Advanced and Sustainable Technologies
ASET
E-ISSN 2976-2294
UNIVERSITI MALAYSIA PERLIS

*Volume 3, No 1, June 2024 [71-79]*

# Analysis of Tesla CyberTruck Speed on the Velocity and Pressure Distribution Using SimFlow Software

Shankarsana Paramasivan[1*]

[1]Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis (UniMAP), 02600 Arau, Perlis, Malaysia

Received 29 March 2024, Revised 24 April 2024, Accepted 2 May 2024

**ABSTRACT**

*This work presents a comprehensive Computational Fluid Dynamics (CFD) analysis of the external flow around the Tesla CyberTruck, focusing on its aerodynamic characteristics under varying conditions. The primary objectives are to study velocity and pressure distributions. The simulation considers two independent parameters: vehicle speed (20, 40, and 80 m/s) and the type of turbulent flow (k-ω SST and k-ε). The simulation provides insights into complex flow patterns through meticulous meshing, boundary condition setup, and solver configuration, highlighting areas of interest such as flow separation, recirculation, and turbulence. Parametric variations are analyzed to determine how turbulent flow type and speed affect critical parameters like pressure and velocity. The results of this CFD analysis offer valuable information about the vehicle's aerodynamic performance, contributing to design optimization, handling and stability enhancements, and improved fuel efficiency. The findings from this study are expected to enhance visualization and understanding of the aerodynamic aspects of the Tesla CyberTruck.*

**Keywords:** Tesla CyberTruck, Turbulence model, Aerodynamic, Computational Fluid Dynamics.

## 1. INTRODUCTION

Recent advancements in automotive technology have been driven by minimizing aerodynamic drag to enhance safety, particularly with the expansion of highways and roads [1]. Innovations in this field demonstrate a trend towards producing faster, more powerful, and lighter vehicles. A key consideration in modern car design is the emphasis on general safety, which encompasses stability and control of the car [2]. Aerodynamics, the study of airflow interactions with moving objects, is crucial in this context. Aerodynamics can be categorized into two main areas: (a) external aerodynamics [3], which deals with the interaction of airflow around the exterior of a solid body with various shapes, and (b) internal aerodynamics [4], which focuses on the flow that passes through the internal compartments of a solid body. The velocity and design of an automotive vehicle significantly impact its aerodynamic properties, affecting overall performance, fuel efficiency, and safety.

The use of computational fluid dynamics (CFD) to predict aerodynamic flow around vehicles has increased significantly due to the advancements in computing power, making CFD a viable tool for simulating aerodynamic effects [5,6,7,8]. CFD software allows for the easy setup of aerodynamic characteristics of complex bodies, including those with moving walls and components, enabling the implementation of real-world conditions early in the design process. The exponential rise in computing power has driven the growth of the CFD industry, which primarily uses the Navier-Stokes equations [9] to describe fluid flow behavior in a continuum approximation. In contrast, the Boltzmann equation [10] examines the macroscopic behavior of

*Corresponding author: paramasivanshankarsana@gmail.com

a fluid. The distinctions between Navier-Stokes and lattice Boltzmann computational simulations are still being fully explored. CFD is a numerical method used to solve complex fluid dynamics problems cost-effectively by employing a system of partial differential equations that govern fluid behavior. It simulates fluid flows and heat transfer in various engineering applications [11, 12, 13], and continuous advancements in computer technology further enhance simulation capabilities. The CFD process involves pre-processing, solving, and post-processing [14].

A study conducted by Wang et al. [15] analyzed the aerodynamic performance of the Tesla Model S using CFD simulations. They utilized 3D laser scanning to acquire point cloud data of the vehicle and employed the STAR-CCM+ software for the analysis. The study focused on the impact of different working conditions, such as ground clearance, tyre rotation, and tread pattern, on the vehicle's drag coefficient. The findings revealed that air drag significantly affects the cruising ability of pure electric cars, particularly at high speeds. The simulations showed that various vehicle components influenced the overall drag coefficient of the Tesla Model S. The study identified the drag coefficients of different parts and examined the effect of tyre rotation on the overall drag coefficient. It was observed that the airflow acceleration due to tyre rotation led to a pressure decline at the bottom of the car, reducing the vehicle's coefficient of lift.

