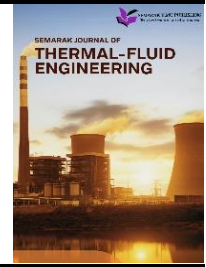




Semarak Journal of Thermal-Fluid Engineering

Journal homepage:
<https://semarakilmu.my/index.php/sjotfe/index>
ISSN: 3030-6639



External Flow Study Around a Hexagonal Cylinder

Chai Yu Pin^{1,*}, Ahmad Danish Syazwan Mazzri¹, Alicia Lai Jia Qi¹, Calvin Fredrick¹

¹ Department of Mechanical Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

ARTICLE INFO

Article history:

Received 15 July 2026

Received in revised form 28 August 2026

Accepted 8 September 2026

Available online 29 September 2026

Keywords:

Hexagonal cylinder; computational fluid dynamics (CFD); drag force; flow separation; external flow

ABSTRACT

The external flow around hexagonal cylinder is significant as this type of bluff body can generate high pressure drag and a well-defined wake region. The present work was performed by analyzing the flow phenomena around a 3d hexagonal cylinder by performing CFD simulation in the ANSYS Fluent software with a variation of inlet velocity. The steady state pressure-based solver and SST k- ω turbulence model were used. A grid independence test was conducted with coarse, medium and fine mesh before making the comparisons of the two main cases. The result indicates that there is only a small difference in drag coefficient (1.18%) between the medium and fine mesh and so the medium mesh was chosen to reduce the time whilst maintaining acceptable accuracy. The inlet velocity is compared between three velocities, namely 5 m/s, 10 m/s and 15 m/s. The drag coefficient stayed almost constant around 2.03, while the drag force increased from 0.6202 N to 5.6043 N. The pressure contour, velocity contour, and pathline result also depicts the front stagnation point, separated wake and the recirculation region behind the cylinder. This implies that the aerodynamic load is greatly increased with wind speed, despite only a small variation in drag coefficient.

1. Introduction

In the study of fluid flow around a bluff body, it is essential to use the engineering design approach. A bluff body is any object which is not streamline and causes the flow to separate from the surface. In practical use, the hexagonal column shape could be used to model some engineering components including column for buildings, bridge pier, tower leg, support member for a sign, etc. The flow around these parts is not simple, but these parts may appear simple. A high drag load was generated, and a large wake occurs when the airflow passes over the body and the sharp corners may disrupt the flow. Therefore, knowledge of the bluff-body flow is beneficial to design that is safer and more reliable [1]. Flow around cylinders has also been studied in general in the classical paper by Zdravkovich [2].

The flat faces and corners of the hexagonal cylinder make it easier to separate compared to the circular cylinder. The velocity in the air decreases in the vicinity of the stagnation point while the

* Corresponding author.

E-mail address: ad220146@student.uthm.edu.my

static pressure increases. The flow is then forced to flow around the top and bottom corners. The pressure after the cylinder is reduced due to the loss of energy and separation of the flow in the wake region. Pressure drag results from the difference of pressures over the front and back face of the object. The bluff body pressure drag is typically larger than the skin friction drag. The shape of the body and the separation location is important for external flow study, hence.

The wake behind a bluff body can be unstable at a certain range of Reynolds numbers and shedding of vortices can occur. This behaviour is often referred to as von Karman vortex street [3,4]. The shedding of vortices can cause varying forces on the body and also, can induce vibration issues for some structures [5]. While this assignment was run using a steady-state solver, the overall flow behaviour—including the stagnation area, side acceleration, separation and wake—can still be seen from the contour and pathline results. The steady state method is also simpler to handle with student level work, and it is adequate for comparing the drag coefficient and drag force as a function of inlet velocity.

A wind tunnel has been traditionally used to test this type of flow problem. Wind tunnel experiment is useful because it provides physical measurement; however, in the case of a wind tunnel, a model, test section, pressure tapping, and force balance are usually required, and correction for blockage effect is necessary. Additionally, it will require a longer time if many cases would need to be tested. As a result, it has become a standard approach to engineering study early in the design process, known as Computational Fluid Dynamics (CFD). Using CFD the flow field can be calculated numerically and the outcome displayed as drag coefficient, drag force, pressure contour, velocity contour, streamline and pathline [6]. It can cut down the number of physical tests, but the result is still dependent on the quality of geometry, mesh boundary condition, solver setting and post processing method.

In CFD, mesh quality is one of the more significant components since the flow variables are evaluated inside each of the small cells of the domain. For coarse meshes, the pressure and velocity gradients around the cylinder and in the wake region could not be resolved appropriately. The finer the mesh, the more time will be required for the simulation and the more difficult it will become to run the simulation, particularly in student software version. Thus, a Grid Independence test (GIT) become performed prior to the main comparison. GIT verifies that the result is not much sensitive to the mesh refinement. In this assignment, the drag coefficient was selected as the main value for GIT because it is directly related to the aerodynamic performance of the hexagonal cylinder.

