

Simulation Of Flow Around A Bullet-Shaped Object

Nur Danish Syahmi Mohammad Ali¹, Ishkrizat Taib^{1*}

1. Department of Aeronautical Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia (UTHM), 86400 Parit Raja, Johor, Malaysia

CORRESPONDENCE

Ishkrizat Taib

iszat@uthm.edu.my

Received: 10 July 2026

Revised: 01 August 2026

Accepted: 31 August 2026

Published: 30 September 2026

The aerodynamic performance of external flow around streamlined bodies is an important aspect in many engineering applications, including aerospace, automotive, and projectile systems. The interaction between airflow and body geometry influences pressure distribution, velocity variation, wake formation, and aerodynamic drag. Although streamlined bodies are designed to minimise aerodynamic resistance, flow separation and wake development may still occur, especially at higher flow velocities. These phenomena can significantly affect aerodynamic performance and energy efficiency. Therefore, an investigation is required to evaluate the influence of inlet velocity on the flow behaviour around a bullet-shaped object. The aim of this study is to investigate the aerodynamic characteristics of a bullet-shaped object subjected to different inlet velocities. A numerical approach was employed to analyse pressure distribution, velocity distribution, streamline behaviour, and drag coefficient around the geometry. Three inlet velocities, namely 10 m/s, 20 m/s, and 30 m/s, were considered. The results showed that the highest pressure occurred at the nose region of the bullet-shaped object due to flow stagnation, while a low-pressure wake region developed behind the rear section. Velocity contours revealed flow acceleration around the curved surface of the body followed by flow separation downstream. Streamline analysis indicated the formation of recirculation zones and vortex structures behind the object. The drag coefficient increased from 2.6717 at 10 m/s to 8.4498 at 20 m/s and 16.8668 at 30 m/s. Furthermore, the grid independence test demonstrated that the difference between the two finest meshes was only 2.82%, indicating that the numerical solution had approached mesh-independent behaviour. In conclusion, inlet velocity significantly influenced the aerodynamic characteristics of the bullet-shaped object. Higher velocities generated larger wake regions, stronger recirculation zones, and greater drag coefficients. The study successfully demonstrated the relationship between velocity variation and aerodynamic performance of a bullet-shaped geometry.

Keywords: External Flow, Bullet-Shaped Object, Aerodynamics, Drag Coefficient, Wake Formation

1. INTRODUCTION

Computational Fluid Dynamics (CFD) has become one of the most widely used numerical techniques for analysing fluid flow behaviour in engineering applications. CFD enables engineers to investigate pressure distribution, velocity fields, aerodynamic forces, wake development, and flow separation around solid bodies without relying entirely on experimental testing. The capability of CFD to provide detailed flow visualisation has led to its extensive application in

aerospace, automotive, and external aerodynamic studies. Recent studies have demonstrated that CFD can accurately predict external flow characteristics and provide valuable information for aerodynamic optimisation and performance improvement [1–4]. External flow refers to fluid motion around an immersed body. The behaviour of external flow is highly dependent on body geometry, Reynolds number, and flow velocity. Numerical investigations have shown that streamlined bodies generally experience lower aerodynamic drag due to reduced flow separation and smaller wake regions compared with bluff bodies. Conversely, bluff bodies generate significant recirculation zones and vortex shedding phenomena that contribute to higher drag forces [5–7]. Therefore, understanding the interaction between fluid flow and body geometry is important in aerodynamic design and optimisation.

A bullet-shaped object represents a streamlined geometry commonly used in projectile and aerodynamic applications. The aerodynamic performance of such a body is strongly influenced by pressure distribution at the nose section, boundary layer development along the surface, and wake formation behind the rear section. Previous investigations on projectile aerodynamics reported that flow separation, wake development, and drag characteristics play important roles in determining the overall aerodynamic behaviour of bullet-type geometries [8–11]. Variations in geometry and flow conditions may significantly alter pressure distribution and aerodynamic forces acting on the body. Aerodynamic drag is one of the most important parameters used to evaluate external flow performance. Drag force consists primarily of pressure drag and skin friction drag generated by the interaction between the fluid and body surface. Previous studies reported that drag characteristics are strongly affected by body shape, flow velocity, and wake formation mechanisms. Streamlined geometries generally exhibit lower drag due to reduced pressure differences between the front and rear surfaces of the body [12–16]. Furthermore, streamline analysis provides valuable information regarding flow attachment, separation, recirculation zones, and vortex formation occurring behind the object.

In addition to pressure and drag characteristics, wake dynamics and flow separation are important aspects of external aerodynamics. Flow separation occurs when the boundary layer loses momentum and detaches from the surface, producing a wake region characterised by recirculating flow structures. Previous studies highlighted that wake size and vortex structures increase aerodynamic resistance and significantly influence aerodynamic performance [7], [10], [15]. Therefore, analysing streamline behaviour and wake development is essential for understanding flow characteristics around streamlined bodies. The accuracy of CFD simulations is highly dependent on the selection of turbulence models. Among the available models, the SST $k-\omega$ turbulence model has been widely adopted for external aerodynamic applications because it provides reliable predictions of boundary layer behaviour, flow separation, and wake formation. Recent validation studies reported that the SST $k-\omega$ model offers good agreement with experimental data and is suitable for external flow simulations involving aerodynamic bodies [17–20].

Therefore, this study investigates the flow behaviour around a bullet-shaped object subjected to inlet velocities of 10 m/s, 20 m/s, and 30 m/s. A numerical simulation approach was employed to analyse pressure distribution, velocity distribution, streamline behaviour, wake formation, and drag coefficient under different flow conditions. The simulation results showed that increasing inlet velocity produced larger wake regions and higher drag coefficients, indicating stronger aerodynamic resistance at higher flow velocities. The study focuses on analysing the pressure distribution around the bullet-shaped object, examining the velocity distribution and streamline behaviour surrounding the geometry, and evaluating the aerodynamic characteristics of the object under different inlet velocity conditions.

2. MATERIAL AND METHODS

This section describes the methodology used to investigate the flow behaviour around a bullet-shaped object. The procedure includes geometry development, mesh generation, boundary condition specification, numerical setup, and grid independence testing to ensure solution accuracy.

2.1. Geometry Setup

A bullet-shaped object was designed as the test geometry for this study. The geometry consisted of a rounded nose section and a flat rear section to represent a streamlined body subjected to external flow conditions. The overall length of the geometry was 30 mm with a maximum height of 4.5 mm. The nose profile was generated using arc radii of 3.6675 mm and 53.732 mm to produce a smooth aerodynamic contour, while a straight section with a length of 6.5 mm was incorporated at the rear. The detailed dimensions of the geometry are presented in Figure 1. The rounded nose was intended to reduce flow separation at the front section, whereas the flat rear surface promoted wake formation for aerodynamic analysis.