CFD systems are built with numerical algorithms and equations to solve fluid flow problems, particularly vehicle aerodynamics. The Reynolds-Averaged Navier-Stokes (RANS) models with k-ω SST [16] and k-ε turbulence models [17] are commonly used due to their ability to predict surface flow visualizations, provide accurate pressure distributions, and maintain low computational costs. These turbulence models offer differential reliability and performance, making them essential in CFD software for determining the aerodynamic features of vehicles and passenger cars.

The importance of reviewing issues related to the concept and design considerations of car models, which influence simulation effectiveness and aerodynamic performance, especially in terms of drag forces, is well-documented [18]. Based on Bernoulli's principle, CFD simulation results have been thoroughly validated for three main types of car model geometries by analyzing flow separations in static pressure contours. The reviews highlight the differences in characteristics, results, and applications among simple bodies, basic car shapes, and production (series) cars. According to these reviews, simple body geometries are preferable for simulations due to their ease of meshing, minimal computer resource requirements, and ability to yield reliable simulation results. These attributes make simple bodies an optimal choice for initial aerodynamic analysis in car design, providing a good balance between accuracy and computational efficiency.

The aerodynamic efficiency of a car is determined by its drag coefficient, which is influenced by several factors, including the car's shape, the angle of the windshield, and the presence of spoilers or other aerodynamic features. Das and Riyad [19] conducted a CFD analysis of a passenger vehicle at various rear-end spoiler angles and found that the drag coefficient decreased as the spoiler angle increased. Similarly, Srinivasarao and Lakshamaih [20] conducted CFD research on car bodies. They found that optimizing the car's shape can significantly improve aerodynamic efficiency. These studies underscore the importance of aerodynamic design in reducing drag and enhancing vehicle performance.

The primary focus of this project is to utilize Computational Fluid Dynamics (CFD) to examine the external aerodynamics of the Tesla CyberTruck [21]. A 3D CAD model of the CyberTruck was created using CAD software. The domain and mesh were established around the CyberTruck geometry using appropriate meshing techniques. CFD simulations were conducted using Reynolds-Averaged Navier-Stokes (RANS) turbulence models such as k-ω and k-ε to investigate overall drag and downforce at various speeds. The results of the CFD simulations were postprocessed in Paraview to visualize the velocity and pressure contours, which provide a comprehensive understanding of the CyberTruck's aerodynamic performance.

## 2. MATERIAL AND METHODS

Computational Fluid Dynamics plays a crucial role in car aerodynamics analysis by simulating airflow around a vehicle. This technology has numerous applications in the automotive industry, aiding in the design and optimization of cars for better performance, fuel efficiency, and safety. The geometry of the simplified Tesla CyberTruck car body is shown in Figure 1. Using the SimFlow software, the geometry of the car body was imported for meshing and simulation was done to estimate the maximum velocity and maximum pressure when the inlet velocity varies from 20m/s, 40m/s, and 80m/s, using the RANS k-ω SST and RANS k-ε turbulent model. In Simflow, the mesh refinement was enabled with a minimum of 3 and a maximum of 5 to enhance the element created around the Tesla CyberTruck model.

The base mesh parameters used are minimum (m) (-6, -14,-0.06) and maximum (0, 12, 8) to create the domain of the flow. The mesh division of (50, 15, 20) was set on the domain to generate the hexahedral dominant meshing on the domain and model. The cell size at the x-, y- and z-axis is 0.12, 1.73 and 0.4 m, respectively. The total generated mesh elements are 108851. The symmetrical model was considered to reduce the number of elements, and the symmetry plane was set at the Z+ axis. The boundary conditions, such as inlet velocity, pressure outlet and walls, were defined on the domain surfaces. In this simulation, the steady state, incompressible flow and SIMPLE discretization solver were considered based on the assumptions. The results visualization was conducted using Paraview post-processing software.