There have been a few studies on the flow around a bluff body and the influence of sharp-edged bodies. Bhattacharyya *et al.*, [7] simulated the flow and heat transfer around a hexagonal cylinder numerically. Other research on square, rectangular, triangular, circular and polygonal cylinders have revealed that drag, wake size and vortex behaviour can also be influenced by Reynolds number, body shape, angle and blockage condition [8-19]. Ghosh *et al.*, [20] performed wind tunnel experiments for one hexagonal cylinder and reported on the pressure distribution and drag coefficient. Their result is helpful for engineering comparison, but the present assignment is not exactly as they did.

From the current available literature, it can be concluded that there is still a need for a simple CFD study to be carried out focusing exclusively on the external flow around one hexagonal cylinder at different inlet velocities. The case is not meant to be a perfect validation work. It is more to the point of understanding the pattern of flow and how the drag force on the body varies with the speed of the wind. For this assignment, a three-dimensional hexagonal cylinder was modelled in the modeler ANSYS Design Modeler with characteristic diameter (d) of 100 mm and a span length (L) of 200 mm. ANSYS Fluent simulation was performed under steady-state condition to simulate the flow. The main goal of this assignment is to simulate the external air flow around a hexagonal cylinder and compare the aerodynamic result at inlet velocity of 5 m/s, 10 m/s and 15 m/s. In addition, the study

incorporates GIT to choose an appropriate mesh prior to the last cases being analysed. Drag coefficient, drag force, the Reynolds number, pressure contour and velocity contour and pathline are used to compare the results. These outputs will explain the relation between wind speed, pressure difference, flow separation and drag load in a basic engineering approach.

2. Methodology

2.1 Geometry Development

The 3D geometry of the hexagonal cylinder and external fluid domain was created using ANSYS Design Modeler. The characteristic diameter of hexagonal cylinder (D) was fixed at 100 mm and span length (L) in Z-direction was 200 mm (Figure 1). To simulate the external air domain, a rectangular enclosure was created around the solid cylinder similar to a basic virtual wind tunnel. The outer dimension is significant as it is important that the flow around the cylinder is not affected by the outer boundary. From the set-up, the size of the total fluid domain is X-direction of 2000 mm, Y-direction of 1000 mm and Z-direction of 200 mm. The flow is parallel to the X-direction (right). The distance between the inlet and the center of the cylinder is $5D$, which is 500 mm. The distance from the cylinder to the outlet is longer and the wake can be adequately captured as $15D$ (1500 mm). Top clearance is $5D$ (500 mm) and the bottom clearance is $5D$ (500 mm). Then Boolean subtract was applied to remove the solid cylinder leaving only the fluid volume for the CFD calculation.

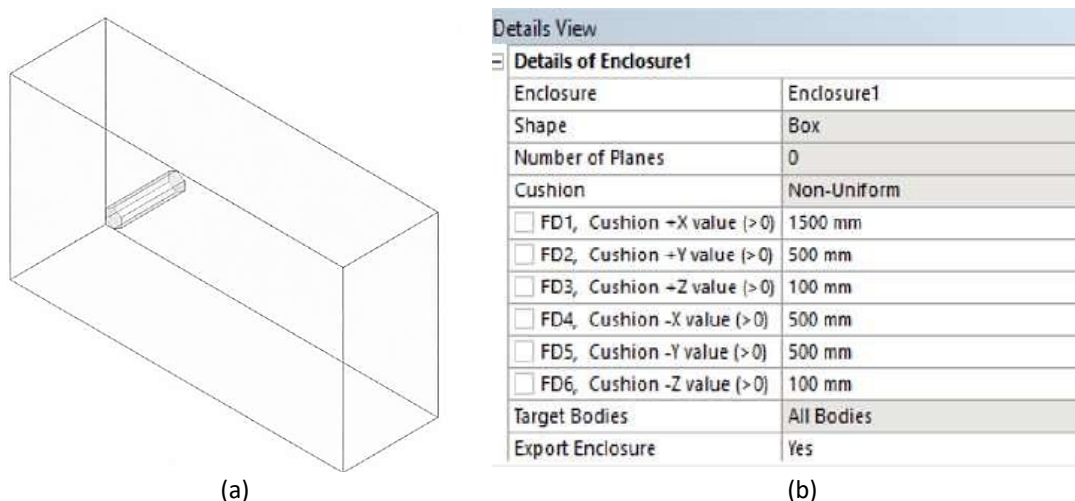


Fig. 1. Computational domain and enclosure setup in ANSYS Design Modeler (a) Fluid domain (b) Dimension of fluid domain

2.2 Mesh Generation

A structured and unstructured meshing was used for the fluid domain. To get a better representation of the near wall region and the flow separation, inflation layer was used near the hexagonal cylinder wall. To reduce the mesh error and to test the impact of mesh density on CFD results, a grid independence test (GIT) was performed. This step is crucial to not alter the solution unduly as the mesh is refined more in the future [8]. Three types of meshes were made; coarse, medium and fine meshes. The same boundary condition and inlet velocity (5 m/s) were applied to all three simulations (Figure 2). The drag coefficient (C_d) of each mesh was measured and compared.

