YEDİTEPE UNIVERSITY
DEPARTMENT OF MECHANICAL ENGINEERING
ME333 FLUID MECHANICS LABORATORY

CFD Analysis of Unsteady Flow Around a 2D Circular Cylinder

Instructor: Ali Bahadır Olcay
Group Id: 4G1
Group Members:

İlker Kuzey Alkaşi	20210705076
Neşet Fatih Biçkin	20200705006
Artun Yağar	20200705055
Ömer Dolaş	20200705001

LAB DATE: 29/11/2023
DUE DATE: 24/12/2023

Table of Contents

1)Abstract

Computational Fluid Dynamics (CFD) is used in this study to investigate unsteady flow around a 2D circular cylinder. We navigate challenges in determining an optimal time step aligned with the Strouhal number using steady-state, incompressible, turbulent simulations and subsequent transient analysis. This study aims to unravel complexities in unsteady flow and contribute insights applicable to engineering practices by balancing computational efficiency and precision in vortex shedding representation.

2)Introduction

Understanding the complexities of fluid flow, particularly in unsteady conditions, is critical in a wide range of engineering applications. In this study, we use Computational Fluid Dynamics (CFD) to investigate the unsteady flow around a 2D circular cylinder. The research includes steady-state, incompressible, and turbulent simulations, as well as a transient analysis.

The theoretical framework guiding our research includes compressibility classifications expressed by the Mach number (Ma) and the Strouhal number (St), a dimensionless parameter representing the frequency of vortex shedding. The Strouhal number becomes critical in our transient analysis, where the challenge is determining an appropriate time step size that aligns with the system's inherent dynamics.

Our procedural framework starts with a meticulously prepared mesh file, laying the groundwork for accurate simulations. The first phase consists of a steady-state, incompressible, and turbulent simulation that provides information about the mean flow characteristics around the circular cylinder.

The transition from steady-state to transient analysis is a key focus in our study of unsteady flow around a 2D circular cylinder. A critical challenge is determining an optimal time step that is aligned with the Strouhal number. Iterative processes are essential for unraveling the dynamic complexities of fluid flow by balancing computational efficiency and accurate representation of vortex shedding patterns.

The transient analysis that follows builds on the steady-state results by introducing the complexities of unsteady phenomena. In this phase, calculating an optimal time step size that is aligned with the Strouhal number becomes a critical challenge. The process of iteration attempts to find a balance between computational efficiency and accurate representation of vortex shedding patterns.

Finally, this research aims to not only unravel the complexities of unsteady flow around a 2D circular cylinder, but also to address the difficulties associated with transitioning from steady-state to transient simulations. We hope to contribute valuable insights into the field of fluid dynamics and its applications in engineering by navigating these challenges.

