

CFD Analysis Of Drag Force On A Flat Plate

Nurul Hidayah Razlin¹, Ishkrizat Taib^{1*}

1. Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, MALAYSIA

CORRESPONDENCE

Ishkrizat Taib

iszat@uthm.edu.my

Received: 01 July 2026

Revised : 05 August 2026

Accepted : 25 August 2026

Published: 30 September 2026

This study presents a Computational Fluid Dynamics (CFD) analysis of airflow over a flat plate to investigate the effect of inlet velocity on flow characteristics and drag force. Numerical simulations were performed using ANSYS Fluent 2026 R1 with inlet velocities of 5 m/s, 10 m/s, and 15 m/s. A two-dimensional computational domain containing a 100 mm long and 5 mm thick flat plate was developed, and a grid independence test was conducted using mesh sizes of 10 mm, 8 mm, and 5 mm. The 5 mm mesh was selected for the subsequent simulations due to its more stable and accurate solution. The results show that increasing the inlet velocity significantly influences the velocity distribution, pressure distribution, wake development, Reynolds number, and drag force around the flat plate. The Reynolds number increased from 34,237 at 5 m/s to 102,711 at 15 m/s, while the drag force increased from 0.109 N to 0.970 N. Meanwhile, the drag coefficient remained approximately constant at 0.007 for all investigated cases. At the highest inlet velocity, stronger velocity and pressure gradients as well as a longer and more pronounced wake region were observed. Overall, the numerical results demonstrate that increasing inlet velocity leads to greater flow disturbance and a substantial increase in drag force, consistent with the theoretical relationship between drag force and the square of flow velocity.

Keywords: Computational Fluid Dynamics, Drag Force, Flat Plate, Inlet Velocity, Pressure Distribution, Reynolds Number, Grid Independence

1. INTRODUCTION

A numerical method called Computational Fluid Dynamics (CFD) is employed to investigate the flow behaviour of a fluid around an object. CFD allows engineers to visualise velocity distribution, pressure variation and flow patterns without conducting expensive experimental testing. The study of external flow over a flat plate is one of the most basic applications of CFD.

The formation of a boundary layer in flow over a flat plate is caused by friction between the fluid and the plate. As the fluid moves closer to the wall, the velocity will slowly increase from zero at the wall surface to the freestream velocity away from the wall. This effect is a factor that causes the drag force on the plate.

This study investigates the airflow around a flat plate using ANSYS Fluent 2026 R1 [1]. The objective is to analyse the effect of different inlet velocities on velocity distribution, pressure distribution, streamline behaviour and drag characteristics. In order to test the accuracy of the solution, a grid independence test was performed first, then the flow behaviour with the inlet velocity of 5 m/s, 10 m/s and 15 m/s was compared [2].

2. LITERATURE REVIEW & THEORY

The flat plate geometry is often used in fluid mechanics as a reference geometry for comparison because the flow can be analyzed with well-known solutions and is well understood. When fluid flows over a flat surface, the viscous effects form a boundary layer where the velocity changes from zero at the wall to the free stream velocity.

The Reynolds number is another important parameter governing external flow behaviour and is defined as:

$$Re = \frac{\rho VL}{\mu} \quad \text{Equation 1}$$

Where FD is for drag force, ρ is for fluid density, V is for free stream-velocity, Cd is for drag coefficient while A is for area.

It is the drag or shear force exerted on the plate due to fluid viscosity which produces the wall shear stress. The drag force can be expressed as:

$$F_D = \frac{1}{2} \rho V^2 C_d A \quad \text{Equation 2}$$

Where L is for characteristic length while μ is for dynamic viscosity. Reynolds number is the ratio of inertial forces to viscous forces and tells if the flow is laminar or turbulent.

The drag coefficient is a dimensionless measure of the resistance of an object to air flow. The lower the drag coefficient the better the aerodynamic performance. The numerical simulation is very sensitive to mesh quality. Too coarse a mesh can led to inaccurate results, too fine a mesh can led to a high computational expense. Therefore, a Grid Independence Test is necessary to ensure that the simulation results are independent of mesh density and accurately represent the physical behaviour of the flow and areas where there is significant drag [3-4].

