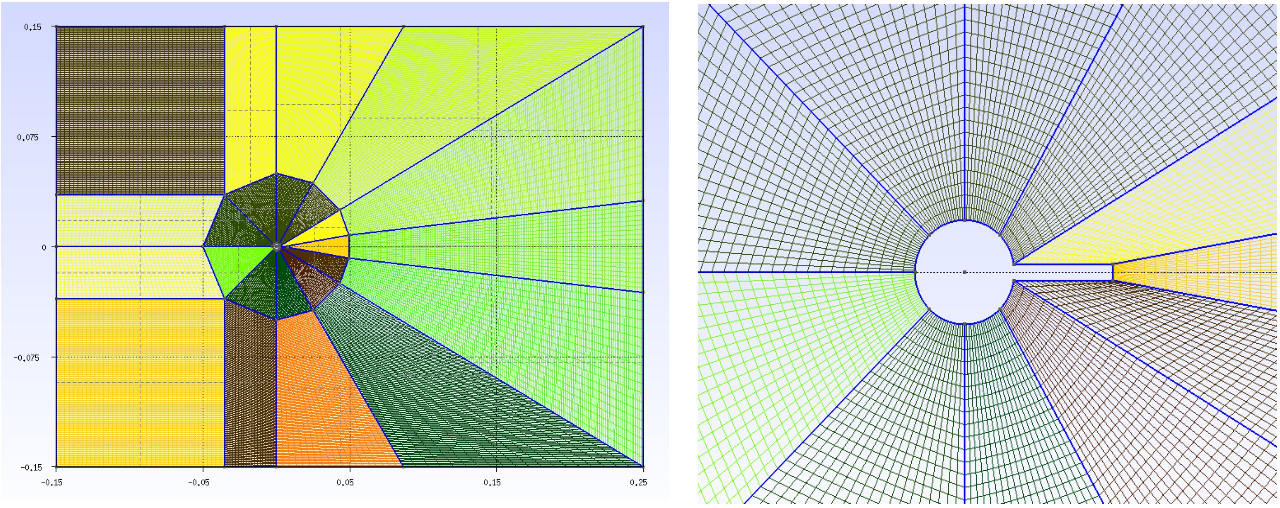

Figure 2.3: Mesh Topology Adopted (For Flow Over Cylinder With Rigid Splitter Plate at 0°)

Mesh Details checked using checkMesh(Cylinder With Rigid Splitter Plate Case):

No. of hexahedra cells: 57584.

Max skewness : 2.57826 OK.

2.3. Solver Augmentation For Adjustable Time Step Implementation:

The existing solver pisoFoam has been slightly modified to read the maxCo from the controlDict file in order to implement global time stepping during the computations. For this in the main pisoFoam.C file we have added the createTimeControls.H & CourantNo.H with setDeltaT.H

```
#include "createTimeControls.H"     // line added

turbulence->validate();

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

Info<< "\nStarting time loop\n" << endl;

while (runTime.run()) // added
{
    #include "readTimeControls.H"
    #include "CourantNo.H"
    #include "setDeltaT.H" // added
    runTime++;            // added
```

Figure 2.4: Adjustable Time step implementation in pisoFoam.C file

```
application     pisoFoam_Vino;
startFrom       latestTime;
//startTime       0;
stopAt          endTime;
endTime         400;
deltaT          0.025;
writeControl adjustableRunTime;
writeInterval   0.5;
//writeControl    timeStep;
// writeControl    adjustableRunTime; // OF-2.3.x
//writeInterval   10;
purgeWrite      0;
writeFormat     ascii;
writePrecision  6;
writeCompression off;
timeFormat      general;
timePrecision   6;
runTimeModifiable true;
maxCo 0.95; // Or other Courant number you wish
adjustTimeStep yes; // Or no
maxDeltaT 0.1; // Maximum deltaT in seconds
```

Figure 2.5: Adjustable Time step implementation in controlDict file

We set a limit for maxCo as 0.95 & the maxTimestep as 0.1s for our computations. It is found based on this limit the time steps were found to be 0.002 seconds during the simulations.