3)Materials and Methods

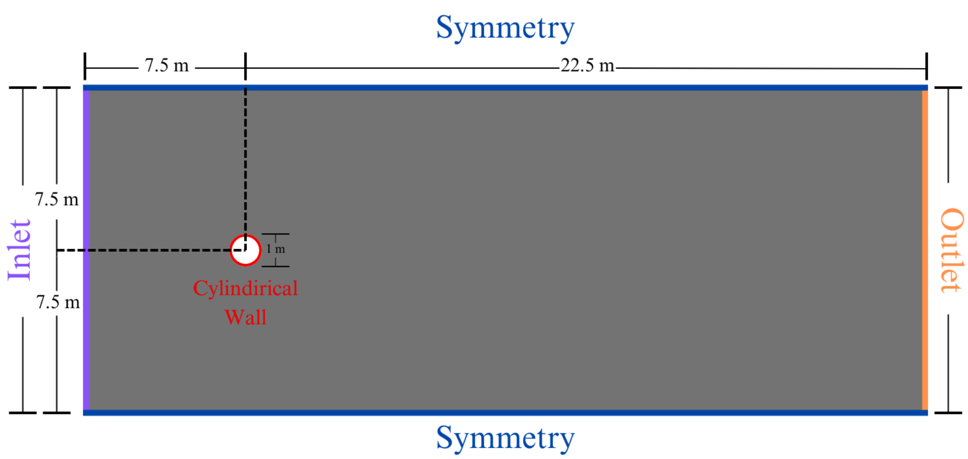

Figure 1: B.C. of numerical model

3.1 Steady flow around a circular cylinder

- The mesh file was given as ready by the instructor and imported to the setup section.
- The Reynolds number of the fluids is given as 80 (Re), so the flow is “Laminar”.
- Time is adjusted to be ‘steady’ in the general section and viscous is adjusted to be ‘laminar’ in the models section.
- Material is set in the following conditions:
  - Density of fluid is given as 1 kg/m^3 (ρ)
  - Viscosity of the fluid is given as 1 Pa.s (μ)
- Characteristic length of the cylindrical wall section is given as 1 m. (L)
- Boundary Conditions is set in the following conditions:
  - The inside cylindrical section is set as “wall”
  - Left wall is setted as “Inlet” with a V(velocity magnitude (m/s))
  - Right wall is setted as “Outlet” with a 0 gauge pressure value (Pa).
  - Top and bottom walls setted as “Symmetry”
- Velocity (V) was calculated as 80 m/s from equation 2. Where $\rho = L = \mu = 1$.
- Mach Number is calculated as 0.233 where the speed of the sound is 343 m/s. We can say that the flow is compressible

- Reference values computed from inlet where:
  - Area $[m^2] = 1$
  - Density $[kg/m^3] = 1$
  - Depth $[m] = 1$
  - Length $[m] = 1$
  - Viscosity $[Pa.s] = 1$
- Report definitions is created under the Force section in the following order:
  - Lift coefficient is calculated around the cylinder zone with a force vector [0,1,0].
  - Drag coefficient around the cylinder zone with a force vector [1,0,0].
- The solver settings were adjusted to use "Least Squares Cell Based" for the gradient, "Second Order" for pressure, and "Second Order Upwind" for momentum handling with the “Coupled” scheme of the Pressure-Velocity coupling.
- The CFD problem was initialized from the inlet using the “Hybrid”Initialization Method with 15 Number of Iterations.
- Then the calculation is runned.

3.2 Unsteady flow around a circular cylinder

- The steady flow solution was duplicated and a few changes were made given in the following:
- Time was changed to be ‘transient’ in the general section.
- Transient formulation was changed to be ‘second order implicement’ in the solution-methods section.
- Strouhal number corresponding to the Reynolds number was found from figure 2 and using equation 5 as 0.178.
- Frequency of the vortex is found from the equation 3 as 14.24 Hz with using the following values as:
  - Strouhal Number = 0.178
  - Characteristic length = 1m
  - Velocity = 80 m/s
- Vortex shedding period (T) is calculated from equation 4 as 0.07s.
- The CFD problem was initialized from the inlet using the “Standard ”Initialization Method.
- The solver settings were adjusted to use "Least Squares Cell Based" for the gradient, "Second Order" for pressure, and "Second Order Upwind" for momentum handling with the “Simple” scheme of the Pressure-Velocity coupling. And the transient formulation is set to “Second Order Implicit”
  - A. The calculation is runned with “User Specified” with followings:
    - a. Time Step Size: 0.007s (found from the T/10)
    - b. Number of time steps: 100
    - c. Max Iterations/Time Step: 30
    - d. Reporting and profile update interval: 1

B. Calculation is runned for number of time steps
   a. Time Step Size: 0.007s (found from the T/10)
   b. Number of time steps: 200
   c. Max Iterations/Time Step: 30
   d. Reporting and profile update interval: 1

Equations used for this simulation are given below:

$$Re = \frac{\rho V L}{\mu} \tag{1}$$

$$V = \frac{Re\mu}{\rho L} \tag{2}$$

$$S_t = \frac{fL}{V} \tag{3}$$

$$T = \frac{1}{f} \tag{4}$$

$$S_t = 0.25 - \frac{0.64}{\sqrt{Re}} \tag{5}$$

$$Ma = \frac{V}{a} \tag{6}$$

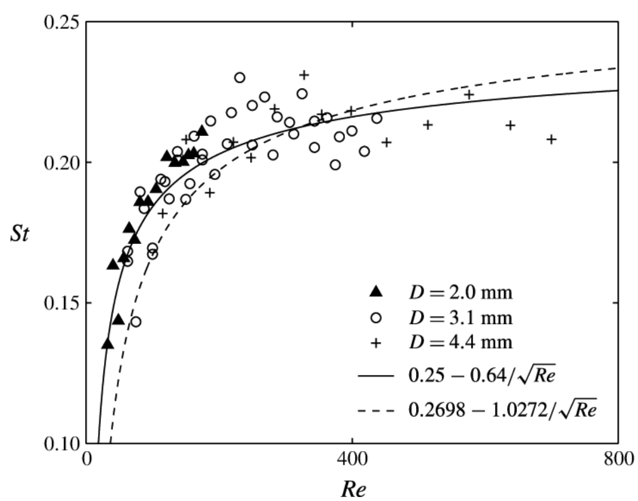

Figure 2: Reynolds Number vs Strouhal Number graph

4)Results

4.1 Steady State solution

Figure 3 shows the pressure change for steady flow of the fluid. The pressure increases in the region where the fluid hits the front surface of the cylinder and drops at the back surface of the cylinder.

