Research on the External Fluid Flow of a Round Cylinder with CFD

Mingxuan Sui

School of Mechanical and Mechatronic Engineering, University of Technology, Sydney, Australia

Mingxuan.sui-1@student.uts.edu.au

Abstract. Fluid mechanics is a frequently applied discipline, and technology advancements have substantially improved how engineers' approach and solve issues, and one of them is CFD. The purpose of the study was to evaluate how closely CFD adheres to accepted engineering principles. Selected CFD conducts computations to compare the magnitude and direction of drag coefficient changes under conditions of rising Reynolds number by simulating conditions in a wind tunnel to gather data on the object's reaction to fluid velocity. Within this range, the drag coefficient decreases with increasing Reynolds number. The simulated Reynolds number and drag coefficient do have a consistent relationship with the facts.

Keywords: Fluid mechanics, CFD, SolidWorks, Reynolds number, drag coefficient, flow pattern.

1. Introduction

Within the numerous subfields of engineering, fluid mechanics is a subject that is frequently applied. For instance, the engineering of pipelines, bridges, oil rigs, dams, and even drainage infrastructure can benefit from knowledge gained through the study of fluid flow around cylinders and other circular objects [1]. As a result, using CFD offers useful abilities and information that can be utilised for real-world engineering tasks.

There are hints that over time, technology development has significantly enhanced how engineers' approach and resolve problems by enabling them to access additional resources and tools to better solutions and designs. For instance, the first CFD simulation program was created in 1957 using the Navier-Stokes equations. Engineers had depended on 2D simulations and models up to now, which did not offer sufficient visual information to address issues successfully. Engineers and designers may now solve complicated issues more quickly and provide more imaginative products because of advancements in CFD technology.

Through this research, SolidWorks 2022 would be utilised to examine CFD in a cylinder. The simulation makes it possible to obtain velocity data as well as visualise the cylinder and the fluid flow around it. These facts allow for the calculation of the cylinder's drag coefficient. To examine their impacts on flow patterns and drag coefficients, the velocities employed in the simulations were derived using various Reynolds numbers.

The studies presented in this study examine computational fluid dynamics by simulating fluid flow in a cylinder using SolidWorks. Through the computer modelling of the situation, relevant fluid dynamics constants and principles, such flow patterns and drag coefficients, were examined and illustrated by entering different Reynolds numbers.

2. Methodology

2.1. Simulation setup

The simulation replicates the conditions of the research in the wind tunnel. In a wind tunnel, the object is put in the midst of a wind that is being produced at a certain speed, and the data collected includes the object's response to speed. A program called SolidWorks, which is generally utilised to produce 3D representations, was employed for this CFD experiment. Then the SolidWorks Flow

Simulation was utilised to simulate. Experiments will be done on the flow outside the cylinder because SolidWorks has real-world flow simulation capabilities [2].

SolidWorks and the given dimensions are used to display a 3D cylindrical structure that is fixed and situated in the enclosing computational domain. The cylinder is 50 mm in height and 60 mm in diameter. Draw a cylinder and position it such that its distance from the entrance and outflow is 900 (15D) and 3000 (50D) mm, respectively. The domain is 720 mm wide in addition, which places the cylinder 360 mm from the top and bottom exits. The simulation changed the wind speed in the program to correspond to the requirement to set the Reynolds number at three stages of 0.8, 100, and 50000. The simulation was able to determine the speed needed for the monitoring tests based on the aforementioned Reynolds number for air conditions at sea level and 15°C.

2.2. Essential quantities

2.2.1 Reynolds number

In fluid mechanics, the dimensionless Reynolds number is critical, and other dimensionless numbers are frequently utilised to give a standard for judging dynamic equality. Even though two flow modes may be in different fluids and at various flow rates, they are said to be dynamically comparable if they have the same value for the dimensionless number. The Reynolds number, which calculates the connection between inertial and viscous forces in a given flow scenario, is the ratio of these two forces in fluid mechanics [3]. The resistance to a fluid's change in velocity that creates motion is known as inertial force. Inertial forces are regarded to be dominating in turbulent fluids. It is a sticky force if not. There are numerous ways to translate this value into airspeed, though. The kinematic and static properties of the fluid, which are reflected in the Reynolds number, must be considered before selecting a numerical model.

