Rear-roof spoiler effect on the aerodynamic drag performance of a simplified hatchback model

S Y Cheng[1,2] and S Mansor[3]

[1]Centre for Advanced Research on Energy, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia

[2]Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia

[3]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia

E-mail: cheng@utem.edu.my

Abstract. The effect of rear-roof spoiler on the aerodynamic drag performance of simple hatchback model has been investigated for a range of spoiler pitch angles. RANS based CFD method was utilized. Validation was done by comparing the CFD results of Ahmed model without spoiler to the experimental data. The spoiler has a beneficial influence at positive inclination angle below 5°. Above that angle, the aerodynamic drag increases with pitch angle. The mechanism of how the spoiler influences the aerodynamic drag was identified through extensive flow visualization.

1. Introduction

A rear-roof spoiler is an external structure fitted to the trailing edge of the roof of a vehicle to modify the airflow going over it. The purpose is to improve the vehicle's aerodynamic performance. Spoiler designs fall into two categories: strips and free-standing wing. The latter could have a simple flat plate profile or an inverted airfoil. The effectiveness of wing-type spoiler has been reported in numerous studies. For instance, five different wing-type spoiler configurations had been tested on a simplified car model based on a HONDA S2000 [1]. All of them had achieved lift reduction. However, the drag coefficient (C_d) of all the cases had increased tremendously as compared to the case without a spoiler.

On a similar note, a numerical study has shown up to 75% reduction in the lift coefficient C_l of a passenger car model when properly manipulated the inclination angle of the flat-plate-type spoiler, i.e. at 50° relative to the ground [2]. However, the C_d had increased by 8%. Meanwhile, a simplified-passenger-car model mounted with an inverted-airfoil type spoiler has experienced 80% reduction in the C_l with an increase of 3% C_d [3]. The increase in drag is a drawback as it is directly linked to fuel consumption. This performance is important because the emission from automobiles is accounting for about 10% of global carbon dioxide emissions, a major contributor to the greenhouse effect [4]. On the other hand, there are also cases where a spoiler is found to reduce the aerodynamic drag of vehicles (e.g. [5]; [6]).

As for the strip-type rear-roof spoiler, it was reported that the highest drag reduction of around 5 to 6% has been achieved on a simple hatchback model, namely, an Ahmed model at 35° slant angle [7].

The spoiler was configured at the length of about 32% of the slant length, and at 0° spoiler angle. However, the study did not cover the range of positive inclination angles, which are more commonly found in real life applications. To fill the gap, hence, the present paper aims to investigate the effect of pitch angle of rear-roof spoiler, in particular, at positive inclination, on the aerodynamic performance of simplified hatchback model.

2. Simplified hatchback model and spoiler configurations

Rear-roof spoilers are commonly found in hatchback-type vehicles. Hence, the Ahmed body with the slant angle of 35°, which is typical for most hatchback cars, was adopted. Figure 1 shows the Ahmed model fitted with a 0° pitch angle spoiler. Also shown in the figure are the spoiler configurations tested, i.e. pitch angles of -15°, 0°, 5°, 10°, and 15°. The length of the spoiler was 66.6 mm, which corresponds to 30% of the length of the slant section. The trailing edge of the spoiler was filleted (5 mm radius) for avoiding highly skewed cells during meshing. For details of the Ahmed model dimensions, the reader is referred to [8].
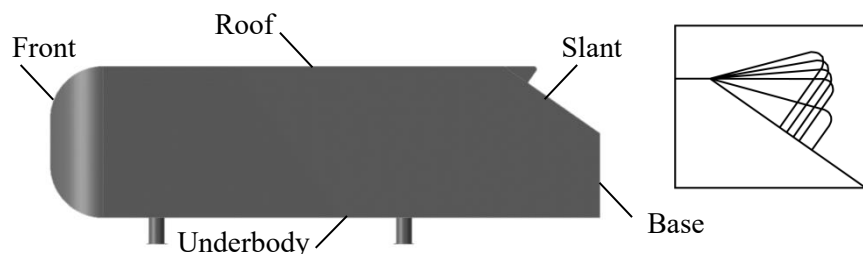

Figure 1. Side view of Ahmed model fitted with a rear-roof spoiler (left), and the spoiler configurations (right).