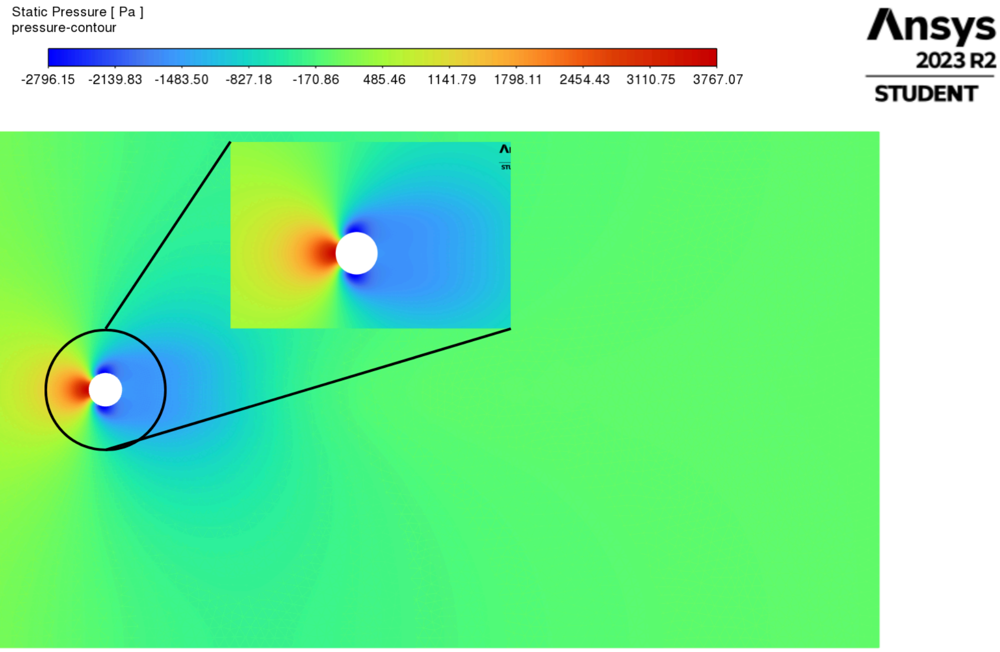

Figure 3: Static pressure contour of steady state flow

Figure 4 shows the velocity of the fluid in vectors. In this figure, the fluid velocity decreases before the fluid hits the cylinder and the fluid velocity decreases behind the cylinder. When viewed from this cross-section, the flow separation from the center of the cross-section upwards was uniform.

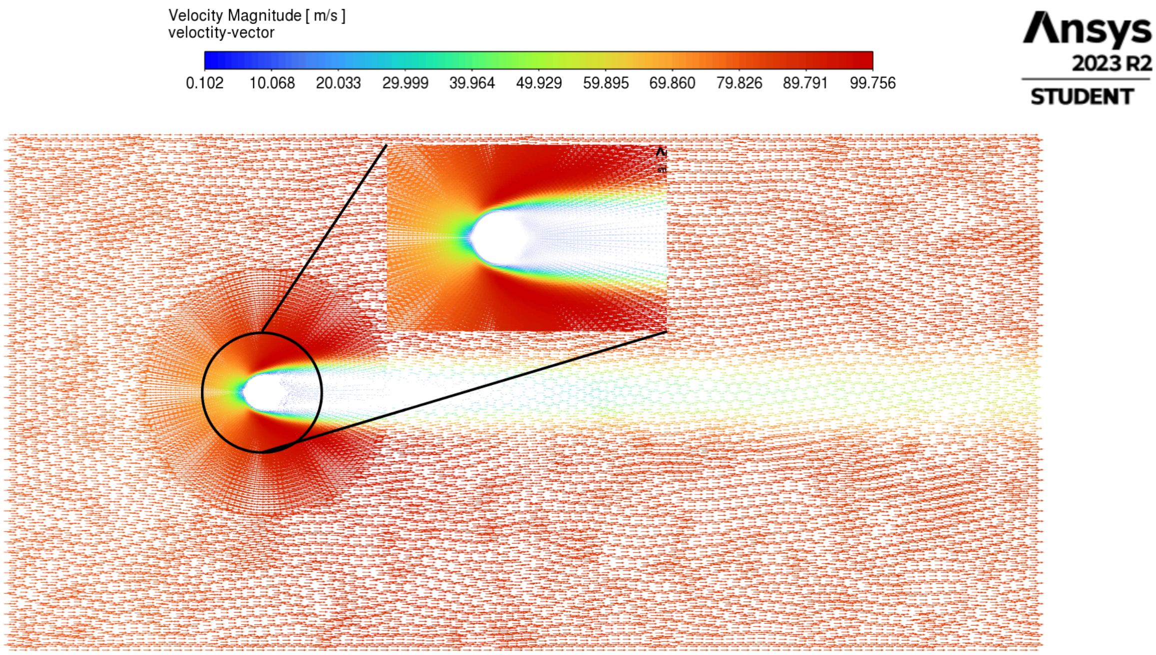

Figure 4: Velocity vector of steady state flow.

