Design and Development of Hybrid Electric Vehicle Rear Diffuser

Rosli Abu Bakar
Devarajan Ramasamy
Fazli Ismail
Ahmad Syahrizan Sulaiman

ABSTRACT

Next in line for alternatives, hybrid electric vehicles (HEV) are the most logical step in reducing pollutions. However, an efficient model needs to be developed which can adapt latest technologies so that the vehicle is more efficient on the road. Under the program of "Series Hybrid Vehicle Cost-Effective Powertrain Components Development" some aerodynamic studies were carried out. The aim was to reduce energy consumption of the battery module by reducing wind drag and weight while retaining the original structure of the model. In the attempt to reduce the drag of the vehicle certain aerodynamic modifications was targeted. The modifications were investigated so that only proven results were chosen for the model. One of these modifications is the rear diffuser. Its idea was to reduce lift of the car and also some weight from the luggage compartment. The task was done by doing a computational fluid dynamic (CFD) analysis for expected vehicle speed of 50-100 km/h. A drag force was found based on Finite Element Method (FEM) based on inputs from CFD. This force was calculated to produce the drag coefficient of the car as a whole. The approach needed to justify the amount of drag that can be reduced by addition of a rear diffuser as compared to the model without a diffuser. Pressure profile of the model was analysed together with velocity plot around the model. The reduction of lift and flow separation is the key results that will be a point of discussion. The justification made enabled the design to proceed on the actual car.

Keywords: *Computational Fluid Dynamic (CFD), diffuser, drag, weight reduction*

Introduction

A series hybrid electric vehicle was developed in Universiti Malaysia Pahang (UMP) as in Figure 1. It is currently designed with efficiency in mind. The importance of the research was to incorporate energy saving by various method of reducing traction friction from the wheel and also to reduce the drag by aerodynamic losses. Aerodynamic losses generally increase in quadratic with speed from fluid mechanics. The platform for the model was started using a Proton Iswara Hatchback model. The model was taken as it is also one of the commonly used passenger car in Malaysia. The car has been optimized by the manufacturer by having rounded body corners, raked windows and hatchback as found by Masaru et al, (2004). However, the passenger car has to be large enough to accommodate people, the power train and support components, making it extremely difficult to realize an aerodynamically ideal body shape. The car is, therefore, obliged to have an aerodynamically bluff, not an ideal streamline shape as seen on fish and birds. Such a body shape is inevitably inefficient in nature to reduce drag along with a flow separation from the rear end of the body. Flow visualizations can help to understand the nature of the flow. Generally tuft and wind tunnels can be used to visualize the flow. Nowadays, CFD can be used as the technology of computer simulation has evolved to produce good preliminary results for designers before the actual fabrication. Based on the visualized flow the drag of the vehicle can be estimated and computed. This will cut down costs as well as time needed to design the preliminary models. Many variations can be included from CFD that is hard to be done in actual conditions. The passenger car body's aerodynamic bluffness, when expressed by the drag coefficient (CD), is generally between 0.2 and 0.5.
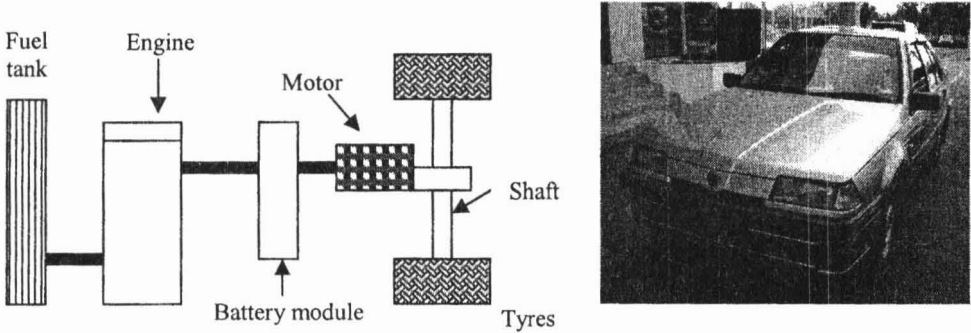
Figure 1: Hybrid Powertrain in UMP a Proton Iswara

In order to reduce the drag, new technologies can be adopted in the current body structure for purpose of maintaining the current design. The available technologies are by Colin and Wael (2005) as front air dam, rear spoiler for the upper body of the car and a diffuser for car lower body. Diffusers are generally used to increase down force of the car and to reduce drag in high performance racing cars such as F1 (C.K. Van Steenberg, pg. 9, 2004). The concept of using a diffuser in a passenger car and how far a reduction of drag is obtained is the key interest in this study.