3. Numerical Method

The present paper employed numerical simulations to investigate the influence of pitch angle of rear-roof spoiler on the aerodynamics of a simplified hatchback model. All results were obtained using the commercial finite-volume solver ANSYS Fluent 16. The Reynolds-averaged Navier-Stokes (RANS) approach was used. The turbulence model was the widely used two-equation model, namely, k-epsilon realizable model with enhanced wall treatment. The steady, pressure-based solver was utilized to achieve steady-state simulations. The effects of compressibility was neglected due to the relatively low Mach numbers (about 0.12). All reported results were obtained using a second-order node-based upwinding discretization scheme.

The inlet boundary condition was set as uniform flow at 40 m/s with the turbulence intensities of 0.2%. The corresponding Reynolds number (Re) was 768,000 based on the model height. The pressure outlet at zero gauge pressure was imposed on the outlet boundary. As for the side and top walls of the domain, symmetry boundary condition was applied. Meanwhile, the ground and model surfaces were set as no slip wall.

The computational domain resembled a rectangular box. Since the model is symmetric, all simulation cases were run for half of the flow domain with a symmetry plane placed at the centerline location. The cross sectional area of the half flow domain was 1738 mm height x 1129.5 mm width. The corresponding blockage ratio was less than 1.5%, which is well within the typically accepted range of 5% in automotive aerodynamic testing [9]. The upstream and downstream extends of the domain were $1.4l$ and $11.4l$, respectively, where l is the vehicle length.

The computational domain comprises unstructured and prismatic cells (see figure 2). The latter was employed around the model and the ground for obtaining better boundary layer resolutions. The first prismatic cell layer thickness was at 0.5 mm. The corresponding y^+ around the model surface was within 1 to 57, which is in the appropriate range for k-epsilon turbulence model. The total nodes and cells for all simulation cases were around 315,000 and 890,000, respectively.
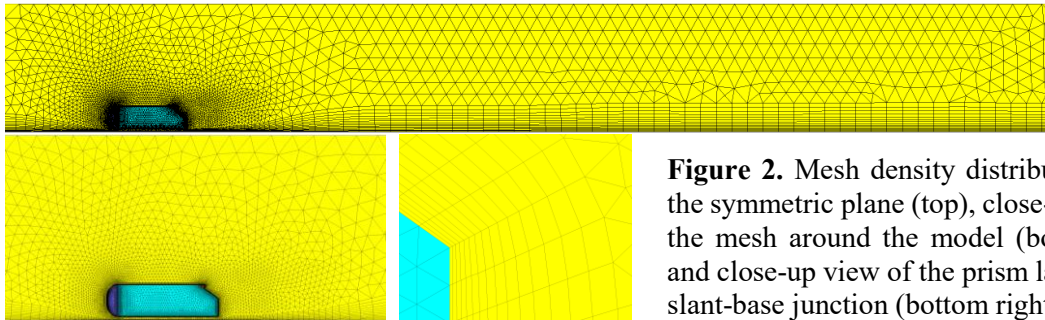
Figure 2. Mesh density distribution along the symmetric plane (top), close-up view of the mesh around the model (bottom left), and close-up view of the prism layers at the slant-base junction (bottom right).

4. Validation

To validate the numerical method, the present study carried up a case of flow simulation for the Ahmed model without the spoiler. The Re was the same as the experimental value by [10], i.e. 768,000 based on the model height. Table 1 summarizes the validation results. As depicted, the total C_d value is in excellent agreement with the experiment, i.e. percentage difference of 2.9%. The drag breakdown shows that the C_d of the base and slant sections are well predicted. Good prediction of C_d in these two sections is important because the influence of rear-roof spoiler is expected to be most prominent at the rear part of the model.