2.4. Post-Processing Methodology:

From the forceCoeffs.Dat file using fft the predominant frequency of vortex shedding is found from the python script & the corresponding maximum Cd is also determined inorder to check if it matches with the literature. The strouhal number estimated as,

$$St = f_s * D / U_\infty$$

The Cd(Drag Co-efficient) is estimated as,

$$Cd = F_d / 0.5 * \rho * U_\infty^2 * D$$

So, After the computations the python script attached has to be put in the post-processing folder & has to be executed to determine the plots over time & fft with Cd & St.

CHAPTER-3: EXPERIMENTAL APPROACH

3.1. Experimental Setup:

An attempt to do experimental validation with the same Re=100 in water tunnel has been made. The water tunnel is a closed loop with dye injection setup to visualise the vortex shedding. For the Re=100, The tunnel has been tuned to 16 mm/s velocity using variable frequency drive.The same procedure is repeated for the cylinder with splitter plate too. The splitter plate is kept at 0° behind the cylinder.

3.2. Methodology Followed:

The above experimental setup has been used to estimate the strouhal numbers for both cylinder with & without splitter plates. The stopwatch method sampling has been used to find the number of vortices shed for time of 50 seconds. Based on this sampling the total number of vortices per second has been estimated.

Drawback: The approach is fully manual observation based & thus the errors due to counting or lag in start/stop of sampling is expected here.

CHAPTER-4: RESULTS & DISCUSSION - COMPARISON OF NUMERICAL RESULTS WITH EXPERIMENTAL DATA

In this section let's compare the results of our numerical & experimental studies with existing Literature data,

Test cases	Flow quantities	Hwang et al. (2003)	Sudhakar & Vengadesan (2012)	Park et al (1998)	Kwon & Choi (1996)	Willia-mson (1989)	Posdziech & Grundmann (2007)	Bao & Tao (2013)
Plain cylinder, Re = 100	$\overline{C_D}$	1.34	1.37	1.33	-	-	1.32	1.335
	St	0.167	0.165	0.164	-	0.164	-	0.164
Single plate Re = 100, L/D=1	$\overline{C_D}$	1.17	1.174	-	1.18	-	-	-
	St	0.137	0.139	-	0.137	-	-	-

Table 3.1: Comparison of characteristic quantities for flow past a circular cylinder with and without splitter plates.

FOR FLOW OVER CYLINDER WITH Re=100 :

