

Reducing Vehicle Drag Force Through a Tapered Rear Side Wall

Behrang Daryakenari
University Kebangsaan Malaysia

Shahrir Abdullah
Universiti Kebangsaan Malaysia

Rozli Zulkifli
UKM

ABSTRACT

Recent fluctuation in oil prices has generated interest in fuel-efficient vehicles, especially their aerodynamic profile. The literature indicates that turbulent wakes that form at the rear end of the vehicle contribute to vehicle drag in a major way. Minor studies have addressed the effects of rear-end wall angle to the drag force through effecting the wake behind the vehicle; however, this study assesses the reduction of drag using angular side walls. A previous simulation of external airflow over Ahmed's body was investigated, utilizing the $k-\omega$ SST models. Different angles of side walls were analyzed, and a maximum 36.85% reduction in drag coefficient was achieved using an angular rear side wall. The turbulent model was validated and the effectiveness of angular rear side walls thus proven. The study then simulated the flow for a road vehicle model to investigate the real world effect of angular rear side walls. The results showed up to 11.39% reduction in drag coefficient, supporting the effect of angular rear walls on reducing drag coefficient.

CITATION: Daryakenari, B., Abdullah, S., and Zulkifli, R., "Reducing Vehicle Drag Force Through a Tapered Rear Side Wall," *SAE Int. J. Commer. Veh.* 6(2):2013, doi:10.4271/2013-01-9020.

INTRODUCTION

Mindful of the ever-increasing cost of engine fuel and its projected dwindling supply in the future under an escalating demand for the commodity, vehicle manufacturers are in a competitive race to find ways and means to optimize mileage for every unit volume of fuel. Apparently, one of the observable means to address the aforesaid objective is to reduce the drag forces generated by a vehicle in motion, thereby considering aerodynamic principles in designing vehicle bodies. As we know, the separation of the flow which leads to the production of a wake behind the vehicle has the major contribution to the vehicle's drag. Wake's complexity and full three-dimensional structure, can cause considerable difficulties for numerical simulations and experimental studies.

Because of a convenient number of experimental results, the Ahmed body (Fig.1) which was first introduced by Ahmed et al. [1] has been used as a validation case for many

numerical investigations. As a result of both the simplicity of the model and the wide range of experimental results, the Ahmed body is an ideal candidate for CFD studies.

Using $k-\epsilon$ turbulent model to simulate flow over Ahmed body, Baxendale et al. [2] results were noticeably higher than the experimental results and Makowski et al. [11] results tended to suggest an excessive CFD prediction of drag coefficient values. Gilliéron and Chometon [4] used the RNG $k-\epsilon$ turbulence model and their results for drag coefficient on average were approximately 25% higher than the Ahmed experimental data. Large eddy simulation was performed by Hinterberger et al. [6], and Krajnovic & Davidson [8, 9]. Although they could get good results but the problem with their work, however, was utilizing the low Reynolds Number (2×10^5) in compare with a similar experiment using 5×10^6 . Hinterberger et al. [6] performed another numeric study using the Finite Volume Code LESOCC2 (Large Eddy Simulation on Curvilinear Coordinates). Although for the front region

the results showed very good agreement but in the back region, there were some differences between the computations and the experiment that mainly concerned the separation point of the flow over the slant part. Fares [3] used the LBM (Lattice Boltzmann Method) to simulate the unsteady flow over two slant angles of the rear part of the Ahmed reference body (25° and 35°), which successfully predicted the flow structure for that case as 35°, but for the other case, it predicted a separation area longer than in the experiment. Performing another numerical study on the same cases, Yao et al. [16] employed two turbulence models (the standard SST model and the k- ϵ model), and due to this study, the SST (shear stress transport) model had better performance in predicting the separation region. As with other studies, the simulation did not capture the initial separation of the case with a 25° angle while they did for the case with a 35° angle. Kapadia et al. [7] used DES (detached eddy simulation) model for the first time for ground vehicle flow analysis which explores the possibility of using such model as a commercial software and turbulence scheme respectively in automobile industries. Comparison of the results with the results obtained by RANS model for same time -steps showed the ability of DES in capturing unsteady structure of the flow with minor flow details than RANS. Guilmineau [5] studied the Ahmed body with 25° and 35° slant back by solving unsteady Reynolds Averaged Navier Stokes Equations (URANSE). All examined turbulence models, predicted the structure of the flow correctly for the 35° slant angle but for the 25° slant angle, all models predicted huge separation zone, while the experiment shows reattachment of the flow on the slant back about half-way down the center of the face. On the other hand, the EASM model (explicit algebraic stress model) gave a better estimation of the drag coefficient. Minguez et al. [13] used high-order large-eddy simulations (LESS) at a lower Reynolds Number than Ahmed's experiment (7.6×10^5). The topology of the flow over the body front shows a recirculation bubble larger than the experimental results that resulted in overestimated drag coefficient.