Literature Review

In a moving common car there are constantly forces acting to the car in which causes drag. It is dependant on the geometry of the body, motion of the body and the fluid in which it is traveling. The general forces found are drag and lift.

Figure 2: Distribution of Lift and Drag for a Car With frontal Area, A
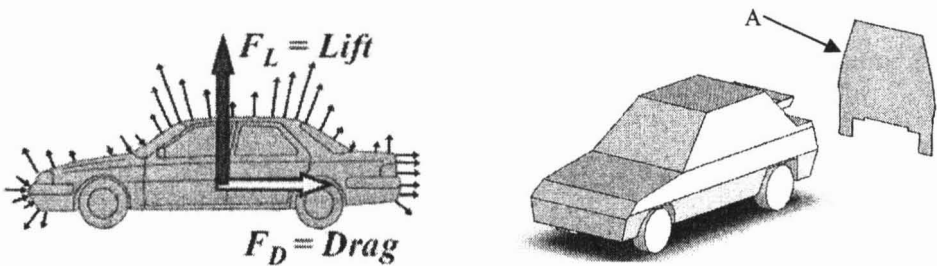
This can be proved from fluid mechanics (Cengel and Cimbala, 2006) as:

$$F_D = C_D \frac{1}{2} \rho V^2 A$$

(1)

$$F_L = C_L \frac{1}{2} \rho V^2 A$$

(2)

Where, F_D and F_L are both the drag and lift forces, C_L and C_D are the drag and lift coefficients, r is the fluid density, V is the object velocity and A being the frontal area projected. All of the annotations are distributed from the car body as shown in Figure 2. As a normal car produces lift at the rear end, a modification of the rear area is needed to reduce the rear lift. This is where the

diffuser can be applied.

CFD generally solves fluid motion by solving the Navier-Stokes equation of mass, momentum and energy equation. The three equations can be written in the conservation form as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}(\rho u_k) = 0 \qquad (3)$$

$$\frac{\partial \rho u_i}{\partial y} + \frac{\partial}{\partial x_k}(\rho u_i u_k - \tau_{ik}) + \frac{\partial P}{\partial x_i} = S_i \qquad (4)$$

$$\frac{\partial(\rho E)}{\partial y} + \frac{\partial}{\partial x_k}((\rho E + P)u_k + q_k - \tau_{ik}u_i) = S_k u_k + Q_H \qquad (5)$$

where u is the fluid velocity, r is the fluid density, S_i is a mass-distributed external force per unit mass, E is the total energy per unit mass, Q_H is a heat source per unit volume,   is the viscous shear stress tensor and q_i is the diffusive heat flux. τ_{ik}

While CFD gives the fluid energy mass and momentum, the forces on a car body   due   to fluid movements can now be identified through integration of CFD and FEM. The results of CFD can be turned into pressure force as an input for FEM.

Methods and Materials

A model of the vehicle body was made by using data obtained from Mark Coombs and Spencer Drayton (2003) and also from raw measurements using combination of meter ruler, L-Ruler, vernier calipers and strings for radius (Amir Shidique, p39, 2007). The model is shown in Figure 3.
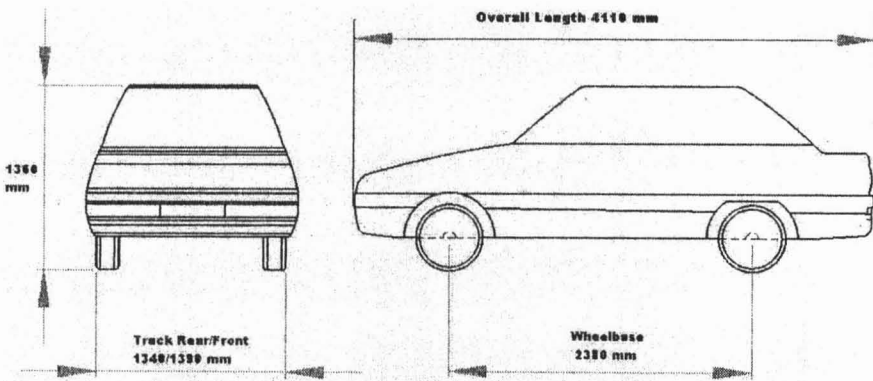