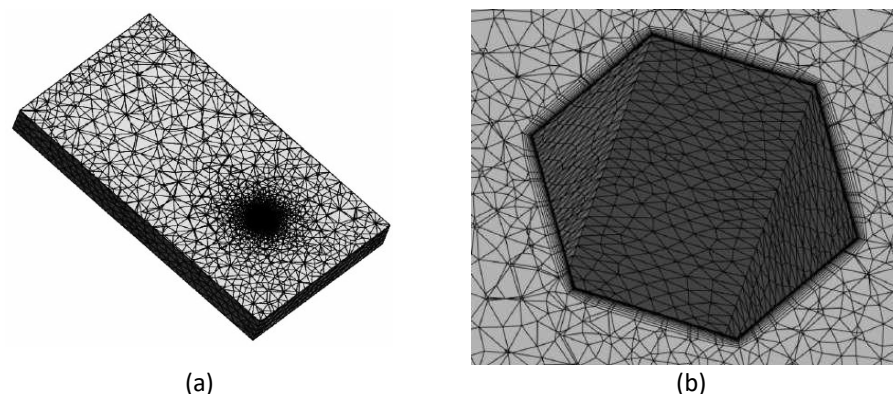


Fig. 2. Mesh detail (a) Section plane of mesh fluid domain with hexagonal cylinder (b) Mesh of hexagonal cylinder

2.3 Grid Independence Test (GIT) Explanation

Before the velocity comparison the Grid Independence Test (GIT) was conducted. This step was performed to determine if the CFD result is highly sensitive to mesh size or not. Three types of mesh were created in this assignment: coarse mesh, medium mesh and fine mesh. The same geometry, same material property, same inlet velocity and same solver setting were used for all three GIT cases. The only change made was the mesh size. This is significant because if multiple settings are altered simultaneously, it would not be possible to clearly evaluate the effect of the mesh.

The inlet velocity 5 m/s had been selected for the GIT as it is the first main case and it provides stable flow result for comparison. Once all the GIT cases were solved, the drag coefficient was noted from Fluent. The drag coefficient was selected as the parameter to check as it is the primary aerodynamic parameter in this assignment. The percentage difference between medium and fine mesh was then compared. If the difference is small, then the solution may be deemed relatively insensitive to mesh refinement.

The fine mesh provides a higher resolution but requires a greater number of elements and more calculation time. Thus, the final selected mesh should be a compromise between accuracy and cost of making calculations. For this assignment the mesh was accepted when the change in the drag coefficient with the changed mesh was less than approximately 2%. This is similar in concept to the grid refinement study, where the solution should not vary significantly with mesh refinement [21].

2.4 Boundary Conditions and Solver Settings

After the meshing process was finished, the model was imported into ANSYS Fluent. Pressure-based and steady-state were selected as the solver option. The Shear Stress Transport (SST $k-\omega$) turbulence model was chosen. This model was chosen due to its ability to provide more accurate predictions of near-wall flow and separation than the standard k -epsilon model for a variety of external flows [22]. The working fluid is the air, density is 1.225kg/m^3 , and dynamic viscosity is $1.7894 \times 10^{-5}\text{kg/ms}$. For boundary condition, the front face was considered as Velocity Inlet with the velocity of air flow of 5 m/s, 10 m/s and 15 m/s was applied. Pressure outlet (0 Pa gauge pressure) was set on the back of the domain. The surface of the hexagonal cylinder was fixed as a stationary Wall with no-slip condition.

The rectangular enclosure was assumed to be symmetrical with respect to its top, bottom and side faces. This was to minimize the additional wall friction at the outer domain boundary. Pressure-velocity coupling was performed using SIMPLE algorithm and the reference area was 0.02 m^2 for the

calculation of drag. The values of all the solvers and the boundary condition are summarized in Table 1.

Table 1

Summary of the solver setup and boundary conditions

Parameter	Setting
Solver type	Pressure-based, steady-state
Turbulence model	Shear stress transport (SST k-omega)
Fluid material	Air (density = 1.225 kg/m ³ , viscosity = 1.7894 × 10 ⁻⁵ kg/ms)
Inlet boundary	Velocity inlet (5 m/s, 10 m/s, 15 m/s)
Outlet boundary	Pressure outlet (0 Pa gauge pressure)
Cylinder surface	Wall (No-slip condition)
Enclosure boundary	Symmetry
Pressure-velocity coupling	SIMPLE scheme
Reference area (A)	0.02 m ²

2.5 Governing Equations

The basic fluid flow equations were used to simulate the external flow around the hexagonal cylinder, and the equations are conservation of mass and conservation of momentum. The working fluid is air at low speed, so the flow was considered as incompressible. The continuity equation is used to make sure that mass is conserved in the computational domain, as shown in Eq. (1). To calculate the velocity and pressure field, the Navier-Stokes equation was used to represent conservation of momentum in the fluid domain, as shown in Eq. (2).

$$\nabla \cdot V = 0 \quad (1)$$

$$\rho V(V \cdot \nabla) = -p + \mu \nabla^2 V \quad (2)$$

where V is velocity vector, ρ is air density, p is static pressure, and μ is dynamic viscosity.