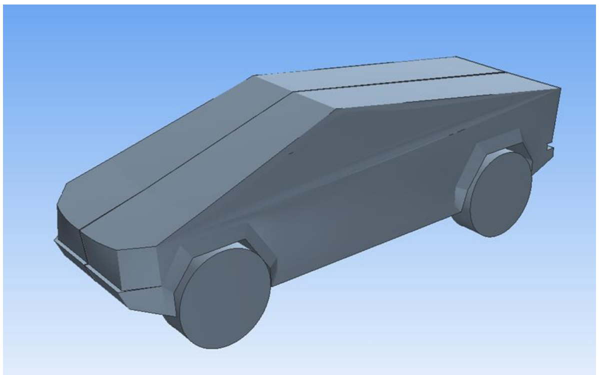

**Figure 1**: Geometry of the simplified Tesla CyberTruck.

## 3. RESULTS AND DISCUSSION

Computational Fluid Dynamics (CFD) was used in this project to simulate the external aerodynamics of a simplified Tesla CyberTruck model. The primary objective was to employ CFD techniques to analyze key aerodynamic parameters, such as the maximum airflow velocity and the maximum pressure distribution around the vehicle. The car geometry was constructed based on a simplified version of the Tesla CyberTruck, which was subsequently imported into the CFD software, SimFlow 4.0. The simulation setup involved defining the computational domain around the car model with appropriate boundary conditions, including an inlet velocity, a pressure-based outlet, and a wall interface to simulate the ground and vehicle surfaces. A hexahedral dominant mesh was generated to discretize the domain, and a steady-state pressure solver was employed to achieve convergence, with the residuals dropping below a threshold of 1e-03 (Figure 2).

The CFD simulation included several pre-processing steps and a detailed analysis of the results. Different turbulence models and varying inlet velocities were tested to assess their impact on the simulation accuracy and convergence behavior. Among the turbulence models evaluated, the k-ω SST (Shear Stress Transport) model demonstrated superior performance, yielding smoother residual convergence compared to the k-ε model. This suggests that the k-ω SST model provides more accurate and reliable predictions for the aerodynamic characteristics of the simplified Tesla CyberTruck model. Moreover, the project highlighted the effectiveness of CFD in analyzing and optimizing vehicle aerodynamics, offering insights into airflow patterns and pressure distributions that are critical for enhancing the aerodynamic efficiency and performance of automotive designs.

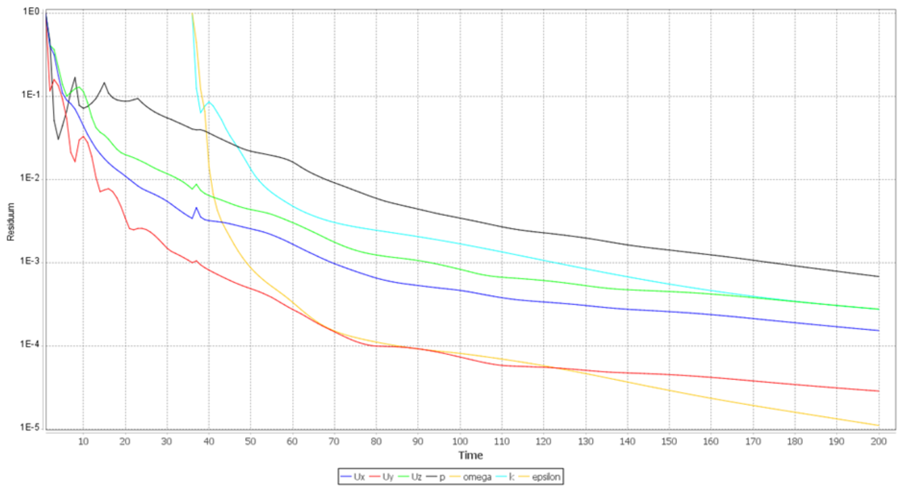


**Figure 2**: Residual plots at 40 m/s for RANS k-ω SST.

The comparison between the two turbulence models, k-ω SST and k-ε, revealed exciting insights into the aerodynamic behavior of the simulated Tesla CyberTruck model (Table 1). While the pressure distributions exhibited marginal differences between the two models across various inlet velocities, the discrepancies in maximum velocity values were striking. At an inlet velocity of 20 m/s, the k-ω SST model yielded a maximum velocity of 28.695 m/s, surpassing the value obtained by the k-ε model, which stood at 23.979 m/s. This trend continued as the inlet velocity increased (Figure 3). At 40 m/s, the maximum velocity recorded with the k-ω SST model reached 57.832 m/s, significantly higher than the corresponding value of 48.023 m/s obtained with the k-ε model. The disparity became even more pronounced at an inlet velocity of 80 m/s, with the k-ω SST model registering a maximum velocity of 116.791 m/s, compared to 91.646 m/s with the k-ε model.