This number is frequently utilised to forecast the flow's behaviour and direction. Additionally, the magnitude of the Reynolds number in pipe flow may be utilised to distinguish between various flow regimes, including turbulent, laminar, and transitional flow. For the flow distribution of cylindrical objects, the laminar flow limit is defined by an upper boundary of Reynolds number 2000, while the turbulent flow is represented by a Reynolds number greater than 4000. Reynolds number stands in for the transition flow between 2000 and 4000. The temperature and pressure of the fluid are two variables that would have an impact on flow measurement.

2.2.2 Drag coefficient

Aerodynamicists utilise the drag coefficient to simulate all of the intricate relationships between form, tilt, and flow conditions and aircraft drag [4]. A cylinder was utilised in the wind tunnel test. However, because of the symmetry of the flow field, the fluid flow only produces drag rather than lift. Drag is therefore primarily discussed and estimated in this research. As the total force is mentioned in this research, it represents the drag force only. Two factors contribute to the overall drag: the first is pressure drag, and the second is skin friction, also known as a frictional drag force. Pressure drag is what causes pressure variations near an object's surface. The viscous friction or adherence of fluids along an object's surface is what causes skin friction. The drag coefficient is defined as the drag to dynamic pressure force multiplied by area. The drag coefficient specifies how much drag force or fluid drag an object has in a fluid environment, such as water or air. It is a dimensionless number, and the geometry will determine the magnitude of the drag coefficient will change. The amount of drag force or fluid drag experienced as an item travel decreases with decreasing drag coefficient values.

2.3. Input data and relationship

For the Relationship between Reynolds number and fluid velocity, the definition of Reynolds number was introduced as follow:

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

Where: Re is the Reynolds number, ρ is the fluid density, u is the speed of the fluid, L is the diameter of the object, μ is the fluid dynamic viscosity.

Hence the velocity of fluid can be calculated by:

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

The drag of coefficient was calculated by the drag equation as follow:

$$C_D = \frac{2F}{\rho \, u^2 \, A} \tag{3}$$

Where: F is the drag force, ρ is the fluid density, u is the velocity of the fluid, CD is the drag coefficient, A is the reference area.

The input data are as follow: Re = 0.8, 100, and 50000, ρ = 1.225 kg/m^{-3}, L = 0.060 m; μ = 1.802*10^{-5} kg/m^{-1}s^{-1}; A = 0.06 x 0.05 = 0.003 m^2.

2.4. Procedure

Utilizing the SolidWorks modelling program, create a model of a cylinder with a 60 mm diameter and 50 mm length.

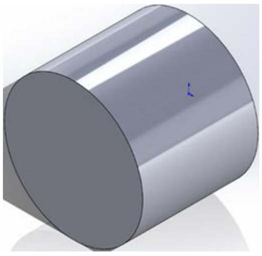

Figure 1. Circular Cylinder Model

SolidWorks Flow Simulation was then loaded by going to the tools tab, add-ins.

To get going and configure the default settings simultaneously, click the wizard tab in the upper left corner.

Add air under gases to the project fluid and choose the SI unit system as the analysis type. Set each Reynolds number’s corresponding value for Speed in X. (0.8, 100, and 50000). Ensure that every other setting in the wizard tabs stays the same. Press the finish button to finish.

Choose computational domain from the input data menu, then size the cylinder to be 0.9m in the negative x-direction, 3m in the positive x-direction, 0.36m in both negative and positive y-direction, and 0.025m in both negative and positive z-direction, respectively.

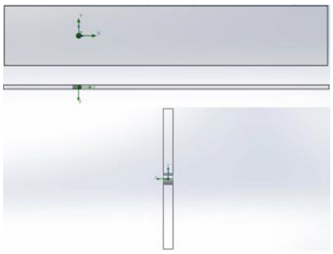

Figure 2. X-Y, X-Z, and Y-Z Plane

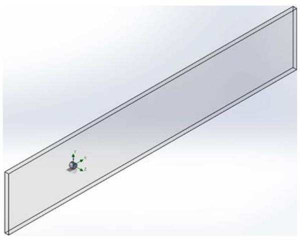

Figure 3. 3D View

Select the goal and equation goal from the input data tab, then write the formula for the drag coefficient. This will make it possible to determine the drag coefficient mathematically. Then, in the preceding goal section, supply the pertinent statistics pertaining to the drag coefficient.

