JOURNAL OF AEROSPACE SOCIETY MALAYSIA

*ARTICLE*

# EXPERIMENTAL PASSIVE FLOW CONTROL ON AHMED MODEL FOR AERODYNAMIC DRAG REDUCTION

**Ananta Kullan [1,*], Shabudin Mat [2] and Rafiqi Daniel Ramadhan [2]**

1. Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Malaysia.
2. UTM Aerolab, Institute for Sustainable Transport, Universiti Teknologi Malaysia, Malaysia.

*Correspondence: nanthakva@live.com

**Abstract:** This study presents an investigation on the effects of yaw and slant angles to the aerodynamic behavior of the Ahmed model. Experiments were conducted in a low-speed wind tunnel at a constant speed of 25 m/s across different yaw angles. Four Ahmed models with slant back angles of 0°, 15°, 25° and 40° were used. All in all, the experimental results indicate that yaw angle significantly influences the aerodynamic behavior of the Ahmed model. As the yaw angle increases, the pressure distribution also becomes increasingly asymmetric due to changes in flow separation and wake formation. Additionally, the findings also highlight the influence of slant angle variations under yawed conditions, an area with limited prior investigation. The obtained knowledge on the aerodynamic behavior of the Ahmed model from this study is very useful in further vehicle design development that is based on this generic model.

**Keywords:** aerodynamic characteristics; Ahmed model; slant angle; vehicle aerodynamics; wind tunnel

## 1. Introduction

Aerodynamic drag significantly affects the energy efficiency and also environmental impact of road vehicles. For instance, a 15% reduction in drag can result in up to a 5% improvement in fuel economy, making aerodynamic optimization vital for internal combustion engine vehicles (ICEVs) and electric vehicles (EVs). Among various vehicle body types, bluff-body shapes with blunt rear ends are prone to pressure drag due to flow separation and wake formation. The Ahmed body, introduced by Ahmed et al. in 1984, serves as the benchmark model to study the bluff-body aerodynamics due to its geometric simplicity and also its relevance to the real-world vehicle designs [1]. It has been widely adopted in both experimental and numerical investigations to explore the flow control strategies aimed at reducing drag.

Flow control techniques and methods have been applied to reduce the drag, which are categorized into active and passive methods. In essence, the active flow control methods involve energy input to manipulate the flow field. Techniques such as steady blowing, synthetic jets and the Coanda effect have shown promising results in wake control and drag reduction. For instance, a conducted study reported a 10% reduction in the base drag using Coanda-induced flow deviation, which extended the wake length and reduced its width [2]. In addition, another study has also demonstrated that localized suction and blowing could effectively control the separation bubble and suppress swirling structures at the rear of the Ahmed body [3]. Although highly effective, active methods require precise control and energy input, making them less suitable for mass-market applications. On the other hand, passive flow control devices such as vortex generators (VGs), base cavities, deflector plates and geometric modifications (i.e. roof-edge rounding) offer energy-free methods to delay flow separation and reduce drag. Bayindirli (2024) demonstrated that the roof-mounted VGs could increase surface adherence and reduce pressure drag by disturbing the laminar separation [4]. Additionally, different VG geometries and their aerodynamic effectiveness have been explored [5], which highlights that their application is not merely aesthetic but also functional.

It has been shown that the flow separation at the blunt rear and nose sections is the primary source of drag forces in bluff bodies [6]. Therefore, by modifying the Ahmed body geometry and validating it against experimental data using standard k-turbulence model, this study will contribute valuable insights into the effects of passive devices on complex wake flows. A critical gap in current research is the lack of studies examining the combined influence of yaw angle and rear slant angle on the aerodynamics of Ahmed body. While many investigations isolate the effects of either yaw or slant angle, few studies have considered their interaction. Real-world vehicles often operate under the crosswind (yawed) conditions, which interact with the vehicle's rear geometry to influence separation and wake behavior. This gap has been highlighted by Sumnu and Eraslan (2025), noting that very few studies simultaneously explore the effects of passive control strategies and shape optimization under yawed flow [7]. This oversight limits the generalizability of findings to actual operating conditions of the vehicles.

