SciEn

Proceedings of 8th International Conference on Mechanical Engineering and Renewable Energy 2025
10-12 December, 2025, Chattogram, BANGLADESH

SciEn Conference Series: Engineering Vol. 4, 2026, pp 295-300 https://doi.org/10.38032/scse.2026.4.139

Effectiveness of Drag Reduction Device on Simplified Vehicle Model: A CFD Study

Md. Bulbul Ahmed Rubel, Prasanjit Das, and Itquan Hossen*

Department of Mechanical Engineering, Chittagong University of Engineering & Technology, Chattogram-4349, Bangladesh

ABSTRACT

Aerodynamic drag significantly influences vehicle performance and fuel efficiency, making its reduction a key focus in automotive design. This study examines the drag reduction of the simplified Ahmed body model by integrating diffusers at 8° and 10° angles. Using Computational Fluid Dynamics (CFD) simulations, airflow analysis was conducted on Ahmed bodies with 25° and 30° slant angles, both in baseline and modified configurations. The results show that diffuser-equipped models achieved a notable reduction in drag coefficient compared to the original design and previous literature. This improvement is primarily due to minimized turbulence in the wake region, as the diffuser slows airflow and enhances pressure recovery, thereby lowering drag. Both 25° and 30° slant configurations demonstrated substantial aerodynamic benefits with diffuser integration, with greater improvements observed at higher slant angles. Overall, the study highlights the effectiveness of diffusers in enhancing aerodynamic efficiency, offering valuable insights for vehicle design aimed at improving fuel economy, performance, and sustainability.

Keywords: Aerodynamic Drag, Wake Region, Diffuser, Turbulence, Optimization.

1. Introduction

In earlier times, vehicle design was primarily influenced by factors such as aesthetics, operational safety, and service accessibility, with aerodynamics receiving relatively little attention [1]. Efficient transportation was not a primary concern. However, in today's fast-paced world, the pursuit of efficient transportation has become a major focus for engineers and researchers. Rising energy demands and environmental challenges have made aerodynamic optimization crucial for improving vehicle performance and fuel efficiency. Drag reduction devices have emerged as an effective means to minimize aerodynamic resistance, leading to better fuel economy, reduced carbon emissions, and improved vehicle handling. Since a vehicle's fuel consumption largely depends on the aerodynamic drag generated by its shape, many researchers have focused on modifying vehicle geometries. A notable example is the work by S. R. Ahmed et al., who developed the Ahmed body, illustrated in Fig. 1.

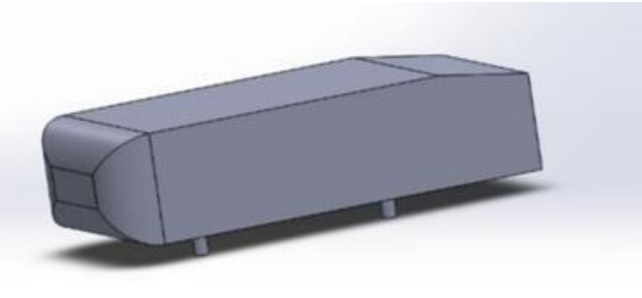


Fig. 1: Ahmed Body with 10 degree slant angle

Their experiments on varying slant angles revealed that the maximum aerodynamic drag occurs due to pressure drag at the rear of the model [1]. To mitigate this, the original Ahmed body has been modified in multiple ways. Sagar et al. performed geometric modifications on the Ahmed body and observed significant reductions in both drag and lift forces [2]. Other studies have explored the addition of devices such as spoilers. Cheng et al., for example, investigated the Ahmed model equipped with a spoiler and found that both lift and drag coefficients increased with higher yaw angles [3]. Harun Chowdhury et al. reported that poorly designed deflectors in countries like Bangladesh and Pakistan could increase drag by up to 22%, whereas aerodynamically optimized designs could reduce drag, fuel consumption, and greenhouse gas emissions [4]. Additionally, numerical simulations by N.A. Siddiqui and M.A. Chaab demonstrated that adding a flap at a 10° angle on the Ahmed body reduced drag by 14% by delaying flow separation, enhancing pressure recovery, and minimizing wake turbulence [5].