3. METHODOLOGY

This simulation has been carried out with the help of the software ANSYS Fluent 2026 R1. A 2D model was developed in DesignModeler. The geometry was a flat plate in a rectangular air domain. The length of the plate was 100 mm, and the thickness was 5 mm. In order to allow the flow to develop to a sufficient extent before and after the plate, the surrounding air domain was set to 500 mm long by 300 mm high.

First the Add Frozen operation was used to create the air domain. The flat plate geometry was then created using Add Material operation. A Boolean subtraction operation was then used to subtract a plate from air domain and create the fluid domain for CFD simulation.

3.1. Geometry & Boundary Conditions

Table 1. Geometry Dimensions

Component		Dimension (mm)
Flat Plate	Length	100
	Thickness	5
Air Domain	Length	500
	Height	300

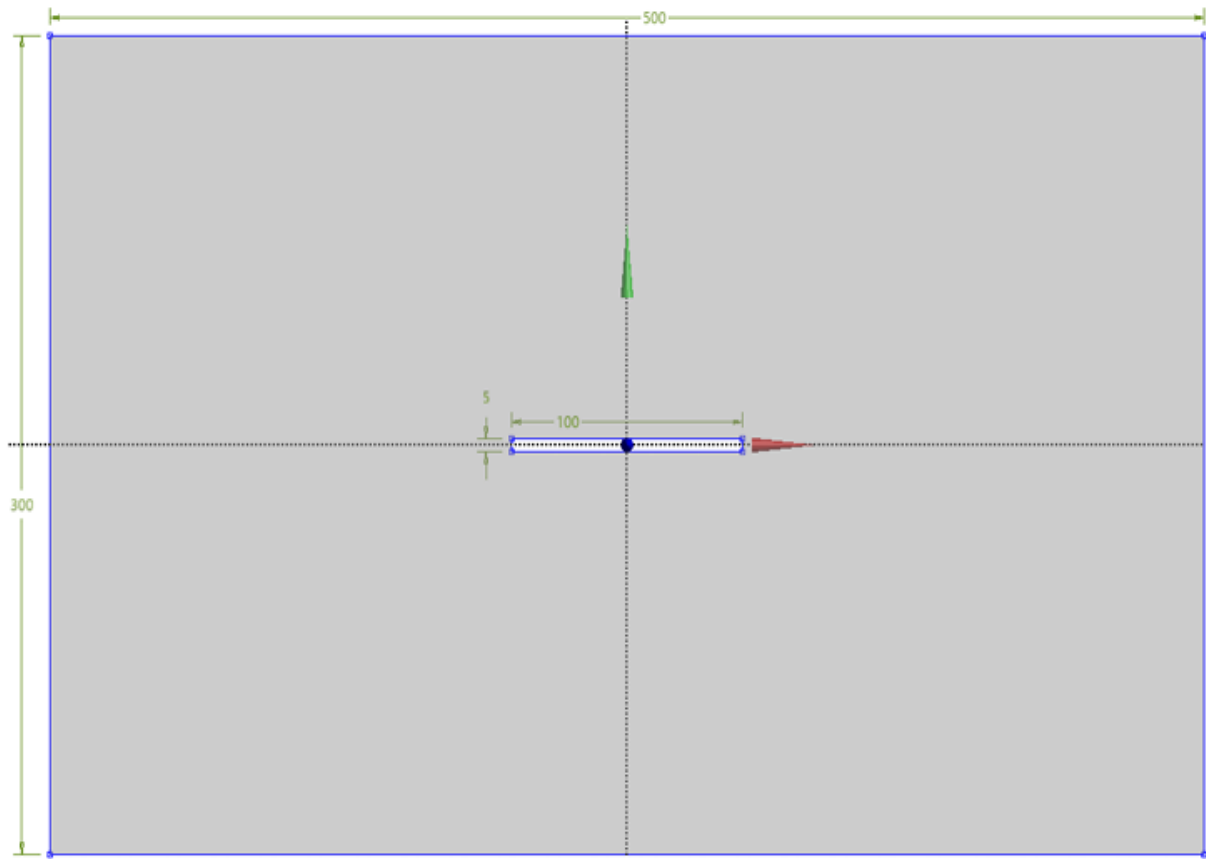


Figure 1. Flat Plate and Air Domain Geometry

Once the geometry was done, the boundary conditions were specified. The left side of the domain was designated as a velocity inlet and the right side as a pressure outlet. The flat plate surface was modelled as a stationary wall having a no slip boundary condition [5].