Figure 5 shows the variation of drag coefficient with the number of iterations for steady state flow. Here the drag coefficient converged at the 64th iteration.

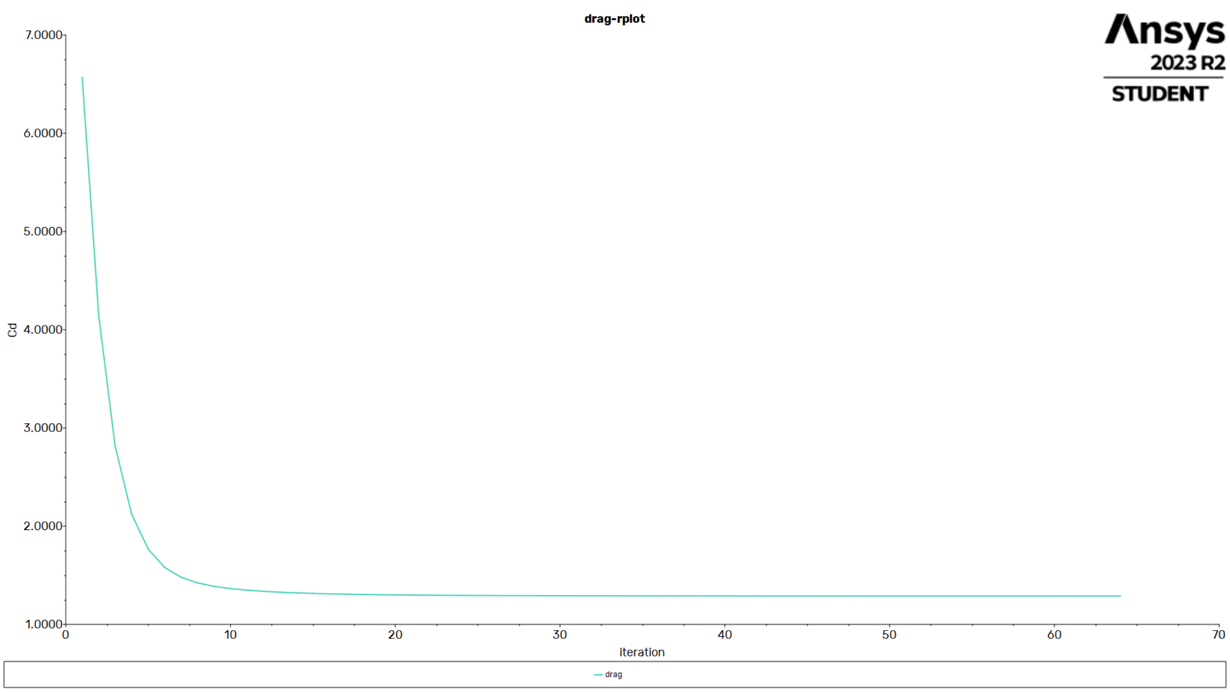

Figure 5: Drag Coefficient vs Iteration graph of steady state flow.

Figure 6 shows the variation of lift coefficient with the number of iterations for steady state flow. Here the lift coefficient converged at 64th iteration.

