Project Report

"CFD - Simulation for Energy Systems"

Karman's Vortex Street

Submitted By: Group 12

Harikrishnan Sabu (Matriculation Number: 03757213)

Vishakh Cheruparambath (Matriculation Number: 03765715)

Submitted on: July 29, 2022

Examiner: Gaderer, Matthias, Prof. Dr.-Ing

Supervision: Gaderer, Matthias, Prof. Dr.-Ing

Huber, Bernhard, M.Sc.

Affidavit

I hereby certify that this project work was prepared without the help of third parties and only with the sources and aids indicated. All passages used have been marked. This work has not yet been submitted to any examination authority in the same or a similar form.

Harikrishnan Sabu

Vishakh Cheruparambath

Straubing, July 29, 2022

Summary

The following document presents a summary of the CFD simulations performed by Mr. Harikrishnan Sabu and Mr. Vishakh Cheruparambath as part of requirement for the completion of course “CFD- Simulation for Energy Systems” during summer semester 2022.

This report is focused on Karman vortex formation, while a medium of water is allowed to flow over a cylinder.

An iterative approach was applied for all the models considered, improving the models step by step towards the optimum solution. Several iterations were repeated, with constructive feedback from our professor.

All the simulations described in this report have been carried out using ANSYS CFD software: Space Claim for geometrical modelling, CFD Prep Post for mesh generation, Fluent for setting boundary and flow conditions, and finally CFD Post for post processing.

Table of Contents

Formula Symbols and Abbreviations

Symbol	*Size*	Unit
Re	*Reynolds Number*	(-)
ρ	*Density*	kg/m^3
μ	*Dynamic Viscosity*	$kg/m * s$
L	*Characteristic Length*	m
v	*Velocity*	m/s
p	*Pressure*	$bar\ or\ Pasacal$
Sr	*Strouhal Number*	(-)
C_D	*Coefficient of Drag*	(-)
C_L	*Coefficient of Lift*	(-)
y	*Height of First layer*	m
f	*Frequency*	Hz

Chapter 1- Introduction

In fluid dynamics, a Karman Vortex is a repeating pattern of swirling vortices, caused by a process known as vortex shedding, which is responsible for the unsteady separation of flow around blunt bodies. Under the correct flow conditions, the flow around a bluff body produces a regular pattern of alternating vortices known as a von Karman vortex street. These vortices produce a sinusoidal force acting on the body perpendicular to the flow. [1]

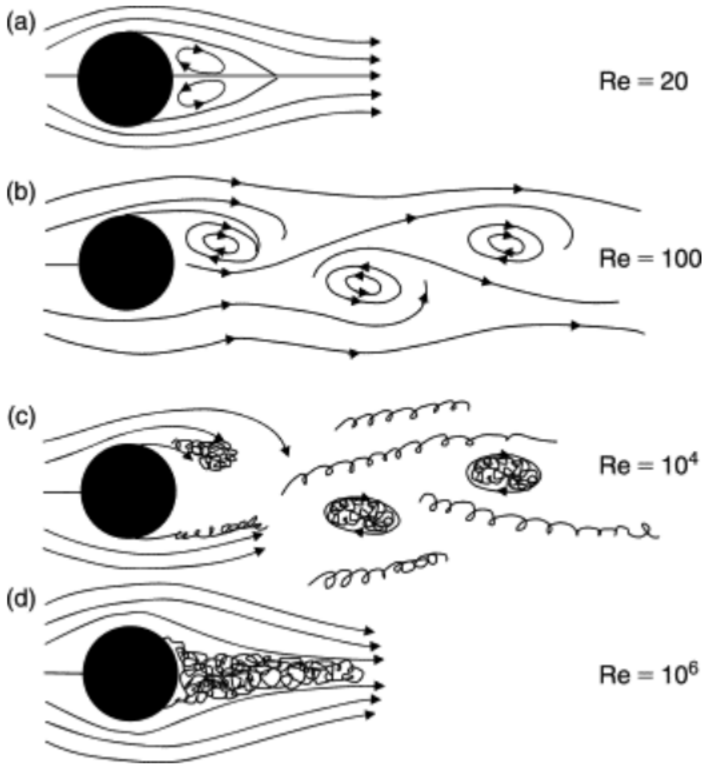

Figure 1: Vortex Forming According to different Reynolds Numbers *[2]*

For a cylinder, the frequency ω (in Hz) of the shedding is related to the diameter D(m) and wind speed v (ms^{-1}) via the Strouhal number S such that: [3]

$$S = \frac{\omega D}{v}$$

If the frequency of the shed vortices is the same as one of the natural frequencies of the structure, then large vibrations can result, also leads to damage of the structure if any.

1.1 Tacoma Narrows Bridge

Tacoma Narrows Bridge, suspension bridge across the Narrows of Puget Sound, connecting the Olympic Peninsula with the mainland of Washington state, U.S. Four months after opening of the first bridge, it suffered collapse in a wind of about 42 miles ($67kmh^{-1}$). The 840-metre main span went into a series of torsional oscillations and broke up.

Figure 2: Collapse of Tacoma Narrows Bridge *[4]*

The bridge collapsed primarily due to the aeroelastic flutter. The wind was forced to move above and below the structure, leading to flow separation, which lead to the development of Karman vortex street, as the flow passes through the object. [5]

1.2 Description of Project Task

Our task is to analyse the flow over a cylinder. The above-mentioned case study is a real-life example in the case of vortex forming, while flow over an object. In our case, a cylinder is situated in the medium of flowing water. The given datas are:

Flowing medium	Water
Condition	Transient, Frictional
Inlet temperature (T)	20^o C
Pressure (P)	1 bar
Dynamic Viscosity (η)	0.001 kg/m^3
Density (ρ)	1000 kg/m^3
Reynolds Number	50
Critical Reynolds number (for flow around a cylinder)	Re_crit > 10^4
Convergence criterion of the residuals	10^{-3}

Table 1: Given Datas

The simulation tasks are as given below:

- Preparation of a 2D geometry for the problem
- Mesh generation and refinement
- Simulation using Ansys Fluent
- Graphical representation of results

Calculation of Diameter and Velocity

$$Reynolds\ Number, Re = 50\ (Given)$$

$$Let\ the\ Diameter, = 20mm\ (Assumed)$$

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

$$From\ Table\ 1\ , substituting\ the\ values, Velocity\ (v) = 2.5\ mm/s$$

Since the Reynolds number is low, the considered turbulence model is $k - \omega$

Chapter 2 - Task A

2.1 Draw the geometry in Space Claim as a 2D fluid domain

The 2D fluid domain is generated by Space claim. The fluid domain is considered such that, the inlet to the cylinder distance is 3 times the diameter of the cylinder, outlet is 8 times the cylinder diameter and the distance from top and bottom walls are 3 times to the diameter of the cylinder.

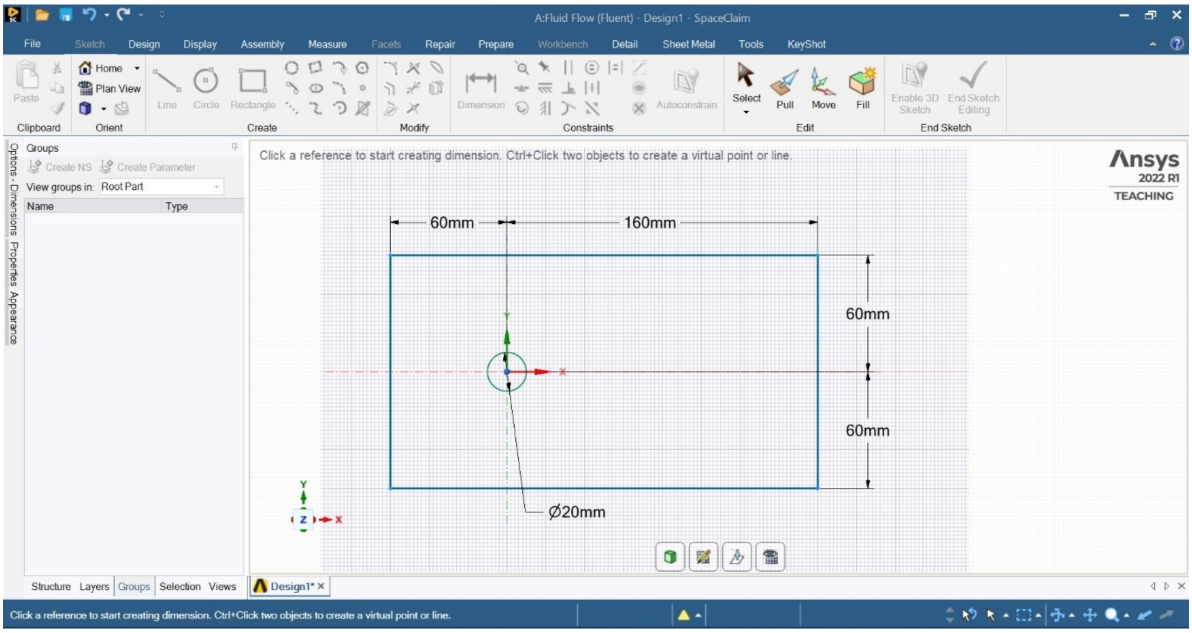

Figure 3: 2D Fluid domain in Space Claim

The boundaries of the fluid domain are also defined in space claim. Considering a fluid particle, it enters into the considered domain via “inlet”, proceeds to the “wall_cylinder” and exits through the “outlet”. The top and bottom boundaries are specified as “wall_top” and “wall_bottom” respectively.