2.6 Post-Processing and Analysis

After the solution had converged, the post-processing was carried out in ANSYS Fluent. The drag coefficient and drag force were taken from the force report by choosing the surface of the hexagonal cylinder. The drag direction was set to the direction of the flow and this is the X-direction. The reference area used for the drag coefficient was 0.02 m² which is derived from the length of span and the characteristic diameter of the cylinder used in this model.

The drag coefficient monitor was inspected for all cases of inlet velocity to assure the result had approached a steady state. The primary values (drag coefficient and drag force) were then noted in a table. The Reynolds number has also been computed to demonstrate the influence of flow condition on the increase in velocity. A reason for the drag force increasing more rapidly than the drag coefficient was given by the drag equation.

Pressure contour, velocity contour and pathlines were then used to prepare visualization of flow. The pressure contours were used to demonstrate the high-pressure contour at the front and the low-pressure contour at the rear of the cylinder. The velocity contour was used to visualize the flow stagnation area, flow acceleration around the corners and low velocity wake. The pathlines result was plotted to demonstrate the flow direction and the recirculation area behind the hexagonal

cylinder. These post processing results were compared with the 5 m/s, 10 m/s and 15 m/s to answer the objective of this assignment.

3. Results

3.1 Grid Independence Test

To make sure the simulation result was not strongly affected by mesh density, the GIT was conducted at $V = 5$ m/s. Three meshes were used, which are coarse (GIT 1), medium (GIT 2), and fine (GIT 3). The drag coefficient (C_d) was compared under the same boundary conditions, as listed in Table 2. As shown in Table 2, the drag coefficient for coarse mesh is 2.0782. When the mesh density is increased to medium, the value becomes 2.0253. Further refinement to fine mesh gives $C_d = 2.0014$. The percentage difference between medium and fine mesh is only about 1.18%. Since the difference is small and less than 2%, the solution can be considered mesh independent for this assignment. Therefore, medium mesh was selected for the detailed analysis because it gives a close result to fine mesh but uses less calculation time.

Table 2

Parameters and results for the grid independence test

Parameter	GIT 1 coarse	GIT 2 medium	GIT 3 fine
Element size (mm)	20	8	3
Number of nodes	9415	47989	297103
Number of elements	36360	168561	919244
Drag coefficient (C_d)	2.0782	2.0253	2.0014

3.2 Evidence Solution Convergence

It is important to demonstrate that the results of the numerical calculation have converged, or have reached a "stable solution", before comparing the results. The drag coefficient (C_d) monitor vs iteration is plotted in Figure 3 for the medium mesh simulation. From Figure 3, the drag coefficient is not stable in the first iterations, from 0 to 200. This is because the flow field around the hexagonal cylinder is still in the process of development. It takes about 300 iterations for the line to become nearly horizontal around 2.03. This flat line indicates that the steady-state solution is already steady. It does not imply perfection of the result, but only that it is acceptable to use to obtain the drag coefficient and to compare the three velocity cases.

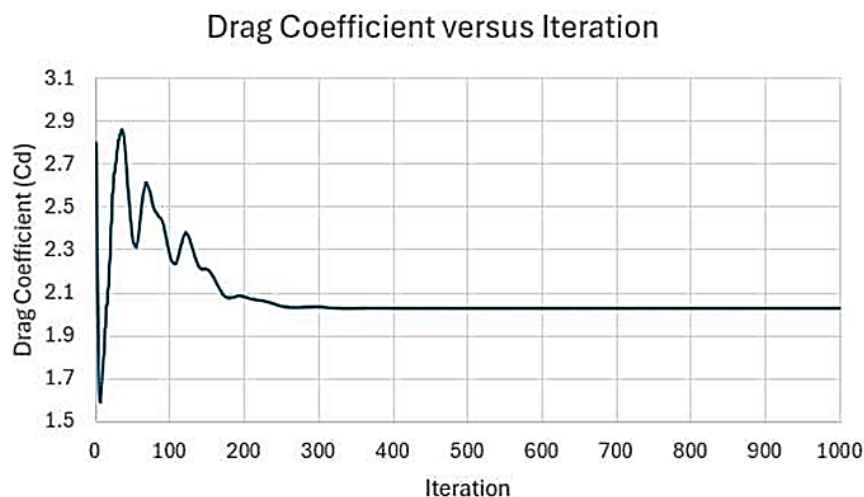


Fig. 3. Drag coefficient versus iteration showing evidence of convergence

3.3 Effect of Inlet Velocity on Drag Coefficient and Drag Force

For the selected medium mesh, the aerodynamic load on the hexagonal cylinder was simulated for three different wind velocities to understand the effect of the wind speed. For each case, the drag coefficient (C_d) and the drag force (F_d) were measured. The drag equation below defines the relationship between drag force and drag coefficient, as shown in Eq. (3).