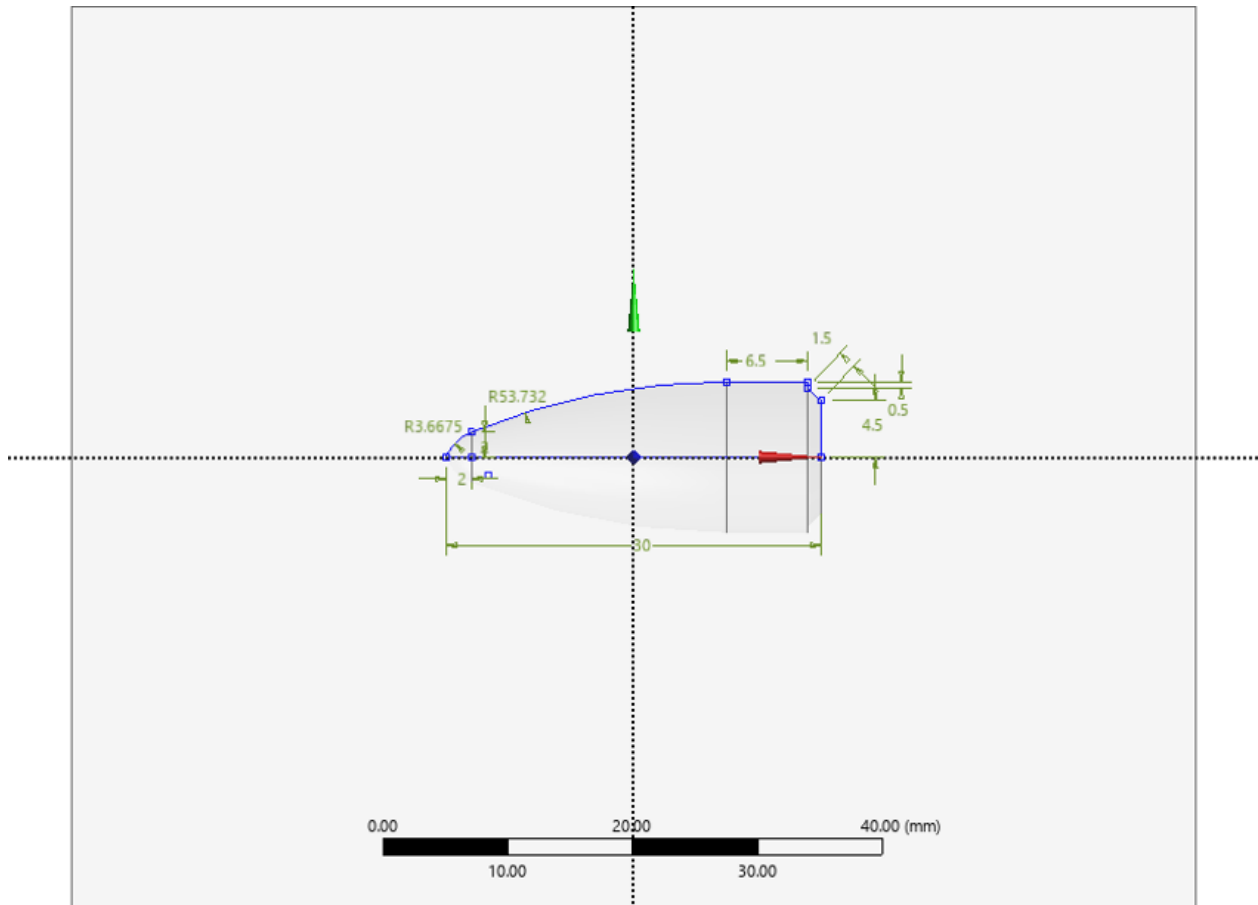


Figure 1. Dimensions of the bullet-shaped object used in the simulation

2.2. Mesh Generation

A two-dimensional computational domain was created around the bullet-shaped object and discretised using an unstructured triangular mesh. Local mesh refinement was applied near the object surface and nose region to accurately capture significant pressure and velocity gradients. Additional refinement was introduced in the wake region behind the object to improve the prediction of flow separation and recirculation behaviour. As shown in Figure 2, the mesh density was considerably higher near the bullet-shaped object than in the outer flow domain. This meshing strategy enabled important flow features to be resolved with greater accuracy while maintaining a reasonable computational cost. Based on the grid independence test, a mesh consisting of 268,379 elements was selected for all simulations because it provided a good balance between computational efficiency and solution accuracy.

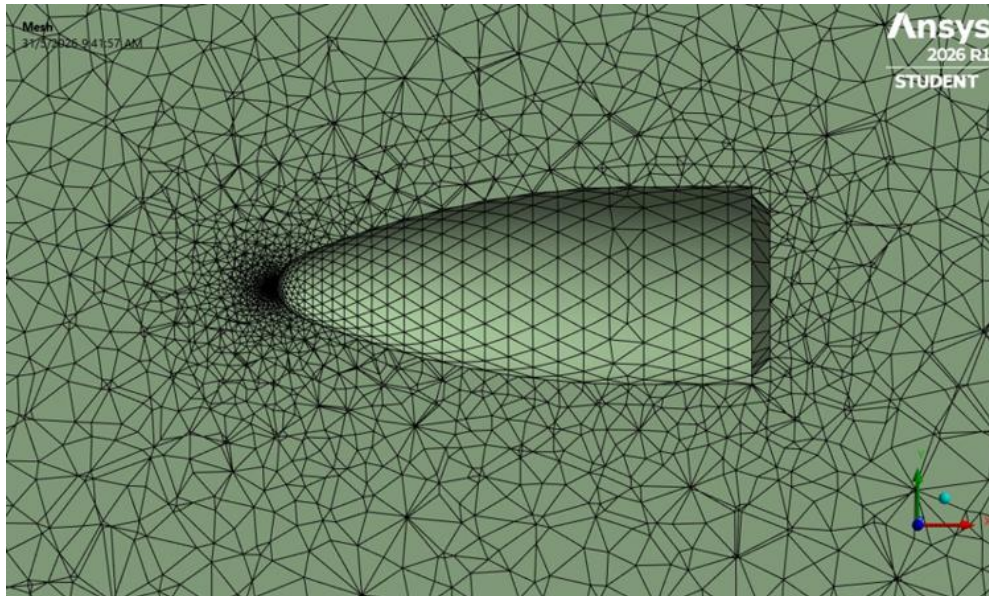


Figure 2. Computational mesh showing local refinement near the bullet-shaped object and wake region

2.3. Boundary Conditions

Appropriate boundary conditions were applied to represent the external flow environment. Air was selected as the working fluid with a density of 1.225 kg/m^3 . Three inlet velocities, namely 10 m/s, 20 m/s, and 30 m/s, were investigated to evaluate the influence of flow velocity on the aerodynamic behaviour of the bullet-shaped object. The outlet boundary was specified as a pressure outlet with atmospheric pressure conditions, while the object surface was treated as a stationary no-slip wall. The boundary conditions used in this study are summarised in Table 1. The selected parameters were maintained throughout all simulations to ensure consistent comparison between the different inlet velocity cases.