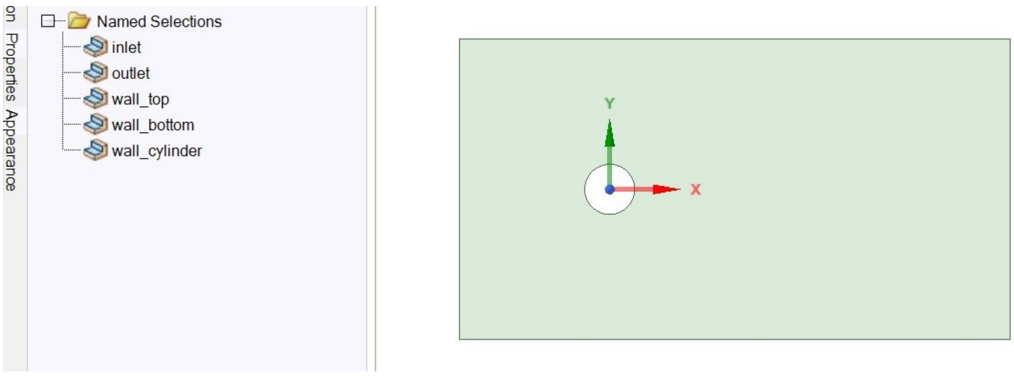

Figure 4: Naming of the Boundaries

Chapter 3 - Task B

The net generation should be unstructured. Depending on the turbulence model, the walls of the cylinder need to be refined with prism layers in such a way that the Y+ values do justice to the turbulence model.

Since the Reynolds number is low (i.e. Re = 50), the considered turbulence model is $k-\omega$. Also, Laminar model can be considered because of the low Reynolds number conditions.

3.1 Inflation layer calculation ($k-\omega$ model)

The assumed Y+ value = 0.75 (for $k-\omega$, y < 1)

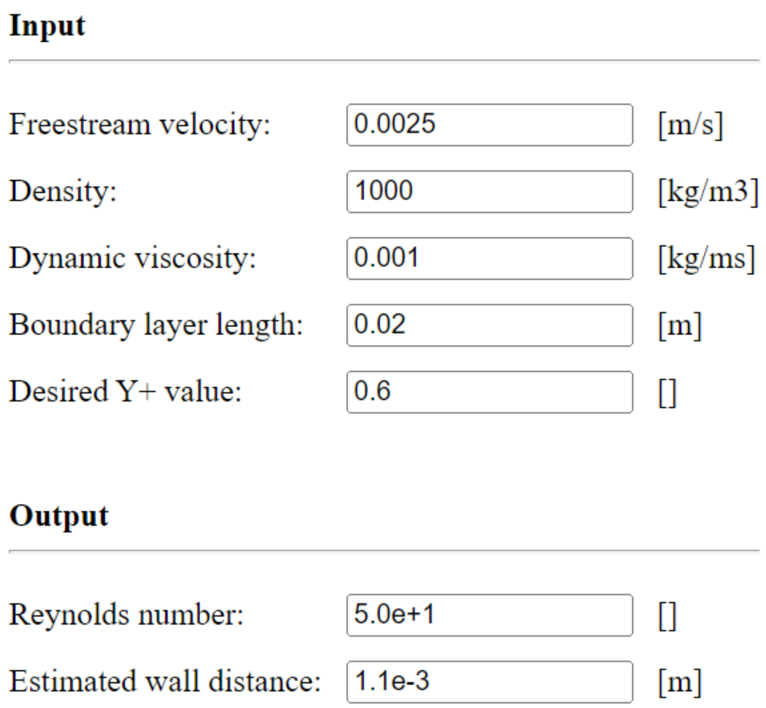

Figure 5: First Layer Calculation

Calculations according to the considered turbulence model $k-\omega$

$$Height\ of\ First\ Layer, y = 1.5\ mm$$

$$Number\ of\ Layers, n = 7\ (Assumed)$$

$$Growth\ Ratio = 1.05$$

$$Height\ of\ Last\ Layer, D = y * Growth\ Ratio^n = 1.5 * \ 1.05^7 = 2.11\ mm$$

$$Global\ Element\ Size = D * Growth\ Ratio = 2.11 * 1.05 = 2.21\ mm$$

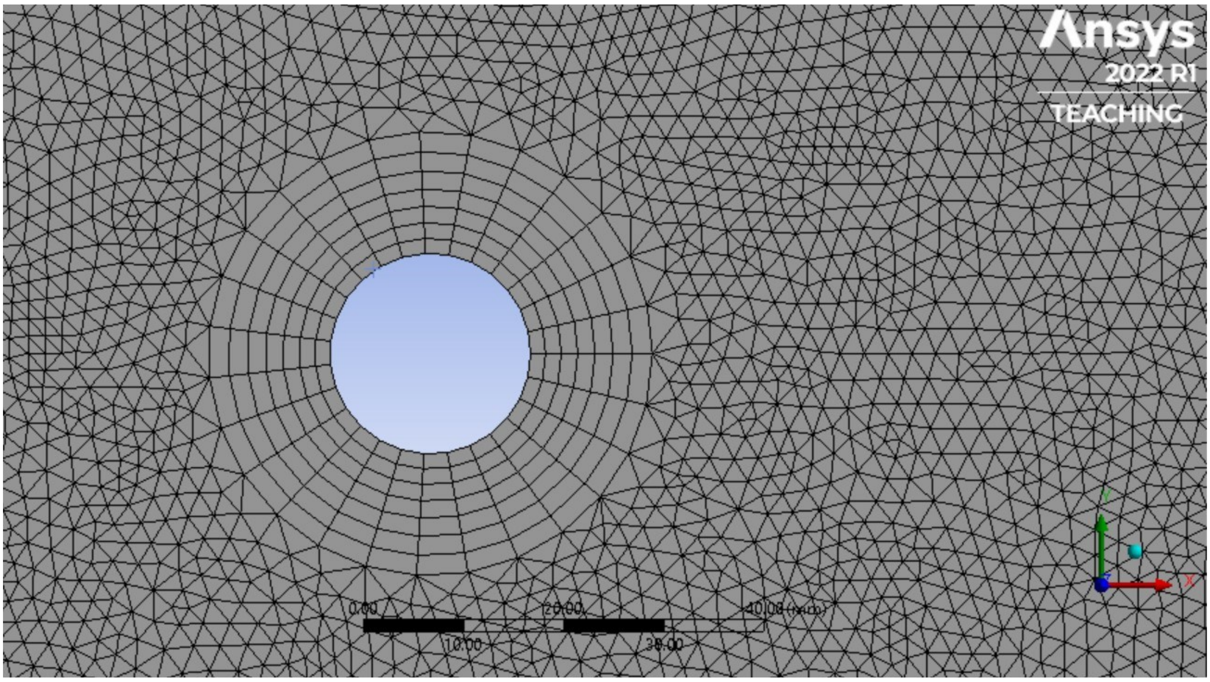

Figure 6: Mesh Generation with Inflation Layers

Even though unstructured meshing is done, we have got a clear main direction of flow. So decided to do simulation on a structured mesh also, and compare the results. Grid independent study was also considered for comparison of results in the future.

3.2 Mesh Generation- Parameters

Mesh Generation (Unstructured)	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Method	All Triangles Method
Element Size	2.3 mm

Inflation Layer Parameters	
Inflation Option	First Layer Thickness
First Layer Height	1.5mm
Maximum Layers	7
Growth Rate	1.05
Inflation Algorithm	Pre

Table 2: Meshing Parameters- Unstructured

Mesh Generation (Structured)		
Physical Preference	CFD	
Solver Preference	Fluent	
Method	All Triangles Method	
Element Size	12.53 mm	
Edge Sizing		
	Inlet and Outlet	Wall_top and Wall_bottom
Type	Number of divisions	
Number of divisions	60	110
Face Meshing		
Method	Triangles: Best Split	

Table 3: Meshing Parameters- Structured

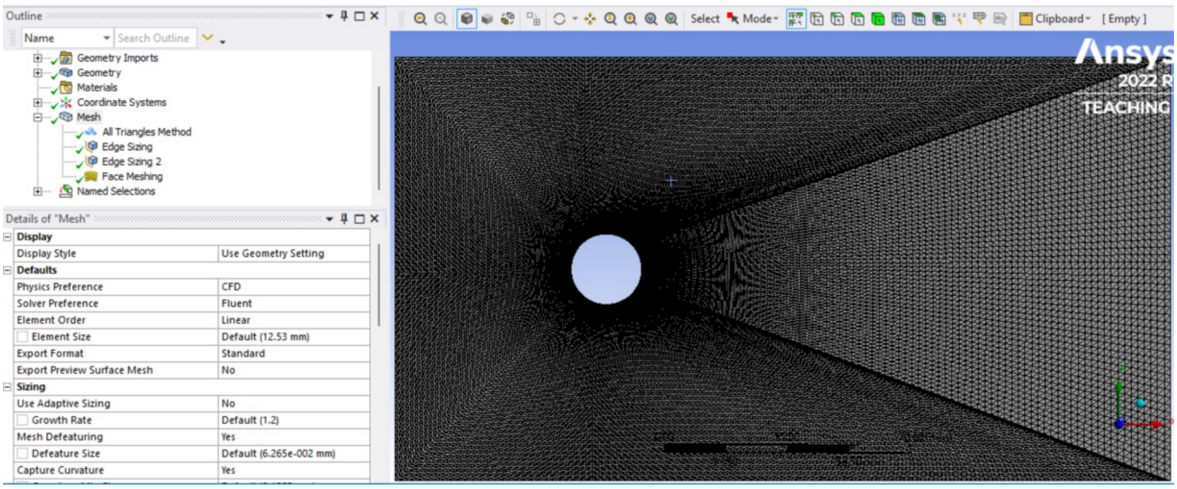

Figure 7: Structured Meshing

3.3 Mesh Quality Analysis

Mesh Quality Analysis- Unstructured Meshing

Analysing the quality of number of numerical errors present in mesh generation is very important for the end results to be more accurate. The basic parameters that define the quality of a mesh is skewness, orthogonality and aspect ratio. The table below is the analysis of mesh quality.