$$F_d = \frac{1}{2} \rho v^2 C_d A \quad (3)$$

where ρ is air density, v is inlet velocity, C_d is drag coefficient, and A is the reference area of the hexagonal cylinder. The summary of simulated results for each velocity is shown in Table 3.

Table 3

Comparison of drag coefficient and drag force at different inlet velocities

Inlet velocity (m/s)	Drag coefficient (C_d)	Drag force (F_d), N
5	2.0253	0.6202
10	2.0303	2.4871
15	2.0333	5.6043

For the selected medium mesh, the aerodynamic load on the hexagonal cylinder was simulated for three different wind velocities to understand the effect of the wind speed. For each case, the drag coefficient (C_d) and the drag force (F_d) were measured. The drag equation (standard form) below defines the relationship between drag force and drag coefficient.

Based on Table 3, the drag coefficient only changes slightly from 2.0253 to 2.0333. This is reasonable as the bluff body is sharp cornered and hexagonal cylinder. The separation point is primarily located at the corner, and in this range of Reynolds numbers, the dimensionless drag coefficient will vary only slightly. The calculated Reynolds numbers are about 3.42×10^4 , 6.85×10^4 , and 1.03×10^5 for 5 m/s, 10 m/s, and 15 m/s respectively. The value of C_d at 2.03 is also near the high drag range obtained from published data for square and rectangular cylinders, but not quite the same [9,20]. Thus, it is not valid to use this comparison as precise validation.

But the real drag force (F_d) on the cylinder becomes quite large. When the inlet velocity is doubled from 5 m/s to 10 m/s, the drag force increases from 0.6202 N to 2.4871 N, which is almost four times higher. An increase in the velocity to 15 m/s results in a drag of 5.6043 N, nearly nine times that of velocity 5 m/s. This trend is like the trend of the drag equation, which is related to drag force being proportional to velocity squared.

The outcome of this is significant for the simple study of the structural aerodynamic. It is the ability of a hexagonal column or similar bluff-like structure to withstand much higher wind load at higher wind speeds even though the drag coefficient remains nearly constant. Therefore, in design, drag force proves to be a more practical tool for determining the actual load, while drag coefficient proves to be a more useful tool for comparing the aerodynamic behaviour between cases.

3.4 Velocity and Pressure Flow Fields

Flow visualization is important to understand the aerodynamic behaviour around a bluff body. From the CFD results, static pressure contour and velocity contour were obtained at stationary situation. Visual results were compared with each other when inlet velocity was 5 m/s, 10 m/s and 15 m/s to see the flow pattern.

The static pressure over the bluff body in Figure 4, confirms the fundamental pressure trend around a bluff body. The flow velocity decreases and the pressures are high at the front stagnation area. This can be seen as the red region on the windward face of the cylinder. The flow increases and the static pressure decreases at the side corners. The pressure behind the cylinder is low due to the wake region. As the velocity of the inlet increases, the pressure difference between the front and back side will increase (Figure 5). This is why the drag force is high at high speeds as demonstrated in Table 3.