Table 1. Comparison of the experimental C_d.

	CFD	Experiment [10]	Percentage difference (%)
Slant	0.095	0.097	2.3
Base	0.094	0.096	4.0
Front	0.039	0.015	161.8
Viscous	0.049	0.055	11.4
Total	0.0265	0.0258	2.9

5. Result and Discussion

Figure 3 indicates that the rear-roof spoiler could have a positive or negative impact on C_d, depends on its pitch angle. At 0° and 5° pitch, its effect is favorable. Figure 4 compares the pressure coefficient (C_p) at the rear section of the model between the with- and without-spoiler cases. As seen, in the latter, the flow accelerates near the rear end of the roof. The higher airflow produces lower surface C_p at the slant section. On the other hand, the one with the 0° pitch spoiler is showing a relatively higher C_p at both the slant and base sections.
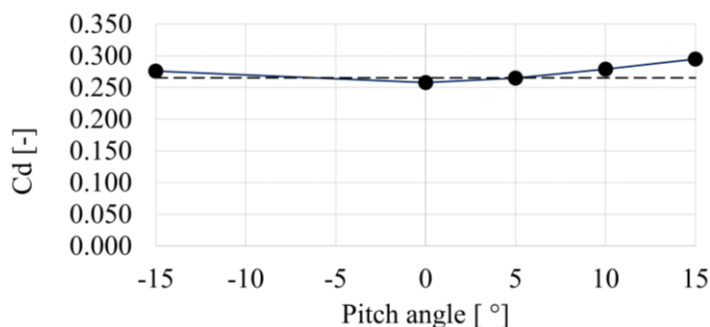

Figure 3. Graph of drag coefficient vs pitch angle of rear-roof spoiler; Dashed-horizontal line is without spoiler.

Another factor that contributes to the higher C_d value in the without-spoiler case is the shedding of longitudinal vortex structure at the side edge of the slant section (see figure 5). This vortex structure

which initiated at the upper corner of the slant has induced lower surface C_p along the side edge of the slant. In the with-spoiler case, however, the onset of the vortex structure has been prevented by the spoiler geometry.
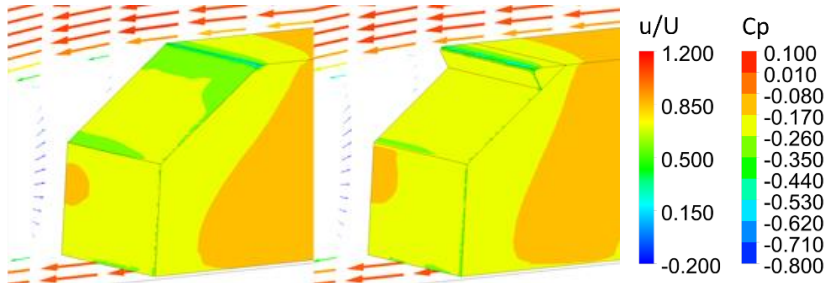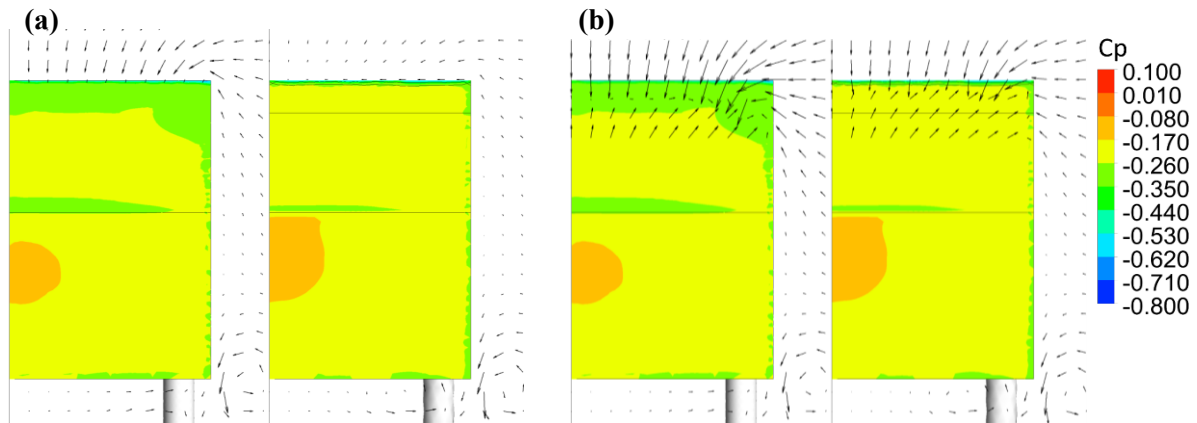