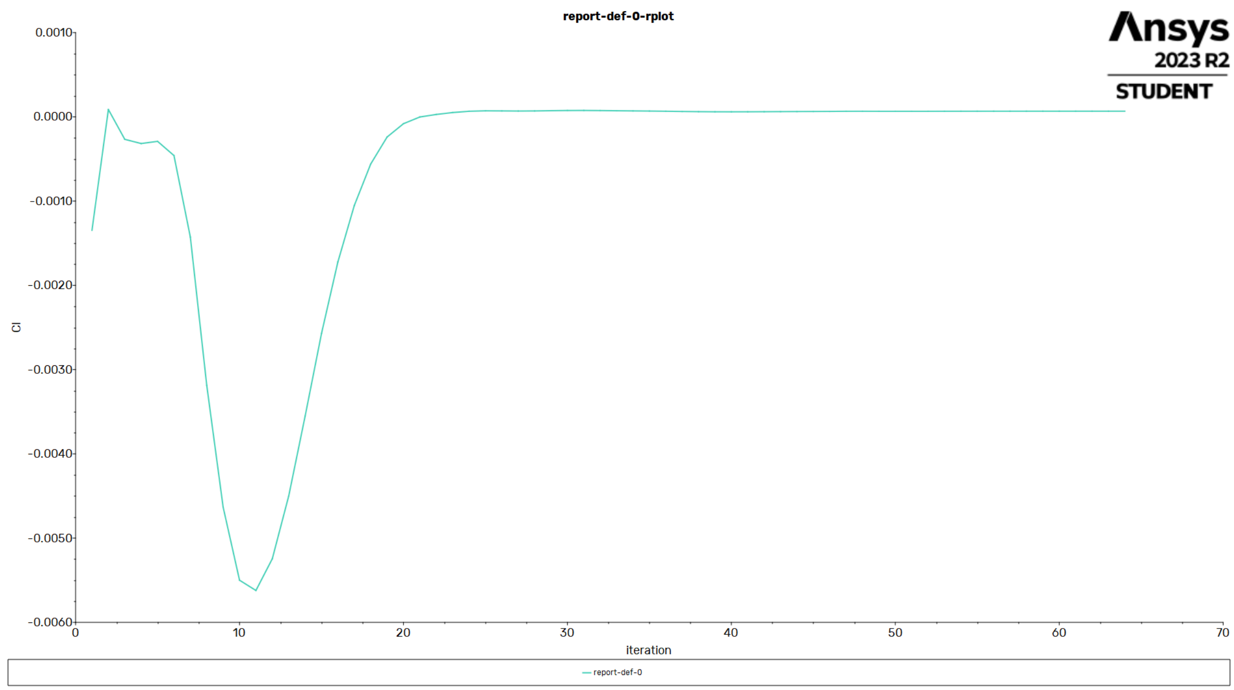

Figure 6: Lift Coefficient vs Iteration graph of steady state flow.

4.2 Transient Solution

Figures 7-8 shows the variation of lift coefficient with the number of iterations for steady state flow. Here the lift coefficient converged at the 1671'st iteration for A and converged at the 3969'th Iteration for B.

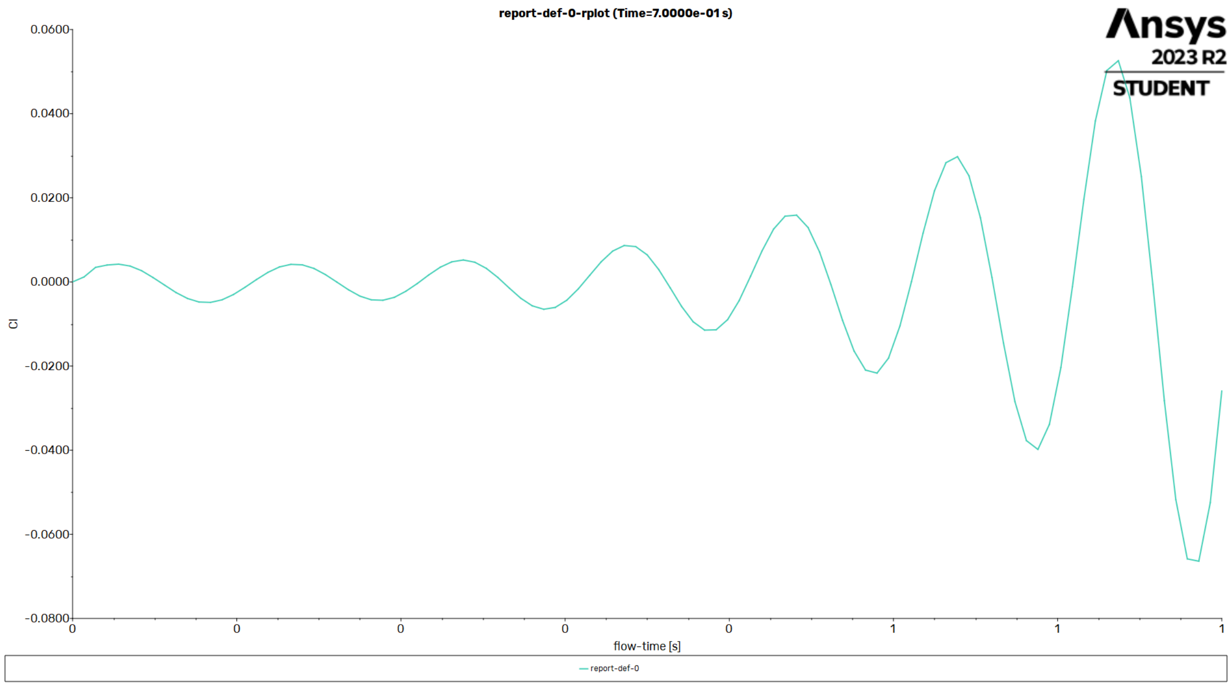

Figure 7: Lift Coefficient vs Iteration graph of transient flow (A).