Skewness

Skewness is the deviation between the optimal cell size to the existing cell size.

Skew	Quality
1	
0,9 ……1	Insufficient, English sliver
0,75 …. 9	Poor
0,5 ……0,75	Satisfactory
0,25 …..0,5	Good
0 ………0,25	Very good
	Optimum

Figure 8a: Recommended Skewness

Orthogonality

Orthogonality means the angle between adjacent element faces of cells.

Orthogonality	Quality
0	
0 ……0,001	Insufficient, English sliver
0,001 …. 0,14	Bad
0,15 ……0,19	Satisfactory
0,20 …..0,69	Good
0,70 …. 0,95	Very good
	Optimum

Figure 8b: Recommended Orthogonality

Aspect Ratio

Aspect ratio is the ratio of a cell’s longest length to the shortest length.

Recommendation: A < 10…100

	Unstructured Meshing	Structured Meshing
Skewness	0.05	0.9
Orthogonality	0.968	9.56413e-05
Aspect Ratio	1.208	3.53363e+03

Table 4 Mesh Quality Analysis

From the table above, it is clear that in the case of unstructured meshing, the mesh quality is optimal. The quality met the standard mesh quality values.

But, in the case of structured meshing, the skewness quality is low. The orthogonality is also below 0.1, which is a low-quality mesh. But due to system limitations and after performing numerous trial and error methods, this is the maximum quality that can be attained.

Chapter 4 - Task C

The RANS equations should be used in Ansys Fluent. The complete setup with selection of turbulence models, boundary conditions and solver should be described in the report. Diagrams of residuals and a monitor point the pressure in the outlet of the cylinder should be created.

The meshed models are simulated in Ansys Fluent as follows. The table below shows the input parameters used for the simulation.

4.1 CFD Fluent- Input Parameters

Fluent Version: 2d, pbns, sst k-ω, transient (2d, pressure-based, SST k-omega, transient)	
Models	
Model	**Settings**
Space	2D
Time	Unsteady, 1st-order Implicit
Viscous	SST k-ω turbulence model
Heat Transfer	Disabled
Multiphase	Disabled
Material Properties	
Material: Water – Liquid (Fluid)	
Property	**Values**
Density	998.2 kg/m^3
Cp (Specific Heat)	4182 J/(kg K)
Thermal Conductivity	0.6 W/ (m K)
Viscosity	0.001003 kg/ (m s)
Molecular Weight	18.0152 kg/kmol
Boundary Conditions	
Zones	**Type**
Inlet	Velocity- Inlet
Outlet	Pressure- Outlet
Wall_Top	Wall
Wall_bottom	Wall

Wall_Cylinder	Wall	
Setup Conditions		
Inlet: Velocity Specification Method X- Velocity [m/s] 0.0025		
Walls: Stationary wall with no slip		
Solver Settings		
Unsteady Calculation Parameters	**Structured Meshing**	**Unstructured Meshing**
Number of Time steps	100	100
Time Step Size [s]	5.012	0.001
Max Iterations/ Time Step	50	20
CFL Number	1	1
Type of Solver	Linear	
Pressure-Velocity Coupling		
Type	**Coupled**	
Flow Courant Number	1	
Discretization Scheme		
Variable	Scheme	
Pressure	Second Order	
Momentum	Second Order Upwind	
Turbulent Kinetic Energy	Second Order Upwind	
Specific Dissipation Rate	Second Order Upwind	

Table 5: Input Parameters_ Ansys Fluent

4.2 Result Analysis- Structured Meshing

The simulation was carried out according to above specified input conditions, and the residuals are oscillating. It may be desirable in a transient flow condition. The solution is not converged, unfortunately. That may be because of the poor mesh quality. The results are calculated with different number of iterations as well.

A converged solution may be attained by specifying the convergence criteria more strictly and also by performing the calculation with a greater number of iterations.