Table 1. Boundary conditions used in the simulation

Parameter	Value
Fluid	Air
Density	1.225 kg/m^3
Inlet Velocity	10 m/s, 20 m/s, 30 m/s
Pressure Outlet	0 Pa
Turbulence Intensity	5%
Turbulence Viscosity Ratio	10
Wall Condition	No-Slip Wall

Figures 3, 4, and 5 show the boundary conditions applied to the computational domain. The velocity inlet was assigned at the upstream boundary to introduce airflow into the domain, while a pressure outlet condition was specified at the downstream boundary to allow the flow to exit the domain. In addition, the bullet-shaped object was defined as a stationary no-slip wall, ensuring that the fluid velocity at the surface was equal to zero.

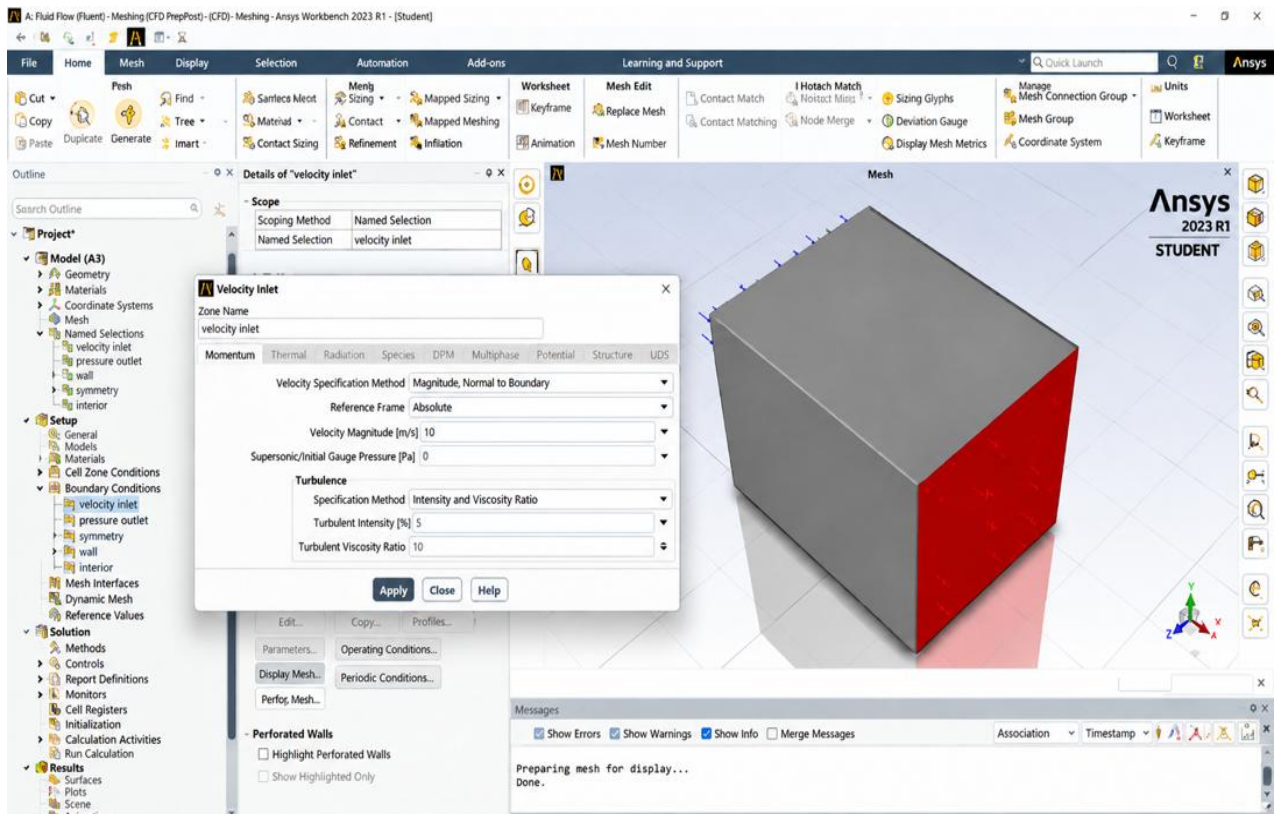


Figure 3. Boundary conditions applied to the bullet-shaped object simulation (Velocity inlet)