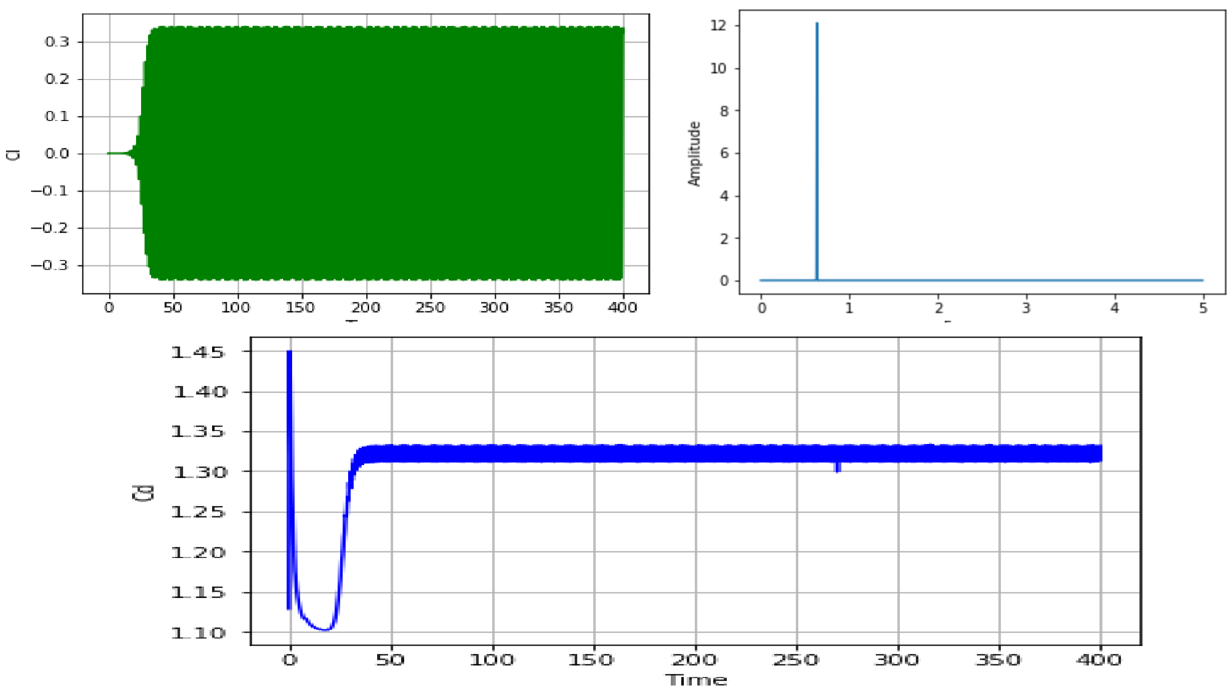

Figure 3.1: C_l, C_d , FFT Numerical Results For Flow Over Only Cylinder @Re=100

THE FREQUENCY OF VORTICES PER SECOND IS :

0.635

THE STROUHAL NUMBER IS :

0.178

THE Cd IS :

1.325

From Literature (Rezvan Abdi et.al), Several other results has been compared where for only cylinder Cd~1.32 which matches with our result of Cd~1.325.

FOR FLOW OVER CYLINDER WITH SPLITTER PLATE BEHIND AT 0° WITH Re=100:

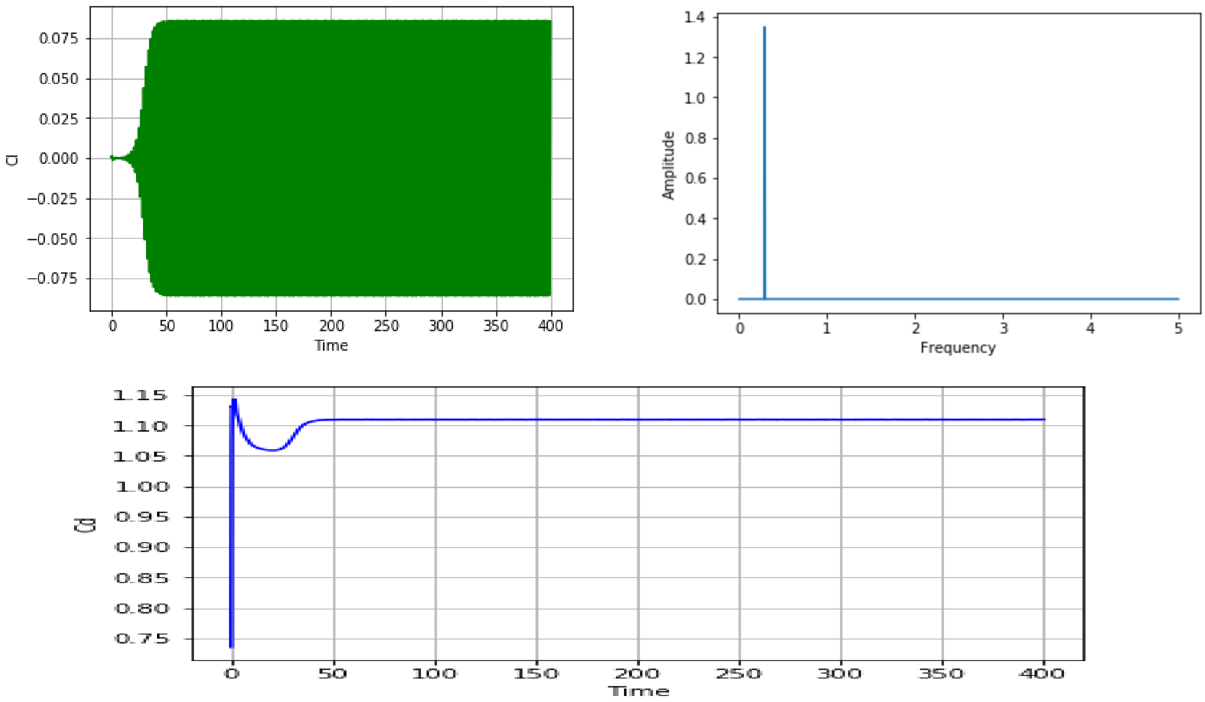

Figure 3.2: C_l, C_d , FFT Numerical Results For Flow Over Only Cylinder With Rigid Splitter Plate Behind at 0° @ Re=100 (NOTE – Plate Length (L) = Cylinder Diameter (D)).