Although computational fluid dynamics (CFD) studies have advanced significantly, many of them still rely on coarse grids or simplified turbulence models. High-fidelity simulations incorporating fine meshes and advanced turbulence models (e.g. LES or DES) under yaw and slant variations are still rare. Meanwhile, experimental studies often lack the comprehensive pressure mapping or wake visualization under combined yaw and slant angle conditions, leading to incomplete understanding of flow dynamics. On the whole, while substantial research has been conducted on the aerodynamic drag reduction using passive and active techniques, the combined influence of yaw angle and slant angle on the aerodynamics of Ahmed body remains inadequately addressed. Furthermore, the interaction between passive devices like VGs and complex flow conditions involving yaw and slant angles is poorly understood. This study aims to address this gap by experimentally investigate the aerodynamic behavior of the Ahmed model under various yaw and slant angles, with and without the passive control devices. The objectives include evaluating the resultant pressure distribution and drag force alteration from the different configurations.

## 2. Methodology

Without changing the projected frontal area of the vehicle, it is still possible to modify its shape in a more streamlined way. External attachments can help minimize the aerodynamic drag based on their external shape, size and placement. Aerodynamic drag (D) depends on the size of a vehicle (projected frontal area, A), drag coefficient ($C_D$) that is a measure of the flow quality around the vehicle, and the square of the vehicle speed (V) as expressed as in Equation 1.

$$D = \frac{1}{2} C_D \rho V^2 A \quad (1)$$

Dynamic pressure (velocity pressure) is the increase in a moving fluid's pressure over its static value due to motion. In incompressible fluid dynamics, it is indicated as q and is defined in Equation 2, where q is the dynamic pressure in Pascal, ρ is the fluid density in kg/m$^3$ and V is the flow speed in m/s.

$$q = \frac{1}{2} \rho V^2 \quad (2)$$

Another important aspect of the dynamic pressure is that the aerodynamic stress (i.e. stress within a structure subject to aerodynamic forces) experienced by a vehicle travelling at speed V is proportional to air density and square of V (i.e. proportional to q). Hence, by looking at variation of q, it is possible to determine how the stress will vary and, in particular, when it will reach its maximum value. The point of the maximum aerodynamic load is often referred to as "maximum q" and it is a critical parameter in many applications such as launch vehicles.

Moreover, pressure coefficient ($C_p$) is the parameter for studying the flow of incompressible fluids such as water and also the low-speed flow of compressible fluids such as air. The relationship between this dimensionless coefficient and the dimensional numbers is given by Equation 3, where P is the static pressure at the point at which the pressure coefficient is being evaluated, Ps is the static pressure in the freestream (i.e. remote from any disturbance), ρ is the freestream fluid density (i.e. for air at the sea level and 15 °C is 1.225 kg/m³) and V is freestream velocity of the fluid, or the velocity of the body through the fluid.

$$C_P = \frac{P - P_S}{\frac{1}{2}\rho V^2} \tag{3}$$

The pressure coefficient describes the relative pressures throughout a flow field in fluid dynamics. It is used in aerodynamics and hydrodynamics, whereby every point in the fluid flow field has its own unique pressure coefficient. In many situations, the pressure coefficient at a point that is near a body is independent of body size. Consequently, an engineering model can be tested in a wind tunnel or water tunnel through which the pressure coefficients can be determined at critical locations around the model and these pressure coefficients can be used with confidence to predict the fluid pressure at those critical locations around a full-size vehicle.

For this study, an experimental study on an Ahmed model is conducted using the wind tunnel. The resulting data is collected and analyzed to test the resulting flow separation from different yawing angle with the slant back angle of 0°, 15°, 25° and 45°. The model has also been set at various angles of yaw of -25°, -20°, -15°, 0°, 25°, 20° and 15°. The model used for this project is an Ahmed model from the UTM Aero lab, which is a 1:16 scale replica constructed from high-strength, vibration-resistant wood. While the material's surface finish can influence the boundary layer behavior and also flow separation, the model's surface is maintained with a smooth finish to reduce roughness effects in this study. This model is built for use in wind tunnel testing and has been made to place pressure vessels to be connected to the pressure scanner. This model has a weight of almost 2 kg and can withstand the wind speed and pressure when the test in the wind tunnel is carried out. Figure 1 shows the overall view of the applied generic Ahmad model in this study and Table 1 tabulates its main dimensions.