1.1 Problem Statement And Objectives Of The Study

Despite extensive advancements in vehicle aerodynamics, drag reduction remains a critical challenge in improving energy efficiency and overall performance. The Ahmed body, a widely used benchmark model for automotive aerodynamic studies [7], provides an ideal platform to investigate the influence of geometric modifications on flow behavior and drag characteristics. However, the effect of diffuser configurations on the aerodynamic performance of the Ahmed body has not been fully optimized [8, 9]. In particular, the relationship between diffuser angle, flow separation, and pressure distribution requires deeper understanding to achieve meaningful drag reduction [10, 11].

The primary objective of this study is to investigate the influence of various diffuser configurations on the aerodynamic performance of the Ahmed body, with a particular focus on reducing its drag coefficient. The research aims to analyze how different diffuser designs alter the airflow characteristics around the model, thereby affecting overall aerodynamic behavior. Through detailed computational analysis, the study seeks to identify optimal diffuser parameters that contribute to significant drag reduction, ultimately supporting the development of more aerodynamically efficient and sustainable vehicle designs with enhanced performance.

1.2 General Information About The Study

In this study, the drag on the modified Ahmed body was observed. Drag is a force exerted by a fluid (liquid or gas) on an object moving through it, opposite to the direction of the object's motion. When an object goes through a fluid, it creates a vacuum space behind it. In this vacuum space low pressure is created as a result the object going through the fluid experiences a backward force. This backward force is simply named as drag force. The drag force can be calculated using the Eq. (1). [6]

$$F_d = \frac{1}{2} C_d \rho v^2 A \tag{1}$$

A diffuser works based on Bernoulli's Principle. When the air enter the diffuser the velocity remain high equal as free stream velocity. As the air passes through the diffuser the velocity decreases with the traversed distances as a consequences the pressure increases at the rear end of the model and this help to reduce the backward force known as drag. This can be expressed by Eq. (2).

$$p + (\tfrac{1}{2})\rho v^2 + \rho g h = \text{Constant} \tag{2}$$

1.3 Computatioal Modeling

The CFD simulations were conducted using ANSYS FLUENT 2024R1. In this study Reynolds Averaged Navier Stokes (RANS) equation was used in the ANSYS Software. This equation involves Continuity equation, X momentum equation, Y momentum equation, Z momentum equation.
The Reynolds Averaged Navier Stokes (RANS) equation is given by Eq. (3).

$$\frac{\partial}{\partial x_j}\left(\overline{\rho u_i u_j}\right) = -\frac{\overline{\partial p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\overline{\partial u_i}}{\partial x_j} + \frac{\overline{\partial u_j}}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\overline{\partial u_i}}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}\left(-\overline{\rho u_i u_j}\right) \tag{3}$$

2. Research Methodology

The study employs Computational Fluid Dynamics (CFD) simulations using ANSYS Fluent, beginning with accurate geometry modeling in SOLIDWORKS, followed by structured mesh generation and finite volume discretization. Three-dimensional flow simulations are conducted using the control volume method to precisely evaluate aerodynamic behavior. After simulation, ANSYS Fluent facilitates comprehensive post-processing, including the generation of graphs and charts for detailed visualization.

This streamlined methodology provides an effective framework to assess the impact of diffusers on drag coefficients in various modified Ahmed body configurations.

2.1 Simplified Model and Target Models

The dimensions of the Ahmed body were sourced from the journal published by Ahmed et al. This was done to ensure that accurate and realistic measurements were incorporated, thus providing a comprehensive presentation of solutions and analysis. The dimensions of Ahmed body are shown in Fig. 2.

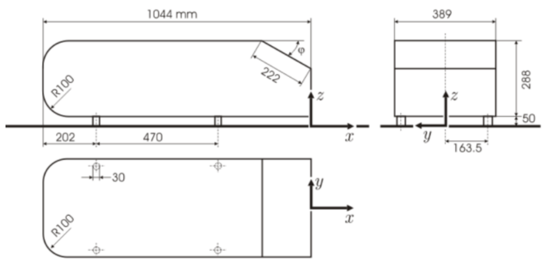


Fig. 2: Dimensions of Ahmed Body [1]

In the modified model the rear end was equipped with a diffuser. Ahmed body with slant angle 25 degree and 30 degree was modified by using diffuser installed at 8 degree and 10 degree. The target model with 25 degree slant angle with diffuser at 8 degree and 10 degree is shown in Fig. 3 and Fig. 4.