In this data menu, click run, then select the new calculation option. Check to see if resolve is selected.

In the flow simulation window, choose the checked black and yellow flag tab to display the results when the computation has completed.

In the sidebar below the findings, expand the cut graph to see the velocity distribution.

Follow steps (V) through (IX) again for different Reynolds numbers and the associated velocities.

3. Result

3.1. Re = 0.8

When Reynolds number, Re = 0.8, the relative speed would be calculated as follow:

$$u = \frac{\mu\, Re}{\rho\, L} = \frac{1.802*10^{-5}(0.8)}{1.225*0.060} = 0.0001961\ m\ s^{-1} \tag{4}$$

According to the calculation, the inlet velocity in the x-direction may be applied at 0.0001961 m/s. The simulation's settings are displayed in Figure 4 below.

Parameter	Value
Parameter Definition	User Defined
Thermodynamic Parameters	
Parameters	Pressure, temperature
Pressure	101325 Pa
Temperature	288.15 K
Velocity Parameters	
Parameter	Velocity
Defined by	3D Vector
Velocity in X direction	0.0001961 m/s
Velocity in Y direction	0 m/s
Velocity in Z direction	0 m/s
Turbulence Parameters	

Figure 4. Parameters for Reynolds Number = 0.8

The drag coefficient values with the velocity distribution and flow pattern are displayed in figures 5-8 below.

Figure 5. Drag Coefficient for Reynolds Number = 0.8

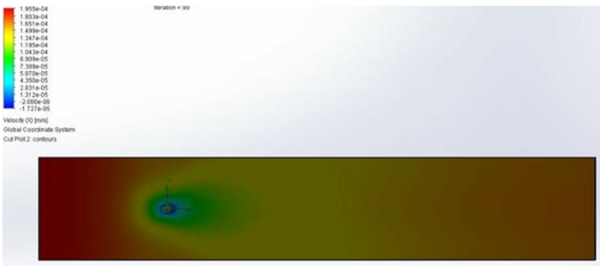

Figure 6. Velocity for Reynolds Number = 0.8

Figure 7. Flow Pattern for Reynolds Number = 0.8

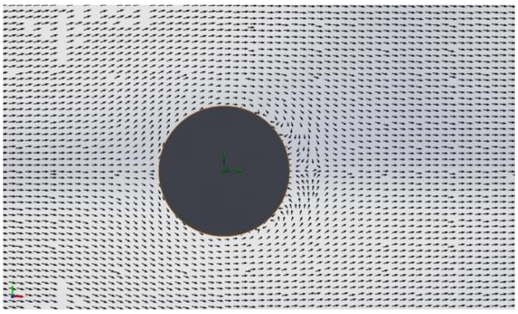

Figure 8. Close-up of Flow Pattern for Reynolds Number = 0.8

3.2. Re = 100

When Reynolds number, Re = 100, the relative speed would be calculated as follow:

$$u = \frac{\mu\, Re}{\rho\, L} = \frac{1.802*10^{-5}(100)}{1.225*0.060} = 0.0245170\ m\ s^{-1} \tag{5}$$

According to the calculation, the inlet velocity in the x-direction may be applied at 0.0245170 m/s. The simulation's settings are displayed in Figure 9 below.

Parameter	Value
Parameter Definition	User Defined
Thermodynamic Parameters	
Parameters	Pressure, temperature
Pressure	101325 Pa
Temperature	288.15 K
Velocity Parameters	
Parameter	Velocity
Defined by	3D Vector
Velocity in X direction	0.024517 m/s
Velocity in Y direction	0 m/s
Velocity in Z direction	0 m/s
Turbulence Parameters	

Figure 9. Parameters for Reynolds Number = 100

The drag coefficient values with the velocity distribution and flow pattern are displayed in figures 10-13 below.