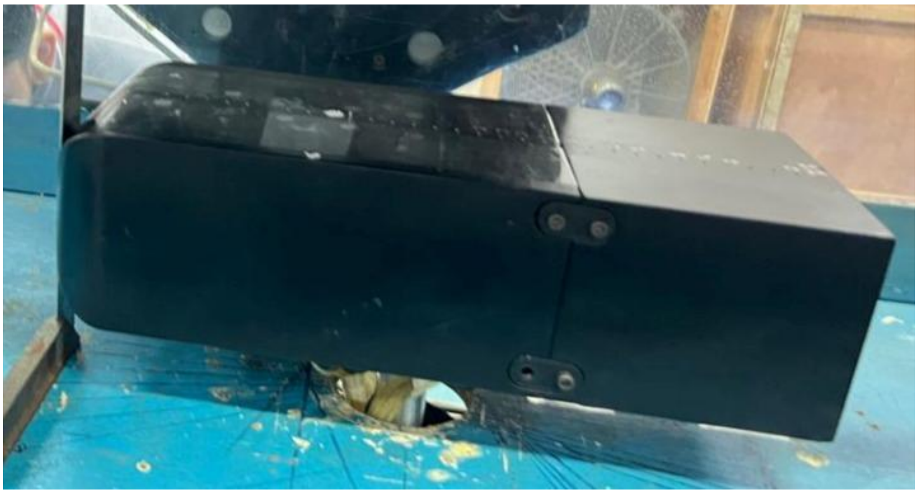

Figure 1: Generic Ahmed model used in the wind tunnel testing

Table 1: Dimensions of the Ahmed model

| Part | Description | Dimension (mm) |
|---|---|---|
| A | Overall Length | 350 |
| B | Overall Width | 120 |
| C | Overall Height | 130 |

The wind tunnel experiment is conducted at Universiti Teknologi Malaysia Aeronautics Laboratory (UTM Aerolab). The test section for putting the model is 2.0 m (width) x 1.5 m (height) x 5.8 m (length) and the wind tunnel is a closed-return low speed type. In this study, this model is tested on 3.2 x $10^6$ and 3.7 x $10^6$ Reynolds Numbers. This wind tunnel can generate airflow up to a Mach speed of 0.23 or 80 m/s in the test section area. It has the flow and temperature uniformity of less than 0.15% to 0.20%, respectively, and less turbulence from 0.06%. Figure 2 shows example of the Ahmed model used with different slant angles while Figure 3 depicts the pressure scanner used in the testing.

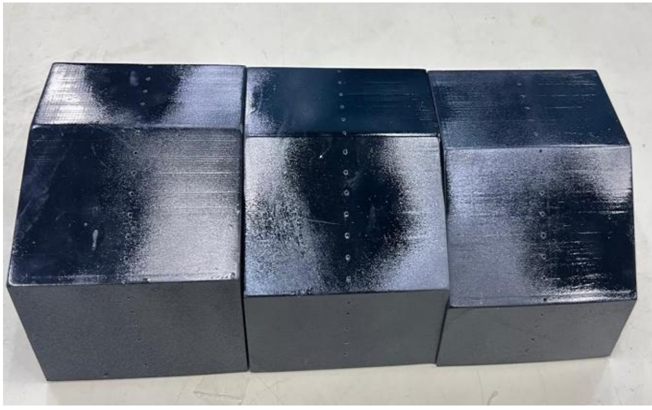

Figure 2: Ahmed Model slant angles of 15°, 25° and 45°

Figure 3: Pressure scanner used in this study

In short, the experiment is conducted in a subsonic wind tunnel and the wind speed has been set within a controlled range to analyze both pressure and force. The Ahmed model is mounted in the test section of the wind tunnel and a series of pressure sensors (approximately 2 mm in diameter) have been installed at specific locations on the model's surface to capture the pressure variations due to the vortex interactions. These sensors are connected via urethane tubing to a data acquisition system for real-time recording. Depiction of this wind tunnel setup can be seen in Figure 4. It should be noted that the usage of a closed-return low speed wind tunnel for this study allows for a spacious environment to minimize the wall effects and ensure uniform airflow across the model. All in all, the experimental matrix for this study consists of testing four different Ahmed models with slant back angles of 0°, 15°, 25° and 40° at seven settings of yaw angles of -25°, -20°, -15°, 0°, 15°, 20° and 25°. Note that the wind tunnel test for each configuration is repeated three times to ensure data repeatability, with the average value used for analysis. Error margins have been quantified through standard deviation calculations. Prior to each test, sensors and equipment are calibrated to minimize measurement uncertainties.