Figure 4. Rear section surface C_p distribution and centerline velocity vectors; Without spoiler (left); With 0° pitch angle spoiler (right).

(a)   **(b)**

Locations of visualization planes

Figure 5. Rear section surface C_p distribution and cross flow velocity vectors; Without spoiler (left); With 0° spoiler (right); Visualization plane locations (a) x = -80 mm from the base, and (b) -180 mm.
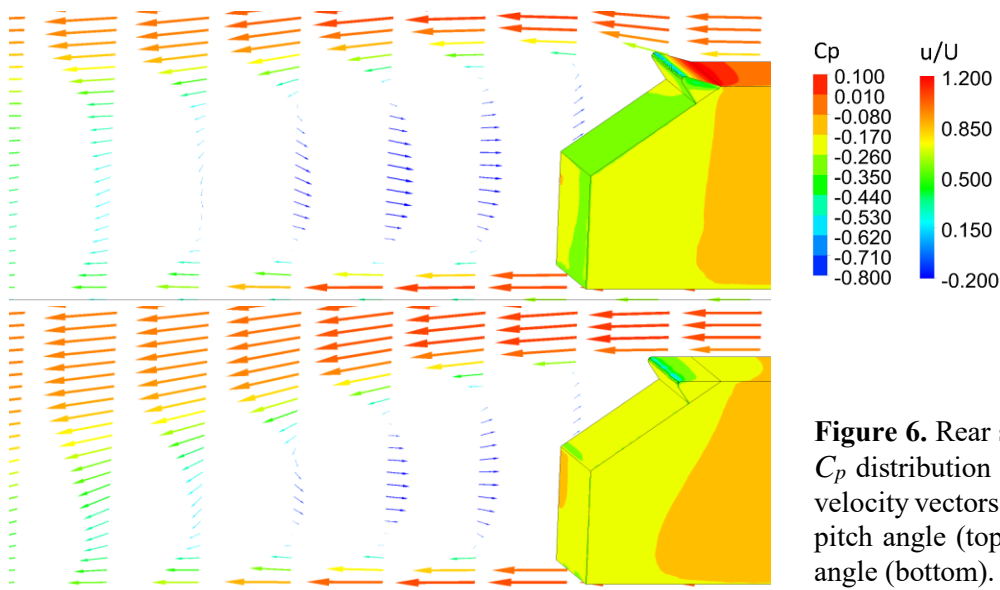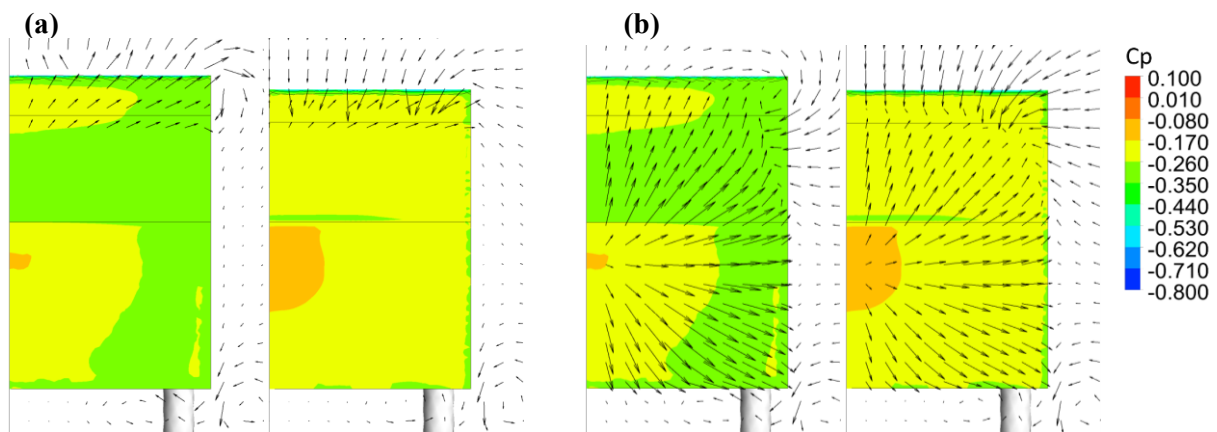