Symmetry boundaries were used for the upper and lower boundaries to reduce the wall interference effects. A structured quadrilateral mesh was created over the computational domain. To get a better prediction of the boundary layer development, additional mesh refinement was applied to the flat plate region [6], [7]. Three different mesh sizes were tested to determine the mesh sensitivity [8]. The Grid Independence Test consisted of mesh sizes of 10 mm, 8 mm and 5 mm [9].

3.2. Grid Independence Test

The variation between the medium and fine meshes was less than 0.3%, indicating mesh-independent results. The 5 mm mesh produced the highest accuracy and was selected for all velocity comparison simulations [10].

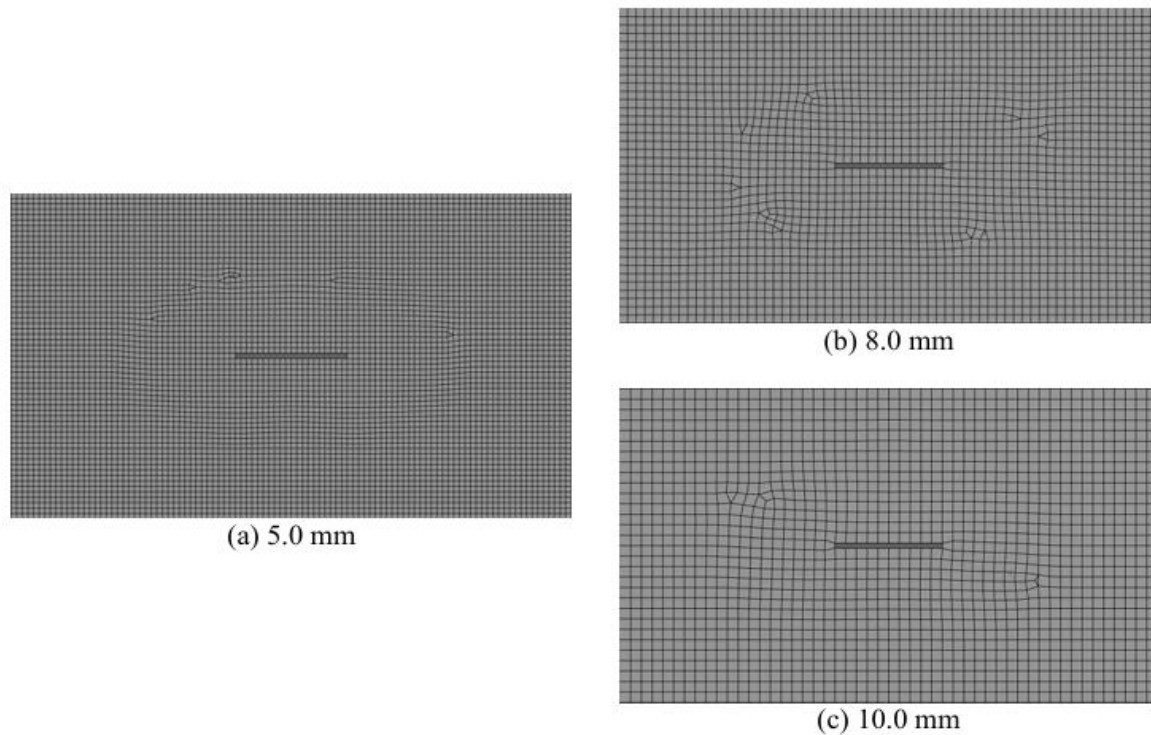


Figure 2. Computational Mesh Generated for Flat Plate Model at (a) 5.0 mm, (b) 8.0 mm, (c) 10.0 mm