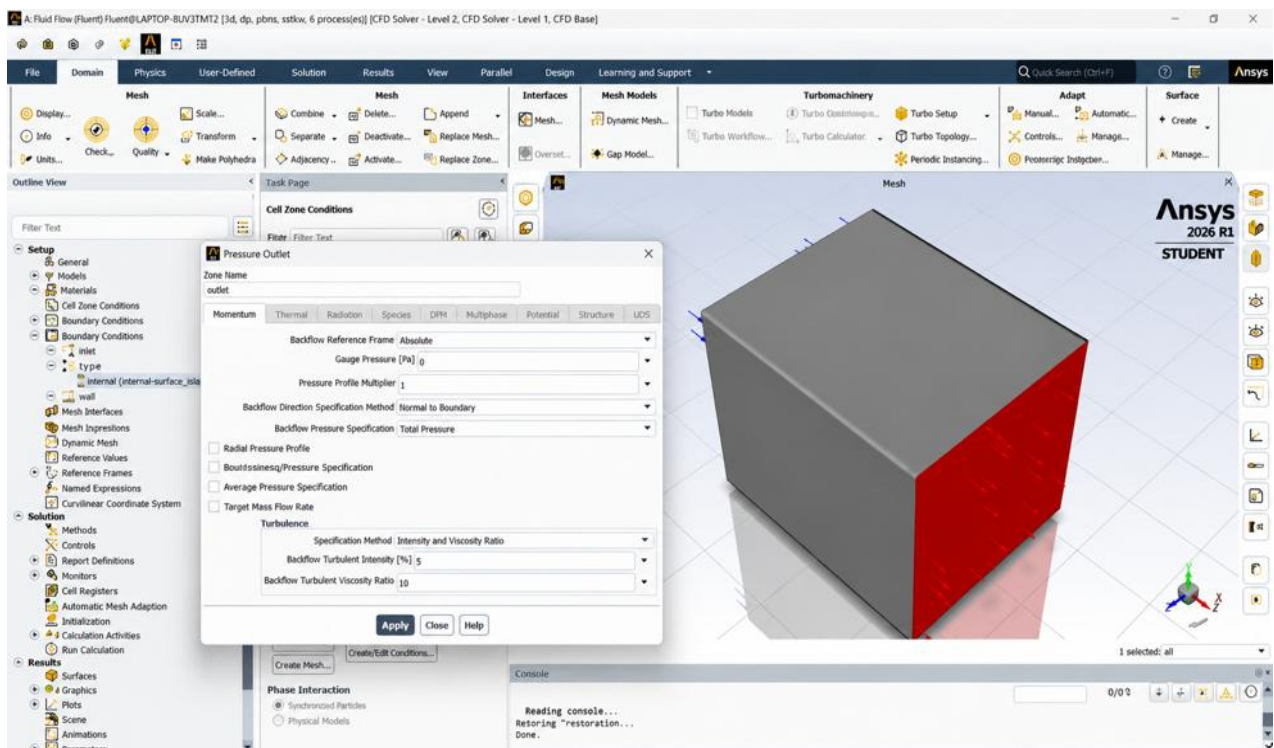


Figure 4. Boundary conditions applied to the bullet-shaped object simulation (Pressure outlet)

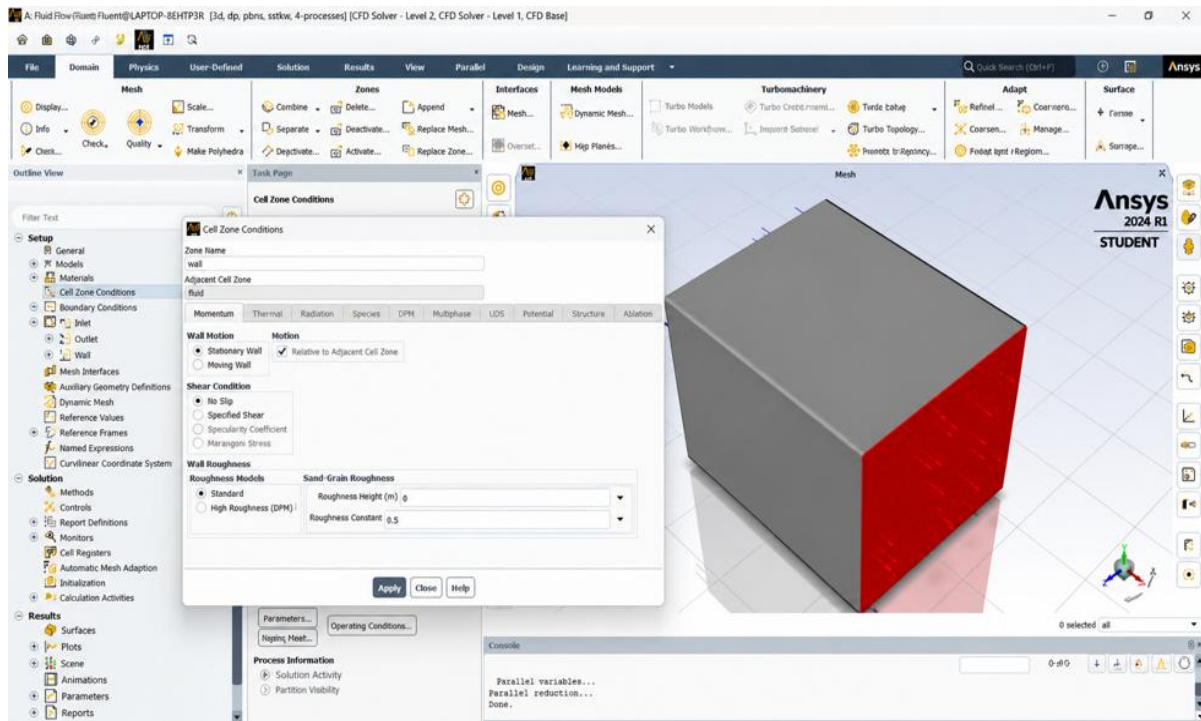


Figure 5. Boundary conditions applied to the bullet-shaped object simulation (No-slip wall condition)

2.4. Numerical Setup

The SST $k-\omega$ turbulence model was selected because of its capability to accurately predict boundary layer development, flow separation, and wake formation in external aerodynamic applications. Pressure–velocity coupling was achieved using the SIMPLE algorithm. Second-order discretisation schemes were applied for pressure, momentum, turbulent kinetic energy, and specific dissipation rate equations to improve numerical accuracy and solution stability. The simulations were performed for inlet velocities of 10 m/s, 20 m/s, and 30 m/s. The numerical settings adopted in the present study are shown in Figures 6 and 7.

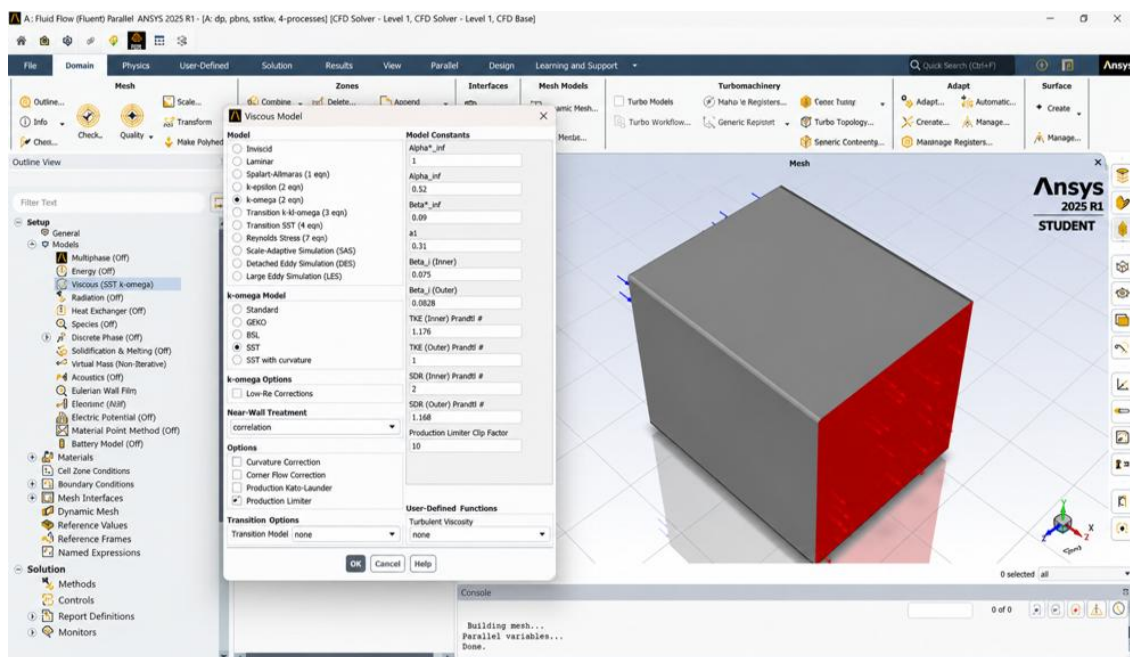


Figure 6. Numerical setup and solution methods used in the simulation (SST $k-\omega$ turbulence model setting)