Figure 6. Rear section surface C_p distribution and centerline velocity vectors; Spoiler of 15° pitch angle (top), and 0° pitch angle (bottom).

In general, the C_d of the model is found to increase with larger spoiler angle for the angle range tested. However, the trend is relatively steeper from 5° onwards. The reason for the higher C_d value at larger pitch angle was partly due to the larger separation bubble behind the model (see figure 6). The larger separation bubble comprises stronger recirculating flow and upwash (see figure 7). Hence, lower surface C_p is induced on the slant, particularly around the lower part. Also notable in Figure 7 is the formation of the trailing vortices at the side edge of the spoiler at higher pitch angle. These vortices only occurred for spoiler angle larger than 5°, and are associated with strong upwash at the middle part of the slant.

The present study included a case of spoiler at negative inclination angle, i.e. -15°. This test case was carried out to verify the results found in the literature [7]. As shown in figure 3, similar tendency was observed in which the case of negative pitch angle does exhibit a higher C_d value than the 0° case.

The reason for the higher C_d value in the case of negative spoiler angle is to be twofold. First, the declining spoiler surface augments the flow near its leading edge (see figure 8). Consequently, the C_p at the entire upper surface of the spoiler drops. Secondly, similar to the slant section of the model, the declining spoiler surface generates side edge vortices. Strong side edge vortices are the characteristic of a high drag generating flow [8].
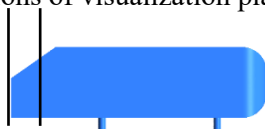

Figure 7. Rear section surface C_p distribution and cross flow velocity vectors; 15° spoiler angle (left), and 0° (right); Visualization plane locations (a) x = -120 mm, and (b) 10 mm.
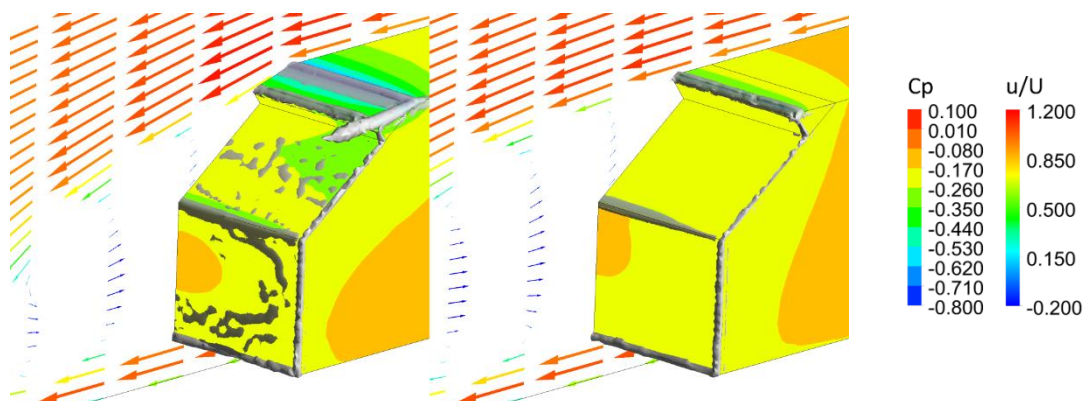