THE FREQUENCY OF VORTICES PER SECOND IS :

0.2912

THE STROUHAL NUMBER IS :

0.0817

THE Cd IS :

1.11

For Cylinder with rigid splitter plate behind at 0 degrees, The Cd value obtained from our computation is 1.1 & from literature its 1.17.

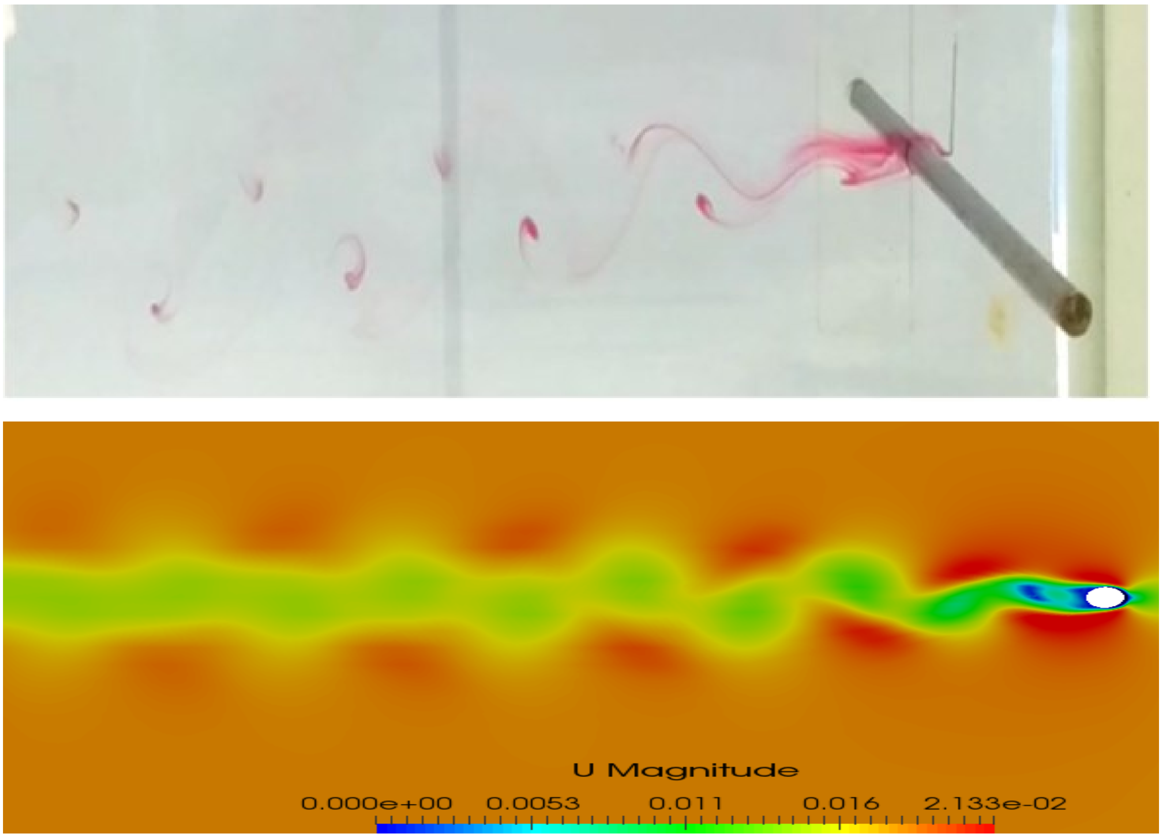

Figure 3.3: Numerical & Experimental Visualization Of Flow Over Cylinder @ Re=100