Firstly, the generic Ahmed model is properly mounted on the wind tunnel test section. Next, the pneumatic piping of the pressure scanner is mounted at the surface body of the model. In addition, the load cell is also mounted at the model's surface body. Once these have been securely mounted on the model, the wind tunnel test can be conducted. In this study, the test is conducted at seven different yaw angles of -25°, -20°, -15°, 0°, 25°, 20° and 15° while the wind speed in the tunnel during the testing has been set to 25 m/s. Once the test is completed, the model is changed with the next Ahmed model with a different slant back side angle and the wind tunnel test is conducted again. This is repeated until all of the four Ahmed models with the different slant back angle have been successfully tested. The collected experimental data is then analyzed to establish the effects of the different model configurations. In this case, effects on adverse pressure gradient and flow separation are observed using the pressure scanner while effects on force are observed using the load cell's digital sensor. Moreover, the flow visualization data is done through the pressure distribution method.

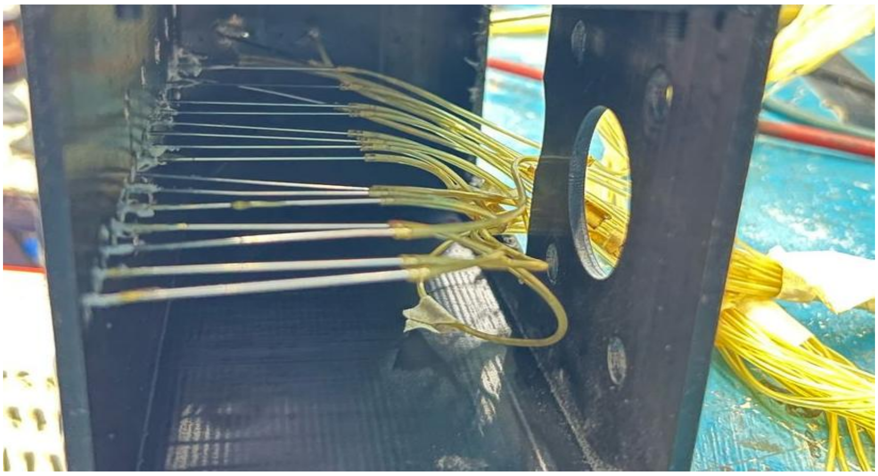

Figure 4: Pressure tap located at the model's top surface and back slant

It should be noted that all of the experimental procedures in the conduct of this study have been documented to allow replication. The model specifications, wind tunnel conditions and data processing techniques ensure that similar results can be achieved in future conduct of the similar procedures. This methodology provides a structured experimental approach to analyze aerodynamic behavior of Ahmed model's pressure and force with different yawing angles and slant back side angles.

## 3. Results and Discussion

The obtained experimental results are shown in Figure 5 to Figure 8, which provide the behaviors of impact pressure and aerodynamic drag force that can contribute towards a better understanding of the maneuvering of the Ahmed model. The experimental data can also be used to aid in the validation of simulation models. In automotive applications, the chosen yaw angles represent the typical vehicle maneuvering scenarios such as turning or side-slip conditions, which are critical for understanding flow separation and pressure distribution. The slant back angles have been selected to simulate different rear vehicle geometries that influence aerodynamic stability and drag. Moreover, Reynolds numbers of $3.2 \times 10^6$ and $3.7 \times 10^6$ have been targeted to emulate the realistic driving conditions under turbulent flow regimes, consistent with standard operating speeds for vehicles. These parameters collectively facilitate a comprehensive analysis of the model's aerodynamic performance across practical scenarios.