Figure 8. Rear section surface pressure distribution, centerline velocity vectors, and isosurface of Q criterion; Spoiler of -15° pitch angle (left), and 0° pitch angle (right).

6. Conclusions

The RANS-based CFD simulations have been carried up to investigate the effect of rear-roof spoiler on the aerodynamic drag performance of a simplified hatchback model. The results show that the simple strip-type spoiler could have a beneficial influence from 0° to 5° pitch angle. Beyond this range, the C_d increases with larger pitch angle. The spoiler relies on two main mechanisms to reduce aerodynamic drag. First, by preventing the airflow from accelerating at the leading edge of the slant section. Second, by preventing the formation of longitudinal vortices at the side edge of the slant section.

The present results were obtained from stationary simulations in which the motion of the vehicle body has not been considered. In practice, motion of vehicle body is common, and could change the inclination angle of the spoiler when the motion mode is of pitching. Hence, it helps to improve the realism of the simulation if the numerical method that incorporates the pitching mode of motion into the flow simulation [11] could be employed in the future studies.

References
[1]   Tsai C H, Fu L M, Tai C H, Huang Y L and Leong J C 2009 Computational aero-acoustic analysis of a passenger car with a rear spoiler *J. Appl. Math. Model.* **33** 3661-73
[2]   Daryakenari B, Abdullah S, Zulkifli R, Sundararajan E and Mohd Sood A B 2013 Numerical study of flow over ahmed body and a road vehicle and the change in aerodynamic characteristics caused by rear spoiler *Int. J. Fluid Mech. Res.* **40** 354-72
[3]   Kodali S P and Bezavada S 2012 Numerical simulation of air flow over a passenger car and the influence of rear spoiler using CFD *IJATP* **01** 6-13
[4]   Metz N 2001 Contribution of passenger cars and trucks to CO2, CH4, N2O, CFC and HFC Emissions *SAE Technical Paper* 2001-01-3758
[5]   Hu X X and Wong T T 2011 A numerical study on rear-spoiler of passenger vehicle *WASET* **57** 636-41
[6]   Sharma R B and Bansal R 2013 Drag and lift reduction on passenger car with rear spoiler *IJAuERD* **3** 13-22
[7]   Menon D P, Kamat G S, Mukkamala Y S and Kulkarni P S 2014 To improve the aerodynamic performance of a model hatchback car with the addition of a rear roof spoiler *16th Annual CFD Symposium* (Bangolore)
[8]   Ahmed S R 1981 An experimental study of the wake structures of typical automobile shapes *J. Wind Eng. Ind. Aerodyn.* **9** 49-62
[9]   Hucho W-H and Sovran G 1993 Aerodynamics of road vehicles *Annu. Rev. Fluid Mech* **25** 485-537
[10] Lienhart H, Stoots C and Becker S 2000 Flow and turbulence structures on the wake of a similified car model (Ahmed model) *DGLR Fach. Symp. der AG ATAB* (Stuttgart University)
[11] Cheng S Y, Tsubokura M, Nakashima T, Nouzawa T and Okada Y 2011 A numerical analysis of transient flow past road vehicles subjected to pitching oscillation *J Wind Eng Ind Aerod* **99** 511-22
      Cheng S Y, Tsubokura M, Nakashima T, Okada Y and Nouzawa T 2012 Numerical quantification of aerodynamic damping on pitching of vehicle-inspired bluff body *J Fluid Struct* **30** 188-204