As can be seen in the previous literature, despite its simplicity, the Ahmed body still constitutes a challenge to numerical algorithms and turbulence modeling due to its complex three-dimensional wake vortex interaction. In this paper after an accurate simulation of flow over Ahmed body using k- ω SST turbulence model, the effect of tapered rear side wall on reducing the size of the wake that consequently will reduce the drag force of the vehicle was studied. Upon getting considerable results, the same method was utilized to investigate the effect of angular rear side wall in for a real car model. The results that show significant reduction of drag coefficients of the simple car model and a real road car proved the angular real side wall's positive performance to modify the drag force of a vehicle.

THE TURBULENCE MODEL

Previously, the SST model was designed for use in aeronautics applications; later as its efficiency was proven it has been used in most industrial applications. This model is a two-equation, eddy-viscosity model that combines the best of the k- ω and k- ϵ models. In a comparison, the k- ω model is noticeably more accurate than the k- ϵ model inside the boundary layer, whereas outside boundary layer, the k- ϵ model is a better choice. This capability of the k- ω SST model enables it to predict the separation point accurately [12], and since for our case separation of the flow from the side walls and the slant part of the model are very important to determine the drag force, we preferred to use this turbulent model. Using a blended function enables the k- ω SST model to switch between these two models without user influence and achieve an accurate result. The numbers of coefficients used in this study are as follows:

$$\sigma_{k1} = 0.85034, \sigma_{k2} = 1.0, \sigma_{\omega1} = 0.5$$

$$\sigma_{\omega2} = 0.8561, \alpha_1 = 0.5532, \alpha_2 = 0.4403$$

$$\beta_1 = 0.075, \beta_2 = 0.0828, \beta^* = 0.09$$

THE PHYSICAL CASE AND NUMERICAL METHOD

For the simple car model Ahmed model which is 1044 mm long, 288 mm high and 389 mm wide was used. It has a slant back with a 30° angle and 222 mm length (Fig. 1). The velocity of the air was set as 60 m/s, Reynolds number was 4.29 million based on model length as in the Ahmed experimental study, but for the real car model, 33.33 m/s was used because of the speed limit on the highways. The ground was considered constant, the same as Ahmed's experiment. At the inflow section a uniform axial velocity profile U_∞ is imposed, while a non-slip boundary condition at the ground is prescribed.

After validating the flow structure and drag data of this model, an angular side wall was added to the rear end of the model (Fig. 2). The distance from the rear vertical base of the model (334 mm) was chosen based on the relative rear seat distance of our real car model from its rear base. We considered not reducing the seating area for the passengers. The effect of this angle (α) was studied for five angles, namely, 2.5°, 5°, 10°, 15° and 20°. Since the performance of such an angular side wall in reducing the drag coefficient was approved, the same method was used to study the effect of manipulating the angle of the side wall on drag reduction of a real road vehicle (Fig 3). This model was a quarter size scale of the real hatchback car that includes the entire engine and under body components in order to obtain the most accurate results. Every angle from 1° to 6° with a 1° increment (Fig 4) were studied for this model and promising results in drag reduction were achieved. The models were first generated in

Catia and OpenFoam which is a CFD software was used respectively to generate the mesh and simulate the flow over the body.

The volume mesh model is triangle mesh. The domain size for both models is $4 \text{ m} \times 20 \text{ m} \times 4 \text{ m}$ with very tense mesh, which has about 11 million cells in order to achieve the required accuracy.