Figure 3: HEV Model Dimensions

The best chosen angle for the diffuser was 5° as found by Wolf-Heinrich Hucho(1998). This angle is seen as the best angle for reduced drag coefficient and lift coefficient in a car body end. The model rear was cut according to height and length available in the actual vehicle. A cut on the spare wheel housing was targeted to meet the 5° angle of the diffuser. The initial shape of the diffuser is as shown in Figure 4.
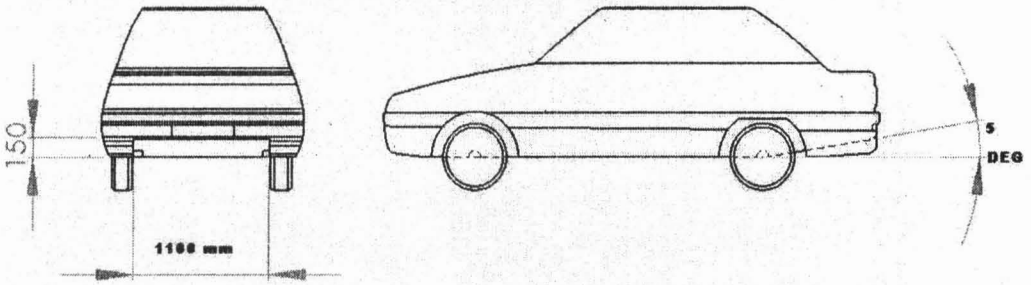
Figure 4: HEV Diffusser Dimensions

After the model parameter were finalized, the analysis moved on to CFD. Here, the model was laid a boundary condition for external flow. The low is considered both as laminar and turbulent with a CFD model. Figure 5 shows the model with the boundary condition.
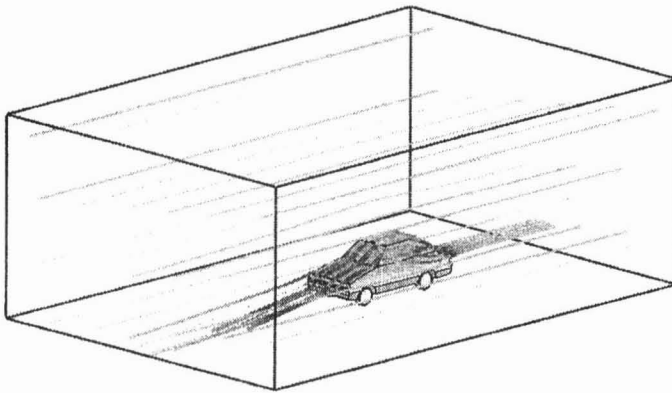

Figure 5: CFD Boundary Condition with Stream Lines

For the FEM analysis a rigid body was assumed. The restrictions were given to the tyres and assumed the whole body was tied as in a wind tunnel. The boundary for the FEM analysis is as Figure 6. The boundary makes sure that the vehicle tyres can not move vertically or sideways, but the vehicle tyres can move backwards. The rear part of the vehicle is modeled as a weld joint so that the body is supported by the horizontal forces from the X-axis only. The pressure from the CFD simulation is applied and a force reading is obtained from the rear support. This is the drag force exerted by the fluid and is used to find the drag coefficient of the car.
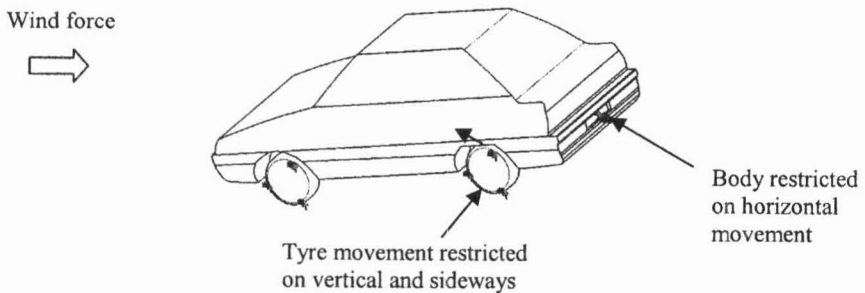