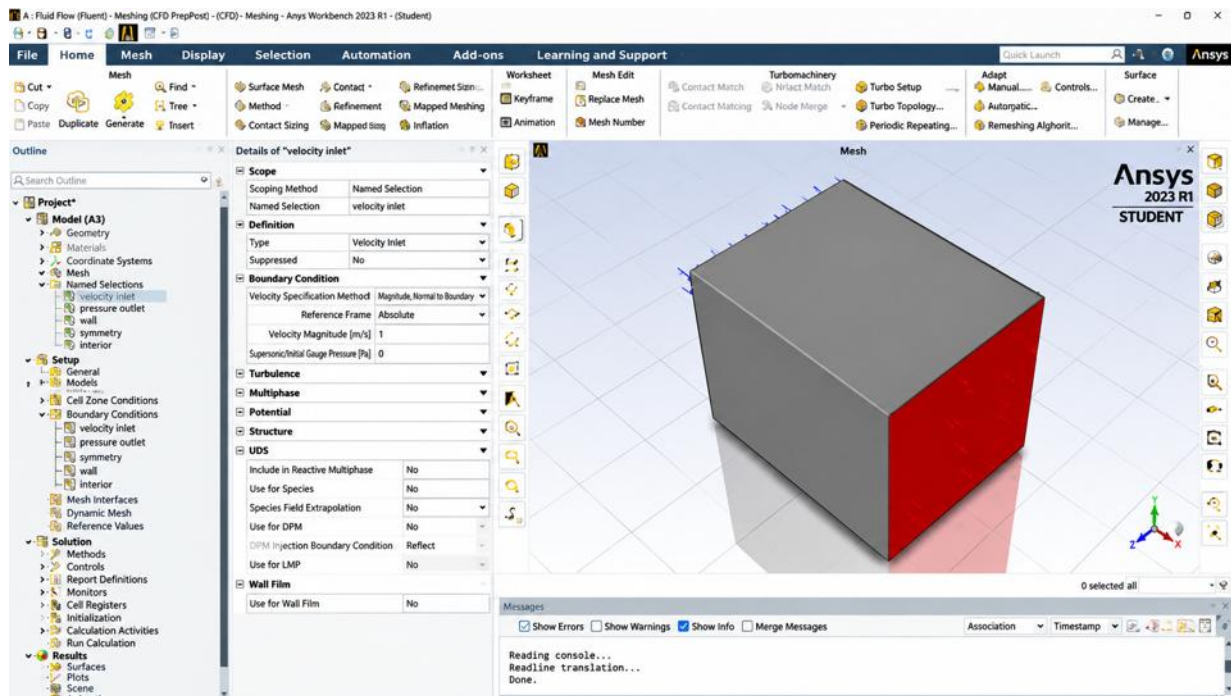


Figure 7. Numerical setup and solution methods used in the simulation (Solution methods and discretization schemes)

3. RESULTS AND DISCUSSION

3.1. Pressure Distribution

Figure 8 shows the pressure contour distribution around the bullet-shaped object at inlet velocities of 10 m/s, 20 m/s, and 30 m/s. The highest pressure was observed at the nose region of the object, where the incoming airflow directly impinged on the surface and formed a stagnation region. As the inlet velocity increased, the pressure magnitude at the front section became more pronounced. In contrast, a low-pressure region developed behind the flat rear surface due to flow separation and wake formation. The size and intensity of the wake region increased with increasing inlet velocity, indicating stronger pressure differences between the front and rear sections of the body. The pressure contours also demonstrate that the overall aerodynamic effects became more significant at higher velocities, resulting in greater aerodynamic resistance.

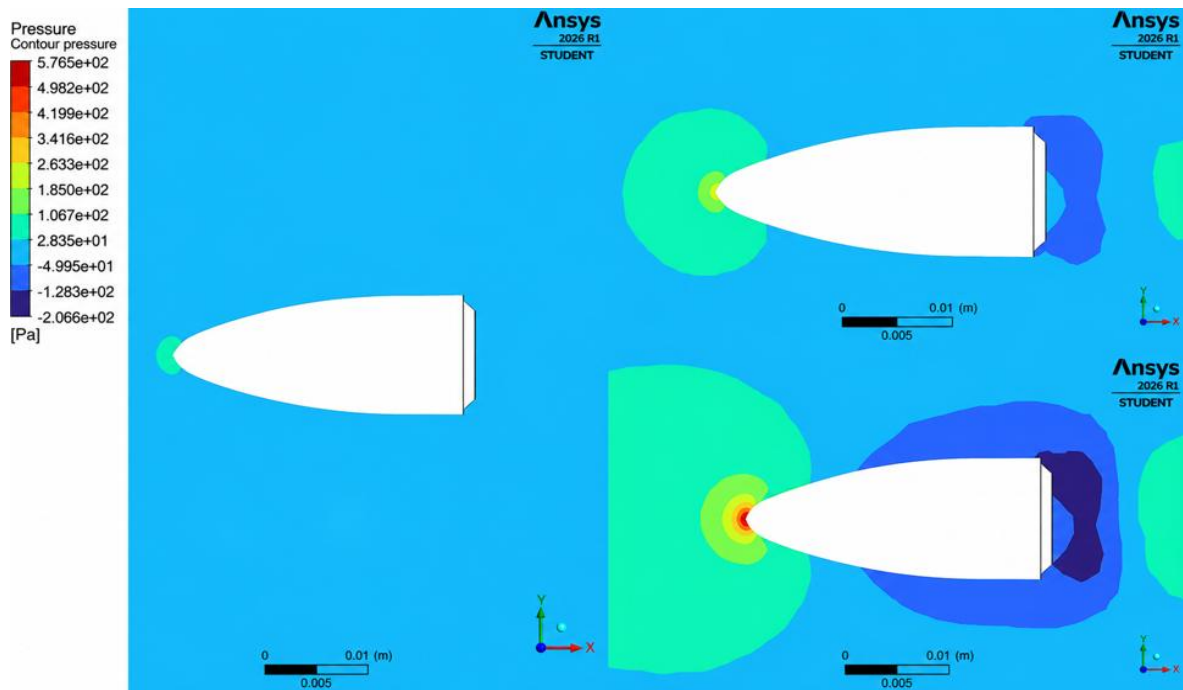


Figure 8. Pressure contour distribution around the bullet-shaped object

3.2. Velocity Distribution

Figure 9 presents the velocity contour distribution around the bullet-shaped object at inlet velocities of 10 m/s, 20 m/s, and 30 m/s. The airflow accelerated along the curved surface of the bullet-shaped object due to the streamlined geometry. The highest velocity regions were observed near the side surfaces of the object where the flow was forced to move around the body. A low-velocity wake region developed behind the flat rear section as a result of flow separation. As the inlet velocity increased from 10 m/s to 30 m/s, the wake region became larger and more pronounced, indicating stronger flow separation and recirculation effects. The velocity contours also show that the overall flow field intensified with increasing inlet velocity, resulting in greater aerodynamic interaction between the fluid and the object.