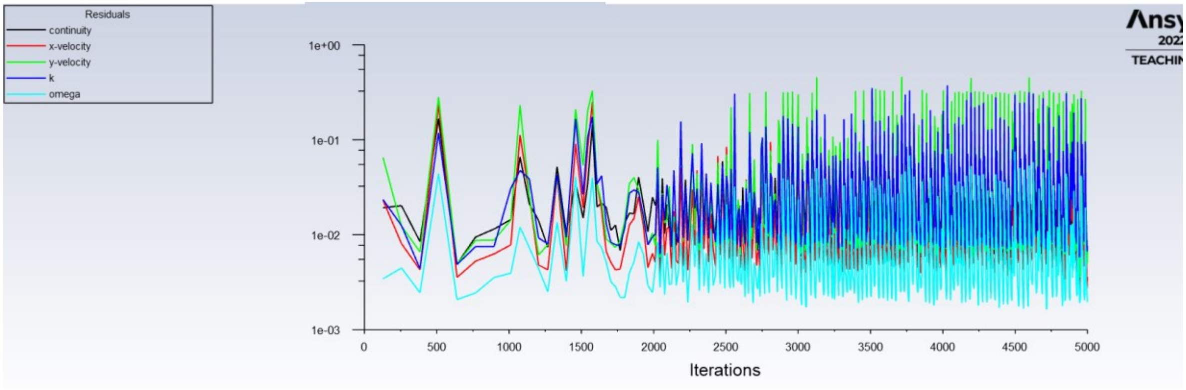

Figure 9: Residuals_Structured Mesh

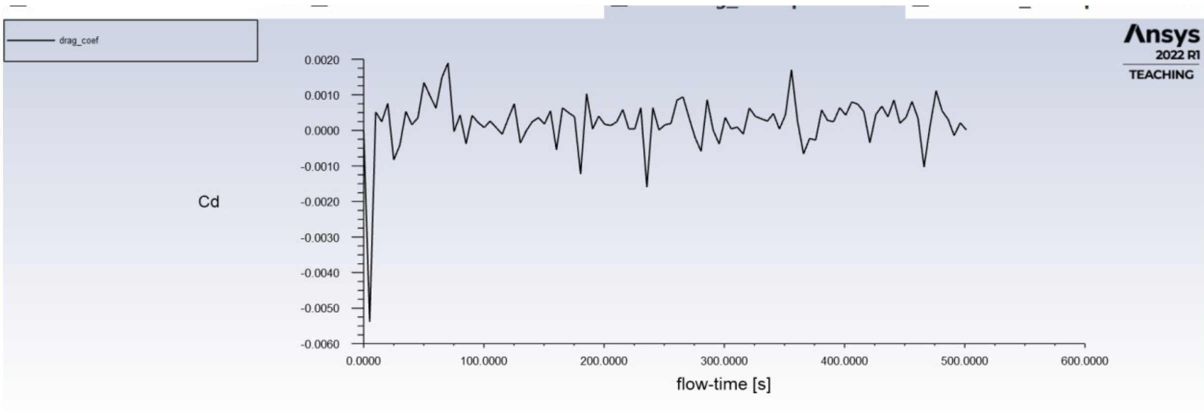

Figure 10: Drag Coefficient_Structured Mesh

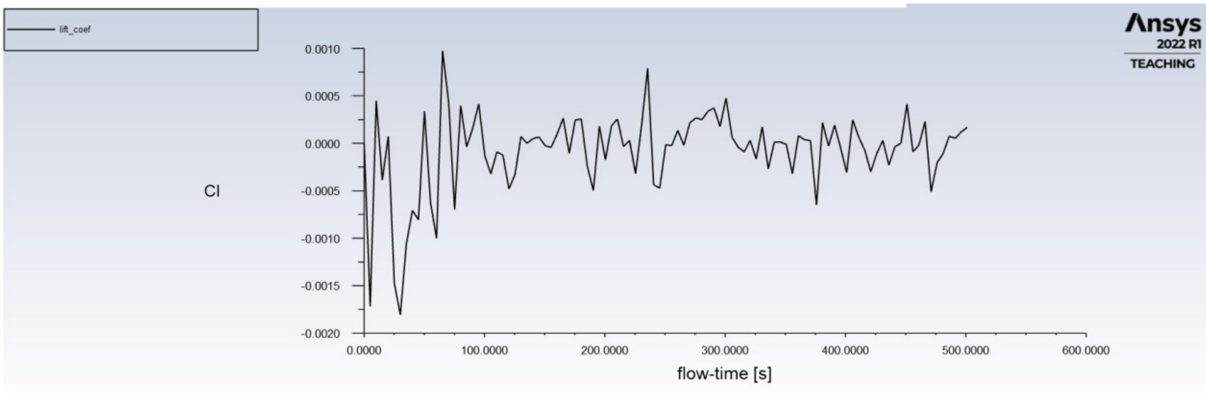

Figure 11: Lift Coefficient_Structured Mesh

4.3 Result Analysis- Unstructured Meshing (k-ω Turbulence Model)

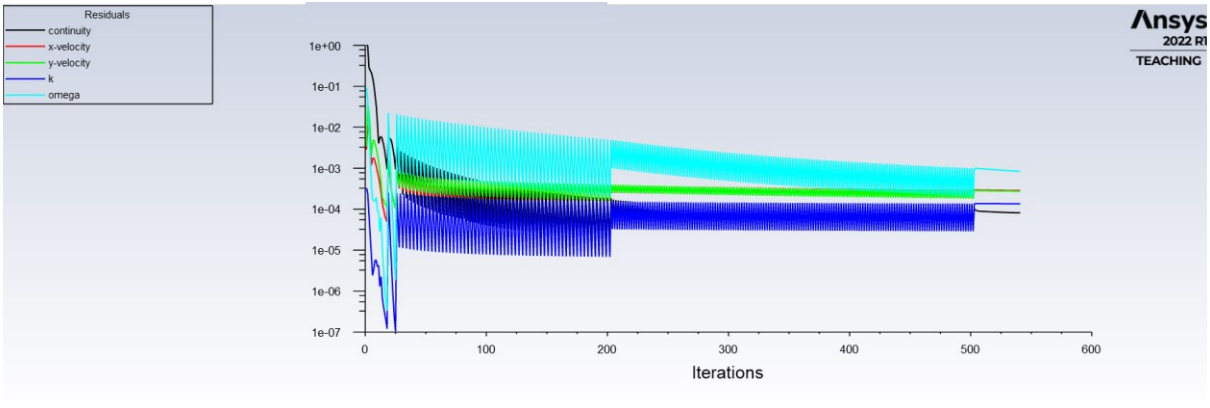

Figure 12: Residuals_Unstructured Mesh

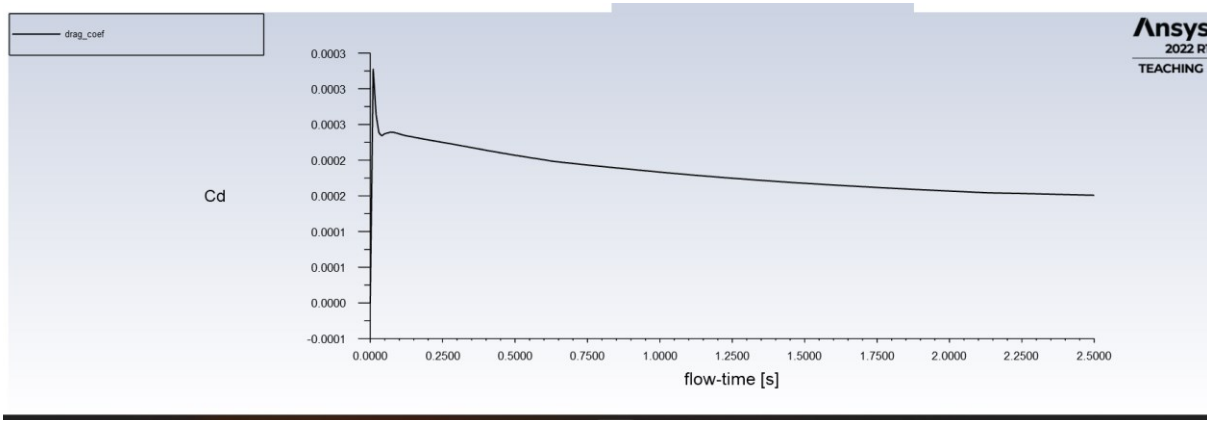

Figure 13: Drag Coefficient_Unstructured Mesh

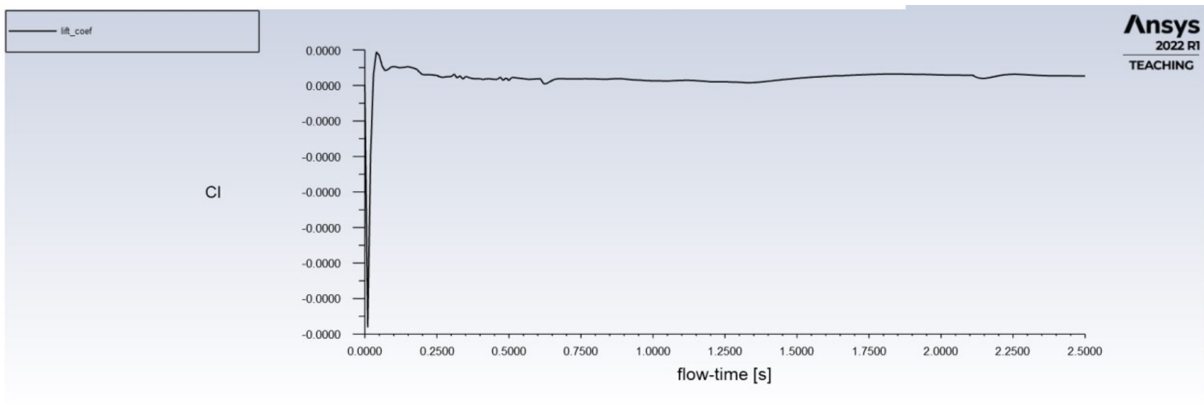

Figure 14: Lift Coefficient_ Unstructured Mesh

4.3 Result Analysis- Unstructured Meshing (Laminar Model)

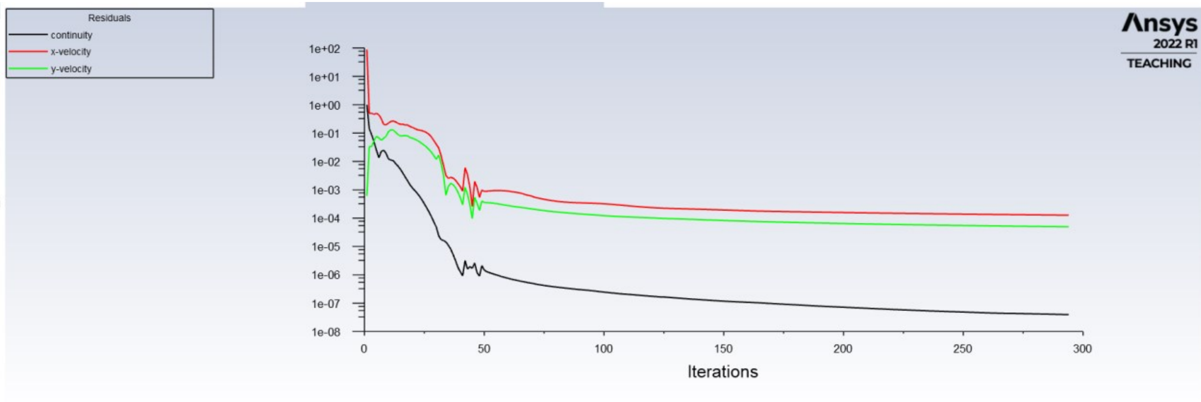

Figure 15: Residuals_Unstructured Mesh with Laminar Flow

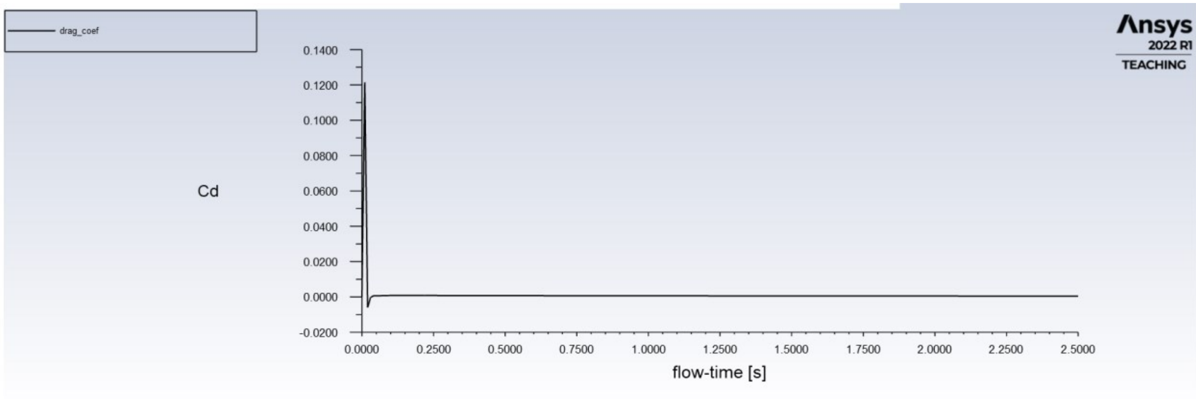

Figure 16: Drag Coefficient_Unstructured Mesh with Laminar Flow

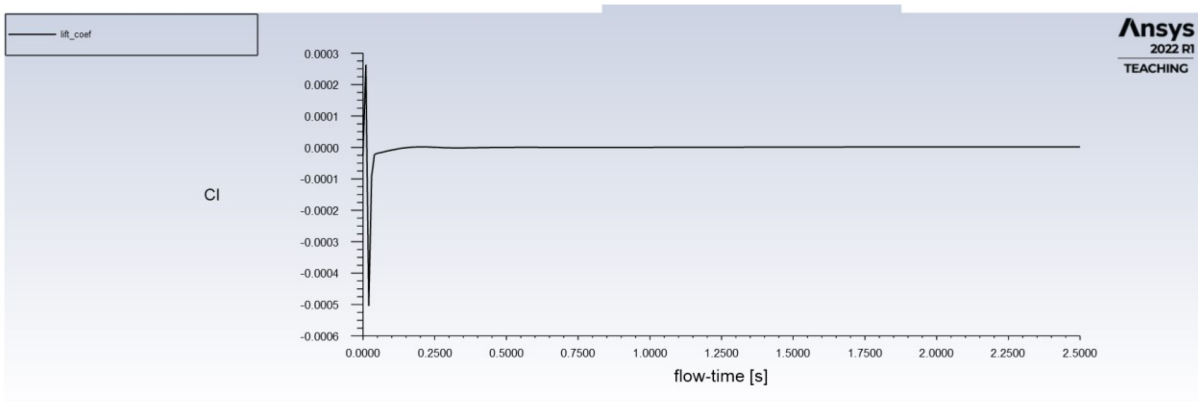

Figure 17: Lift Coeffcient_Unstructured Mesh with Laminar Flow

In k-ω turbulence models, the residuals were fluctuating. On the other hand, in when laminar conditions were applied, the residuals showed a smooth pattern and converged after a certain number of iterations. Since the Reynolds number was very low (i.e Re = 50), a laminar condition may be more preferred.

After simulating the flow over a 2D cylinder for varying Reynolds number (Re) using a steady- state solver with laminar and k-ω turbulence model, the data collected gives the relationship between flow time and Lift and Drag coefficient (C_L & C_D) near the wall of the cylinder. Since the solution have been run for several iterations, we observed oscillatory behaviour for all cases. This is caused due to vortex shedding downstream of the cylinder.