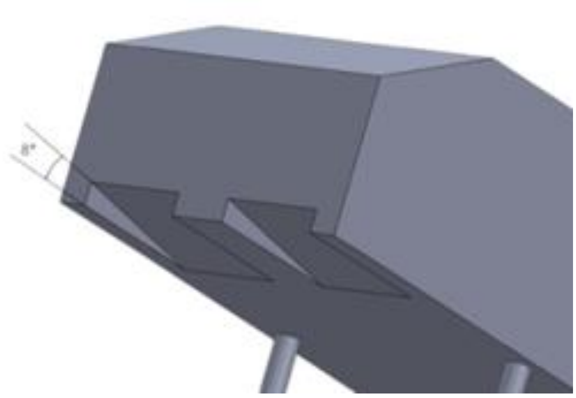


Fig. 3: Ahmed body with 25 degree slant angle and diffuser at 8 degree

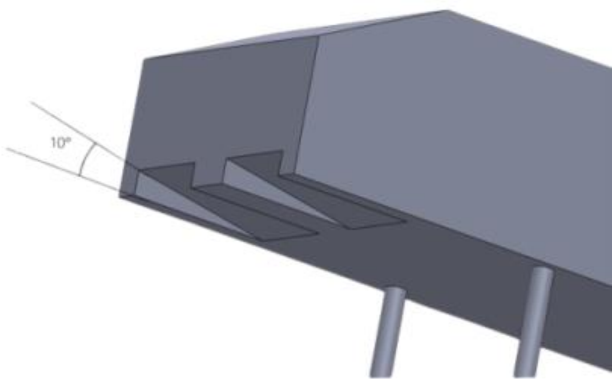


Fig.4: Ahmed body with 25 degree slant angle and diffuser at 10 degree

2.2 Preparing The Simulation Environment

After generating the modified model in SOLIDWORKS software, the model was imported in the ANSYS. In ANSYS, the model was first taken to the design modeler. In the design modeler, a computational domain was designed around the Ahmed body. The dimensions and figure of the computational domain is given in Table 1.

Table 1 Dimension of computational domain

Dimension	Value (m)
Length	4m
Width	0.6m
Height	0.8m

A body of influence was made around the Ahmed body to get the result more accurately. This is done because the flow region around the Ahmed body is more significant as it contribute to the drag force. The dimension and figure of body of influence is shown in the Table 2.

Table 2 Dimension of Body of influence

Dimension	Value (m)
Length	2.7m
Width	0.22m
Height	0.6m

The computational domain and the body of influence is shown in the Fig. 5.

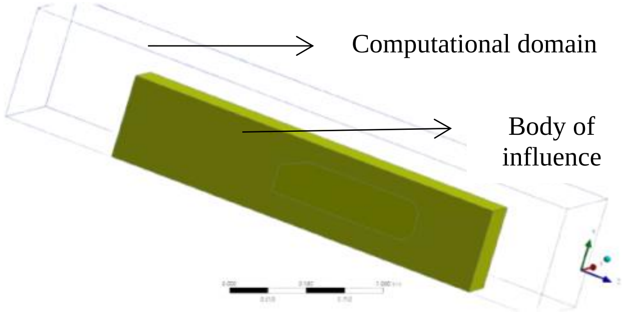


Fig. 5**:** Computational Domain and Body of influence

After this the mesh was generated. There was three kinds of mesh size. The mesh size was small in the body of influence as the adjacent region of the body is more significant for measuring the drag. The faces of the body were also meshed to get more accurate result. The element size of the three type of meshing is given in the Table 3.

Table 3 Mesh Sizing

Type of Mesh	Element size
Global Size	50mm
Body of influence	22mm
Face sizing	10mm

The figure of the different type of mesh is shown in the Fig. 6.