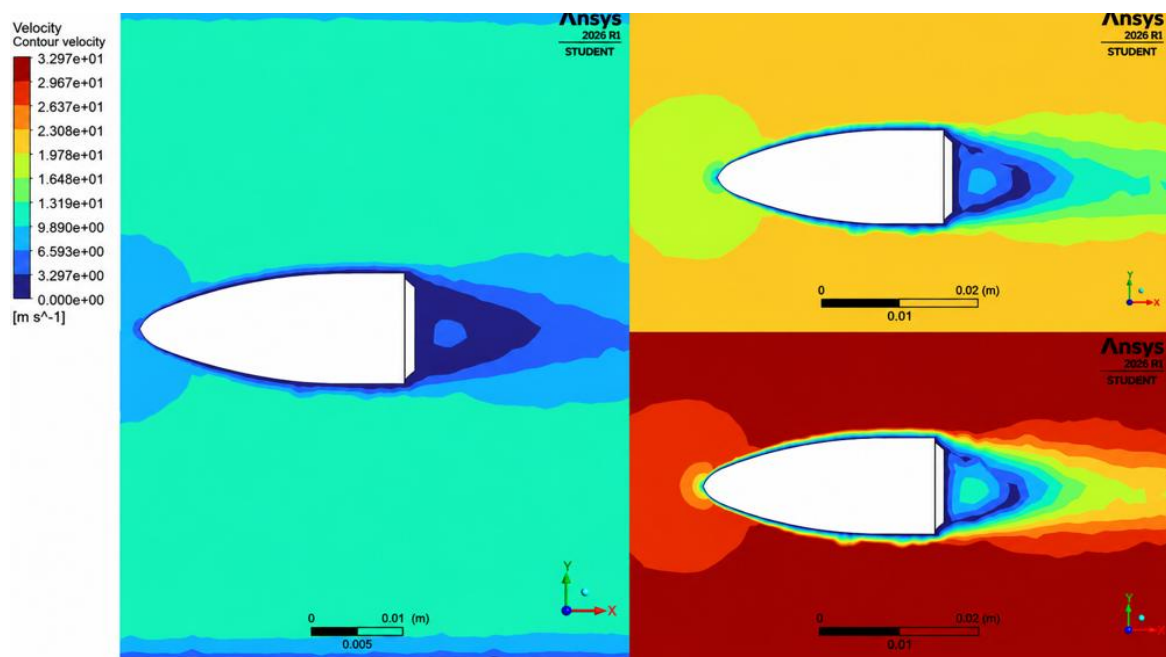


Figure 9. Velocity contour distribution around the bullet-shaped object

3.3. Streamline Analysis

Figure 10 illustrates the streamline behaviour around the bullet-shaped object at inlet velocities of 10 m/s, 20 m/s, and 30 m/s. The streamlines remained attached along most of the curved surface of the object before separating at the flat rear section. Flow separation behind the object resulted in the formation of a recirculation zone characterised by vortex structures in the wake region. At an inlet velocity of 10 m/s, a relatively small wake region was observed behind the object. As the velocity increased to 20 m/s and 30 m/s, the wake region became larger and the recirculation structures became more pronounced. The streamline patterns indicate that increasing inlet velocity intensified the wake development and flow recirculation behind the bullet-shaped object, which contributed to the increase in aerodynamic drag.

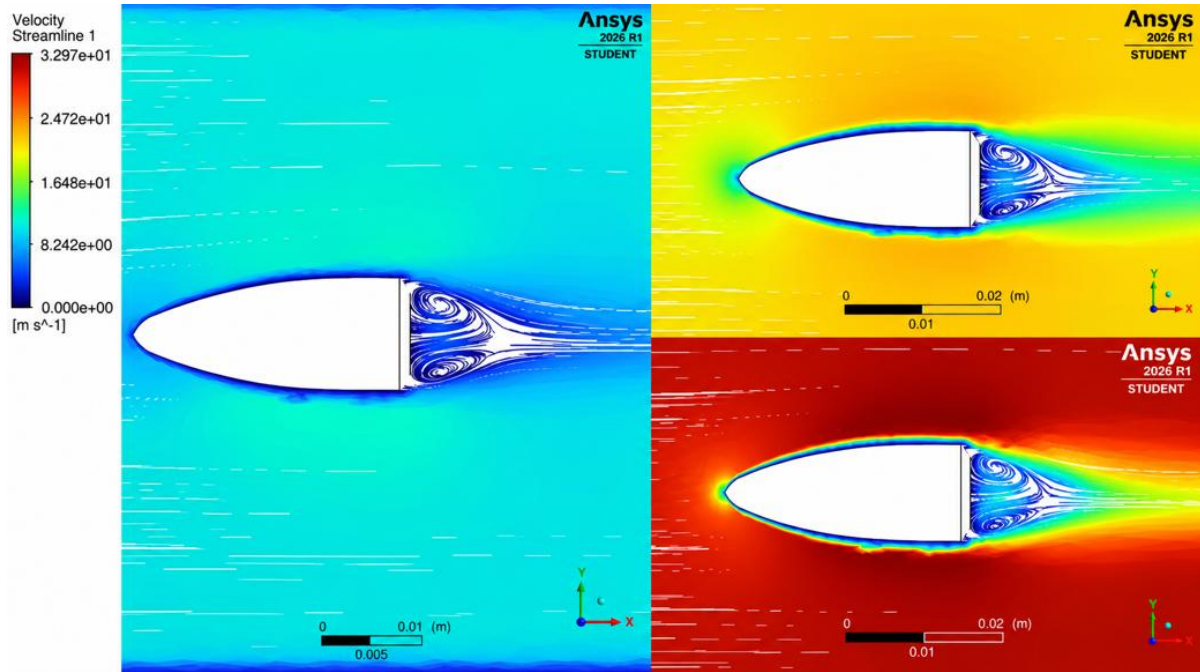


Figure 10. Streamline distribution around the bullet-shaped object

3.4. Drag Coefficient Analysis

Table 2 summarises the drag coefficient obtained for the bullet-shaped object at different inlet velocities, while Figure 8 illustrates the variation of drag coefficient with inlet velocity. The results indicate that the drag coefficient increased significantly as the inlet velocity increased. At an inlet velocity of 10 m/s, the drag coefficient was 2.6717. The value increased to 8.4498 at 20 m/s and further increased to 16.8668 at 30 m/s. The increase in drag coefficient is associated with the stronger aerodynamic effects generated at higher velocities. As the inlet velocity increased, larger pressure differences developed between the front and rear sections of the object, resulting in a more pronounced wake region behind the body. The streamline analysis also showed stronger recirculation zones and vortex structures at higher velocities, which contributed to the increase in aerodynamic resistance.