The values of C_L & C_D have been averaged for all cases except structured meshing to obtain a single value. From the simulation results, it is observed that with change in turbulence model used, the values of C_L & C_D also change.

Chapter 5- Task D

To analyse Karman's Vortex street, Strouhal-Number is used. The Strouhal-Number is defined as:

$$Sr = \frac{f * d}{c}$$

Where, Sr is the Strouhal-number, f is the vortex separation frequency, d is the diameter of the cylinder and c is the velocity of attack.

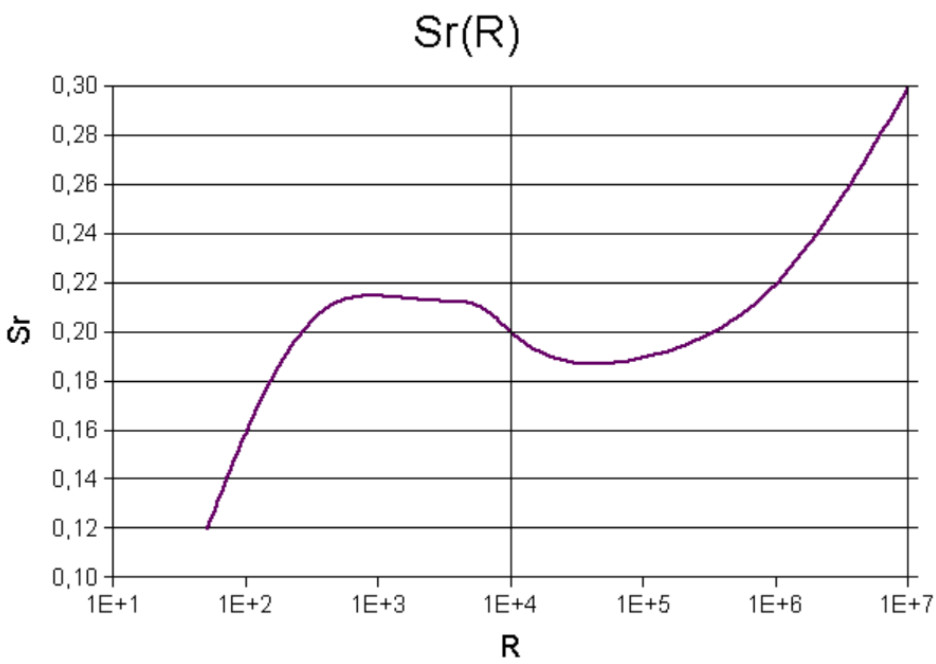

Figure 18: Sr - Re Graph

In our case, the Reynolds number is given as 50. From the graph, Strouhal number is defined as 0.12.

From the equation:

$$Sr = 0.12$$

$$d = 20mm$$

$$c = 2.5\ mm$$

So, Vortex Shedding Frequency, $f = 0.015\ Hz$

Chapter 6- Task E

Create a line through the centre of the domain and around the circle. Plot the velocity and pressure distribution and determine the vortex separation point at the wall of the circle.

6.1 Pressure Distribution on Measuring line

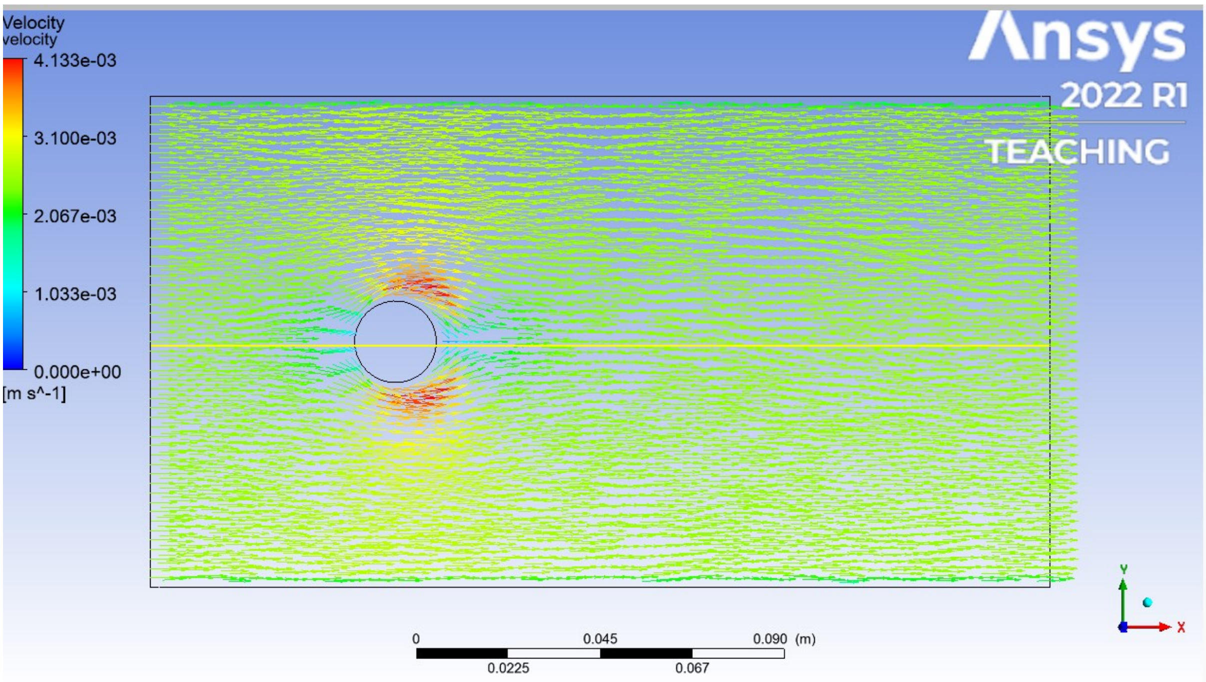

Figure 19: Measuring Line for Velocity and Pressure Distribution

A measuring line is created through the centre of the cylinder and the pressure and velocity distribution on the measuring line is calculated. The pressure and velocity variations both in k-ω as well as in the laminar model are described and compared below.