Table 2. Grid Independence Test Results

Mesh Size (mm)	Number of Nodes	Number of Elements	Mesh Type	Maximum Velocity (m/s)
5	6193	6015	Fine	10.96
8	2516	2404	Medium	10.78
10	1608	1518	Coarse	10.47

According to Table 2, the maximum velocity was slightly higher as the mesh became finer. The maximum velocity obtained was 10.47 m/s with the use of 10 mm mesh, 10.78 m/s with 8 mm mesh and 10.96 m/s with 5 mm mesh respectively. The 8 mm and 5 mm mesh was not as big a difference compared to the 10 mm and 8 mm mesh. This indicates that the solution gets more stable when the mesh is refined. As such, the velocity difference was compared for the velocity in the 5 mm mesh to be closer to the flow around the flat plate [11].

The fluid was taken to be air with a density of 1.225 kg/m^3 and dynamic viscosity of $1.789 \times 10^{-5} \text{ kg/m}^3$. The upstream boundary condition was set as a velocity inlet and the downstream boundary condition was set as pressure outlet. Symmetry boundaries were used for the upper and lower boundaries to minimise wall effects [12].

4. RESULT & DISCUSSION

The result obtained from the CFD simulation around the flat plate for the three inlet velocities 5 m/s, 10 m/s and 15 m/s is being presented in this chapter. The comparison was done in terms of velocity contour, pressure contour, streamline pattern, Reynolds number and drag force. All velocity cases were performed with mesh 5 mm since it produced most stable result from the grid independence test.

4.1. Velocity Contour

The velocity contour obtained from the simulation at different inlet velocities are shown in Figure 3.