Figure 10. Drag Coefficient for Reynolds Number = 100

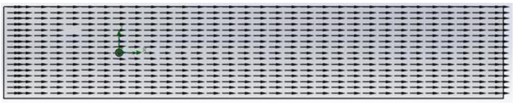

Figure 11. Velocity for Reynolds Number = 100

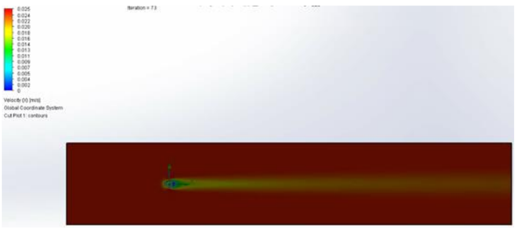

Figure 12. Flow Pattern for Reynolds Number = 100

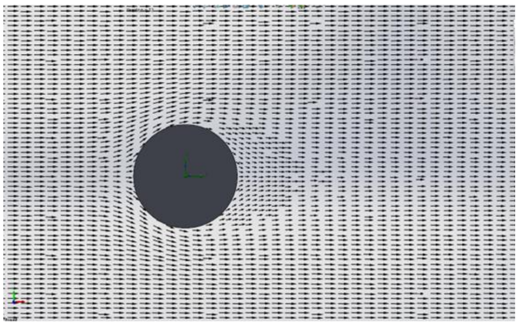

Figure 13. Close-up of Flow Pattern for Reynolds Number = 100

3.3. Re = 50000

When Reynolds number, Re = 100, the relative speed would be calculated as follow:

$$u = \frac{\mu\, Re}{\rho\, L} = 12.2585034\, m\, s^{-1} \tag{6}$$

According to the calculation, the inlet velocity in the x-direction may be applied at 12.2585034m/s. The simulation's settings are displayed in Figure 14 below.

Parameter	Value
Parameter Definition	User Defined
Thermodynamic Parameters	
Parameters	Pressure, temperature
Pressure	101325 Pa
Temperature	288.15 K
Velocity Parameters	
Parameter	Velocity
Defined by	3D Vector
Velocity in X direction	12.2585034 m/s
Velocity in Y direction	0 m/s
Velocity in Z direction	0 m/s
Turbulence Parameters	

Figure 14. Parameters for Reynolds Number = 50000

The drag coefficient values with the velocity distribution and flow pattern are displayed in figures 15-18 below.