On the whole, the results from this wind tunnel testing of the UTM Generic Ahmed Model reveal that yaw angle plays a dominant role in influencing the aerodynamic behavior of the model. Specifically, changes in the yaw angle lead to significant variations in the pressure coefficient ($C_p$) and aerodynamic drag coefficient ($C_D$) across top and rear surfaces of the model. As the yaw angle increases, the pressure distribution becomes increasingly asymmetric, which directly correlates with the changes in the flow separation and wake formation. In addition, it has been also observed that the position of the pressure taps, particularly on the top and rear surfaces, is crucial for capturing accurate variations in the pressure. These areas exhibit the highest sensitivity to changes in yaw angle and can provide a clear representation of how surface pressures are redistributed under non-zero yaw conditions. In contrast, the Reynolds number is shown to have minimal influence on both pressure and force coefficients within the tested flow velocity range. This suggests that the aerodynamic performance of the Ahmed model is primarily governed by yaw angle effects rather than changes in the speed of the flow.

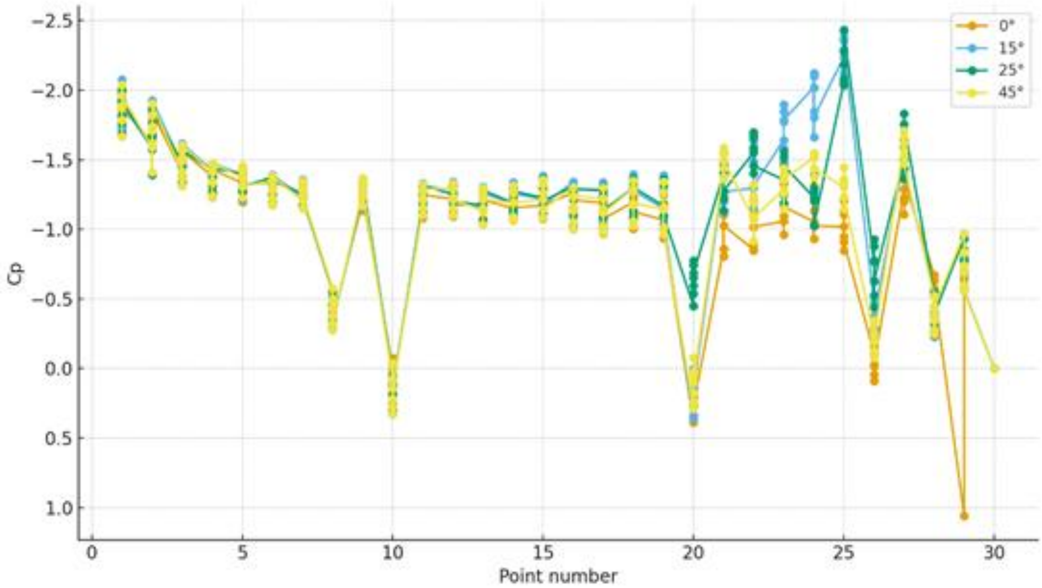


Figure 5: Pressure coefficient distribution over Ahmed model in different back slant angles at yaw angle of -25° and velocity of 25 m/s