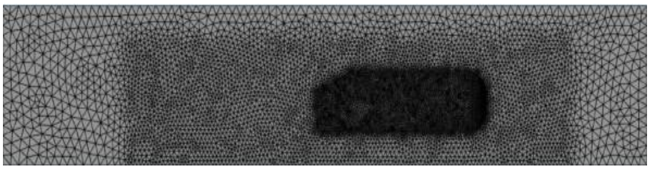


Fig.6: Type of Mesh

An inflation layer was created around the faces of the Ahmed body. Between the inflation layers structured mesh was created. It helps to catch the flow around the body which helps to get the accurate result. In the simulation there 20 inflation layers was made and the first layer thickness was 7×10^{-3} mm. The figure of the inflation layer is shown in the Fig. 7.

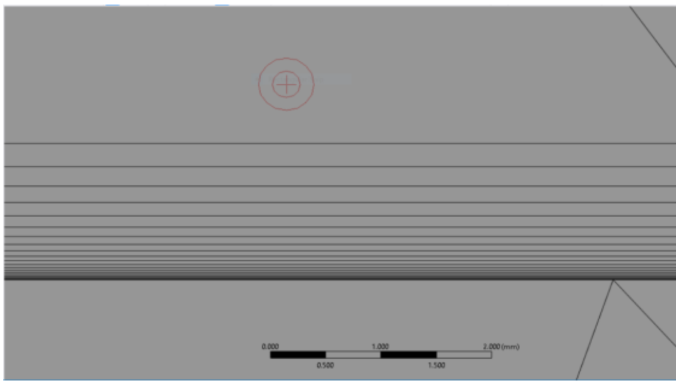


Fig.7: Inflation Layer

After completing the meshing, the case was set up in the ANSYS Fluent. The setup done in the ANSYS Fluent is shown in the Table 4.

Table 4 Physical Properties and Condition

Physical properties	Parameter
Fluid	Air
Flow Characteristics	Incompressible and Turbulent
CFD Simulation	3D
Solver Type	Pressure Based
Turbulent Model	SST k Omega
Inlet Velocity	40m/s
Temperature	288K
Kinematic Viscosity	1.7894×10^{-5} Kg/(ms)
Density	1.225 Kg/m^3

3. Result and Discussion

3.1 Result

After setting up the environment in the ANSYS FLUENT properly, the simulation was run and necessary result was obtained. It was found that the drag coefficient was reduced due to the installation of the diffuser. The results of the simulation are given in the Table 5 and Table 6.

Table 5 Comparison of result Ahmed Body with 25 slant angle with diffusers

Base Model	**Base model with 25 degree slant angle**	**Base model with 25 degree slant angle**
Diffuser Angle	8 Degree	10 Degree
Drag coefficient of base model (literature value)	0.2875	0.2875
Drag coefficient of base model (Simulated value)	0.279	0.279
Drag Coefficient with Diffuser	0.26160768	0.25581866
Reduction of drag coefficient (Compared to literature value)	9%	11%
Reduction of drag coefficient(Compared with simulated value)	6.23%	8.30%

Table 6 Comparison of result Ahmed Body with 30 slant angle with diffusers

Base Model	**Base model with 30 degree slant angle**	**Base model with 30 degree slant angle**
Diffuser Angle	8 Degree	10 Degree
Drag coefficient of base model (literature value)	0.378	0.378
Drag coefficient of base model (Simulated value)	0.285	0.285
Drag Coefficient with Diffuser	0.26519335	0.27331931
Reduction of drag coefficient (Compared to literature value)	29.85%	27.69%
Reduction of drag coefficient(Compared with simulated value)	6.94%	4.09%

3.2 Discussion

As we know that drag force is created due to the vacuum created at the wake region, we need to reduce the vacuum region for reducing the drag force. From the simulation this is also proved. The wake region or turbulent region reduces for the models equipped with diffuser installed at 8 degree and 10 degree.

This incident is shown in the Fig. 8, Fig. 9, and Fig. 10 for Ahmed body with 25 degree slant angle and diffuser at 8 degree and 10 degree.