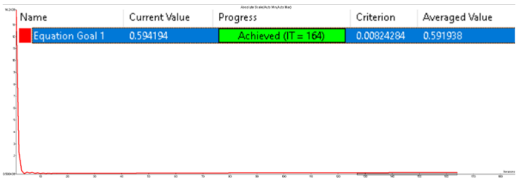

Figure 15. Drag Coefficient for Reynolds Number = 50000

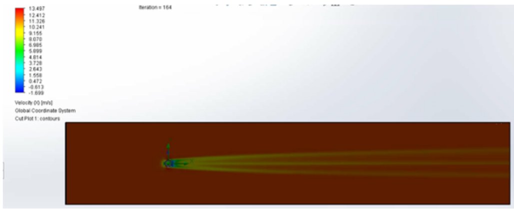

Figure 16. Velocity for Reynolds Number = 50000

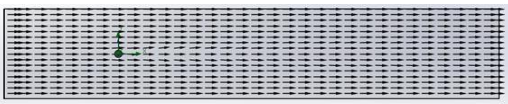

Figure 17. Flow Pattern for Reynolds Number = 50000

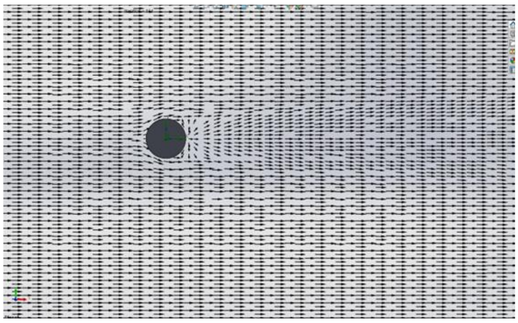

Figure 18. Close-up of Flow Pattern for Reynolds Number = 50000

4. Discussions

4.1. Reynolds number

The variation in the Reynolds number (Re) mimics the variation in airspeed in the wind tunnel while other parameters are set for each simulation. This is because the Reynolds number changes in the simulated data. The vast range of options for the Re value employed in this simulation provides a thorough grasp of what the value denotes for the kind of fluid flow. In the simulation, the Re value can be a variable parameter with a known value, but in a wind tunnel, the Re value is produced by measuring the surface pressure of the object together with the air density and viscosity of certain other variables. By assuming that the Re value varies because of air density and viscosity rather than pure air velocity, one may also utilise changing the Re value to study the impact of pressure or altitude on fluid flow. This suggests that atmospheric circumstances outside of a lab, including deep ocean depth or high altitude, may be easily examined using the simulated model.

The velocity graph displays a cylinder encircled by varying air velocity for a Re value of 100 or 0.0245170m/s of air velocity. It can be imagined as items flowing through a fluid and leaving a wake behind the cylinder. In these situations, CFD can simulate airflow at extremely low air velocities, which may not be achievable in a wind tunnel since the fans will have an effective minimum speed and any really low fan speed may generate airflow that is pulsing rather than at a continuous rate. With windspeed, the mean inter-pulse time decreases and the mean frequency of pulsation increases [5].

With a Re value of 50000 and an equivalent air velocity of 12.2585034m/s, the figures in section 3 more clearly demonstrates the typical pattern of airflow traveling around a moving item in the fluid, with separation points and wakes apparent.

It is evident that when the Reynolds number rises, the drag coefficient drops.

4.2. Drag coefficient

The impact of the Re number on the drag coefficient (Cd) is demonstrated via a graph. Between Re = 0.8 and Re = 100, values fluctuate significantly, while between 100 and 50000, values change less noticeably. This is to be expected since any drag produced by the airflow around the cylinder must be present. Additionally, it has been discovered that the drag force increases with an increase in air velocity, which was caused by the increased Reynolds number, for cylinders of a specific diameter [6], and its consequences also increase. Numbers might represent calculations that are extremely little or insignificant.

The Cd levels for Re = 100 and Re = 50000 were 0.975752 and 0.594194, respectively, and these values were in line with what was anticipated. A smooth cylinder with a Re value of 50000 should have a Cd just over 0.5, according to the graphic below.

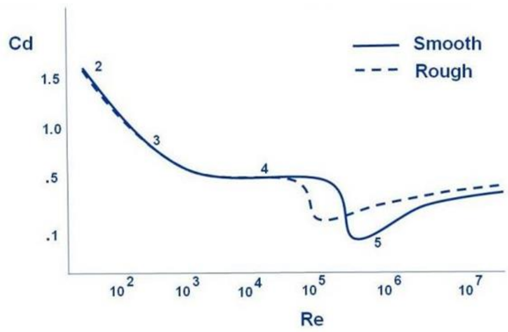

Figure 19. Drag coefficient vs Reynolds number for a smooth and rough surface cylinder [7]

The graph shows a distinct decrease in Re values from 100 to 100000. The change in Cd levels between these Re numbers are compatible with the following graph.

Any experiment requiring precise data should be conducted in a lab setting even if CFD has proven to be a good qualitative depiction of airflow since the predicted values can be inaccurate.

4.3. Flow Pattern

The uniformity of the airflow before it impacts the cylinder's surface, the air flow's initial velocity, and the cylinder's material characteristics, specifically its roughness, are some of the significant physical factors we take into account when studying the flow of fluid around a cylinder's surface.

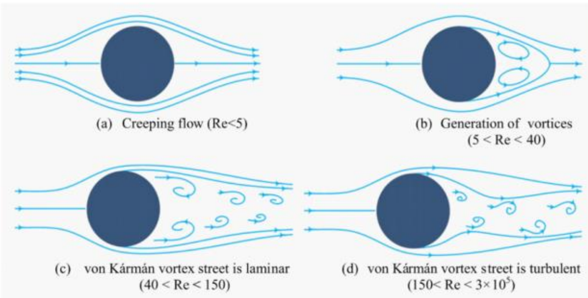

Figure 20. Regimes of flow past a smooth two-dimensional cylinder [8]

For Re of 0.8, 100, and 50000, a number of physical characteristics and flow behaviour of cylindrical objects are assumed. Flow paths like Models A and B are produced conceptually by low Reynolds numbers, such as Re of 0.8, which represent low flow rates and drag coefficients. The flow pattern of Model C correlates with a Reynolds number of 100, whereas Model D's flow behaviour coincides with a Reynolds number of 50000.