The differences in maximum pressure values between the two turbulence models were relatively minor (Figure 4). At an inlet velocity of 20 m/s, the k-ω SST model produced a maximum pressure of 241.128 Pa, slightly lower than the value of 241.285 Pa obtained with the k-ε model. Similarly, at higher inlet velocities of 40 m/s and 80 m/s, the variations in maximum pressure between the two models remained small, with the k-ω SST model consistently exhibiting slightly lower pressure values compared to the k-ε model. These findings suggest that while both turbulence models adequately captured the pressure distribution around the vehicle, the k-ω SST model consistently outperformed the k-ε model in predicting maximum airflow velocities. This superiority is particularly evident at higher inlet velocities, where the k-ω SST model demonstrated a more accurate representation of the flow dynamics, highlighting its effectiveness in simulating complex aerodynamic phenomena associated with the Tesla CyberTruck model.

Table 1: Results of CFD simulation.

| Inlet Velocity (m/s) | Turbulence Model | Velocity (m/s) | Pressure (Pa) |
|---|---|---|---|
| 20 | | 28.695 | 241.128 |
| 40 | k-ω SST | 57.832 | 965.102 |
| 80 | | 116.791 | 3862.29 |
| 20 | | 23.979 | 241.285 |
| 40 | k-ε | 48.023 | 966.12 |
| 80 | | 91.646 | 3907.06 |