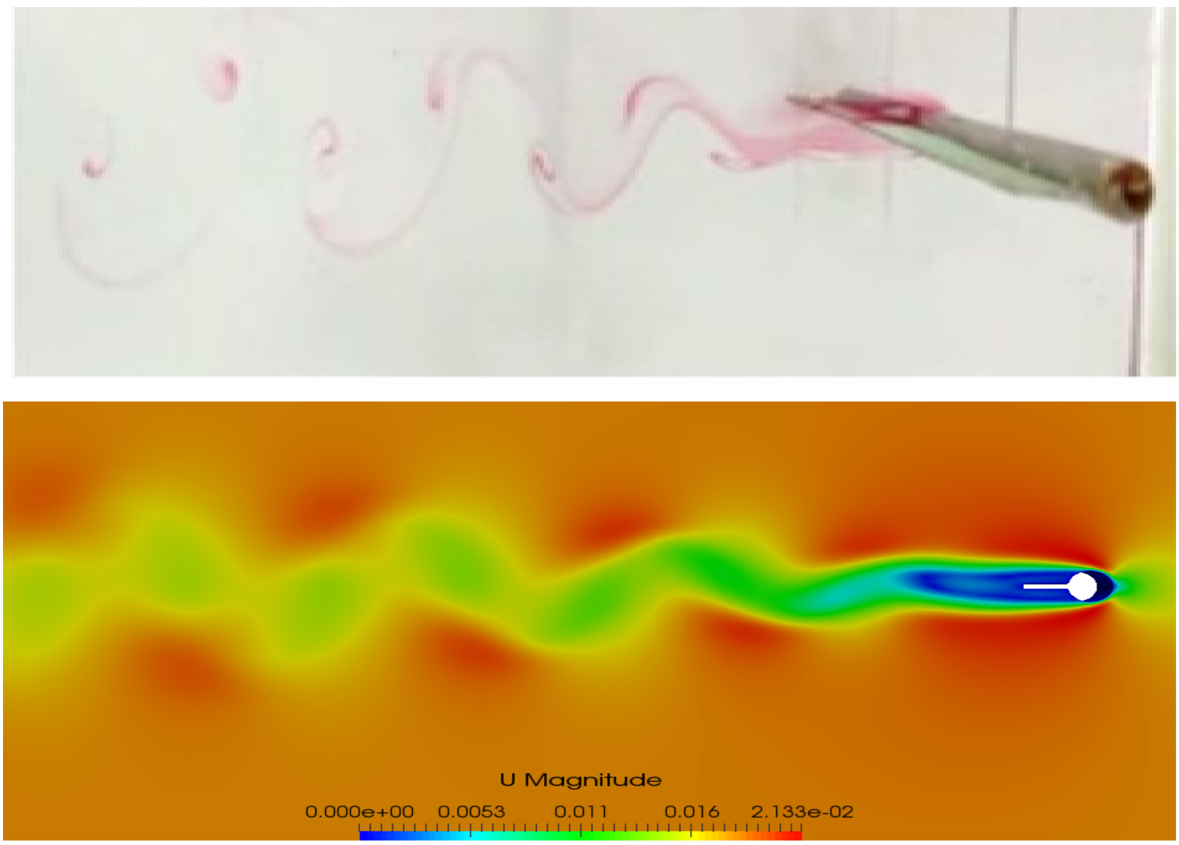

Figure 3.4: Numerical & Experimental Visualization Of Flow Over Cylinder With Rigid Splitter Plate @ Re=100

 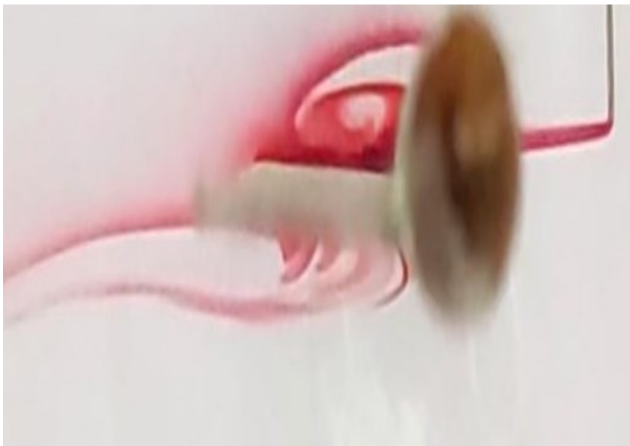

Figure 3.5: Experimental Visualization Of Flow Over Cylinder With & Without Rigid Splitter Plate @ Re=50 – Additional Results (Only Experimental) To Observe The Re-Circulation Zone Formed

The small differences between the present results and the numerical results from literature could be attributed to the different numerical approaches which have been implemented in the studies. Other possible reasons for the discrepancy of the numerical results in the different studies can include the blockage effect, the proximity of the cylinder to the inlet and the low cylinder distance from the outlet. The slight variation between the numerical & the excperimental results are mostly due to the manual error in noting down the number of vortices shed during the experiment.Also, The thickness of the plate is also an influencing parameter in the aspect of vortex shedding suppression.