The pattern was seen to be symmetric along a central line with regard to the body cross-section in all flow channels because of the symmetric topology of the object shape. The obtained findings are in accordance with this theory, as shown in Figure 8, which shows a constant flow pattern around the cylinder.

4.4. Uncertainties

The variation in object smoothness between virtual and actual objects is one potential explanation for the discrepancy. While the software simulation will be modelled on an object that is absolutely smooth, the actual cylinder may actually have some surface irregularities that affect airflow even if it may be predicted to be smooth [9]. Numerical results may change as a result of these and other deviations between ideal and actual circumstances, such as the accuracy of laboratory measurement equipment [10].

5. Conclusion

In order to examine computational fluid dynamics, the experiments in this research employ SolidWorks to model fluid flow in a cylinder. By entering different Reynolds numbers, the computer modelling of the situation allowed us to examine and illustrate relevant fluid dynamics variables and concepts, such as drag coefficients and flow patterns. The significance of various numbers and other factors that might influence how fluids move relative to objects is discussed in computational fluid dynamics. Computational testing of flow simulations allows for a full understanding of fluid flow thanks to "real-time" simulations that emphasise the impact of the Reynolds number on the fluid flow as well as the values of the resisting force and the drag coefficient. With changing Reynolds numbers, changes would also occur on the flow of the fluid on the cylinder. Higher Reynolds numbers indicate turbulence with few vortices and considerably off-balanced streamlines, whereas lower Reynolds numbers indicate laminar flow type with tighter streamlines. For laminar flow, a Reynolds number of 0.8 suggests a greater drag coefficient, but for turbulent flow, a Reynolds number of 50000 indicates a lower drag coefficient.

References

[1] Dodgson, M., Gann, D., & Salter, A. The impact of modelling and simulation on engineering problem solving. Technology Analysis & Strategic Management. 2007. 19(4). 471–489.

[2] Rodriguez-Martin, M., Rodriguez-Gonzalvez, P., Sanchez-Patrocinio, A., & Sanchez, J.R. Short CFD Simulation Activities in the Context of Fluid-Mechanical Learning in a Multidisciplinary Student Body. Applied Sciences. 2019. 9(22). 4809.

[3] Chisholm, N.G., Legendre, D., Lauga, E., & Khair, A.S. A squirmer across Reynolds numbers. Journal of Fluid Mechanics. 2016. 796. 233-256.

[4] NASA, Glenn Research Centre. Website: www1.grc.nasa.gov.

[5] Martell, B.C., Strobel, L.R., & Guerra-Garcia, C. DC-driven positive streamer coronas in airflow. Plasma Sources Science and Technology. 2022. 31(8). 085014.

[6] Mallick, M., & Kumar, A. Study on Drag Coefficient for the Flow Past a Cylinder. International Journal of Civil Engineering Research. 2014. 5(4). 301-306.

[7] Yuce, M.I., & Kareem, D.A. A numerical analysis of fluid flow around circular and square cylinders. Journal-American Water Works Association. 2016. 108.

[8] Gao, W.J. Modelling of windage and churning losses in high speed rolling element bearings. Ph.D. thesis. INSA de Lyon. 2018.

[9] Wang, M.Y., Hashmi, S.A., Sun, Z.X., Guo, D.L., Vita, G., Yang, G.W., Hemida, H. Effect of surface roughness on the aerodynamics of a high-speed train subjected to crosswinds. Acta Mechanica Sinica. 2021. 37(7). 1090-1103.

[10] Gerhart, A.L., Hochstein, J.I., & Gerhart. P.M. Munson, Young and Okiishi's fundamentals of Fluid Mechanics: Si version. Wiley. Estats Units d'Amèrica. 2021.