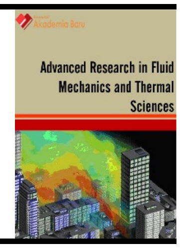
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Simulation Study on the Effect of Rear-Wing Spoiler on the Aerodynamic Performance of Sedan Vehicle

Open Access

Muhammad Zaid Nawam[1], Mohd Afzanizam Mohd Rosli[1,2,*], Nor Azwar Syahila Rosli[1]

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

ARTICLE INFO

Article history:
Received 6 June 2018
Received in revised form 5 July 2018
Accepted 6 August 2018
Available online 11 September 2018

ABSTRACT

There is a fact that the design of the vehicle affects 11% of fuel to overcome the drag force on a high-speed driving. The relatively high value of fuel consumption had urged the engineers in the field to improve the aerodynamics aspect of the vehicle. Furthermore, the shape of the vehicle especially sedan type has greatly affected by the lift force. Undesirable high value of lift force on an automotive vehicle could cause lack of stability and safety of the vehicle. In this study, the improvisation of the aerodynamic characteristics of a vehicle only is discussed in term of rear-wing spoiler design. There are four spoiler designs proposed, analyzed and compared to each other. The drag and lift coefficient value of the vehicle with an attached spoiler and the pressure and velocity distribution are evaluated in determining the best spoiler design. The flow characteristics were analyzed using ANSYS FLUENT®. The CFD technique used is validated by using the Ahmed body as a benchmark. It shows a good agreement which is less than 10% percentage difference. Besides, a grid convergence study is also conducted to test the mesh resolution and examine the turbulence model uncertainties. All generic car and spoiler are created in CATIA®. The base car model is attached with various spoiler designs and tested to gather its data and result. The analysis conducted shows that there is an increment in the drag coefficient of the car of 47.20% when attached with a spoiler. However, the lift coefficient value shows a tremendous declination from 0.03957 to -0.16254. The result of this study show that an attached spoiler could increase the drag coefficient and decrease the lift coefficient of a car. The best spoiler should have a least increment in the drag coefficient and considerable decrement in the lift coefficient.

Keywords:
CFD, rear-spoiler, drag coefficient, lift coefficient

1. Introduction

When a vehicle is moving, it is cutting through an air. By imagining walking in the swimming pool, it is the same thing happens when moving in the air, just the effect is not so significant compared to moving in water. But when saying about a high-speed movement even in the air, for example, 100km/h that is an average speed of today's car, the design of the car plays an important role for the efficiency and stability of the movement. According to Das and Riyad [1], the aerodynamic drag is the

* *Corresponding author.*
E-mail address: afzanizam@utem.edu.my (Mohd Afzanizam Mohd Rosli)

governing form of resistance when the vehicle runs at speeds of 80km/h or greater, especially considering the fact that 65% of the power required at 110 km/h is consumed due to overcoming aerodynamic drag [2]. Too many drag forces on the car body could cause high waste in engine energy consumption while too much lift forces could cause the car lacks in the stability and dangerous to be driven. According to Agarwal [3] approximately 50% of the mechanical energy of the vehicle is wasted to overcome drag at highway speed of nearly 88.5 to 96.5 km/h. In 1997, the fuel consumption of the Class 8 trucks reached 18 billion gallons in which 65% of this fuel consumption was wasted to overcome the aerodynamic drag [4].

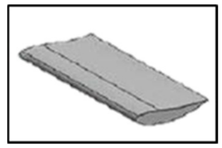
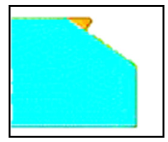
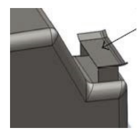
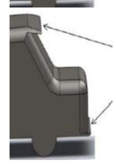
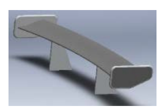
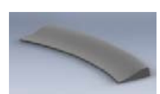
Usually, the lift force affects mostly on the rear of the car especially a sedan car because of its shape. Moving at high speed causes the high-speed air to move following the body shape of the car causing high pressure on the front hood follows by front windshield. But arriving at the rear of the car, the air suddenly loses its track and causing the airflow to be turbulence and producing a low-pressure region at the car's rear. This low pressure is dangerous as it could cause the rear of the car to lift and the tires lose its traction to the road and can cause an accident.