CHAPTER-5: CONCLUSIONS & FUTURE SCOPE

In this extensive study we have adopted both computational & experimental approaches to study the vortex shedding suppression from cylinder using passive control method of attaching rigid splitter plate behind the cylinder. We have tried to validate with one configuration where the rigid plate is attached at 0°.

In this case study we have numerically estimated the Cd & St for flow over cylinder with Re=100 & had compared those results with experimental results with the water tunnel facility in our department. However, the numerical values using pisoFoam solver (Augmented for Adjustable time step) were exactly matching with that of the literature values while the experimental value of St number has mismatch of around 20% this could be due to the thickness to length ratio of the rigid splitter plate as in the numerical model we have used around 0.3D & in experiment its close to 0.05D. The study can further be extended by mounting the plates at different configurations say 15°,30°,45°,60°,90° Etc... Also by increasing the number of blades to 1,2,3 respectively enabling us to make a detailed observation on vortex shedding suppression by Bi & Tri-Splitter blades.

REFERENCES

1. http://ffden-2.phys.uaf.edu/211.fall2000.web.projects/c.%20Schaefer/aero4.htm

2. https://aerospaceengineeringblog.com/boundary-layer-separation-and-pressure-drag/

3. Kwon, K., & Choi, H. (1996). Control of laminar vortex shedding behind a circular cylinder using splitter plates. Physics of Fluids (1994-present), 8(2), 479-486.

4. Sudhakar, Y., & Vengadesan, S. (2012). Vortex shedding characteristics of a circular cylinder with an oscillating wake splitter plate. Computers & Fluids, 53, 40-52.

5. Reduction of Fluid Forces and Vortex Shedding Frequency of a Circular Cylinder using Rigid Splitter Plates by Rezvan Abdi , Niki Rezazadeh, Meisam Abdi,Sabzevar, Iran.

APPENDIX

The Post-Processing Python script mentioned below has been used to generate Cd,Cl & Estimate St Number. The script has to be placed in the Post-Processing folder & to be executed.

```
#!/usr/bin/env python2

# -*- coding: utf-8 -*-

"""

Created on Sun Sep 22 22:24:43 2019

@author: user

"""

import numpy as np

import pylab as pyl

import matplotlib.pyplot as plt

import csv as csv

import pandas as pd

from numpy.fft import fft

from itertools import groupby

import xlrd

import math as math

import scipy.signal as signal

IMPORT_DATA = np.genfromtxt('forceCoeffs.dat',unpack=True, usecols=[0,1,2,3,4,5])

Cl=IMPORT_DATA[3]

TIME=IMPORT_DATA[0]

Cl1=[]

for i in range(np.size(Cl)):

    Cl1.append(Cl[i])

signal_values1= fft(Cl1)

fftvalues=np.fft.fftshift(signal_values1)
```

```
plt.figure(1)

plt.plot(TIME,Cl1,'g')

plt.xlabel('Time')

plt.ylabel('Cl')

plt.grid()

plt.figure(2)

freq, Cl_amp = signal.welch(Cl, 1./0.1, nperseg=20000)

plt.plot(freq, Cl_amp)

plt.xlabel('Frequency')

plt.ylabel('Amplitude')

plt.show()

Cl_max_fft_idx = np.argmax(abs(Cl_amp))

freq_shed      = freq[Cl_max_fft_idx ]

print("THE FREQUENCY OF VORTICES PER SECOND IS")

print(freq_shed)

St=(freq_shed*0.005/0.0178)

print("THE STROUHAL NUMBER IS")

print(St)

Cd=IMPORT_DATA[2]

TIME=IMPORT_DATA[0]

Cd1=[]

for i in range(np.size(Cd)):

    Cd1.append(Cd[i])

signal_values11= fft(Cd1)

fftvalues1=np.fft.fftshift(signal_values11)

plt.figure(3)

plt.plot(TIME,Cd1,'b')
```

```
plt.xlabel('Time')

plt.ylabel('Cd')

plt.grid()
```