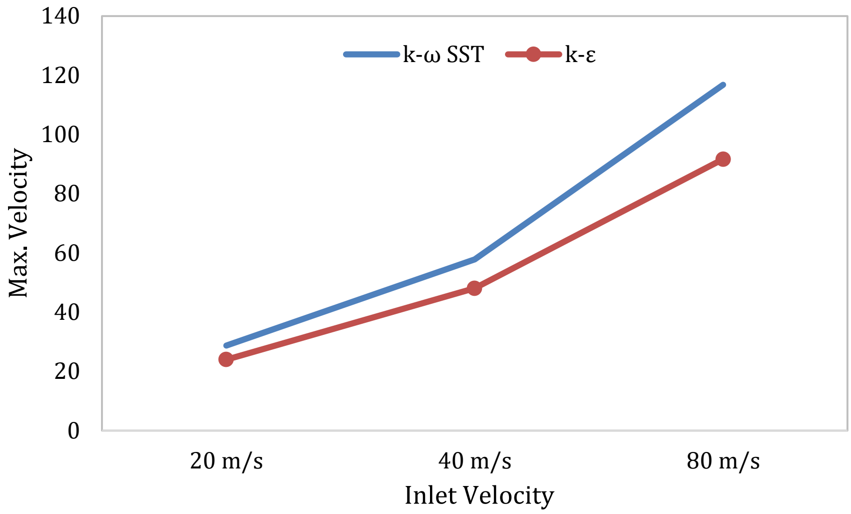


**Figure 3**: Maximum Velocity versus inlet velocity for two turbulent models.

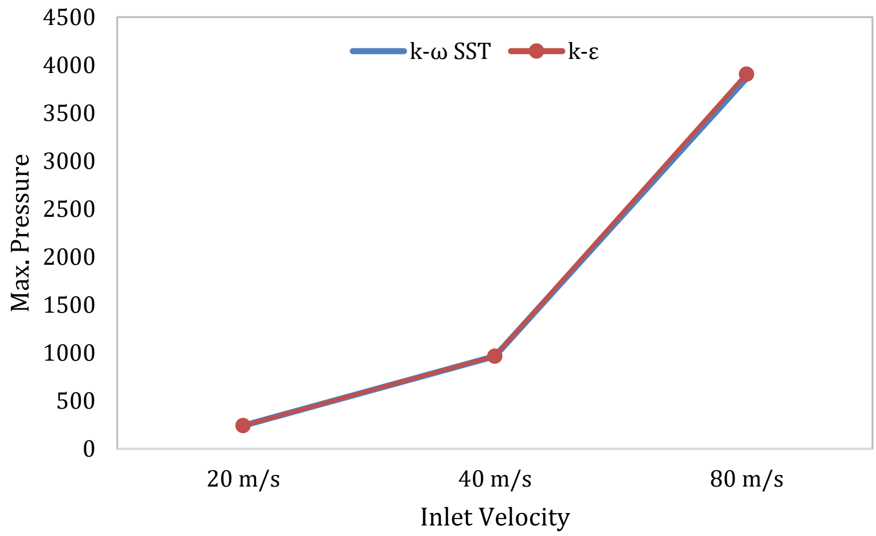


**Figure 4**: Maximum pressure versus inlet velocity for two turbulent models.

The analysis of the results highlights a fundamental principle of fluid dynamics: as the inlet velocity escalates, both the maximum velocity and pressure exhibit a corresponding rise. This phenomenon is clearly shown in Figures 5 and 6. This correlation, rooted in fluid motion laws, underscores the dynamic nature of airflow around the simulated Tesla CyberTruck model. However, beneath this primary trend lies a nuanced interplay of factors stemming from the selection of turbulence models. These models, with their distinct mathematical formulations and underlying assumptions, cause variations in the simulated airflow characteristics [22]. The observed differences in maximum velocity and pressure across varying inlet velocities stem from the intricacies of these turbulence models.

Engineers engaged in computational fluid dynamics (CFD) endeavors must navigate this complexity judiciously. Selecting the turbulence model that best aligns with the flow's inherent dynamics and the intended precision of the simulations is paramount. Such a decision hinges on a comprehensive understanding of flow physics and meticulously considering the turbulence model's strengths and limitations. The choice of turbulence model is a fundamental determinant in the fidelity and reliability of CFD simulations. Engineers must discern the trade-offs between computational efficiency and accuracy, ensuring that the selected model effectively captures the intricate flow phenomena pertinent to the CyberTruck's aerodynamics.

The maximum velocity and pressure variations emphasize the imperative for thoughtful turbulence model selection. By aligning the choice of model with the specific characteristics of the flow regime under investigation, engineers can enhance the predictive capabilities of CFD simulations, thus facilitating informed decision-making in vehicle design and optimization.

The findings of this study carry significant practical implications for engineers and designers involved in automotive aerodynamics. Firstly, the observed dependence of maximum velocity and pressure on inlet velocity highlights the importance of considering operating conditions during vehicle design and optimization processes. Understanding how velocity changes impact airflow characteristics is crucial for enhancing vehicle performance and efficiency across a range of driving scenarios. Moreover, the discernible differences between turbulence models in predicting airflow behavior highlight the necessity of selecting the most appropriate model for a given application. Engineers must weigh the trade-offs between computational efficiency and accuracy, ensuring simulations reliably capture the complex aerodynamic phenomena pertinent to vehicle design. By integrating these insights into the design process, automotive manufacturers can effectively streamline development efforts, optimize vehicle performance, and ultimately deliver enhanced driving experiences to consumers.

