ISSN 2963-7511
Journal of Engineering Researcher and Lecturer, Vol. 3, No. 2, pp. 65-72, 2024

Study of the variation of the leading edge of a railway crane on the aerodynamic characteristics

Willy Hardi Vernando[1*], Andre Kurniawan[1], Randi Purnama Putra[1] and Mirta Widia[2]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Negeri Padang, Padang, **INDONESIA**
[2] Faculty of Industrial Sciences and Technology, Malaysia Centre for Advanced Industrial Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, **MALAYSIA**

Abstract: The front body design of the railway crane is flat or bluff type. The design causes a large drag, so the railway crane requires high fuel consumption to drive. Therefore, this study aims to modify the front body of the railway crane by applying the shape of the leading edge to reduce drag. This research also investigates the effect of leading-edge angles on aerodynamic characteristics. The method used is computational fluid dynamics, using the flow simulation feature of the Solidworks research licence software. This study considered three variations of leading edge angle (40°, 45° and 50°). The simulation results show that the larger the leading edge angle, the lower the drag coefficient value. In addition, the simulation shows that there is a high air pressure at the front of the railway crane with the bluff shape, while the modified railway crane with the leading edge applied has a lower air pressure at the front. Furthermore, the results and discussion in this article present the simulation results showing the velocity streamline and pressure contour of each model.

Keywords: Industry, innovation and infrastructure; Responsible consumption and production; Simulation research; Train design

*Corresponding Author: willyhardivernando@gmail.com

Received: April 05st 2024; Revised: June 14th 2024; Accepted: July 04th 2024

https://doi.org/10.58712/jerel.v3i2.145

Reference to this paper should be made as follows: Vernando, W. H., Kurniawan, A., & Putra, R. P. Study of the variation of the leading edge of a railway crane on the aerodynamic characteristics. *Journal of Engineering Researcher and Lecturer*, 3(2), 65–72. https://doi.org/10.58712/jerel.v3i2.145

1. Introduction

Railway cranes are a type of train used to carry out railway repairs. Railway cranes are equipped with a crane behind the cab and are classified as low-speed repair trains, capable of speeds of up to 100 km/h. The design of the front of the railway crane is of the flat or bluff type. This design creates a large air resistance. The large air resistance affects the movement of the railway crane, which slows it down, causing it to consume more power. Power is directly proportional to fuel consumption, so the more power, the more fuel consumption.

Research has been published on the optimisation of train body design, focusing on passenger or high-speed trains. The previous research investigated the unsteady flow structure around Next-Generation Trains (NGT) under crosswind conditions using the Computational Fluid Dynamics (CFD) method by modifying the front part of the NGT (Arafat & Ishak, 2022). The drag of a high-speed train was reduced by modifying the nose section (Cheng et al., 2018). The more aerodynamic a vehicle is, the smoother it moves because the wind resistance is reduced (Luo et al., 2023).

However, no research has been reported on optimising the car body design of railway cranes. Therefore, this study is a novelty that fills the research gap on railway cranes. In this study, drag reduction is achieved by modifying the cabin body of the railway crane through the application of a leading edge. The application of leading-edge for drag reduction is generally performed on buses (Gan et al., 2020; Nath et al., 2021; Ragavan et al., 2014), so the study focusing on railway cranes is the latest innovation. The modification of the railway crane cabin is done because the cabin requires a large space for the crew, considering that the rear part is a crane. This study not only presents the drag coefficient profiles of each railway crane design with modifications to the leading edge but also presents the airflow characteristics based on velocity contour, velocity streamline and pressure contour.

2. Material and methods

Three methods can be used to test changes in railway car body design: wind tunnel testing, computational fluid dynamics (CFD) and road testing (Katz, 2006). The CFD method has advantages in terms of speed of analysis, no need for costly prototyping and can visualise the characteristics of the fluid flow that occurs. This research uses the CFD method to see the effect of modifying the front part of the railway crane. The CFD method in this research uses Solidworks Research Licence 2021-2022.