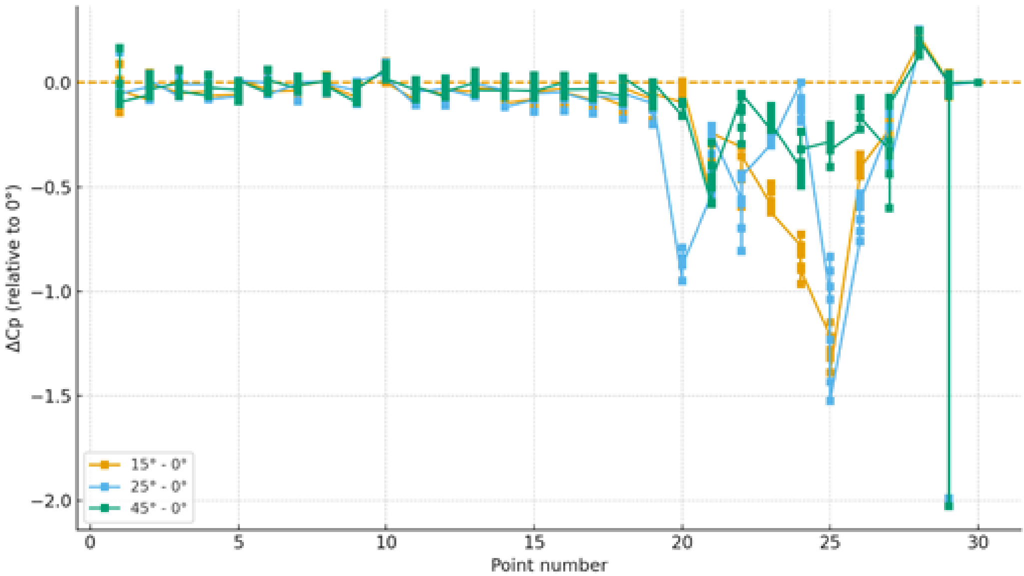


Figure 6: $C_p$ value due to slant angle negative (stronger suction)

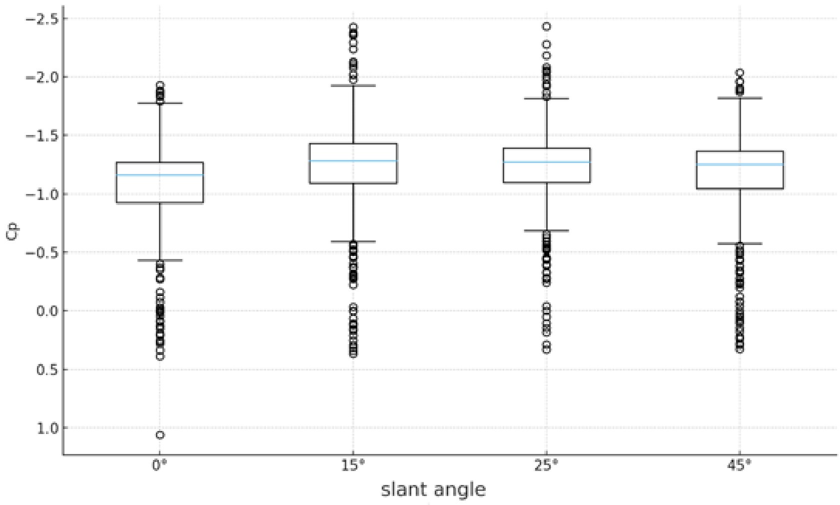


Figure 7: Distribution of $C_p$ by slant angles

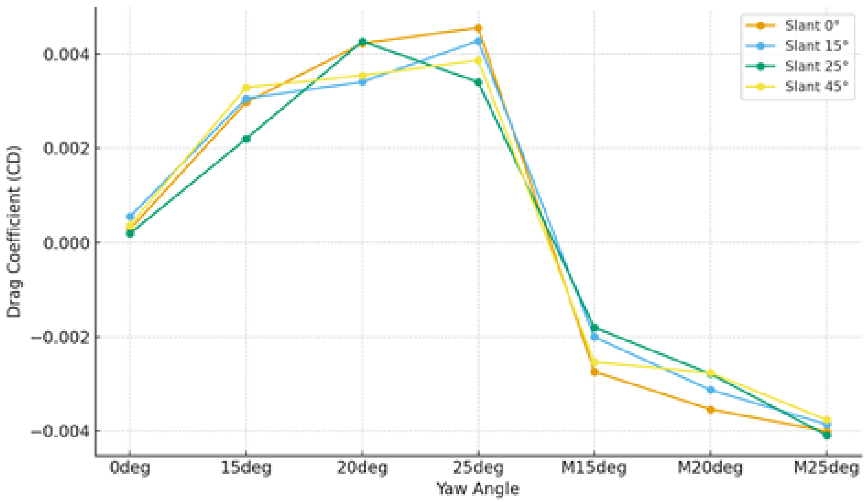


Figure 8: $C_D$ at different yaw and slant back angles

While the experimental setup in this study has provided a valuable insight into the impact of yaw angle on flow behavior, several limitations are also acknowledged as listed below:

- The tests were conducted in a low-speed wind tunnel, restricting the range of Reynolds numbers and limiting comparison to full-scale vehicle conditions.
- Pressure measurements were obtained only from the top and rear surfaces of the model. The flow features on the side surfaces and underbody, which are also affected by the yaw angle, were not captured.
- The study was based on a simplified Ahmed body and did not include real-world features such as rotating wheels, undertray effects or ground clearance variations.
- No flow visualization or turbulence measurements via smoke, tuft grids or PIV were performed, which limits interpretation of three-dimensional flow behavior.