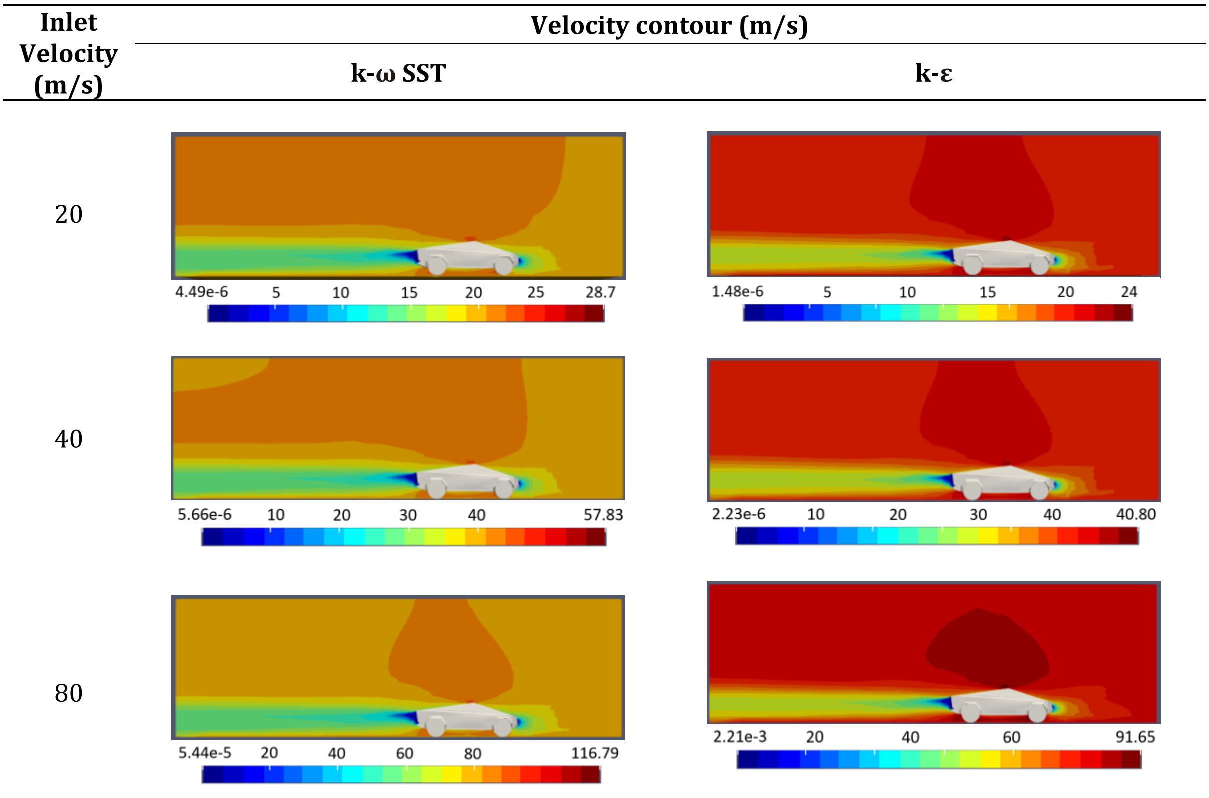


**Figure 5**: Comparison of velocity contour between different inlet velocities and turbulent models.

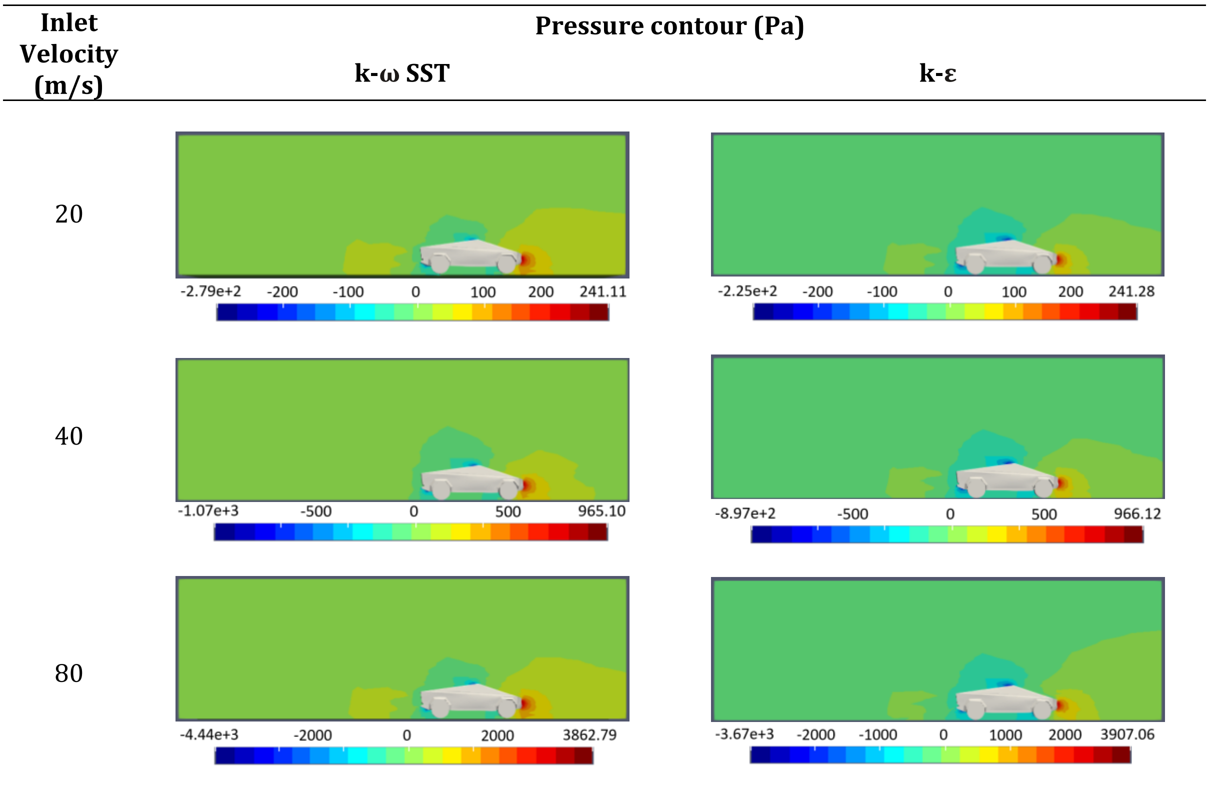


**Figure 6**: Comparison of pressure contour between different inlet velocites and turbulent models.

## 4. CONCLUSION

In conclusion, the Computational Fluid Dynamics simulation conducted on a simplified car model has provided invaluable insights into the complex dynamics of fluid flow. Utilizing Catia software, a detailed CAD model of the car was meticulously created. Subsequently, employing SimFlow 4.0 software, the airflow surrounding the simplified Tesla CyberTruck was precisely simulated. The study focused on elucidating the behaviors of maximum velocity and maximum pressure distributions under varying inlet velocities and turbulence models. The results directly correlate with increasing inlet velocity and augmented maximum velocity and pressure. Moreover, the apparent discrepancies observed among turbulence models, specifically RANS (k-ω SST) and RANS (k-ε), emphasize the critical significance of careful model selection during pre-processing. Each turbulence model manifests distinct characteristics, leading to varied simulation outcomes, a phenomenon