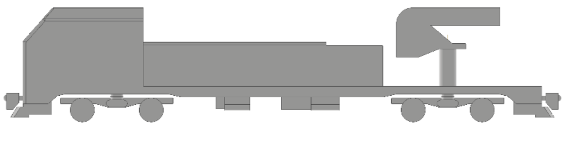
2.1 Design of railway crane

The railway crane is designed to a scale of 1:25 of the actual size (Table 1). The design of the railway crane with leading-edge variation is shown in Figure 1.

Table 1. Dimension of railway crane

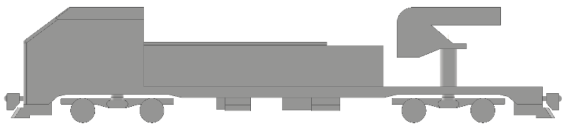
	Length	Hight	Width
Dimension railway crane	17,590.04 mm	3,660.39	2,435.95 mm
Prototipe for simulation	703.601 mm	146.42 mm	96.44 mm

(a)



(b)

(c)



(d)

Figure 1. (a) the standard model of railway crane, (b) Leading Edge 40°, (c) Leading Edge 45° dan (d) Leading Edge 50°

2.2 Simulation setup

The simulation setup parameters are shown in Table 2. The computational domain used for the railway crane simulation is shown in Figure 2. The distance between the railway crane and the wall was set based on the height (H) of the train (Iwnicki, 2006). The distance from the front wall boundary to the front nose of the railway crane was 2H and the distance from the rear of the railway crane to the end wall boundary was 4H. The distance between the left and right side walls of the railway crane and the left and right side wall boundary was 1.6H. The distance between the top of the crane and the top of the wall was 1.6H.

Table 2. Parameter state of the simulation

Analysis type	External analysis
Gravity	Y component: -9.81 m/s^2
Default fluid	Air
Initial condition	- Velocity in X direction - Velocity 1.11 m/s - Wall: No-slip condition

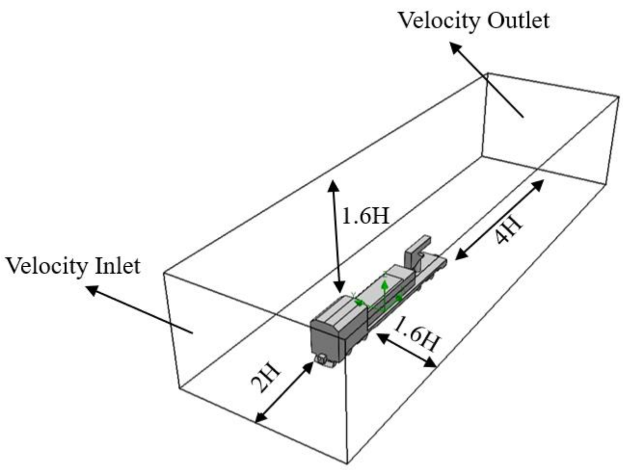


Figure 2. Computational domain

2.3 Meshing

Based on the literature review conducted, the authors did not find any research that discusses the results of drag tests on railway cranes, either experimentally or using CFD methods. Research on NGT using the CFD method uses a number of cells of 1.2 million (Arafat et al., 2023; Arafat & Ishak, 2022), this number of cells is validated with experimental research data (Sun et al., 2021). In this study, the number of cells is 1.2 million with three layers. The fluid close to the railway crane body uses fine cells, the second layer is medium and the third grid is large (Figure 3). The use of local regions for this mesh is commonly done in CFD studies on trains (Fragner & Deiterding, 2018; Sun et al., 2021; Weinman et al., 2018). Some studies relevant to this CFD method state that the finer the mesh size used, the closer the simulation results are to the experimental results, but it increases the computational time required (Aldio et al., 2023; Salmat et al., 2023; Volk et al., 2018).