Present automotive vehicle shape had gone through a few decades of studies and development. This including in the modification of the vehicle frame and chassis as the evolution of the car shape from tear-drop shape to streamline design until the common shapes of the car that are present today. In fact, the effect of CL mainly come from the spoiler and roof [5]. Besides, the refinement of the car bodies such as smoothen underbody, reduced sharp edges and corners also have been applied in designing the vehicle bodies. Moreover, the alternatives of adding some aerodynamic devices to the vehicle’s body also had been applied. One of the devices is a rear-wing spoiler. According to [6], these improvisations had greatly improved the aerodynamic performance of the vehicle in terms of better fuel consumption, minimized noise, preventing undesired lift force and minimizing other causes of aerodynamic instability at high speed. According to Pankajakshan [7], every 0.01 reduction in the CD figure can cut fuel consumption by up to 0.4-litres/100km at 130km/h.

A rear-wing spoiler is a device that is equipped at front or rear of the vehicle to improve the aerodynamic and stability of the vehicle by minimizing the unfavorable movement of airflow around the vehicle’s body [1]. According to Cheng and Shuhaimi [5], there are two types of common rear spoiler equipped on a car. The spoiler types are free-standing wing spoiler and strip spoiler. A rear spoiler is commonly installed upon the trunk lid of the passenger vehicle [8]. However, they also may be equipped at the rear edge of the vehicle’s roof. The attached spoiler could minimize the turbulent airflow occurred at the rear of the vehicle. In addition, the spoiler nowadays is no longer only a decoration but it has a significant and measurable effect on aerodynamic characteristics optimization of the vehicle.

This paper focused on determining the best spoiler design based on the selected variation of spoilers from the previous studies. The base car model used in this paper is a sedan-type as it is the most commonly driven vehicles nowadays [8]. Hence, a simplified generic sedan model with an attached spoiler was used for the simulation. Referring to the literature study on the effect of the rear spoiler, most of the study was conducted by attaching the spoiler with the positive inclination angle. The finding of the studies shown an improvement in the performance of the car in term of the drag and lift characteristics. However, in this study, the spoiler angle was positioned at 00 inclination angle to study the consequences. All of the four spoilers are attached at 00 angle and fixed clearance height and position from the rear window. The results of attaching rear spoiler from the previous study are shown in Table 1.

Table 1
Comparison of spoiler design

Title	Angle of attack (^{0})	Design	% Reduction C_D	% Reduction C_L	Remark
A Numerical Study on Rear-spoiler Of Passenger Vehicle (2017) Xu-Xia Hu, Eric T.T. Wong	5, 10		1.74	3.90	Sedan
Influence of rear-roof spoiler on the aerodynamic performance of hatchback vehicle (2016) S. Y. Cheng, M. Shuhaimi	-10, -5, 0, 5, 10		3	488	Ahmed Body
Drag Reduction of Passenger Car Using Add-On Devices (2014) Ram Bansal and R. B. Sharma	12		2.02	6	Sedan
			3.87	16.62	Sedan
CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction (2013) R. B. Sharma, Ram Bansal	12		3.87	16.62	Sedan
A Numerical Study on Rear-spoiler Of Passenger Vehicle (2012) Mustafa Cakir	5		17.24	7.66	Sedan
			6.47	20.72	Sedan

2. Model and Computational Domain

2.1 Passenger Vehicle Generic Model

The process of creating the generic car model involves a computer-aided design (CAD) software, CATIA. The uses of the CAD software assist in defining the topology of the fluid flow region of interest. The software is majorly used in designing and optimizing process during conducting this study. A generic model of a passenger vehicle is shown in Figure 1. The dimension of the car was created based on the available dimension of the Ahmed body to ensure the relevance of the car dimension. In this study, the car model is simplified by removing the wheels and rear-view mirrors while the bottom part of the car is assumed flat surface. There are four different types of spoiler designs that were attached to the car individually one at a time with a zero-degree inclination angle. The generic car, as well as the four spoilers, are shown in Figure 1 as follows:

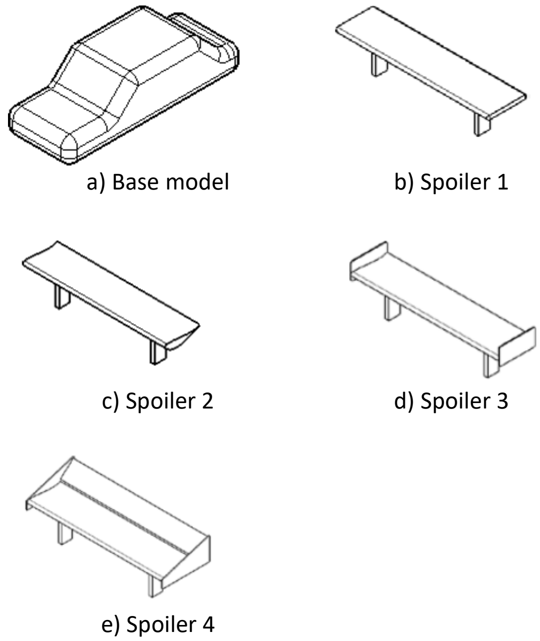


Fig. 1. (a) 3-dimension generic passenger vehicle model (b) spoiler model 1 (c) spoiler model 2 (d) spoiler model 3 (e) spoiler model 4

2.2 The Computational Domain

In ANSYS Design modeler, a box enclosure as shown in Figure 2 is created around the model after subtracting the generic car model from the air domain using Boolean. The purpose of this large air domain is to simulate the wind tunnel while avoiding artificial air acceleration due to squeezing of air through the narrow gap between the car's body and the wall. The velocity inlet surface of the car is situated at 5 times the vehicle length (5000mm) and 7.5 times (7500mm) at the outlet following as the previous study [6]. Furthermore, the top and side wall of the air domain is set at 3 times length (3000mm) of the car. Also, there is symmetry surface generated in the YZ plane to save the resources in processing stage as the generic model is symmetry in shape. Besides, the top and side surfaces are also set as symmetry while the bottom wall that renamed as "road" is set as non-slip wall condition. The generic model is set at 50mm above the floor.

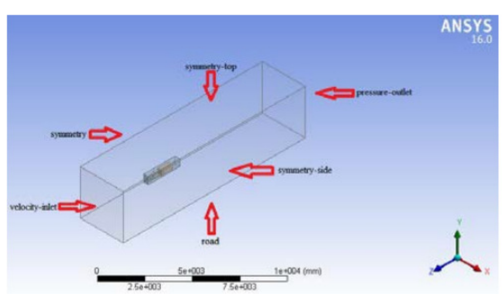


Fig. 2. Boundary condition of the computational domain

3. Formulations

3.1 Meshing

In this case, the “finite element method” technique is used. The ANSYS Meshing tool is used to set up the meshing. A proper meshing and iteration needed to obtain the optimum result are applied to the generic car. Since the simulation is more focused on the rear side of the vehicle where the wake of the airflow phenomenon occurs, there is necessary enough space kept in the rear portion of the car. Coarse meshing is applied to the air domain. Then, refinement is applied to the area and edges needed. The final mesh result of the 3D grid as shown in Figure 3 contained about 398,004 number of nodes and 1,830,143 tetrahedral elements.