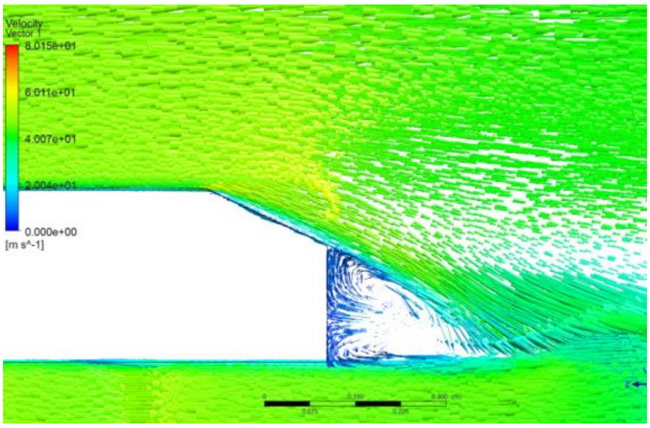


Fig. 8: Wake region behind Ahmed body with 25 degree slant angle

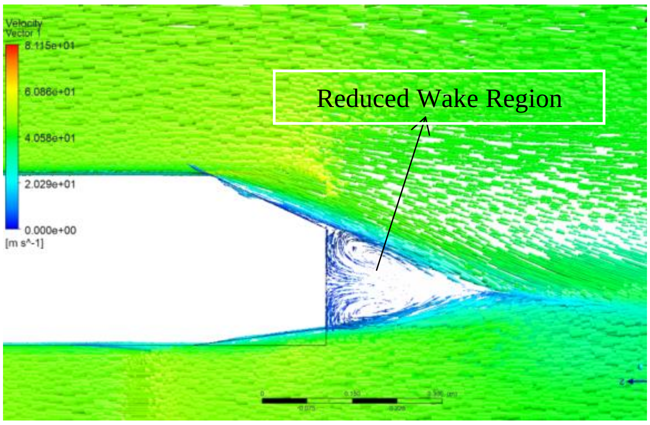


Fig. 9: Wake region behind Ahmed body with 25 degree slant angle and diffuser at 8 degree

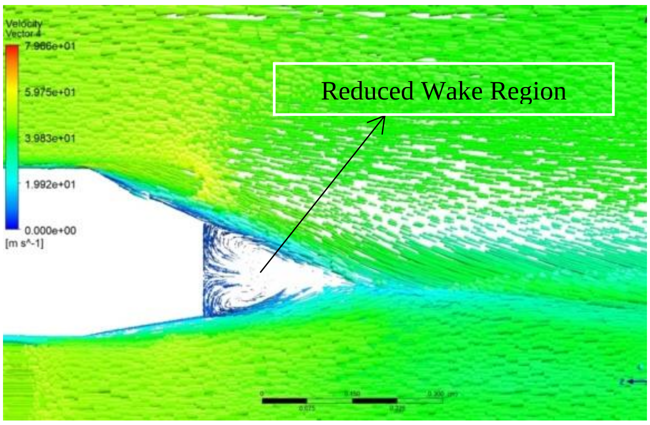


Fig.10: Wake region behind Ahmed body with 25 degree slant angle and diffuser at 10 degree

This scenario is also observed for the model with 30 degree slant angle and diffuser.

To visualize the effect more clearly three plane was taken in the rear wake region at different distances. In these planes we observed the velocity vectors to understand the flow characteristics. The positions of the planes are shown in Fig. 11. The velocity vectors on the plane for the model with 25 degree slant angle is shown in the Fig. 12. The velocity vectors for the models equipped with diffuser at 8 degree and 10 degree is shown in Fig. 13 and Fig, 14.

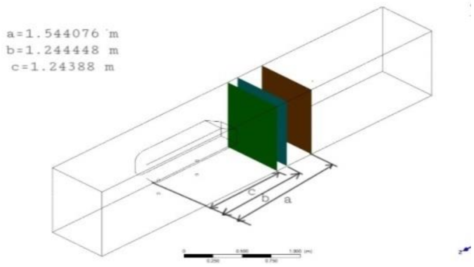
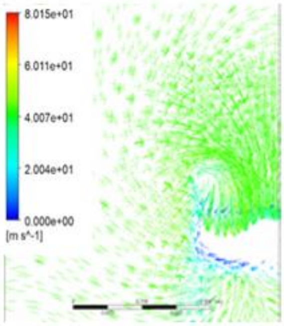
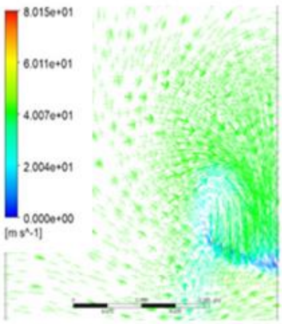
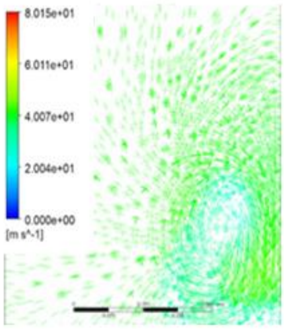