ISSN 2963-7511
Journal of Engineering Researcher and Lecturer, Vol. 3, No. 2, pp. 65-72, 2024

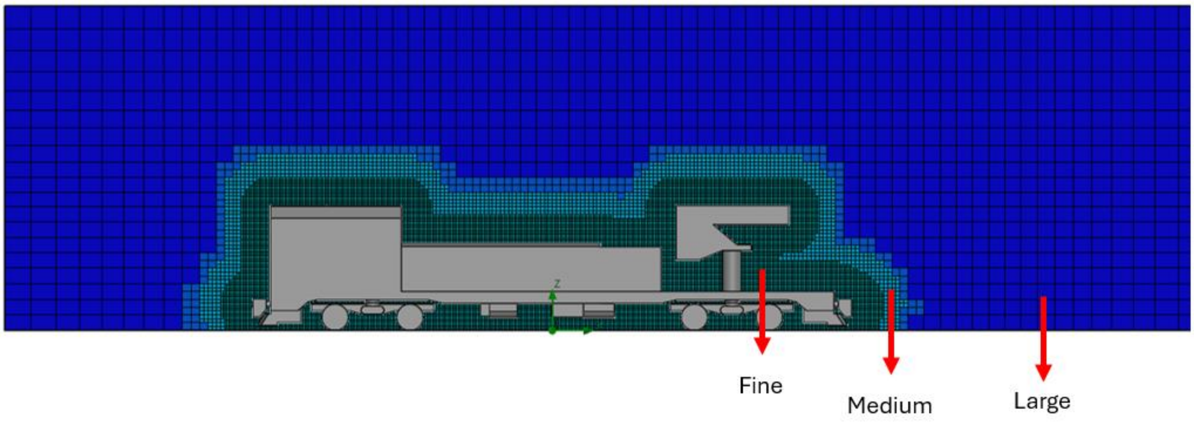


Figure 3. Meshing

3. Results and discussion

The drag coefficients of the CFD simulation results of the four railway crane models without a leading edge and with the application of a 40°-50° leading edge are shown in Figure 5.