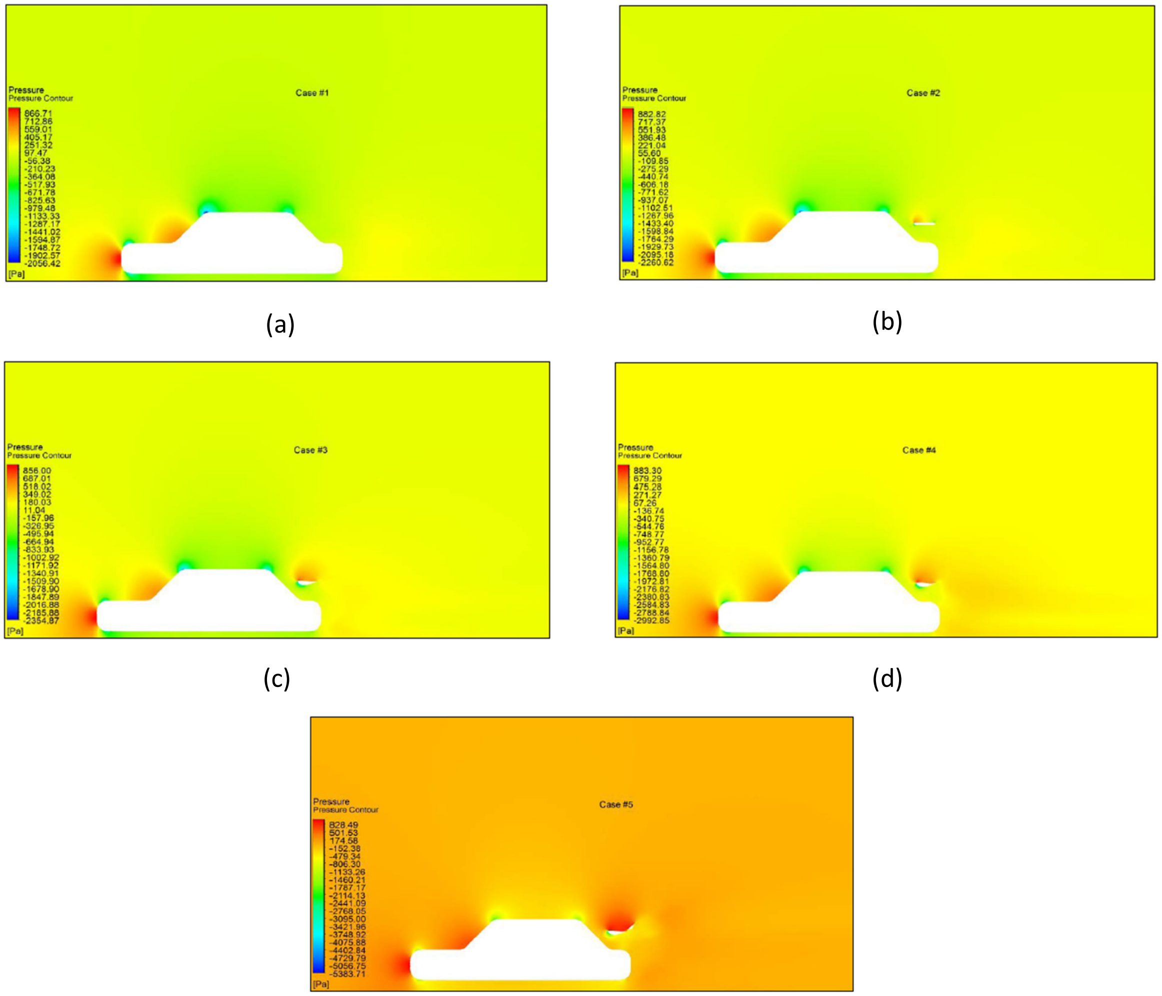
3.2 Numerical Simulation

Steady-state Reynolds Average Navier-Stoke equations are solved using ANSYS-FLUENT®. According to Sitlani [9], the benefit of the Realizable k-ε model is that it more accurately predicts the spreading rate of both planar and round jets. It also likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separations and recirculation. The fluid medium is air with an inlet velocity of 40m/s at a Reynolds number equal to $10*10^5$. The pressure at the outlet is considered constant at atmospheric pressure. The inlet and outlet turbulence level is set at 1% and 5%. The generic car and the road are set as “no-slip wall” boundary condition. The turbulence used in this case is Realizable k-epsilon with a non-equilibrium wall functions. Because of the capability to partly account for the effects of pressure gradients and departure from equilibrium, the non-equilibrium wall functions are recommended for use in complex flows involving separation, reattachment, and impingement where the mean flow and turbulence are subjected to severe pressure gradients and change rapidly [9]. The first order upwind scheme is set for pressure discretization in the initial iterations and changed to the second order upwind for better convergence and stability.