Fig. 11: Position of the planes



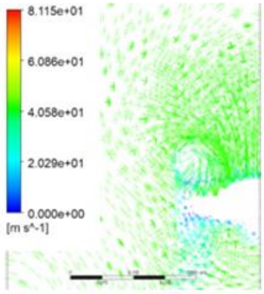


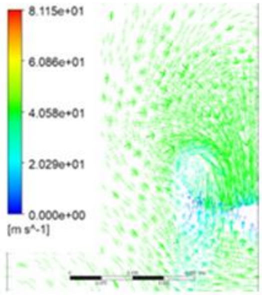
Plane'c' Plane'b'



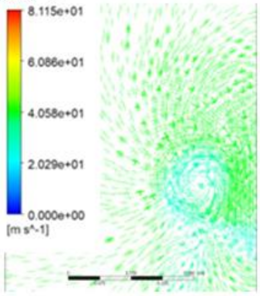
Plane'a'

Fig. 12: Velocity Vector in specified planes for Ahmed body with 25 degree slant angle



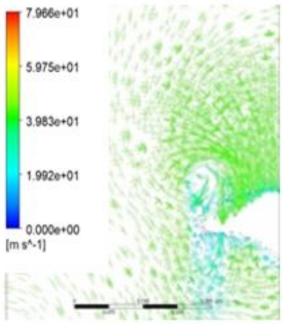


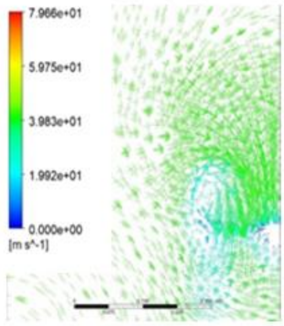
Plane 'c' Plane 'b'



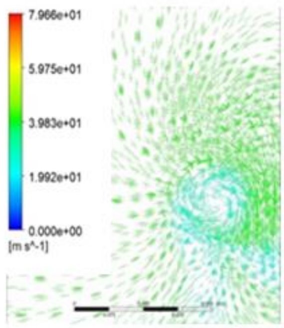
Plane'a'

Fig. 13: Velocity Vector in specified planes for Ahmed body with 25 degree slant angle and diffuser at 8 degree





Plane 'c' Plane 'b'



Plane 'a'

Fig. 14: Velocity Vector in specified planes for Ahmed body with 25 degree slant angle and diffuser at 10 degree

From the above figures, it is clear that in Plane ‘a’, the installation of a diffuser significantly reduced the vortex formation behind the model. In contrast, the base model exhibits a large and intense vortex with pronounced reverse flow and low-velocity circulation, which corresponds to higher pressure drag. When diffusers are applied, these effects become less dominant, resulting in a noticeable reduction in drag.

The diffuser plays a crucial role in reducing aerodynamic drag by decelerating the airflow, which consequently increases the static pressure at the rear end of the vehicle model. This rise in pressure minimizes the pressure differential between the front and rear regions, thereby reducing the overall pressure drag acting on the model. This phenomenon was consistently observed across all models investigated in the study. A comparison of the velocity vector plots for the base model and the diffuser-equipped models across three different planes revealed a significant reduction in flow velocity in the rear region of the latter. According to Bernoulli’s principle, this reduction in velocity corresponds to an increase in static pressure in the downstream flow, leading to a noticeable decrease in drag force.

4. Conclusion

The investigation into the effectiveness of drag reduction device like diffuser on the Ahmed body has provided valuable insights into the complex dynamics of aerodynamic drag and the potential for mitigating its effects. Through comprehensive experimentation and analysis, this study has shed light on the performance of diffuser in altering the flow characteristics around the Ahmed body and consequently reducing its drag coefficient.