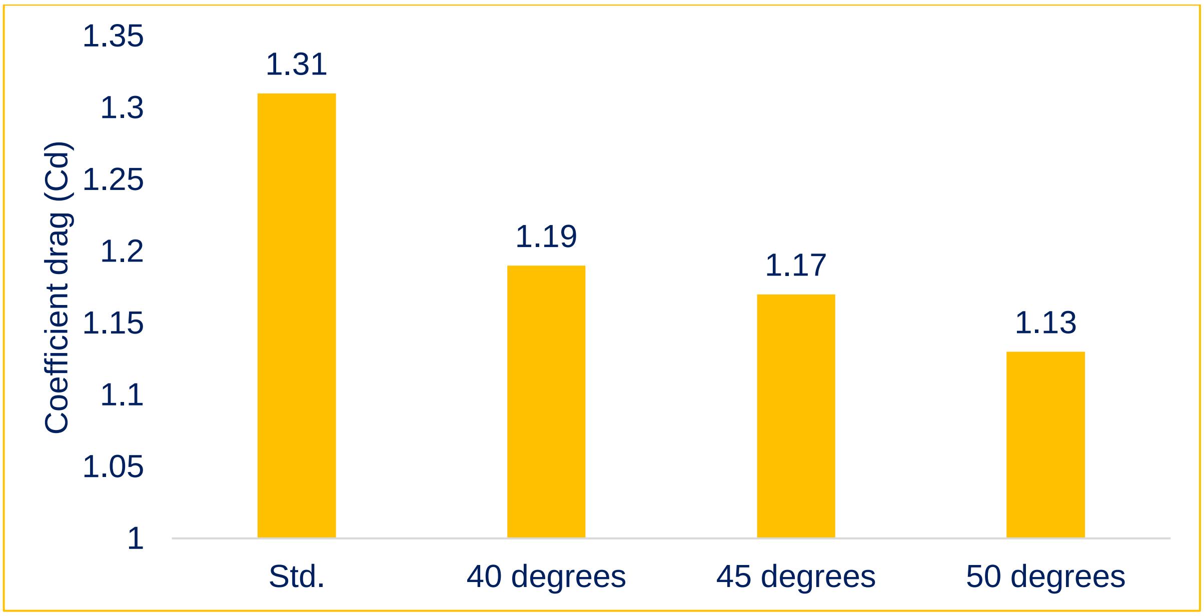


Figure 5. Effect of the leading edge on the drag coefficient

The simulation results show that the use of the leading edge can reduce the Cd value of the railway crane during drive. The railway crane without a leading edge has a Cd value of 1.31. However, when a 40° leading edge is used, the Cd value drops by 9.06%. Similarly, when 45° and 50° leading edges are used, the Cd value drops by 10.68% and 13.74% respectively. These simulation results show that the larger the leading edge angle, the lower the Cd value.

The velocity streamline visualisation of each railway crane model is shown in Figure 6. The standard crane model experiences flow separation from the top of the cab to the rear of the cab, in the area under the railway crane and at the bottom. Meanwhile, the modified railway crane model experiences flow separation in the area behind the cab, under the railway crane and at the bottom. This flow separation is a condition where air flows along the surface of an object and can no longer adhere to the surface of the object, resulting in a decrease in pressure distribution and a pressure difference between the area where flow separation occurs and the other areas (Camp & Figliola, 2018; White, 2010).