Table 2. Drag coefficient obtained at different inlet velocities

Velocity (m/s)	Drag Coefficient
10	2.6717
20	8.4498
30	16.8668

As shown in Figure 11, a positive relationship exists between inlet velocity and drag coefficient. The highest drag coefficient was recorded at 30 m/s, indicating that the bullet-shaped object experienced the greatest aerodynamic resistance under this flow condition. These findings demonstrate that inlet velocity plays an important role in determining the aerodynamic performance of the bullet-shaped object.

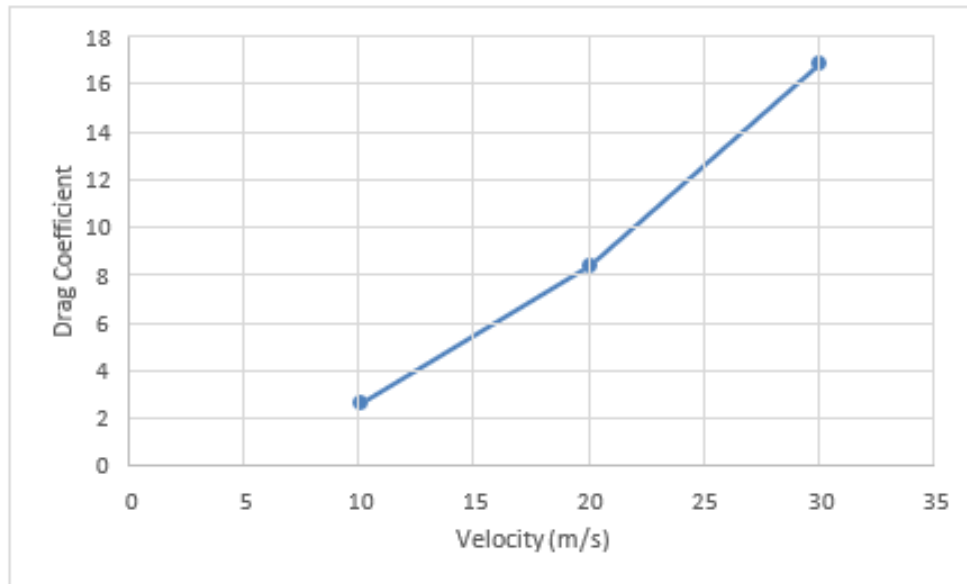


Figure 11. Drag Coefficient Variation with Inlet Velocity

3.5. Grid Independence Test

A grid independence test was performed to evaluate the influence of mesh refinement on the simulation results. The drag coefficient was selected as the comparison parameter because it represents the aerodynamic performance of the bullet-shaped object. Several mesh densities were investigated to determine whether further mesh refinement would significantly affect the numerical solution.

Table 3. Grid independence test results based on drag coefficient

Number of Elements	Drag coefficient
49,870	1.8339
55,311	1.9252
66,669	2.1106
114,440	2.4094
216,278	2.6172
268,379	2.6716
405,254	2.7490

$$Difference = \frac{|2.7490 - 2.6716|}{2.7490} \times 100 = 2.82\%$$

Equation 1

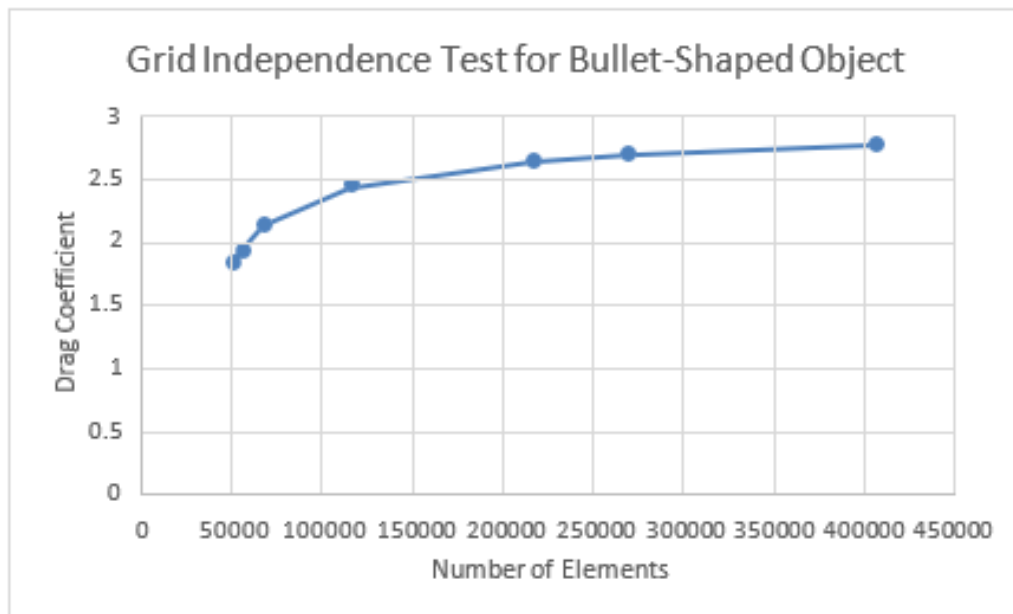


Figure 12. Grid independence test based on drag coefficient

Figure 12 presents the grid independence test based on the drag coefficient obtained from different mesh densities. The results show that the drag coefficient increased gradually with mesh refinement and eventually approached a converged value. As the number of elements increased from 49,870 to 405,254, the drag coefficient changed from 1.8339 to 2.7490. The variation became smaller at higher mesh densities, indicating that the numerical solution was approaching mesh-independent behaviour. The difference between the two finest meshes, namely 268,379 and 405,254 elements, was only 2.82%, demonstrating that further mesh refinement had a minimal effect on the drag coefficient prediction. Therefore, the mesh containing 268,379 elements was selected for the final simulation because it provided a good balance between computational cost and solution accuracy.