## 4. Conclusion

This study is aimed to experimentally investigate the aerodynamic behavior of the Ahmed model under various yaw and slant angles, with and without the passive control devices. The focus of the study is on the pressure distribution and drag force alteration from the different configurations. Based on the results of the conducted wind tunnel testing of the Ahmed Model, it has been demonstrated that yaw angle has a dominant role in influencing the aerodynamic behavior of the vehicle model. Specifically, changes in the yaw angle lead to significant variations in the pressure coefficient ($C_p$) and aerodynamic force coefficients ($C_D$). As yaw angle increases, pressure distribution becomes increasingly asymmetric, which directly correlates with the changes in flow separation and also wake formation. To build upon the findings of this research, the following future directions are recommended:

- Expand the Reynolds number range to include higher-speed conditions and evaluate scale effects more comprehensively.
- Incorporate side and underbody pressure taps, or utilize advanced measurement techniques such as Particle Image Velocimetry (PIV) or Laser Doppler Anemometry (LDA), to visualize complex flow structures and quantify turbulence characteristics.
- Introduce active and passive flow control devices, such as vortex generators, Coanda-effect nozzles, or synthetic jets, to assess their effectiveness in mitigating yaw-induced aerodynamic penalties.
- Investigate the effects of rotating wheels, ground proximity (ride height), and underbody flow, which play a crucial role in real vehicle aerodynamics.
- Combine CFD simulations with experimental data to develop a more comprehensive understanding of the aerodynamic phenomena and validate computational models under yaw conditions.

**Acknowledgement**

The authors would like to acknowledge that this study is funded by Universiti Teknologi Malaysia through their research grant scheme (Grant No. 22H81). In addition, special acknowledgement to the technical staff of Aerolab UTM (Universiti Teknologi Malaysia) for the use of their assistance.

**References**

[1] S. R. Ahmed, G. Ramm and G. Faltin, "Some Salient Features of the Time-Averaged Ground Vehicle Wake," SAE Transactions, vol. 93, pp. 473–503, 1984.

[2] N. A. Siddiqui and M. A. Chaab, "A Simple Passive Device for the Drag Reduction of an Ahmed Body," Journal of Applied Fluid Mechanics, vol. 14, no. 1, pp. 147-164, 2020.

[3] B. Harinaldi, R. Tarakka and S. P. Simanungkalit, "Computational Analysis of Active Flow Control to Reduce Aerodynamics Drag On A Van Model," International Journal of Mechanical & Mechatronics Engineering, vol.11, no. 3, pp. 24-30, 2011.

[4] C. Bayındırlı and M. Çelik, "Drag Reduction of Ground Vehicle by Decreasing Flow Separation With A Passive Flow Control Part," Uluslararası Mühendislik Tasarım ve Teknoloji Dergisi, vol. 6, no. 2, pp. 96-102, 2024.

[5] H. Viswanathan, "Aerodynamic Performance of Several Passive Vortex Generator Configurations on an Ahmed Body Subjected to Yaw Angles," Journal of the Brazilian Society of Mechanical Sciences and Engineering, vol. 43, no. 3, 131, 2021.

[6] C. García-Baena, J. M. Camacho-Sánchez, M. Lorite-Díez, C. Gutiérrez-Montes and J. I. Jiménez-González, "Drag Reduction on a Blunt Body by Self-Adaption of Rear Flexibly Hinged Flaps," Journal of Fluids and Structures, vol. 118, 103854, 2023.

[7] A. Şumnu and Yüksel Eraslan, "Aerodynamic Shape Optimization of Simplified Ground Vehicle (Ahmed Body) using Passive Control Devices," Mechanics, vol. 31, no. 1, pp. 5-14, 2025.