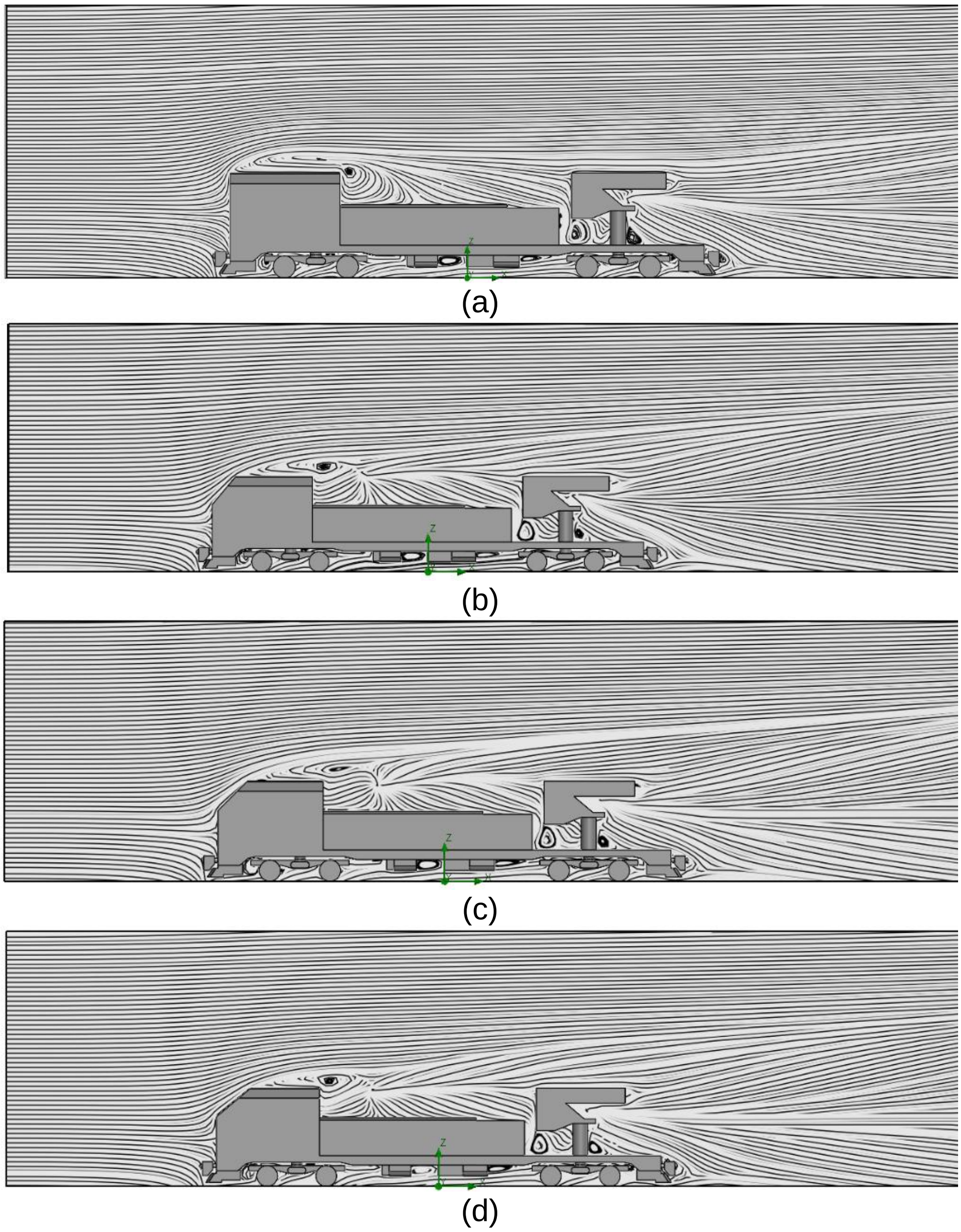
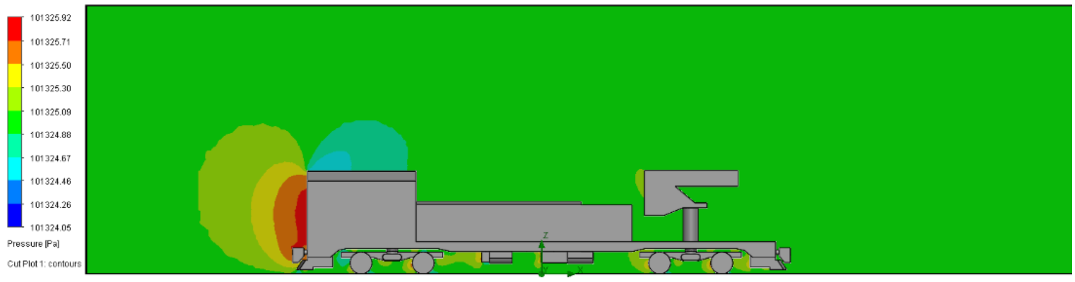
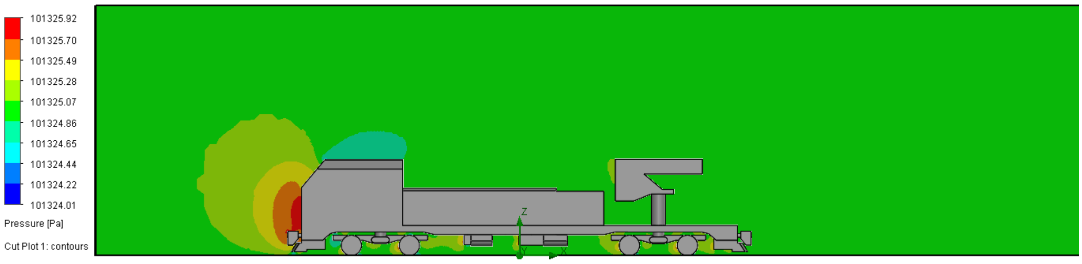


Figure 6. Velocity Streamline: (a) Standard railway crane (b) Leading Edge 40° (c) Leading Edge 45° and (d) Leading Edge 50°