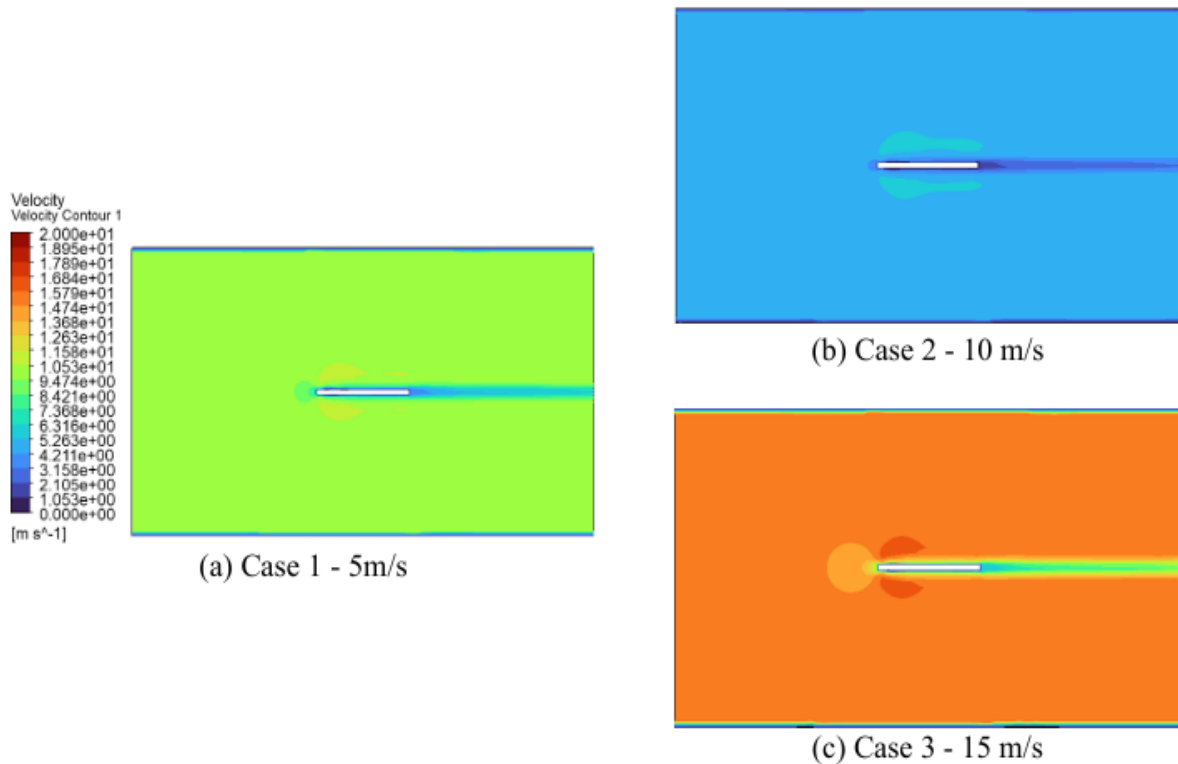


Figure 3. Velocity Contour Around Flat Plate Obtained at Inlet Velocities (a) 5 m/s, (b) 10 m/s, (c) 15 m/s

From the velocity contours the flow velocity was seen to rise with the increase of the inlet velocity. The air flow over the plate was less and the wake area behind the plate was smaller at a speed of 5 m/s. As the speed increased to 10m/s, the velocity gradient around the plate was more noticeable particularly in the vicinity of the leading edge and upper surface of the plate. When the flow speed was 15 m/s, the airflow has the largest acceleration around the plate and the wake area behind the plate is the longest and clearest.

At the flat plate surface there was the slowest velocity due to the no slip wall condition. This implies that at the wall the velocity of air was zero. Off the wall, the velocity rose slowly until it got to the free stream region. This behaviour is indicative of the formation of a boundary layer over the flat plate.

4.2. Pressure Contour

Pressure contours were used to analyse the pressure distribution around the flat plate for each inlet velocity.