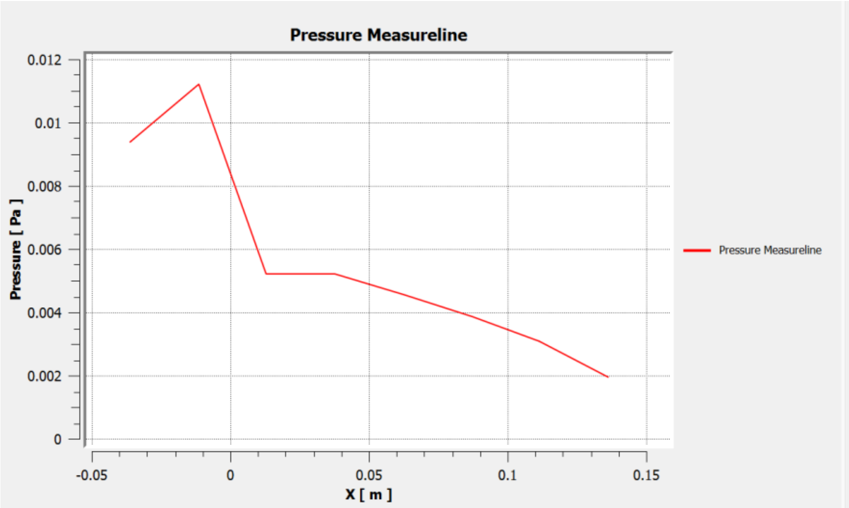

Figure 20: Pressure Distribution_ Laminar Flow

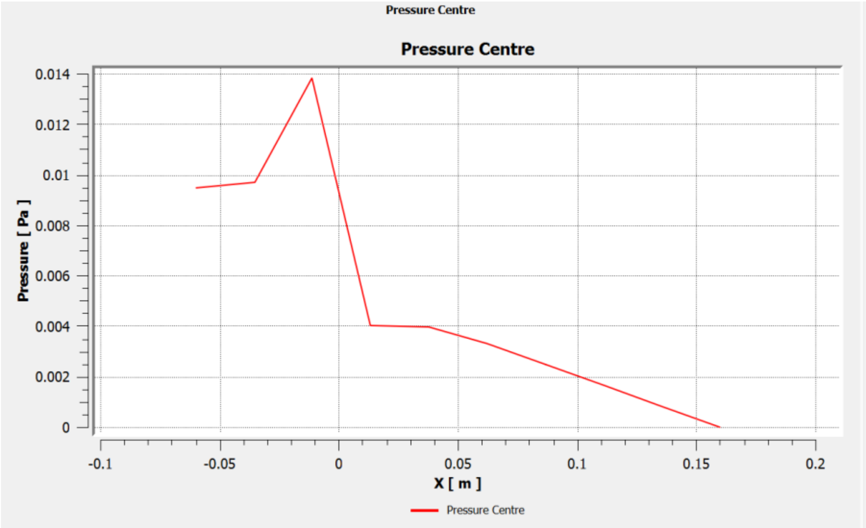

Figure 21: Pressure Distribution_ k-ω Turbulence Model

In laminar flow, as compared to k-ω turbulence model, the pressure distribution is somewhat low with respect to the distance across the flow domain. The minimum and the maximum pressure reached is more in k- ω turbulence model. In both models, the pressure distribution starts from a value of approximate 0.0975 and reach a maximum when the flow touches the cylinder. Then the pressure drops significantly and again after moving the pressure starts to drop linearly.

6.2 Velocity Distribution on Measuring Line

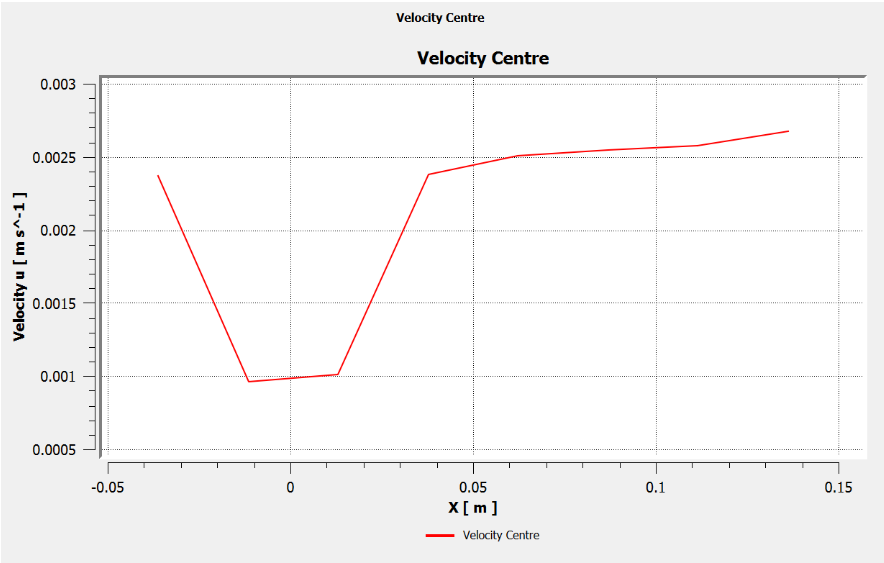

Figure 22: Velocity Distribution_ Laminar Flow

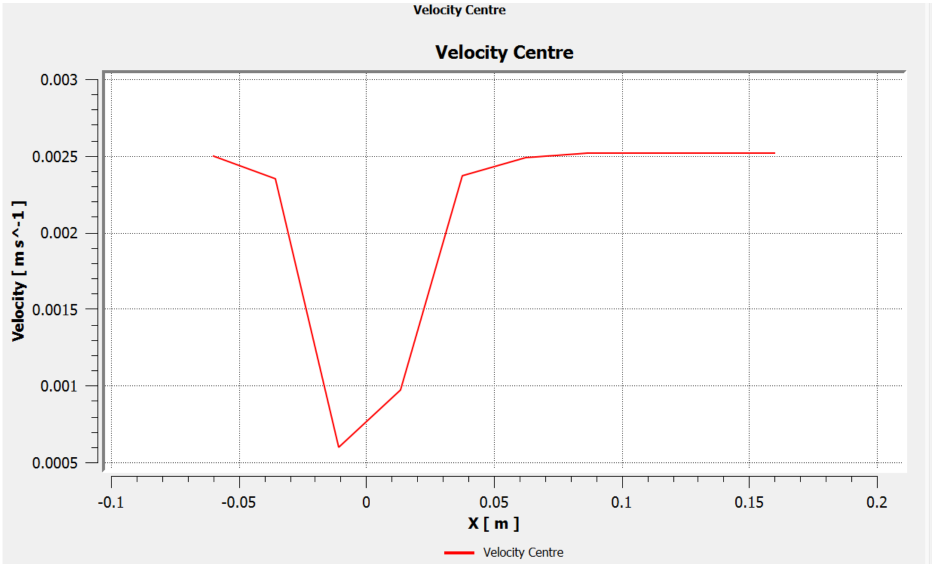

Figure 23: Velocity Distribution_ k- ω Turbulence Model

Looking into the graphs of velocity distribution, the velocity of both models starts from 0.0025 m/s and when the flow reaches the cylinder walls, it significantly drops to a minimum value. In this case also, the velocity drop is maximum in k-ω turbulence model.

As it is seen, the flow velocity of vortices decreases at the beginning and then increases to the stable value equal to the axial velocity of the flowing fluid.

Chapter 7- Task F

It is to be conducted a grid independence study. For this purpose, a rough, unstructured network should be started and at least 2 further refined networks should be calculated. It's supposed to be created a diagram for Strouhal- Number and vortex separation frequency in the wake of the cylinder.

Grid independent study is carried out in this section.