4. CONCLUSIONS

The pressure distribution around the bullet-shaped object was successfully analysed. The highest pressure occurred at the nose region because of flow stagnation, while a low-pressure wake region developed behind the object. The velocity distribution and streamline behaviour around the bullet-shaped object were successfully analysed. Flow acceleration occurred along the body surface, followed by flow separation and recirculation downstream. Larger wake regions and stronger vortex structures were observed at higher inlet velocities. The aerodynamic characteristics of the bullet-shaped object were successfully evaluated under different inlet velocities. The drag coefficient increased from 2.6717 at 10 m/s to 8.4498 at 20 m/s and 16.8668 at 30 m/s, indicating increased aerodynamic resistance with increasing velocity.

ACKNOWLEDGEMENTS

The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support.

REFERENCES

- [1] Pirbastami S, Pepper D, Carrington DB. TURBULENCE MODELING OF SHOCK BOUNDARY LAYER INTERACTION IN HYPERSONIC FLOW. *Computational Thermal Sciences: An International Journal*. 2024;16(2). <http://dx.doi.org/10.34917/28340360>
- [2] Dinesan D, John B. Conjugate heat transfer analysis of mach 2.5 flow over a cylindrical dome with surface arc plasma actuator flow control. *International Journal of Aeronautical and Space Sciences*. 2026 Mar;27(2):1334-53. <https://doi.org/10.1007/s42405-025-01055-w>

- [3] Fernandes E, Pachpande N, Chourushi T. Low reynolds number flow over non-rectangular bluff bodies using OpenFOAM. In AIP Conference Proceedings 2026 Mar 18 (Vol. 3369, No. 1, p. 040009). AIP Publishing LLC. <https://doi.org/10.1063/5.0317385>
- [4] Shirzadi M, Tominaga Y. Comprehensive evaluation of the SST k- ω model for building environment applications: Comparison with k- ϵ models, LES, and wind tunnel experiments. Building and Environment. 2026 Mar 30:114566. <https://doi.org/10.1016/j.buildenv.2026.114566>
- [5] Kumar P, Singh SK. Flow past a bluff body subjected to lower subcritical Reynolds number. Journal of Ocean Engineering and Science. 2020 Jun 1;5(2):173-9. <https://doi.org/10.1016/j.joes.2019.10.002>.
- [6] Shirzadi M, Tominaga Y. Comprehensive evaluation of the SST k- ω model for building environment applications: Comparison with k- ϵ models, LES, and wind tunnel experiments. Building and Environment. 2026 Mar 30:114566. <https://doi.org/10.1016/j.buildenv.2026.114566>.
- [7] Hao X, He X, Zhang Z, Li J. A review of simulations and machine learning approaches for flow separation analysis. Aerospace. 2025 Mar 14;12(3):238. <https://doi.org/10.3390/aerospace12030238>
- [8] Tuan NQ, Minh NH, Xuan SB. Numerical investigation on the supersonic flow around a sabot bullet. Vojnotehnički glasnik. 2024;72(2):676-94. DOI: [10.5937/vojtehg72-48837](https://doi.org/10.5937/vojtehg72-48837)
- [9] SHEN C, ZHOU KD, HE L, LU Y, LI JS. The Influence of Bore Damage on Bullet's Exterior Ballistics Process. Transactions of Beijing institute of Technology. 2020;40(6):623-31. DOI: DOI: [10.15918/j.tbti1001-0645.2019.147](https://doi.org/10.15918/j.tbti1001-0645.2019.147)
- [10] Mustak R, Mashud M. Recent advances in flow separation control of airfoils through surface grooves. AIP Advances. 2025 Nov 1;15(11). <https://doi.org/10.1063/5.0290804>
- [11] Agarwal C, Prakash R, Mahajan V, Chhetri A, Vinaygamurthy G, Bhushan A. Investigation on the Influence of Vortex Generators on Aerofoil. In Conference on Fluid Mechanics and Fluid Power 2021 Dec 27 (pp. 263-270). Singapore: Springer Nature Singapore. https://doi.org/10.1007/978-981-19-6270-7_46
- [12] Götten F, Havermann M, Braun C, Marino M, Bil C. Improved form factor for drag estimation of fuselages with various cross sections. Journal of Aircraft. 2021 May;58(3):549-61. DOI: [10.5937/vojtehg72-48837](https://doi.org/10.5937/vojtehg72-48837)
- [13] Sabirzyanov AN, Akhmetzyanov AR, Golovin AV, Bykov VV. Reducing the Drag of a Prolate Body by Modification of Its Nose Shape. Russian Aeronautics. 2024 Jun;67(2):311-23. <https://doi.org/10.3103/S1068799824020119>
- [14] Hachimy FZ, Omar A, Elsayed O, Bouchaala K. Parametric optimization for van drag reduction using a side flap. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering. 2024 Jul;238(8):2312-26. <https://doi.org/10.1177/09544070231164200>
- [15] Taylor M, Butcher D, Crickmore C, Walker AD. Practical computational fluid dynamic predictions of a cyclist in a time trial position. Sports Engineering. 2024 Dec;27(2):34. <https://doi.org/10.1007/s12283-024-00475-3>
- [16] Wang X, Liu T, Fang M, Chen X, Hu Y, Ouyang J, Liang X, Zhang H. Effects of air blowing in the constant cross section region on skin friction drag and aerodynamic performance of high-speed trains. Physics of Fluids. 2026 May 1;38(5). <https://doi.org/10.1063/5.0319899>
- [17] Raje P, Sinha K. Anisotropic SST turbulence model for shock-boundary layer interaction. Computers & Fluids. 2021 Oct 15;228:105072. <https://doi.org/10.1016/j.compfluid.2021.105072>
- [18] Badran O, Azad Z, Virk M, Masalha I, Alahmar A. Aerodynamic Performance Evaluation of the NACA 65-421 Airfoil Using CFD and the k- ω SST Turbulence Model. In ASME Aerospace Structures, Structural Dynamics, and Materials Conference 2025 May 5 (Vol. 88759, p. V001T01A027). American Society of Mechanical Engineers. <https://doi.org/10.1115/SSDM2025-157721>
- [19] Zeng Y, Wang H, Xiong D, Yang Y, Sun M, Liu W. A multi-physics correction of the shear-stress transport turbulence model for supersonic flows. Physics of Fluids. 2024 May 1;36(5). <https://doi.org/10.1063/5.0205391>
- [20] Ibrahim MD, Ananthan AA, Mahmod DS, Sunami Y, Barroy P, Chin CP, Bakri SR. Friction measurement of modified polydimethylsiloxane (PDMS) surfaces inspired by Malayopython Reticulatus. Biotribology. 2023 Jun 1;33:100240. <https://doi.org/10.1016/j.biotri.2023.100240>