The volume mesh model used triangle mesh. Since the flow over a vehicle has adverse pressure gradients or separation regions that need to be properly resolved a y^+ (Dimensionless wall distance) of 1 was considered. For determining the mesh size over the vehicle for getting the accurate results, several simulations were run for the sizes between one and three millimeters. The drag coefficient result for the 3 millimeters mesh size showed more than 5 percent disparity with Ahmed's experimental result [1]. For the sizes below 2 millimeter the disparity was below 3 percent. The results did not change significantly for smaller mesh size, therefore we chose 2 millimeter mesh size that beside reasonable error has the lower convergence time. Surface mesh is square with about 550,000 cells. This size makes it impossible to present here because of its density. The number of surface cells for the real car model is about 850,000 since it includes more components in compare with Ahmed Body. The model was situated 4 meters from the front of the domain.

NUMERICAL RESULTS AND DISCUSSION

The Ahmed Body

Since producing a steady flow is essential for stability and accuracy requirements, the simulation was run several hundred of time steps for convergence of the results. The symmetry of the time-averaged flow around the vertical symmetrical plane was used as another indicator of the convergence in time of the average flow.

To validate the model, the experimental results for the Ahmed body with a 30° slant angle obtained by Vino et al. [15] and Lienhart [10] were chosen. Vino utilized smoke injection to visualize the flow pattern on the slant part of the body. The flow structures obtained by the numerical results are in positive agreement with the experimental results by Vino (Fig.5) and captured the flow separation on the slant part of the body. The results obtained are also in agreement with the work of Spohn and Gilliéron [14], who reported the detached flow region on the slant which are close and mixed with the flow in the wake.

In Figure 6, the separation of flow over the slant part is obvious and in agreement with Vino and Ahmed's findings over the slant at a 30° angle.

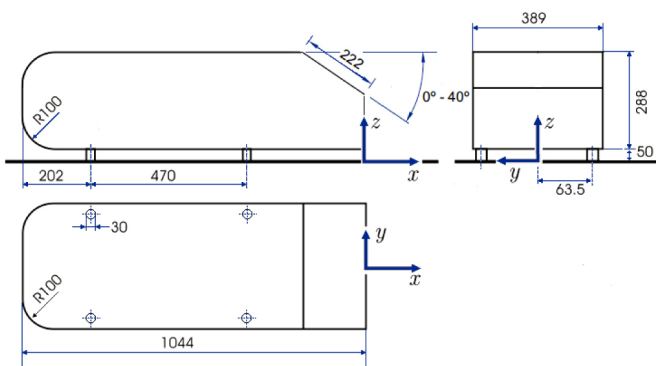


Figure 1. The Ahmed body, dimensions are in mm

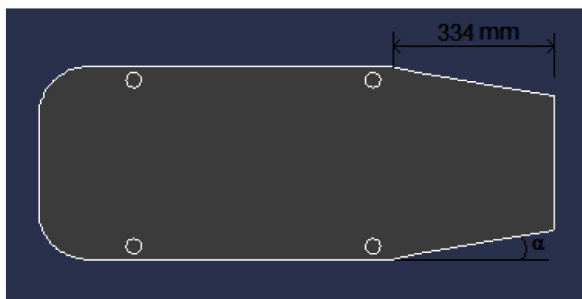


Figure 2. Ahmed model with angular side wall

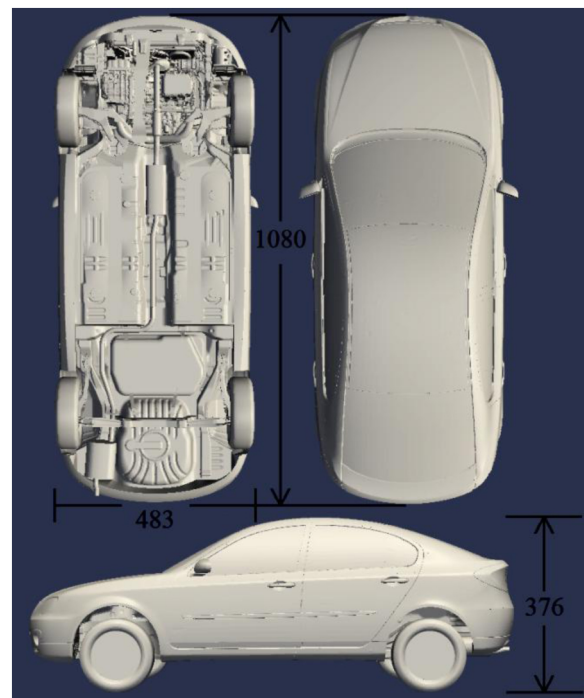


Figure 3. Real road vehicle ($1/4^{\text{th}}$ scale model), dimensions are in mm