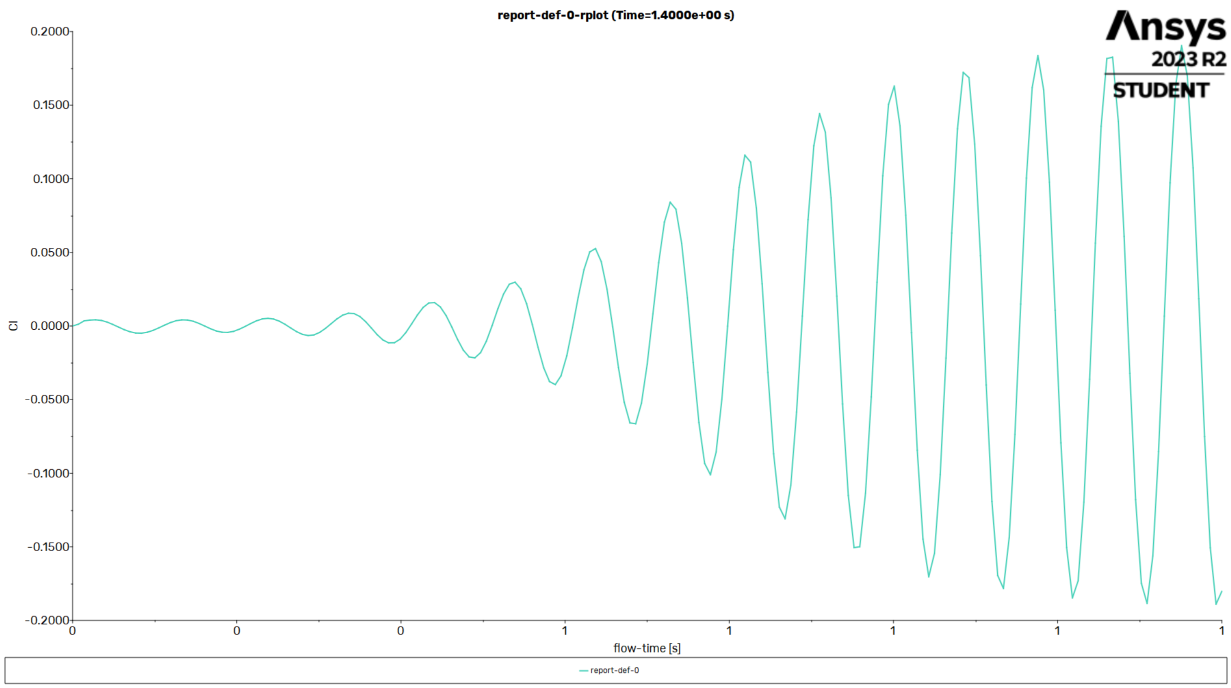

Figure 8: Lift Coefficient vs Iteration graph of transient flow (B).

Figures 9-10 shows the variation of drag coefficient with the number of iterations for transient flow. Here the drag coefficient converged at the 1671’st iteration for A and converged at the 3969’th Iteration for B.