Wind force

Body restricted on horizontal movement

Tyre movement restricted on vertical and sideways

Figure 6: FEM Model With Restriction as in a Wind Tunnel

Results and Discussion

The process of results collection started from CFD. The car body was subjected to vehicle speed ranging from 60kmh to 100kmh. The plot was seen from the center line of the car as the results are clearer and are generally a point of discussion from various researchers (Zhigang and Bahram, 2005). The plots in Figure 7 show changes in the flow stream that indicates a slightly smaller flow separation condition at the vehicle rear. The blue color regions are the separation regions where the velocity of the flow is reversed this is also know as the wake area. This is as found by Kevin and Geoffrey (2005) by doing oil topology tests at the wake area. There is only a slight difference of the condition at the low speed of 60kmh. This may be due to the aerodynamic force not being strong enough at these speeds. As the speed increased the rear flow separation is curved upwards from the flow higher, flow velocity obtained from the diffuser 5° angle. Clearly this is seen on the 100kmh model. The reduction of the separation angle is a good indication that the vehicle can move more efficiently in the wind at the same velocity as the model without diffuser. This will reduce the drag of the car.
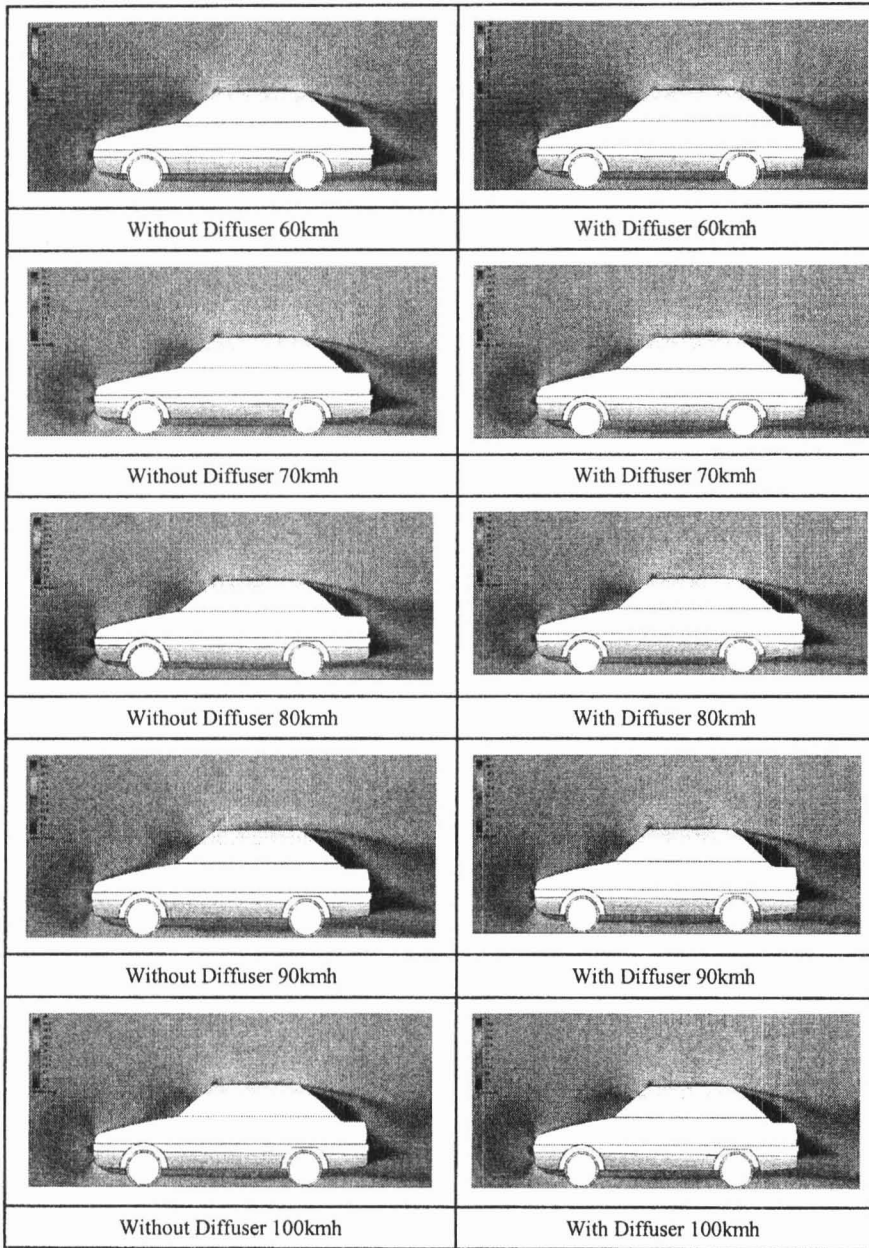
Figure 7 Velocity Plot on the Vehicle Center Line at Various Speeds

The velocity plot is also seen for the vehicle model from the under body of the car. As seen on the center plot the velocity from underbody is pushing some rear flow to become not separated as compared to the model without the diffuser. Generally, the under body flow is changed by this and it is wise to look into the region. Figure 8 shows how the region is affected. There is more separation at the rear wheel in the diffuser model as compared to the model without.