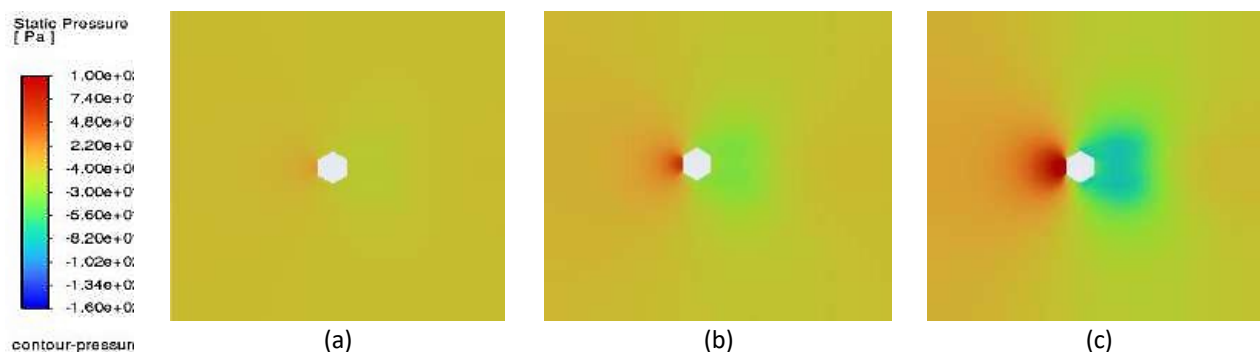


Fig. 4. Static pressure contour (a) 5 m/s (b) 10 m/s (c) 15 m/s

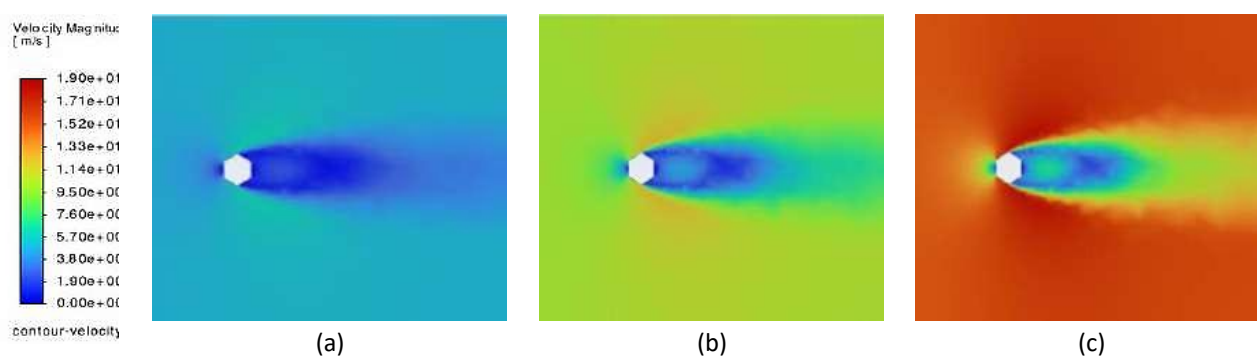


Fig. 5. Velocity contour (a) 5 m/s (b) 10 m/s (c) 15 m/s

3.5 Streamlines / Pathlines and Flow Separation

The pathlines were shown to further check on the wake formation and flow separation. This is a steady-state simulation, so that pathlines are used as streamlines to indicate the direction of flow around the cylinder. The pathlines corresponding to three velocities 5 m/s, 10 m/s and 15 m/s were taken and compared. The separation of the flow can be clearly observed in Figure 6. Hexagonal cylinder is a non-streamline bluff body having sharp corners and the boundary layer cannot remain attached after traversing the sharp corners of the bluff. The flow detaches from the solid wall and forms a shear layer. The pathlines behind the cylinder are curved and create a recirculation area within the wake.

The pathlines of the three velocities differ in colour due to the increased velocity of the free stream pathlines, which goes from blue at 5 m/s to green at 10 m/s to red/orange at 15 m/s. The wake behind the cylinder is very much the same in all three cases, however. This agrees with the drag coefficient result which has remained approximately 2.03. The flow pattern is comparable, but the strength of the flow and the actual drag force rise as a function of flow velocity.

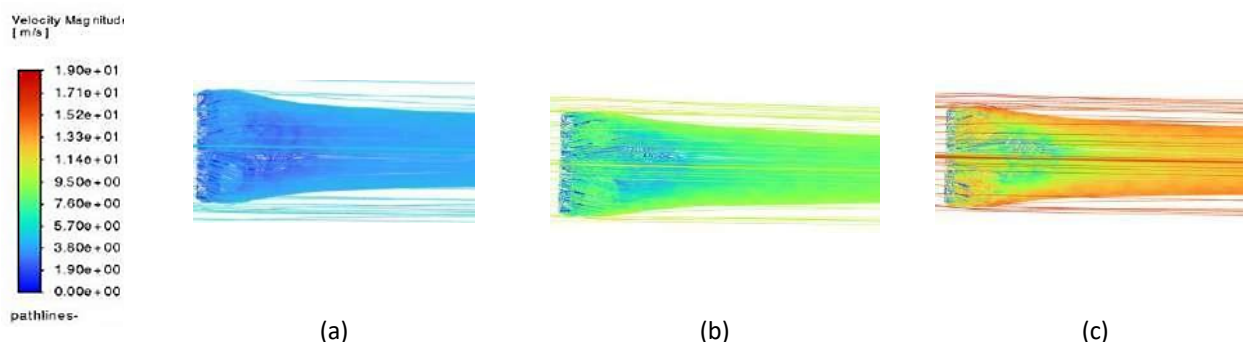


Fig. 6. Streamlines/pathlines around the hexagonal cylinder (a) 5 m/s (b) 10 m/s (c) 15 m/s

3.6 Discussion and Accuracy Analysis

The accuracy of the CFD result was verified through the grid independence test, convergence test, and comparison with an already published experimental study. As can be seen from the data of Table 1, the difference in the drag coefficient of medium and fine mesh is only 1.18%. It indicates that the outcome is not greatly influenced by the resolution of the mesh. Hence, the medium mesh was chosen as it provides a result that was close to the fine mesh with less computational time.