From the study it was found that Ahmed body with 25 degree slant angle has 9% and 11% less drag coefficient when equipped with diffuser at 8 degree and 10 degree at the rear end. Same result was found for Ahmed body with 30 degree slant angle. The drag coefficient reduced by 29.85% and 27.69% percent when Ahmed body was equipped with diffuser at the same angle at the rear end. This reduction was calculated with respect to the literature value. Then the results was compared with simulation of the base model the reduction of drag was also observed. This reduction was 6.23% and 8.30% for ahmed body with 25 degree slant angle equipped with diffuser at 8 degree and 10 degree respectively. For Ahmed body with 30 degree slant angle equipped with diffuser at 8 degree and 10 degree, the drag coefficient was decreased by 6.94% and 4.09% respectively. The reduction was observed due to reduced turbulence, flow separation and increase of pressure at the wake region.

These findings corroborate previous research while also offering new perspectives on the efficacy of drag reduction device in real-world applications. Furthermore, the experimental data presented in this study provide a solid foundation for future research endeavors in this field.

The findings presented in this study underscore the effectiveness of drag reduction devices in improving the aerodynamic performance of the Ahmed body. By leveraging these insights, researchers and practitioners can advance the development of more efficient and sustainable transportation systems in the pursuit of a greener and more technologically advanced future.

5. References

[1] Ahmed, S.R., Ramm, G. and Faltin, G. (1984) *Some salient features of the time-averaged ground vehicle wake*. SAE Technical Paper Series, February.

[2] Sharma, S., Banga, S., Dungriyal, R., Zunaid, M., Ansari, N. and Lal, S. (2015) ‘CFD analysis and optimization of geometrical modifications of Ahmed body’, *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, 12(5), pp. 37–49.

[3] Cheng, S., Mansor, S., Abdullah, M. and Zakaria, M.S. (2018) ‘Numerical study of aerodynamic performance of hatchback vehicle fitted with a strip spoiler: Effect of yaw angle’, *Journal of Mechanical Engineering*, 47, pp. 108–118.

[4] Chowdhury, H., Juwono, R., Zaid, M., Islam, R., Loganathan, B. and Alam, F. (2019) ‘An experimental study on the effect of various deflectors used for light trucks in Indian subcontinent’, *Energy Procedia*, 160, pp. 34–39.

[5] *Journal of Applied Fluid Mechanics* (2021) ‘A simple passive device for the drag reduction of an Ahmed body’, 14(1), January.

[6] Çengel, Y.A. and Cimbala, J.M. (2018) *Fluid Mechanics: Fundamentals and Applications*. 4th edn. New York: McGraw-Hill Education.

[7] Buscariolo, F.F., Assi, G.R.S. and Sherwin, S.J. (2020) ‘Computational study on an Ahmed body equipped with simplified underbody diffuser’, *arXiv preprint*. Available at: https://arxiv.org/abs/2001.11074 (Accessed: day month year).

[8] Ahmadinejad, M.A., Mirshekari, A.S. and Omidvar, A.H.M. (2018) ‘Numerical and experimental investigations on aerodynamic behavior of the Ahmed body model with different diffuser angles’, *Journal of Applied Fluid Mechanics*, 11(4), pp. 915–926.

[9] Bayraktar, S. and Bilgili, Y.O. (2018) ‘Effects of under body diffuser on the aerodynamics of a generic car’, *International Journal of Automotive Engineering and Technologies*, 7(2), pp. 99–109. Available at: https://dergipark.org.tr/en/pub/ijaet/article/458901 (Accessed: day month year).

[10] Aulakh, D.J.S. (2020) ‘Effect of underbody diffuser on the aerodynamic drag of vehicles in convoy’, *International Journal of Vehicle Structures & Systems*, 12(1), pp. 1–6.

[11] Unni, A. (2019) ‘Parametric investigation of effect of diffuser angle on the flow characteristics of an Ahmed body’, *International Journal of Engineering Research & Technology (IJERT)*, 8(5), pp. 134–138. Available at: https://www.ijert.org/parametric-investigation-of-effect-of-diffuser-angle-on-the-flow-characteristics-of-an-ahmed-body