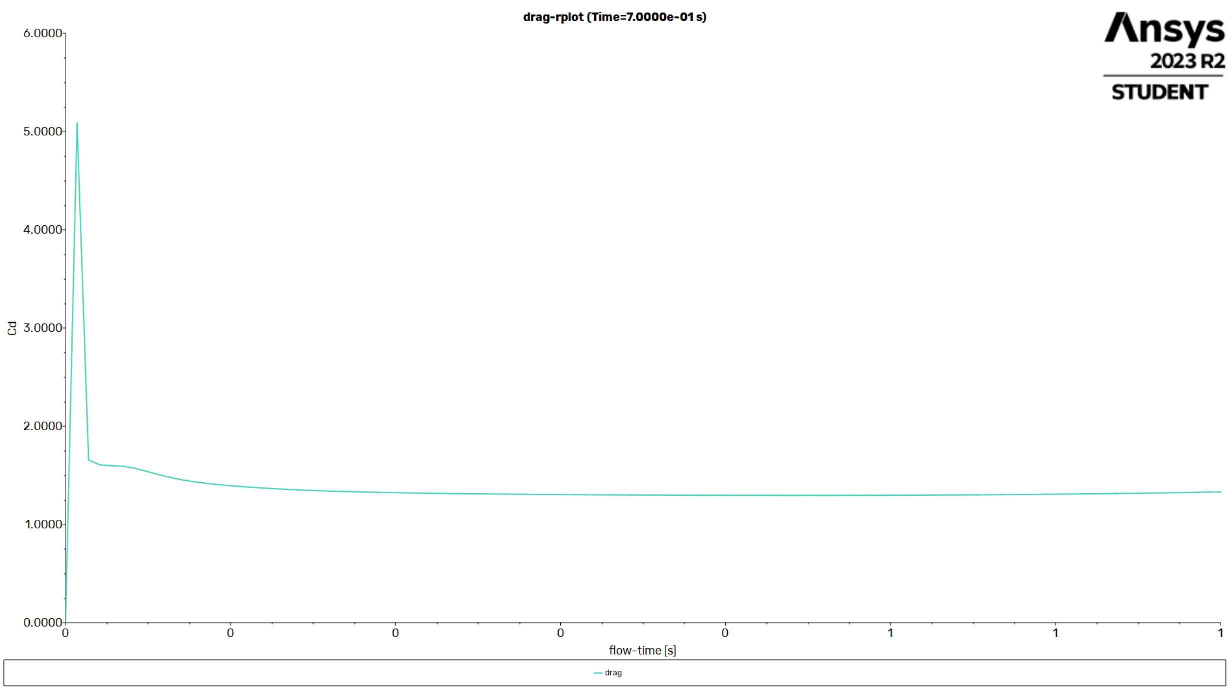

Figure 9: Drag Coefficient vs Iteration graph of transient flow (A)

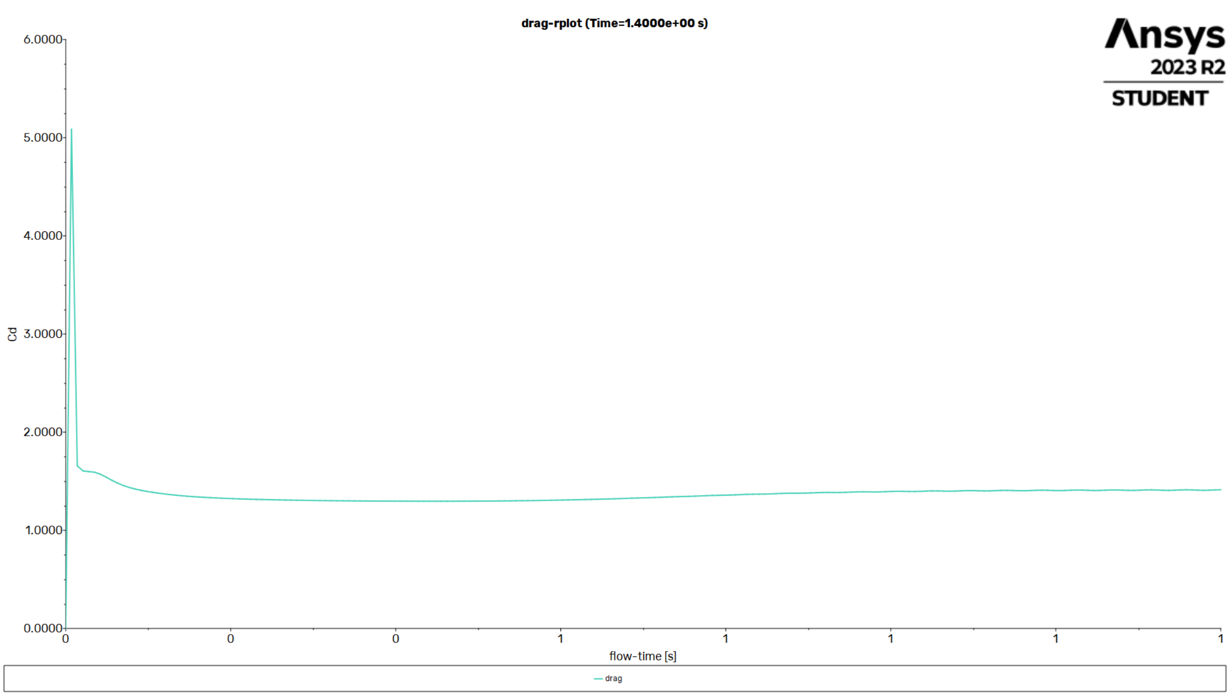

Figure 10: Drag Coefficient vs Iteration graph of transient flow (B)

100 Number of time steps (A) and 200 Number of time steps (B) is used and compared in Figures 11-12 for each figure..

Figure 11 shows the velocity of the fluid in vectors.