The drag coefficient monitor in Figure 3 also becomes almost flat after around 300 iterations. This indicates that the steady-state solver has converged to a steady-state solution. This does not say that the result of the simulation is completely accurate, though. Despite all of these, some error may occur numerically due to the quality of the mesh, the boundary conditions, the size of the domain, the turbulence model, and the turbulence solver setting. The SST $k-\omega$ turbulence model was chosen due to its appropriateness for near-wall flows and the prediction of flow separation.

The Reynolds number and wind speed during the experiment of Ghosh *et al.*, [20] is 4.22×10^4 and 13.5 m/s, respectively, which is used for comparison with the published value. The diameters of the cylindrical Hexagonal Prism are taken as 100 mm in the present CFD study and inlet velocities taken as 5 m/s, 10 m/s and 15 m/s respectively. The result published is used for the engineering comparison, and not as a validating result, due to the different size of the cylinder, the reference area, the orientation of the model, and the test set-up.

Ghosh *et al.*, [20] reported that the drag coefficient of a single hexagonal cylinder at zero angle of attack is about 0.95. In the present CFD result, however, the drag coefficient is approximately 2.03 for all the three inlet velocities. The difference can occur due to different cylinder diameter, reference area used in Fluent, geometry orientation, blockage effect, domain size, mesh setting, and turbulence model. Although the value of C_d is different, the trend of the present result is still reasonable as the drag coefficient will not change significantly with an increase in velocity, but the drag force will increase with a square of velocity.

This simulation has a few drawbacks, in particular, the solver was configured as steady state. In the real bluff-body flow the wake behind the cylinder is normally unsteady and shedding of vortices may be possible. If the vortex shedding frequency and lift fluctuation was to be investigated, a transient simulation will be better. For this assignment, however, the steady-state result is still good enough as we want to compare the drag coefficient, drag force, pressure contour, velocity contour and pathlines for various inlet velocities.

4. Conclusions

The external flow around a hexagonal cylinder was simulated successfully by using ANSYS Fluent under steady-state condition. The main objective of this assignment was achieved because the drag coefficient, drag force, pressure contour, velocity contour and pathlines were obtained and compared for the selected inlet velocities. The GIT was also completed before the final comparison. From the GIT result, the difference of drag coefficient between medium mesh and fine mesh is only 1.18%, so the medium mesh was selected because it gives a close result with less calculation time.

For the velocity comparison, the drag coefficient remains almost constant around 2.03 for 5 m/s, 10 m/s and 15 m/s. However, the drag force increases clearly from 0.6202 N at 5 m/s to 2.4871 N at 10 m/s and 5.6043 N at 15 m/s. This result answers the objective because it shows that the real wind load on the hexagonal cylinder becomes much higher when inlet velocity increases, even when the drag coefficient only changes slightly. The trend also follows the drag equation, where drag force is related to the square of velocity.

The pressure contour, velocity contour and pathlines results show the main flow behaviour around the hexagonal cylinder. High pressure occurs at the front stagnation region, the flow accelerates around the sharp corners, and a low velocity wake forms behind the cylinder. The pathlines also shows flow separation and recirculation behind the body. Based on these results, the CFD simulation is acceptable for this assignment because the mesh effect was checked, the solution had reached a stable condition, and the flow pattern agrees with the basic behaviour of bluff-body flow.