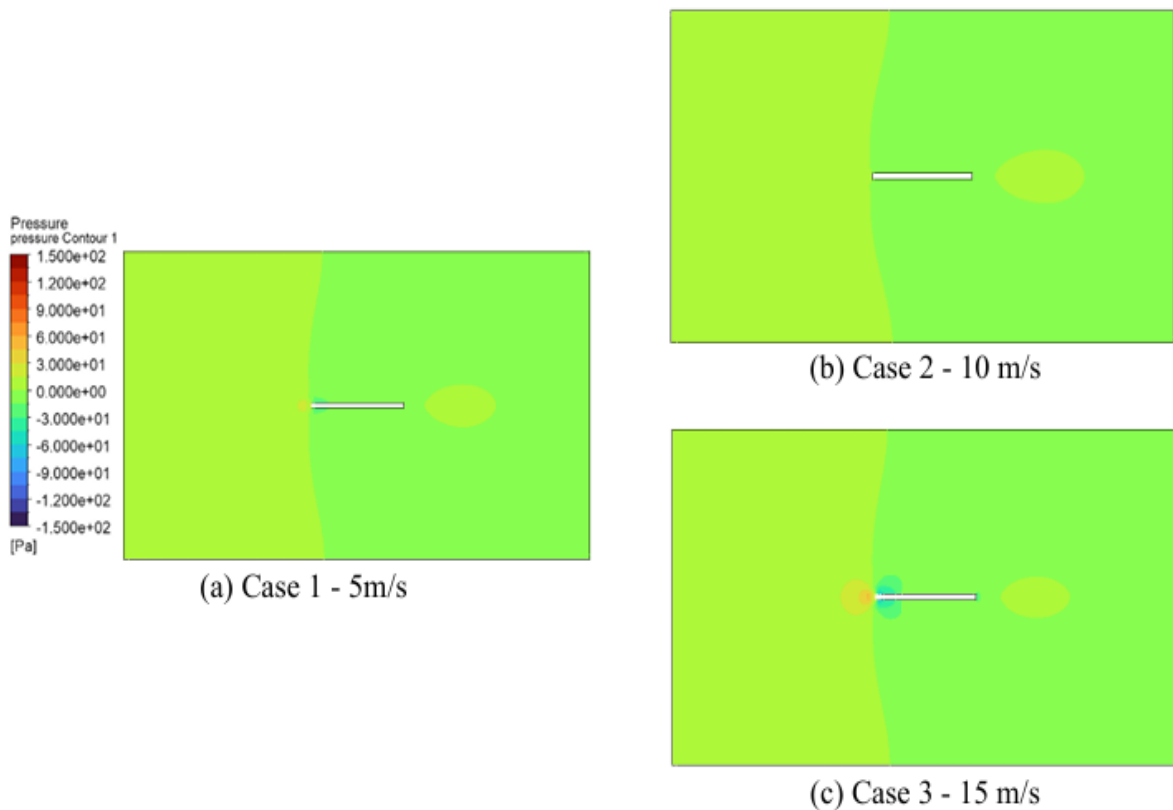


Figure 4. Pressure Contour Around Flat Plate Obtained at Inlet Velocities (a) 5 m/s, (b) 10 m/s, (c) 15 m/s

The pressure contour indicates that a high-pressure region was formed at the front surface of the flat plate where the incoming airflow directly impinges on the plate. The stagnation region is the area. This resulted in a lower pressure behind the plate due to flow separation and the formation of the wake region.

The pressure drops through the plate also increased as the velocity in the inlet to the plate increased. The change in pressure was small at 5 m/s. The pressure difference can be seen at 10 m/s. The pressure gradient was the greatest at 15 m/s because the air had more momentum on impact with the plate. This increased pressure difference led to an increased drag force.

4.3. Streamline Distribution

The flat plate was simulated using the streamlines to represent the airflow path.

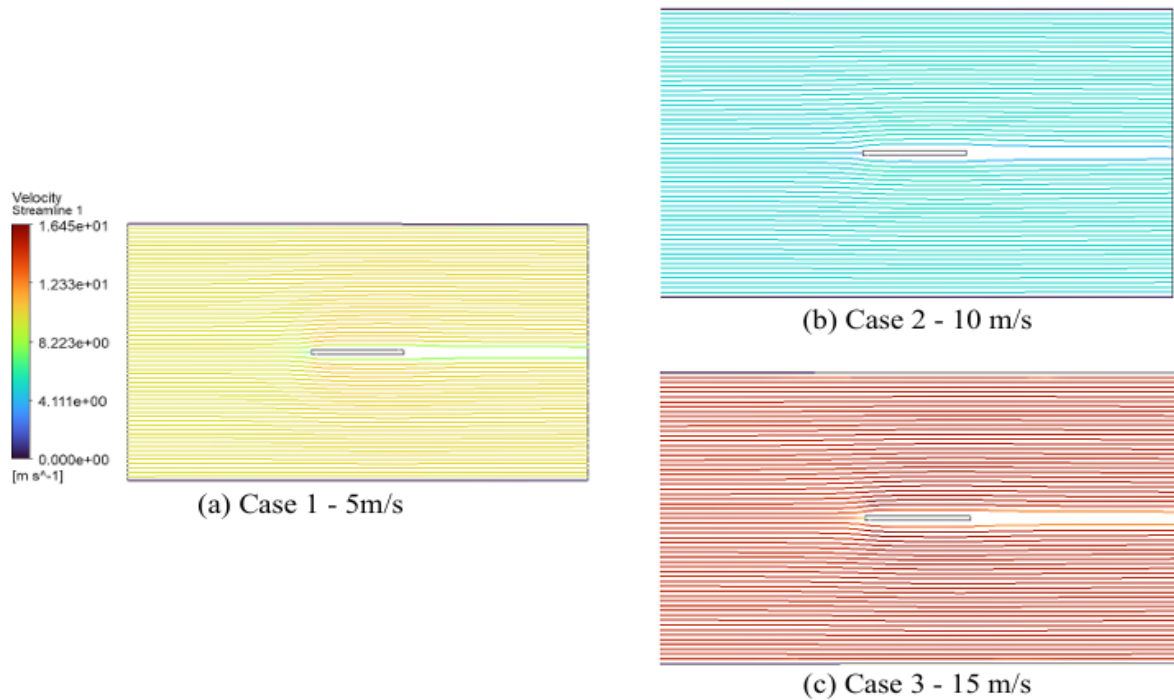


Figure 5. Streamline Contour Around Flat Plate Obtained at Inlet Velocities (a) 5 m/s, (b) 10 m/s, (c) 15 m/s

The streamline results indicate a smooth flow of air towards the flat plate, prior to obstruction. The flow detached from the flat plate and created a wake region downstream of the plate. This wake region is a sign of the momentum loss and the disturbance of the flow due to the presence of the plate.