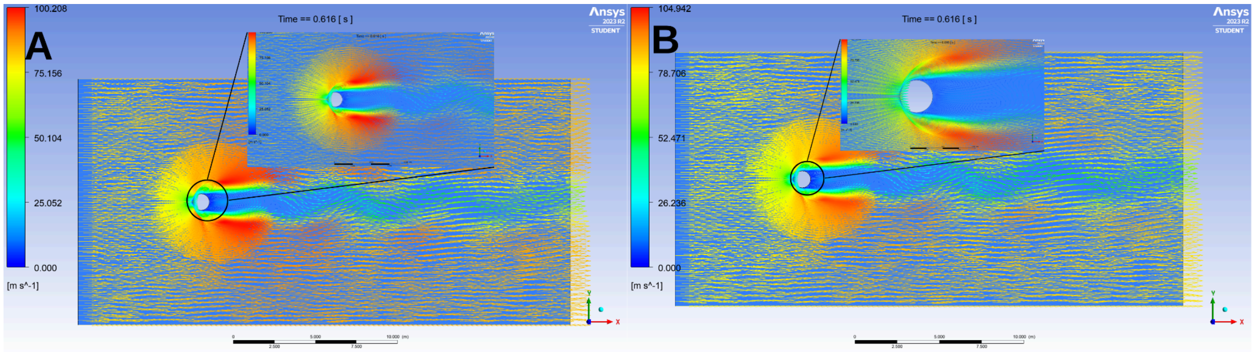

Figure 11: Velocity vector of transient flow for A and B.

Figure 12 shows the pressure change for transient flow of the fluid. The pressure increases in the region where the fluid hits the front surface of the cylinder and drops at the back surface of the cylinder.

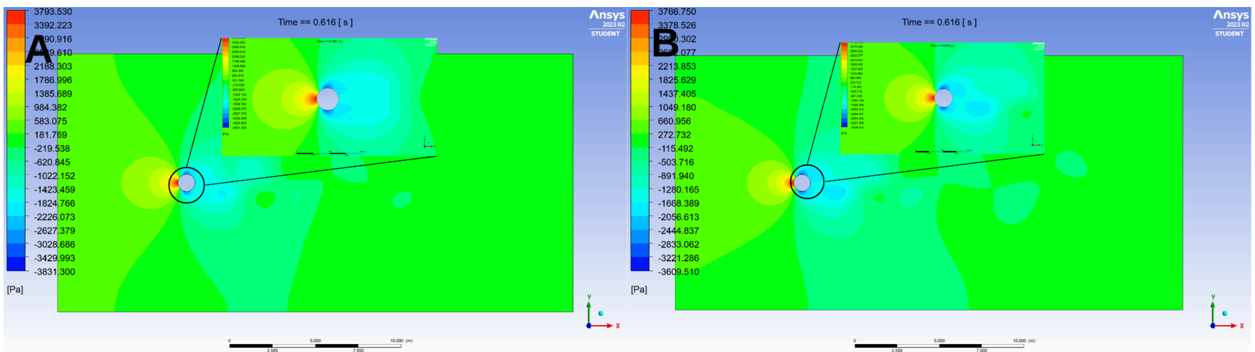

Figure 12: Static pressure contour of transient flow for A and B.

5)Discussion:

In this experiment, the flow around a cylinder was investigated using ANSYS Fluent, with a Reynolds number of 80 for a fluid in an environment with a density and viscosity of 1. The experiments were initially conducted under both steady flow and unsteady flow conditions, and later the unsteady flow solution was repeated using different time step values. The choice of a Reynolds number of 80 indicates that the experiment was conducted for laminar flow.
The obtained Strouhal number, a parameter determining the regularity and vibration frequency of the flow around the cylinder, was found to be 0.178, indicating that the flow behaved regularly and distinctly. The Mach number being less than 0.3 signifies that the fluid is incompressible, suggesting that significant pressure fluctuations caused by accelerations and decelerations are not prevalent during the flow. The interaction of the flow with the cylinder was analyzed through figures, observing high and low-pressure regions, as well as time-dependent pressure variations. The uniform distribution of flow velocity in the interaction with the cylinder in steady solution and vortex shedding in unsteady solution were visualized through animations.

Changes in drag and lift coefficients for both unsteady and steady flows were observed. In conclusion, this experiment provided in-depth insights into the complexity and interactions of the flow around the cylinder. These analyses are crucial for design improvements and the development of flow control strategies. Additionally, the unsteady flow analysis proved valuable in understanding the temporal aspects of the flow.