7.1 Grid Independent study

	Global Element Size
Case 1	2.07 mm
Case 2	1 mm

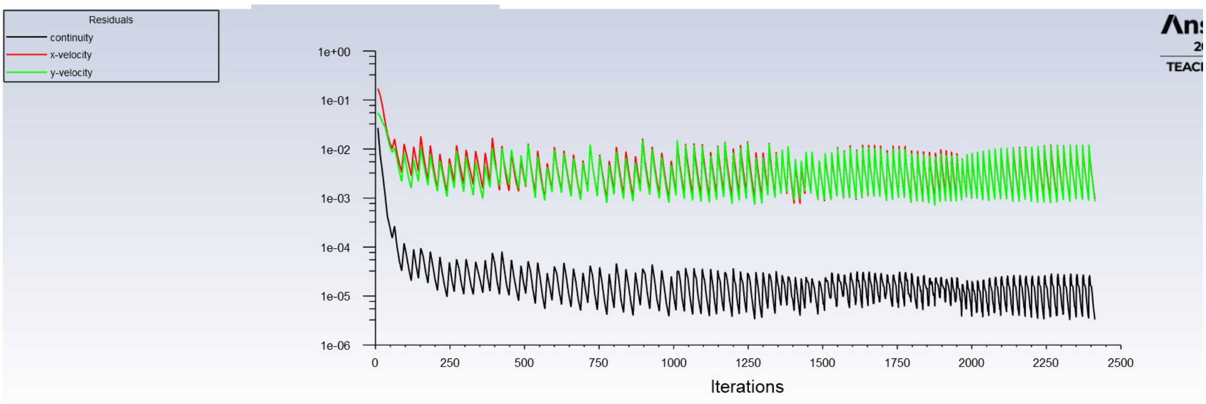

Figure 24: Residuals_ Case 1

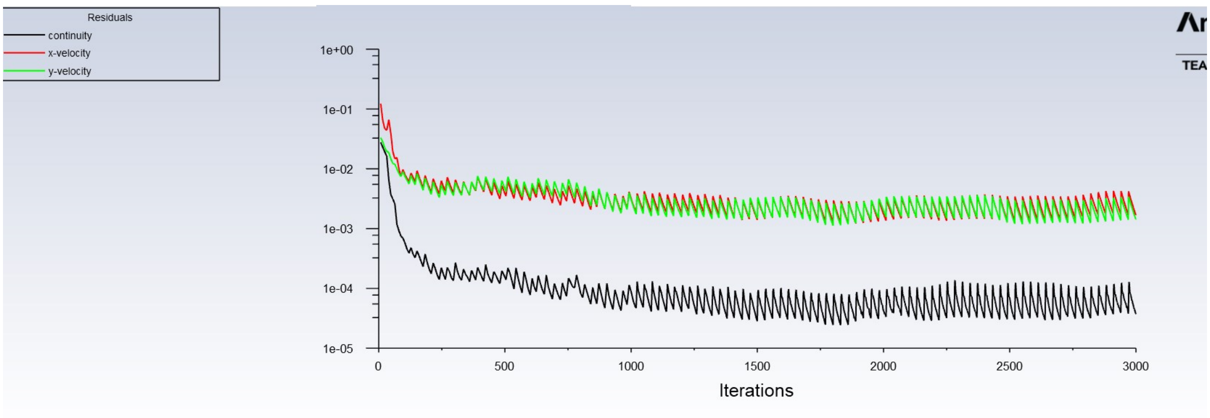

Figure 25: Residuals_ Case 2

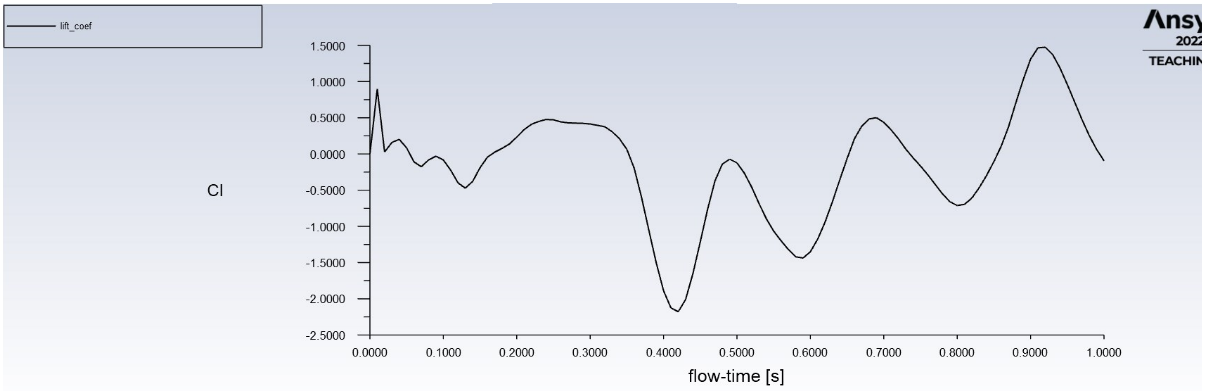

Figure 26: Lift Coefficient_Case 1

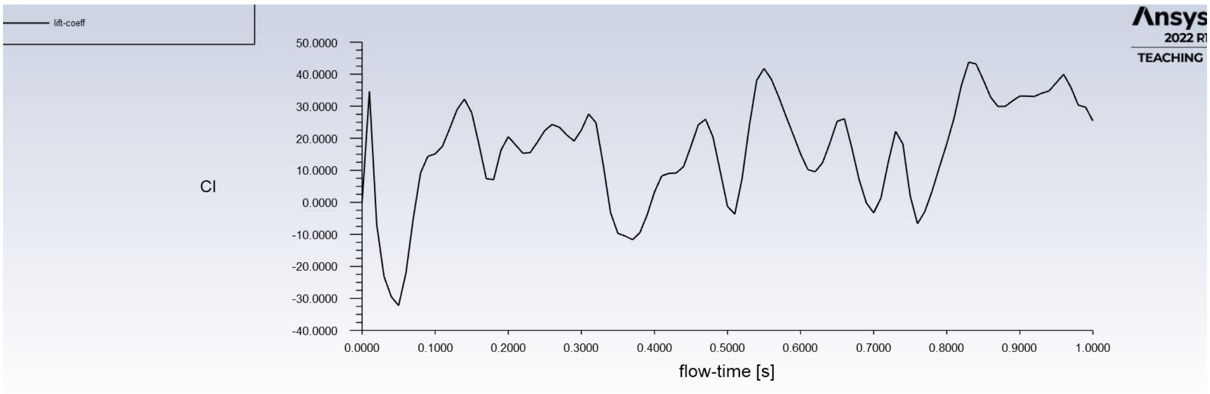

Figure 27: Lift Coefficient_ Case 2

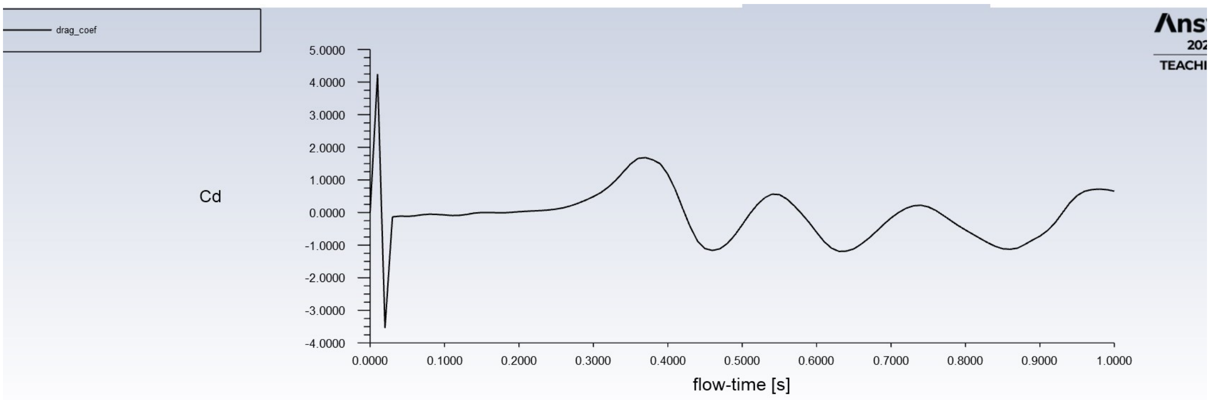

Figure 28: Drag Coefficient_Case 1

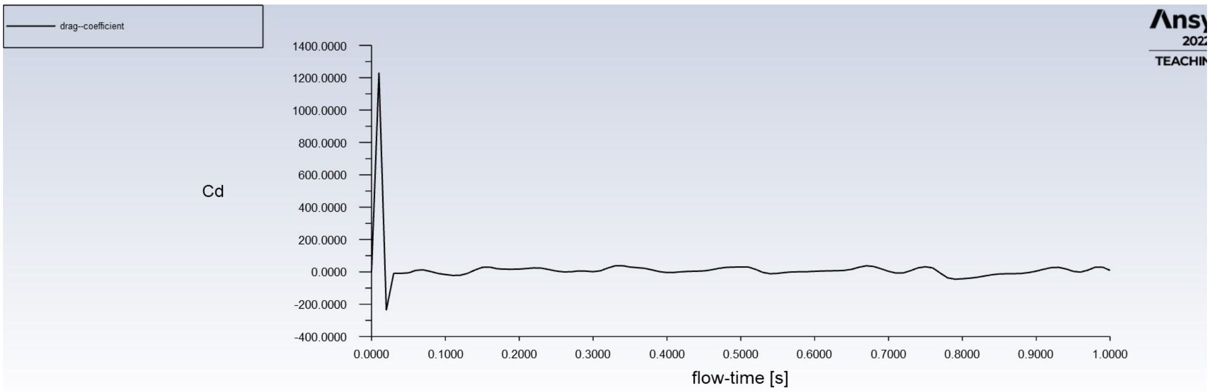

Figure 29: Drag Coeffcient_ Case 2

The change in residuals as well as lift and drag coefficients are relevant from the above graphs, with respect to decrease in mesh size. The drag coefficient became a linear constant when the mesh size reduced to 1mm.

Chapter 8- Task G

8.1 Plausibility Check: Control of the Mass balance

The general equation used for mass balance is:

$$massFlow(\quad)@inlet - massFlow(\quad)@outlet$$

For our calculations, mass balance is extracted from CFD Post as:

Mass Balance	
General Expression: massFlow()@inlet - massFlow()@outlet	
Type	**Value**
For Unstructured Mesh with Laminar Flow	0.000717233 [kg s^-1]
For Unstructured Mesh with k-ω Turbulence Model	0.000718704 [kg s^-1]

In both turbulence models, the mass balance seems to be approximately same value, with minute difference. The difference may be due to poor mesh quality as well as calculation errors.