References

- [1] Williamson, C. H. "Vortex dynamics in the cylinder wake." (1996). https://doi.org/10.1007/978-94-011-0249-0_5
- [2] Zdravkovich, Momchilo M. "Flow around circular cylinders, vol. 1. Fundamentals." *Journal of Fluid Mechanics* 350 (1997): 377-378. <https://doi.org/10.1093/oso/9780198563969.001.0001>
- [3] Roshko, Anatol. *On the development of turbulent wakes from vortex streets*. No. NACA-TR-1191. 1954.
- [4] Khaledi, Hafez A., and Helge I. Andersson. "On vortex shedding from a hexagonal cylinder." *Physics Letters A* 375, no. 45 (2011): 4007-4021. <https://doi.org/10.1016/j.physleta.2011.09.046>
- [5] Siddiqui, Naseeb Ahmed, and Martin Agelin-Chaab. "Nature-inspired solutions to bluff body aerodynamic problems: A review." *Journal of Mechanical Engineering and Sciences* 15, no. 2 (2021): 8095-8140. <https://doi.org/10.15282/jmes.15.2.2021.13.0638>
- [6] Aguirre-López, Mario A., Filiberto Hueyotl-Zahuantitla, and Pedro Martínez-Vázquez. "Advances in numerical data visualization of flow around a square cylinder." *Symmetry* 16, no. 1 (2023): 32. <https://doi.org/10.3390/sym16010032>
- [7] Bhattacharyya, Suvanjan, Supratim Das, Anubhab Sarkar, Anindya Guin, and Ayan Mullick. "Numerical simulation of flow and heat transfer around hexagonal cylinder." *International Journal of Heat and Technology* 35, no. 2 (2017): 360-363. <https://doi.org/10.18280/ijht.350218>
- [8] Gera, B., Pavan K. Sharma, and R. K. Singh. "CFD analysis of 2D unsteady flow around a square cylinder." *International Journal of Applied Engineering Research, Dindigul* 1, no. 3 (2010): 602-610.
- [9] Sohankar, Ahmad, Christoffer Norberg, and Lars Davidson. "Low-Reynolds-number flow around a square cylinder at incidence: study of blockage, onset of vortex shedding and outlet boundary condition." *International Journal for Numerical Methods in Fluids* 26, no. 1 (1998): 39-56. [https://doi.org/10.1002/\(SICI\)1097-0363\(19980115\)26:1%3C39::AID-FLD623%3E3.0.CO;2-P](https://doi.org/10.1002/(SICI)1097-0363(19980115)26:1%3C39::AID-FLD623%3E3.0.CO;2-P)
- [10] Thabet, Senan, and Thabit H. Thabit. "CFD simulation of the air flow around a car model (Ahmed body)." *International Journal of Scientific and Research Publications* 8, no. 7 (2018): 517-525. <https://doi.org/10.29322/IJSRP.8.7.2018.p7979>
- [11] Okajima, Atsushi. "Strouhal numbers of rectangular cylinders." *Journal of Fluid mechanics* 123 (1982): 379-398. <https://doi.org/10.1017/S0022112082003115>
- [12] Eames, Ian, and Tristan Robinson. "Free-surface channel flow around a square cylinder." *Journal of Fluid Mechanics* 980 (2024): A16. <https://doi.org/10.1017/jfm.2023.964>

- [13] Mittal, Sanjay, and Bhaskar Kumar. "Flow past a rotating cylinder." *Journal of Fluid Mechanics* 476 (2003): 303-334. <https://doi.org/10.1017/S0022112002002938>
- [14] Francis, Vivek, Martyn Hann, Keri Collins, Aude Mulard, Saishuai Dai, and Deborah Greaves. "Comparing the performance of different geometries of fixed OWC-type wave energy devices using CFD simulations." *International Marine Energy Journal* 8, no. 5 (2026): 473-479. <https://doi.org/10.36688/imej.8.473-479>
- [15] Zhang, Xuqi, Lian Gan, and Yu Liu. "On the mechanism of dual tones in transitional flow around a heptagonal cylinder." *Journal of Fluid Mechanics* 1026 (2026): A37. <https://doi.org/10.1017/jfm.2025.11031>
- [16] Ozgoren, Muammer, Sercan Yagmur, Sercan Dogan, Muharrem H. Aksoy, Eyub Canli, and Ilker Goktepel. "Flow characteristics around a vortex flow meter via CFD and PIV methods." In *Proceedings of 10th Pacific Symposium on Flow Visualization and Image Processing, Naples Italy*. 2015.
- [17] Yagmur, Sercan, Sercan Dogan, Muharrem Hilmi Aksoy, Ilker Goktepel, and Muammer Ozgoren. "Comparison of flow characteristics around an equilateral triangular cylinder via PIV and Large Eddy Simulation methods." *Flow Measurement and Instrumentation* 55 (2017): 23-36. <https://doi.org/10.1016/j.flowmeasinst.2017.04.001>
- [18] Norberg, Christoffer. "Flow around rectangular cylinders: pressure forces and wake frequencies." *Journal of wind engineering and industrial aerodynamics* 49, no. 1-3 (1993): 187-196. [https://doi.org/10.1016/0167-6105\(93\)90014-F](https://doi.org/10.1016/0167-6105(93)90014-F)
- [19] Lam, K., and Y. F. Lin. "Effects of wavelength and amplitude of a wavy cylinder in cross-flow at low Reynolds numbers." *Journal of Fluid Mechanics* 620 (2009): 195-220. <https://doi.org/10.1017/S0022112008004217>
- [20] Ghosh, Kapil, Md Quamrul Islam, and Mohammad Ali. "Pressure distributions and forces on hexagonal cylinder." *Procedia Engineering* 105 (2015): 835-843. <https://doi.org/10.1016/j.proeng.2015.05.096>
- [21] Roache, Patrick J. "Perspective: a method for uniform reporting of grid refinement studies." *Journal of Fluids Engineering* 116, no. 3 (1994): 405-413. <https://doi.org/10.1115/1.2910291>
- [22] Khan, Sher Afghan, Musavir Bashir, Maughal Ahmed Ali Baig, and Fharukh Ahmed Ghasi Mehaboob Ali. "Comparing the effect of different turbulence models on the CFD predictions of NACA0018 airfoil aerodynamics." *CFD Letters* 12, no. 3 (2020): 1-10. <https://doi.org/10.37934/cfdl.12.3.110>