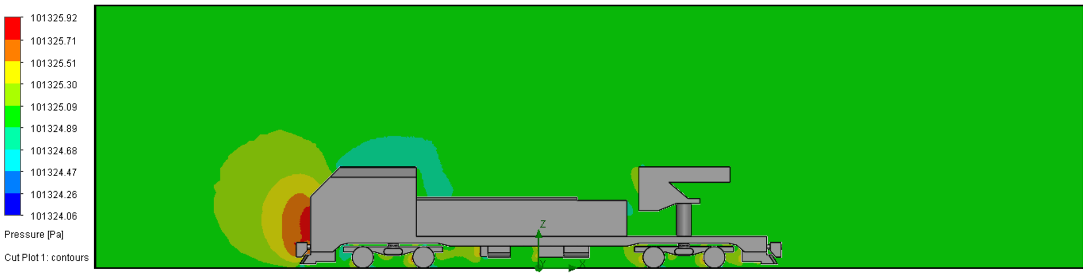
Based on the pressure contour shown in Figure 7. It can be seen that in the standard model of the railway crane, the pressure occurs in the front part of the train which collides directly with the air. The pressure that occurs in the standard model of the railway crane is the highest pressure that can be seen in Figure (a), shown in red because it has a flat and large area compared to the model of the railway crane that is varied. On the other hand, in the modified crane model, the angle of the leading edge can reduce the pressure that occurs because the area of air impact on the modified crane model is reduced. So with the reduced pressure that occurs, the drag that occurs on the tow can be minimised.



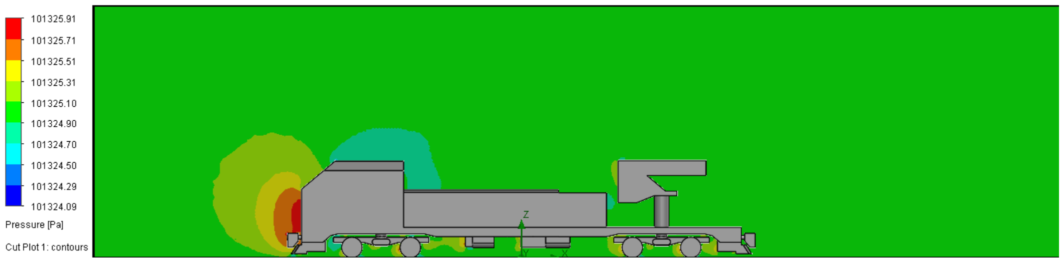
(a)



(b)



(c)



(d)

Figure 6. Pressure contour: (a) Standard railway crane (b) Leading Edge 40° (c) Leading Edge 45° and (d) Leading Edge 50°

4. Conclusion

Based on the CFD simulation results, the Cd value of the standard railway crane is 1.31, at 40° leading edge inclination is 1.18, at 45° leading edge inclination is 1.16, at 50° leading edge inclination is 1.13. The application of varying the slope of the leading edge angle on the standard railway crane model affects the resulting Cd value. The most optimal Cd value reduction is produced at a leading edge angle slope of 55° because the resulting Cd value is the smallest compared to the others, although the difference in the value of the resulting Cd is not much different. Flow characteristics based on pressure contour on the standard and modified railway crane there is stagnation at the front of the railway crane, the largest stagnation occurs on the standard railway crane. In the modified railway crane, the stagnation that occurs is reduced due to the modification of the leading edge tilt angle applied. Likewise, the backflow that occurs on the railway crane is reduced due to the modification of the leading edge tilt angle shown in the velocity streamline.

Acknowledgements

The authors would like to thank the research team at the Manufacturing Laboratory, Universitas Negeri Padang for their assistance during the research.

Declarations

Author contribution

Willy Hardi Vernando: Conceptualization, methodology, formal analysis, software and writing - original draft. Andre Kurniawan and Randi Purnama Putra: Methodology, Validation, data curation and writing – review and editing. Mirta Widia: Methodology, Validation, data curation and writing – review and editing.

Funding statement

This research has not been funded by any person or organisation.

Competing interest

The authors declare no conflict of interest in this study.

Ethical Clerance

There are no human subjects in this manuscript and informed consent is not applicable.