The streamline pattern with a velocity of 5 m/s indicates a smaller and weaker wake. The wake was clearer at 10 m/s. The wake was longer and more intense at a velocity of 15 m/s, as there was more momentum to be carried in the wake. This is an indication that higher inlet velocity results in greater flow separation behind the flat plate.

4.4. Reynolds Number

The Reynolds number was determined for each inlet velocity to find the flow condition. **Equation 1** is used to identify the Reynolds numbers of inlet velocities.

Table 3. Reynolds Numbers Comparison

Case	Inlet Velocity (m/s)	Reynolds Number
1	5	34,237
2	10	68,474
3	15	102,711

The Reynolds number increased with the increase of inlet velocity from Table 3. This is because the Reynolds number is directly proportional to the velocity. It was found that the highest Reynolds number was provided at 15 m/s, where inertial forces were greater than in the lower velocity cases.

4.5. Drag Force

As the inlet velocity increased, the drag force also increased considerably. The drag force at 15 m/s was about 9 times more than the drag force at 5 m/s. This agrees with the drag force equation which states that drag force is proportional to velocity squared.

Table 4. Drag Force Comparison

Case	Inlet Velocity (m/s)	Drag Force (N)	Drag Coefficient, Cd
1	5	0.109	0.007
2	10	0.431	0.007
3	15	0.97	0.007

4.6. Chapter Summary

Table 5 summarises the main CFD results obtained for the three inlet velocity cases. The Reynolds number, velocity distribution, pressure variation, wake development and drag characteristics are compared to assess the effect of inlet velocity on the aerodynamic behaviour of the flat plate.

Table 5. Different Inlet Velocities Summary

Parameter	5 m/s	10 m/s	15 m/s
Max Velocity (m/s)	5.532	11.073	16.628
Min Pressure (Pa)	-10.59	-41.65	-92.859
Max Pressure (Pa)	18.046	71.381	159.918
Drag Force (N)	0.109	0.431	0.97
Drag Coefficient, Cd	0.007	0.007	0.007
Reynolds Number	34,237	68,474	102,711
Velocity Contour Gradient	Small	Moderate	Strong
Pressure Distribution	Low Pressure Differences	Clear Pressure Difference	Highest Pressure Differences
Wake Region	Small Wake	Clear Wake	Strong & Longer Wake
Drag Force	Lowest	Medium	Highest

From Table 5, it can be seen that when the inlet velocity increased from 5 m/s to 15 m/s, the Reynolds number, the pressure difference, the wake formation and the drag force increased. The 15 m/s case resulted in the highest aerodynamic effect, and the lowest aerodynamic effect and drag force occurred in the 5 m/s case. This proves that the external flow behaviour around the flat plate is much influenced by the inlet velocity.

5. CONCLUSIONS

ANSYS Fluent 2026 R1 was successfully employed to simulate the air flow around the flat plate with inlet air velocity of 5 m/s, 10 m/s and 15 m/s. For the first time the grid independence test was performed with the mesh size of 10 mm, 8 mm and 5 mm, the latter one was chosen because of higher accuracy and stable solution.

The results indicated that the aerodynamic characteristics around the flat plate can be significantly changed with the increase of inlet velocity. The velocity gradient increased with velocity, as did the pressure difference, wake formation and drag. At an increased wind speed of 15 m/s, the Reynolds number rose to 102711, reflecting stronger inertial effects on the flow field as compared to 5 m/s.

The 15 m/s case had the maximum drag force in the cases investigated and caused the highest flow disturbance. In overall, the CFD results were found to be in agreement with the theory of fluid mechanics and showed the effect of the inlet velocity on the external flow behaviour around a flat plate.