4. Results and Discussion

4.1 Colour Contour Analysis

The pressure distribution analysis on the surface of the vehicle is done using Bernoulli’s equation. The contour of static pressure distribution shown in Figure 4 indicates the level of pressure effect on the surface of the car. Observation on the contour of the pressure distribution around the base model in Figure 4 obviously shows that there are two areas with positive pressure. The areas are at the front of the car’s body and the other one is in the area between the front hood and the windshield. On the other hand, there are three areas with negative pressure can be found on the generic car model. The areas are at the front and rear end of the roof and at a small area of the front hood. However, when the spoilers are attached to the car, the areas of positive pressure increase at the spoiler area. This improves the lift coefficient of the car as more pressure on the upper part of the car. This result on higher negative lift coefficient of the car.



(a) (b) (c) (d) (e)

Fig. 4. (a) Distribution of pressure on the base model (b) Distribution of pressure on the base model attached with spoiler 1 (c) Distribution of pressure on the base model attached with spoiler 2 (d) Distribution of pressure on the base model attached with spoiler 3 (e) Distribution of pressure on the base model attached with spoiler 4

The concept of pressure drags happen when there are two different areas with different level of pressure. In this case, the area on the bottom of the car's body is relatively higher than the pressure level at the top of the car. The factor of the pressure difference could contribute to the high lift coefficient of the car. The high lift coefficient could cause pitching moment on the rear of the car to lift up from the ground which will make the rear wheels traction on the road hence causing the car to lost control. In the simulation of the base model of the sedan car, the final predicted drag and lift coefficient are 0.19230 and 0.03957.

The airflow over the car body creates a velocity distribution which results in the aerodynamic loads acting on the body of the generic model. The velocity distribution indicates the amount of velocity at different portions of the vehicle. In addition, the purpose of the rear spoiler is to spoil the velocity direction to reduce the lift force on the car thus improving the aerodynamic performance of the car. Moreover, the velocity value is directly proportional to the value of the drag and lift coefficient. Thus, the velocity distribution around the car gives a big impact on the aerodynamic performance of the car. The airflow separation shown in Figure 5 causes the airflow to become turbulent, hence lower pressure region formed at the back of the car. This will cause a negative effect on the car aerodynamic performance and stability. The addition of the spoiler could make the air "see" becomes longer, smoother flow from the roof to the spoiler, that helps to delay the airflow separation. Thus, the aerodynamic performance of the car could be better.