Chapter 9- Task H

9.1 Creation of a Contour- Plots of the Pressure distribution

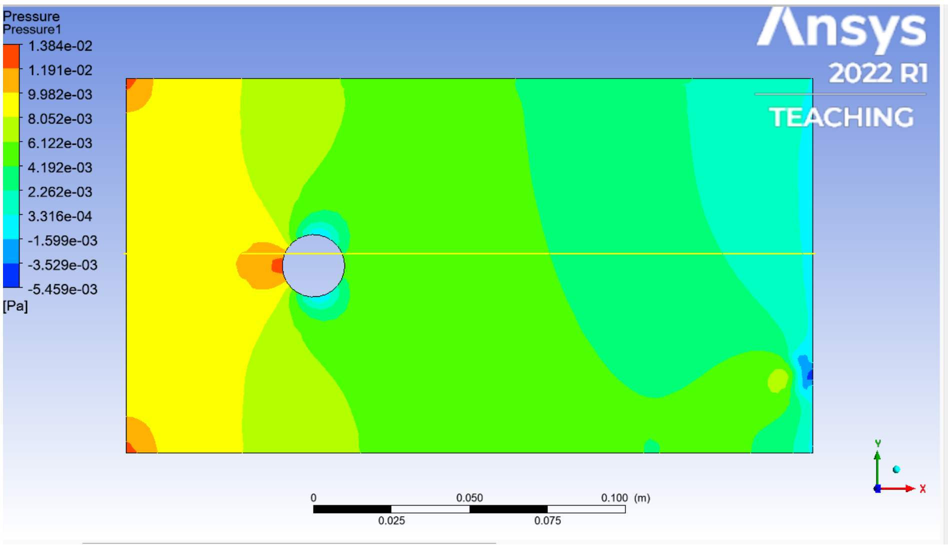

Figure 30: Pressure Contour_ Laminar Flow

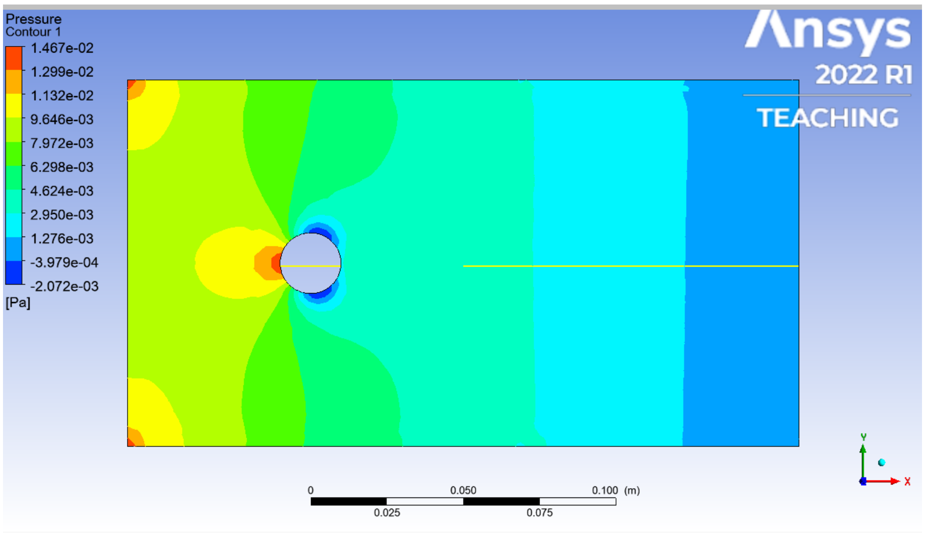

Figure 31: Pressure Contour_ k-ω Turbulence Model

The pressure contours of Laminar as well as k-ω turbulence models are given above. From these diagrams, the overall pressure distribution can be analysed.

Chapter 10- Task I

10.1 Creation of a vector plot of speed distribution

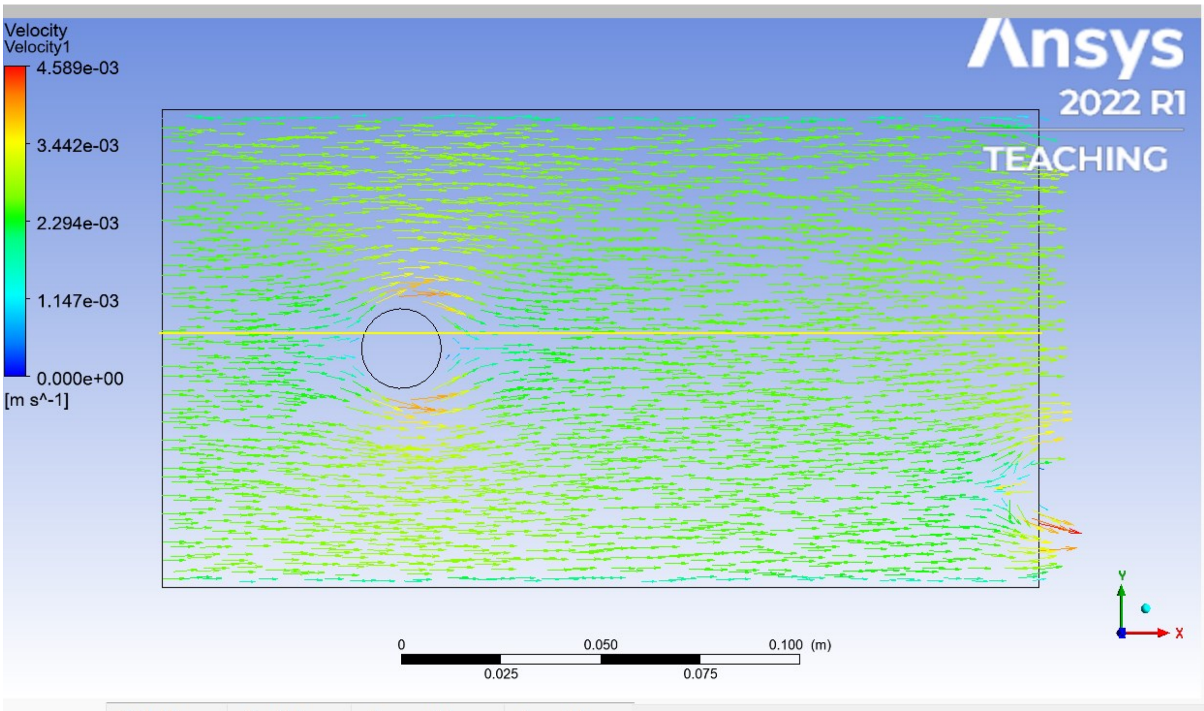

Figure 32: Velocity Vector Plot_ Laminar Flow

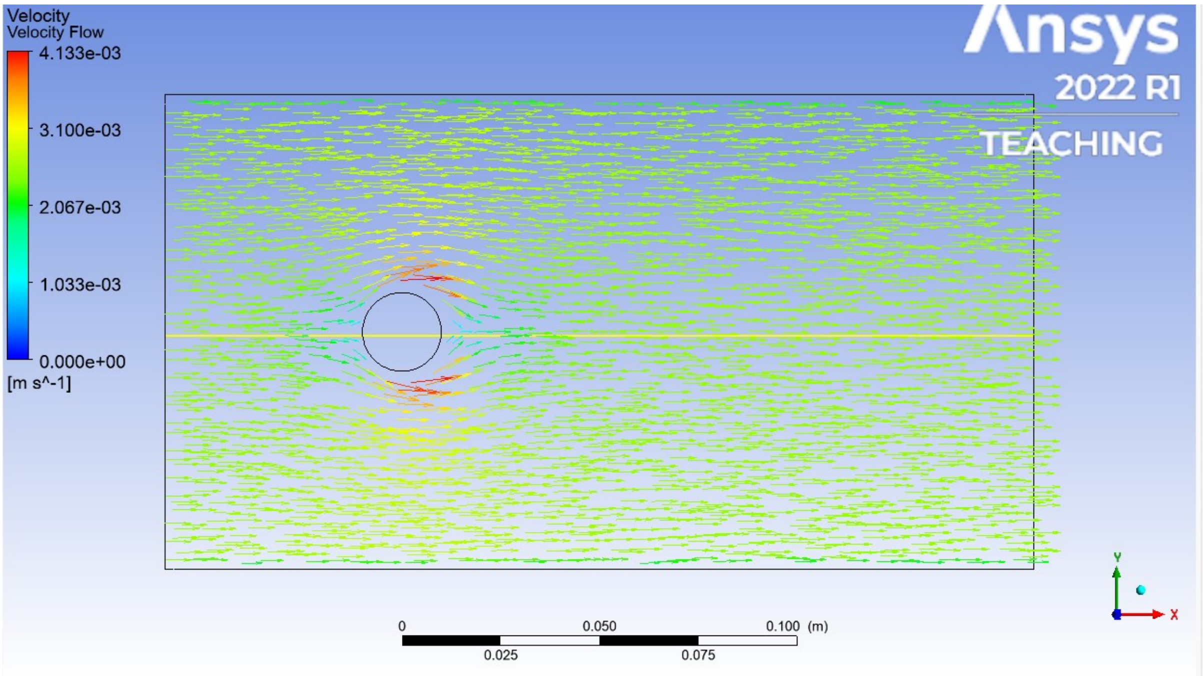

Figure 33: Velocity Vector k-ω Turbulence Model

Conclusion

Task J

While comparing with the given literature, the residuals look similar in both cases. The pressure distribution is more fluctuating in the provided literature, while in our case, it looks normal. Within the frame work of this project, the focus of the observation was not only on the evaluation of vortex shedding using key figures, but also on the graphic processing. [6]

After simulating the flow over a 2D cylinder for varying Reynolds number (Re) using a Steady-State solver with laminar as well as k-ω turbulence model, the data collected gives us the relationship between Reynolds number & Lift & Drag coefficient (C_L & C_D) near the wall of the cylinder. Since the solutions have been run for several iterations, we observed oscillatory behaviour, due to the vortex shedding on the cylinder walls.

Bibliography

[1] Wikipedia, “Wikipedia,” 01 June 2022. [Online]. Available: https://en.wikipedia.org/wiki/K%C3%A1rm%C3%A1n_vortex_street.

[2] S. L and V. W.F, “Progress in Low Temperature Physics,” 2009.

[3] C. J.E, “Encyclopedia of Vibration,” 2001.

[4] L. o. Congress, Liobrary of Congress, DC, 1940.

[5] D. W. Olson, S. F. Wolf and J. M. Hook, “The Tacoma Narrows Bridge Collapse,” 2015.

[6] C. Lodes and D. -. I. R. Sieber, “Numerische Berechnung der Karmanschen Wirbelstraße,” 2017.