stemming from their unique mathematical formulations. Notably, the RANS k-ω SST turbulence model emerges as a compelling choice for future CFD investigations of external aerodynamics, owing to its smoother residual convergence and heightened predictive accuracy.

## ACKNOWLEDGEMENTS

The author sincerely appreciates the invaluable support and resources the Faculty of Mechanical Engineering & Technology provided throughout this research.

## REFERENCES


[1] Hucho, W. H. (Ed.). Aerodynamics of road vehicles: from fluid mechanics to vehicle engineering. Elsevier, (2013).

[2] Koscher, K., Czeskis, A., Roesner, F., Patel, S., Kohno, T., Checkoway, S., ... & Savage, S. Experimental security analysis of a modern automobile. In 2010 IEEE symposium on security and privacy, (2010) pp. 447-462.

[3] Ahmed, A., & Murtaza, M. A. CFD Analysis of car body aerodynamics including effect of passive flow devices–A REVIEW. International Journal of Research in Engineering and Technology, vol 5, issue 3 (2016) pp. 141-144.

[4] Petrov, A. Effect of Inner Air Flow on the Aero-dynamics of the Car. Periodica Polytechnica Transportation Engineering, vol 47, issue 3 (2019) pp. 186-189.

[5] Katz, J. Aerodynamics of race cars. Annu. Rev. Fluid Mech., vol 38, (2006) pp. 27-63.