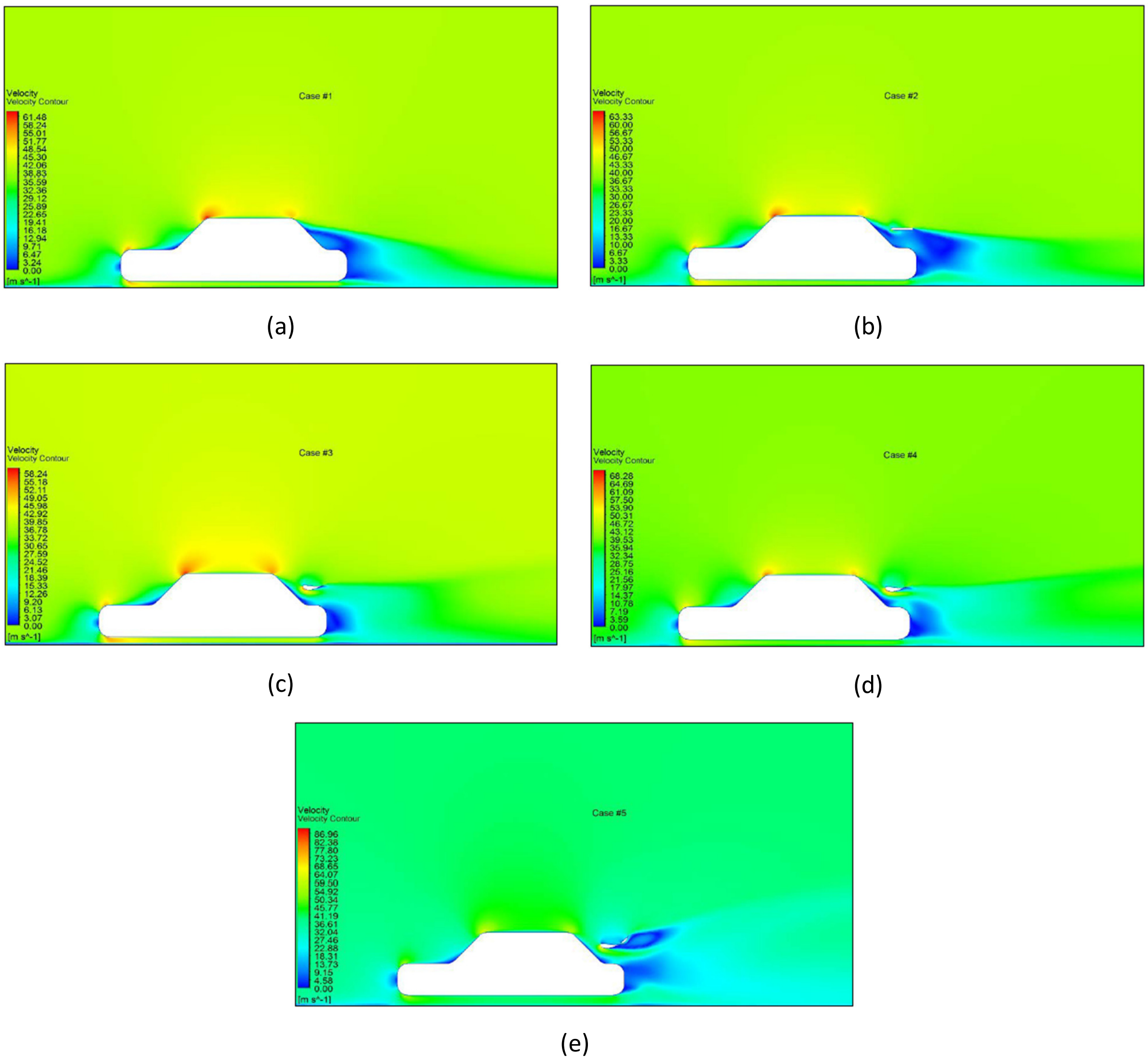


Fig. 5. (a) Distribution of velocity on the base model (b) Distribution of velocity on the base model attached with spoiler 1 (c) Distribution of velocity on the base model attached with spoiler 2 (d) Distribution of velocity on the base model attached with spoiler 3 (e) Distribution of velocity on the base model attached with spoiler 4

4.2 Comparison of Rear-Spoiler Design

In this section, the computational results of the following cases are presented and discussed.

All the results are obtained with the same meshing resolution and setup, the standard realizable *k*-epsilon model and also the same boundary conditions. Drag and lift coefficient for all the three cases are presented in Table 2. The reduction rate of the drag and lift coefficients did not depend on the mesh resolution, turbulence model and air domain size.

Table 2
Drag and lift coefficient reduction rate on different design of rear spoiler

Model	C_D	Percentage change	C_L	Percentage change
Case 1	0.19230	-	0.03957	-
Case 2	0.24361	+26.68%	0.03902	-1.39%
Case 3	0.28283	+47.07%	-0.13464	-440%
Case 4	0.28307	+47.20%	-0.16254	-510.77
Case 5	0.48246	+150.89%	-0.74901	-1992.87%

Case 1: results for the base model, Case 2: results for the base model with attached spoiler 1, Case 3: results for the base model with attached spoiler 2, Case 4: results for the base model with attached spoiler 3, Case 5: results for the base model with attached spoiler 4

5. Conclusion

In conclusion, the drag and lift characteristics of a sedan car with and without a spoiler situation were numerically simulated. The grid independence study shows that the k-epsilon model is the most appropriate turbulence model for external flows around the car body as other model shows an inconsistent C_D and C_L results and lack of converged solutions. Besides, the coarse and medium meshing are also not enough to get precise results. The applied fine meshing with the addition of volume box meshing on the base model has shown the best and the most closer results to the case where the spoilers are attached.

The results of the simulations with the third spoiler design (case 4, an airfoil with end plate) has shown an increment of the drag coefficient from 0.19230 to 0.28307, which is 47.2% of increment. While the C_L value of the car has shown a decrement from 0.03957 to -0.16254 which is about -510.77% reduction. By comparing the velocity distribution on the car body, it was found the recirculation zone above the rear window was almost gone by using a spoiler in case 4. The air flow shows a smooth and gentle sloped above the rear window. Higher negative force is more important than having lower drag coefficient in this case as stability and safety of the car is a priority.