REFERENCES

- [1] Klazly MMA, Bognár G. Computational fluid dynamic simulation of laminar flow over a flat plate. *Design of Machines and Structures*. 2019;9(1):29–47. <https://doi.org/10.32972/dms.2019.004>
- [2] Wang H, Zhai ZJ. Analyzing grid independency and numerical viscosity of computational fluid dynamics for indoor environment applications. *Building and environment*. 2012 Jun 1;52:107-18. <https://doi.org/10.1016/j.buildenv.2011.12.019>
- [3] Hong JW, Simanto RI, Ahn BK, Oh SM, Lee DH. LDV measurements of boundary layer velocity profiles on flat plates with different surface roughnesses. *International Journal of Naval Architecture and Ocean Engineering*. 2024 Jan 1;16:100619. <https://doi.org/10.1016/j.ijnaoe.2024.100619>
- [4] Xiao H, Xu ZZ, Li DY, Lyu SK. Grid independence in the study of boundary layer and its application in Hypergolic Propellants. *International Journal of Precision Engineering and Manufacturing*. 2016 Jul;17(7):887-95. <https://doi.org/10.1007/s12541-016-0108-7>
- [5] Prajapati CR, Donga AP, Babariya PR, Patel AD. Comparative analysis of turbulence models for simulating flow over a flat plate. *Int. J. Eng. Appl. Sci. Technol.*. 2023 May;8(01):278-83. DOI: [10.33564/IJEAST.2023.v08i01.043](https://doi.org/10.33564/IJEAST.2023.v08i01.043)
- [6] Mohamad Sahar A, Mohd Sidek NZMS, Abdul Hadi Ramachandran FS, Anuar NS, Mohd Makhatat NA. CFD Investigation of Boundary Layer Development in Laminar Flow over Flat Plates. *J. Adv. Res. Numer. Heat Trans.* [Internet]. 2025 Mar. 30 [cited 2026 Sep. 27];31(1):23-37. <https://doi.org/10.37934/arnht.31.1.2337>
- [7] Canbolat G, Yıldızeli A, Köse HA, Çadırcı S. Numerical Investigation of Transitional Flow over a Flat Plate under Constant Heat Fluxes. *Academic Perspective Procedia*. 2018 Nov 9;1(1):187-95. DOI: [10.33793/acperpro.01.01.39](https://doi.org/10.33793/acperpro.01.01.39)
- [8] Dobhal S. Numerical study of turbulence models on boundary layer flow over a flat plate using computational fluid dynamics. *Journal of Advanced Research in Applied Mechanics and Computational Fluid Dynamics*. 2020 Jun 12;07(01):6–14. DOI: [10.24321/2349.7661.202002](https://doi.org/10.24321/2349.7661.202002)
- [9] Lee M, Park G, Park C, Kim C. Improvement of grid independence test for computational fluid dynamics model of building based on grid resolution. *Advances in Civil Engineering*. 2020;2020(1):8827936. <https://doi.org/10.1155/2020/8827936>
- [10] Bounds CP, Uddin M. Advancing CFD Simulations Through Machine-Learning-Enabled Mesh Refinement Analysis. *Fluids*. 2026 Jan 30;11(2):43. <https://doi.org/10.3390/fluids11020043>
- [11] Tsega EG. Numerical Simulation of Unsteady 2D Boundary Layer Flow and Heat Transfer Over a Flat Surface Using the Finite Volume Method With a Collocated Grid. *International Journal of Mathematics and Mathematical Sciences*. 2025;2025(1):2275810. <https://doi.org/10.1155/ijmm/2275810>
- [12] Belhocine A, Stojanovic N, Abdullah OI. Numerical simulation of laminar boundary layer flow over a horizontal flat plate in external incompressible viscous fluid. *European Journal of Computational Mechanics*. 2021 Nov 27:337-86. <https://doi.org/10.13052/ejcm2642-2085.30463>
- [13] ANSYS Fluent- 2D Flow Over a Flat Plate [Video]. YouTube; 2022 [cited 2026 Sep 27]. Available from: <https://www.youtube.com/watch?v=SNn7Cu3w1L4>