[6] Diedrichs, B., Berg, M., Stichel, S., & Krajnović, S. Vehicle dynamics of a high-speed passenger car due to aerodynamics inside tunnels. Proceedings of the Institution of Mechanical Engineers, Part F: Journal of Rail and Rapid Transit, vol 221, issue 4 (2007) pp. 527-545.

[7] Watkins, S., & Vino, G. The effect of vehicle spacing on the aerodynamics of a representative car shape. Journal of wind engineering and industrial aerodynamics, vol 96, issue 6-7 (2008) pp. 1232-1239.

[8] Damjanović, D., Kozak, D., Živić, M., Ivandić, Ž., & Baškarić, T. CFD analysis of concept car in order to improve aerodynamics. Járműipari innováció, vol 1, issue 2 (2011) pp. 108-115.

[9] Guilmineau, E. Computational study of flow around a simplified car body. Journal of wind engineering and industrial aerodynamics, vol 96, issue 6-7 (2008) pp. 1207-1217.

[10] Kotapati, R., Keating, A., Kandasamy, S., Duncan, B., Shock, R., & Chen, H. The lattice-Boltzmann-VLES method for automotive fluid dynamics simulation, a review (2009) (No. 2009-26-0057).

[11] Herrando, M., Fantoni, G., Cubero, A., Simón-Allué, R., Guedea, I., & Fueyo, N. Numerical analysis of the fluid flow and heat transfer of a hybrid PV-thermal collector and performance assessment. Renewable Energy, vol 209, (2023) pp. 122-132.

[12] Anjaneya, G., Sunil, S., Kakkeri, S., Math, M. M., Vaibhav, M. N., Solaimuthu, C., ... & Vasudev, H. Numerical simulation of microchannel heat exchanger using CFD. International Journal on Interactive Design and Manufacturing (IJIDeM), (2023) pp. 1-17.

[13] Nishidh, N. B., & Deepakkumar, R. Numerical investigation of suction and blowing effects on fluid flow and heat transfer characteristics of solar air heater. Materials Today: Proceedings, vol 72, (2023) pp. 2846-2853.

[14] Jagadale, P., & Chawdhary, A. B. Computational fluid dynamics, an overview. Int Res J Eng Technol, vol 8, (2021) pp. 1817-1821.

[15] Qi-Liang, W., Zheng, W., Xian-Liang, Z., Li-Li, L., & Zhang, Y. C. Analysis of Aerodynamic Performance of Tesla Model S by CFD. In 3rd Annual International Conference on Electronics, Electrical Engineering and Information Science (EEEIS 2017), Atlantis Press, (2017) pp. 16-21.

[16] Li, T., Zhang, J. Y., Rashidi, M. M., & Yu, M. On the Reynolds-averaged Navier-Stokes modelling of the flow around a simplified train in crosswinds. Journal of Applied Fluid Mechanics, vol 12, issue 2 (2019) pp. 551-563.

[17] Lei, L., Fei, H., Xue-Ling, C., Jin-Hua, J., & Xiao-Guang, M. Numerical simulation of the flow within and over an intersection model with Reynolds-averaged Navier–Stokes method. Chinese Physics, vol 15, issue 1 (2006) p. 149.

[18] Allah, M. Z., Hariri, A., & Mohamed Kamar, H. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, vol 101, issue 1 (2022) pp. 45-58.

[19] Das, R. C., & Riyad, M. CFD analysis of passenger vehicle at various angle of rear end spoiler. Procedia Engineering, vol 194, (2017) pp. 160-165.

[20] Srinivasarao, S., & Lakshamaih, V. M. CFD Research on Car Body. International Journal of Recent Technology and Engineering (IJRTE), vol 8, (2019) pp. 1178-1180.

[21] Maamoun, A. Elon Musk and Tesla: An Electrifying Love Affair. SAGE Publications: SAGE Business Cases Originals. (2021).
[22] Zhai, Z. J., Zhang, Z., Zhang, W., & Chen, Q. Y. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 1—Summary of prevalent turbulence models. HVAC & R Research, vol 13, issue 6 (2007) pp. 853-870.

**Conflict of interest statement:** The author declares no conflict of interest.

**Author contributions statement:** Conceptualization; Methodology; Software; Analysis; Investigation; Writing & Editing, S. Paramasivan.