Acknowledgement

The author would like to thank Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka for allowing the utilization of the Computational Fluid Dynamics (CFD) laboratory.

References

[1] Das, Rubel Chandra, and Mahmud Riyad. "CFD Analysis of Passenger Vehicleat Various Angle of Rear End Spoiler." *Procedia engineering* 194 (2017): 160-165.

[2] Mashud, Mohammad, and Rubel Chandra Das. "Effect of rear end spoiler angle of a sedan car." In *AIP Conference Proceedings*, vol. 1851, no. 1, p. 020017. AIP Publishing, 2017.

[3] Agarwal, Ramesh. "Sustainable Ground Transportation: Technologies, Challenges and Opportunities." In *ASME 2013 7th International Conference on Energy Sustainability collocated with the ASME 2013 Heat Transfer Summer Conference and the ASME 2013 11th International Conference on Fuel Cell Science, Engineering and Technology*, pp. V001T13A003-V001T13A003. American Society of Mechanical Engineers, 2013.

[4] Pankajakshan, Ramesh, Brent Mitchell, and David L. Whitfield. "Full-scale simulations of drag reduction devices for class 8 trucks." In *The Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains*, pp. 339-348. Springer, Berlin, Heidelberg, 2009.

[5] Cheng, See-Yuan, and Shuhaimi Mansor. "Influence of rear-roof spoiler on the aerodynamic performance of hatchback vehicle." In *MATEC Web of Conferences*, vol. 90, p. 01027. EDP Sciences, 2017.
[6] Banga, Saurabh, Md Zunaid, Naushad Ahmad Ansari, Sagar Sharma, and Rohit Singh Dungriyal. "CFD simulation of flow around external vehicle: Ahmed body." *Journal of Mechanical and Civil Engineering* 12, no. 4 (2015): 87-94.
[7] Pankajakshan, Ramesh, Brent Mitchell, and David L. Whitfield. "Full-scale simulations of drag reduction devices for class 8 trucks." In *The Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains*, pp. 339-348. Springer, Berlin, Heidelberg, 2009.
[8] Chu, Julio, and James M. Luckring. *Experimental surface pressure data obtained on 65 delta wing across Reynolds number and Mach number ranges*. National Aeronautics and Space Administration, Langley Research Center, 1996.
[9] Sitlani, Manish P., and Kendrick Aung. "Numerical Simulations on Aerodynamic Drag of Ground Transportation System (GTS) Model." In *ASME 2006 International Mechanical Engineering Congress and Exposition*, pp. 1045-1053. American Society of Mechanical Engineers, 2006.
[10] Meile, Walter, Ewald Reisenberger, M. Mayer, B. Schmölzer, W. Müller, and Günter Brenn. "Aerodynamics of ski jumping: experiments and CFD simulations." *Experiments in fluids* 41, no. 6 (2006): 949-964.
[11] McKay, Noah J., and Ashok Gopalarathnam. *The Effects of Wing Aerodynamics on Race Vehicle Performance*. No. 2002-01-3294. SAE Technical Paper, 2002.
[12] Kim, Min-Ho. "Numerical study on the wake flow characteristics and drag reduction of large-sized bus using rear-spoiler." *International journal of vehicle design* 34, no. 3 (2004): 203-217.
[13] Selvakumar, K., and K. M. Parammasivam. "EXPERIMENTAL INVESTIGATIONS ON OPTIMISATION OF AERODYNMAIC CHARACTERICSTICS IN A HATCHBACK MODEL CAR 3 USING VORTEX GENERATORS." In *The Eighth Asia-Pacific Conference on Wind Engineering, Chennai, India*. 2013.
[14] Bansal, Ram, and R. B. Sharma. "Drag reduction of passenger car using add-on devices." *Journal of Aerodynamics* 2014 (2014).
[15] Shinde, Parashar V., Arshad Shaikh, Saad Mirza, Haidar Khan, and Irshad Shaikh. "Analysis Of the Spoiler."