References

Aldio, M. F., Waskito, W., Purwantono, P., & Lapisa, R. (2023). Optimization of impeller blade number in centrifugal pump for crude oil using Solidworks Flow Simulation. *Journal of Engineering Researcher and Lecturer*, *2*(3), 80–93. https://doi.org/10.58712/jerel.v2i3.116

Arafat, M., & Ishak, I. A. (2022). CFD Analysis of the Flow around Simplified Next-Generation Train Subjected to Crosswinds at Low Yaw Angles. *CFD Letters*, *14*(3), 129–139. https://doi.org/10.37934/cfdl.14.3.129139

Arafat, M., Ishak, I. A., Mohammad, A. F., Khalid, A., Jaát, Md. N. M., & Yasak, M. F. (2023). Effect of Reynolds number on the wake of a Next-Generation High-Speed Train using CFD analysis. *CFD Letters*, *15*(1), 76–87. https://doi.org/10.37934/cfdl.15.1.7687

Camp, T., & Figliola, R. (2018). Fluid Mechanics. In *Mechanobiology Handbook: Second Edition*. https://doi.org/10.1201/9780429444982-2

Cheng, H., Zhou, L. J., & Zhao, Y. Z. (2018). Very large eddy simulation of swirling flow in the Dellenback abrupt expansion tube. *IOP Conference Series: Earth and Environmental Science*, *163*, 012084. https://doi.org/10.1088/1755-1315/163/1/012084

Fragner, M. M., & Deiterding, R. (2018). *Investigating Side-Wind Stability of High Speed Trains Using High Resolution Large Eddy Simulations and Hybrid Models* (pp. 223–241). https://doi.org/10.1007/978-3-319-54490-8_14

Gan, E. C. J., Fong, M., & Ng, Y. L. (2020). CFD analysis of slipstreaming and side drafting techniques concerning aerodynamic drag in nascar racing. *CFD Letters*, *12*(7), 1–16. https://doi.org/10.37934/cfdl.12.7.116

Iwnicki, S. (2006). Handbook of railway vehicle dynamics. In *Handbook of Railway Vehicle Dynamics*. https://doi.org/10.1201/9781420004892

Katz, J. (2006). Aerodynamics of race cars. In *Annual Review of Fluid Mechanics* (Vol. 38). https://doi.org/10.1146/annurev.fluid.38.050304.092016

Luo, J., Li, M., Chen, X., Ye, L., Xu, H., Liu, Z., Tie, Y., Xu, S., & Jiang, C. (2023). Review on aerodynamic characteristics and energy recovery of vehicle platoon. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*. https://doi.org/10.1177/09544070231204104

Nath, D. S., Pujari, P. C., Jain, A., & Rastogi, V. (2021). Drag reduction by application of aerodynamic devices in a race car. *Advances in Aerodynamics*, *3*(1). https://doi.org/10.1186/s42774-020-00054-7

Ragavan, T., Palanikumar, S., Anastraj, D., & Arulalagan, R. (2014). Aerodynamic Drag Reduction on Race Cars. *Journal of Basic and Applied Engineering Research*, *1*(4).

Salmat, S., Yanti Sari, D., Fernanda, Y., & Prasetya, F. (2023). SolidWorks Flow Simulation: Selecting the optimal mesh for conducting CFD analysis on a centrifugal fan. *Journal of Engineering Researcher and Lecturer*, *2*(3), 94–103. https://doi.org/10.58712/jerel.v2i3.104

Sun, Z., Yao, S., Wei, L., Yao, Y., & Yang, G. (2021). Numerical Investigation on the Influence of the Streamlined Structures of the High-Speed Train's Nose on Aerodynamic Performances. *Applied Sciences*, *11*(2), 784. https://doi.org/10.3390/app11020784

Volk, A., Ghia, U., & Liu, G. R. (2018). Assessment of CFD-DEM solution error against computational cell size for flows through a fixed-bed of binary-sized particles. *Powder Technology*, *325*, 519–529. https://doi.org/10.1016/j.powtec.2017.11.051

Weinman, K. A., Fragner, M., Deiterding, R., Heine, D., Fey, U., Braenstroem, F., Schultz, B., & Wagner, C. (2018). Assessment of the mesh refinement influence on the computed flow-fields about a model train in comparison with wind tunnel measurements. *Journal of Wind Engineering and Industrial Aerodynamics*, *179*, 102–117. https://doi.org/10.1016/j.jweia.2018.05.005

White, F. (2010). F_ White Fluid Mechanics 2009. *McGraw-Hill,New York*.