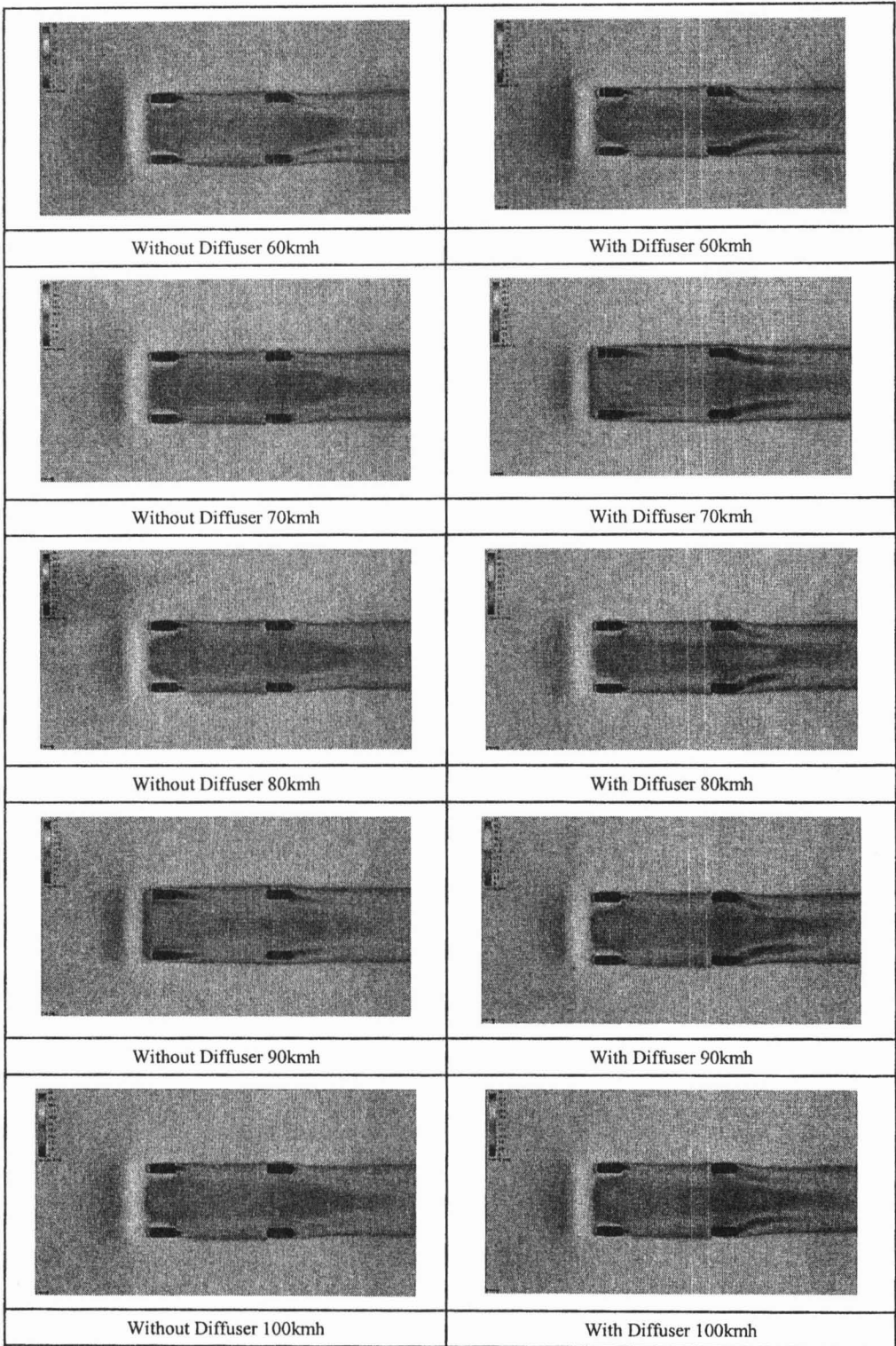|	
Without Diffuser 60kmh	With Diffuser 60kmh
Without Diffuser 70kmh	With Diffuser 70kmh
Without Diffuser 80kmh	With Diffuser 80kmh
Without Diffuser 90kmh	With Diffuser 90kmh
Without Diffuser 100kmh	With Diffuser 100kmh

Figure 8: Velocity Plot on the Vehicle Underbody at Various Speeds

The denser low velocity region with blue colors shows that change has occurred due to addition of the diffuser to the flow field. Apparently, at low speed of 60 kmh and 70 kmh it is also to be noted that the frontal under body plot is having a trend of higher velocity region from the

yellow color for the model without the diffuser. With a diffuser there is a slight lower velocity gain at the frontal section.

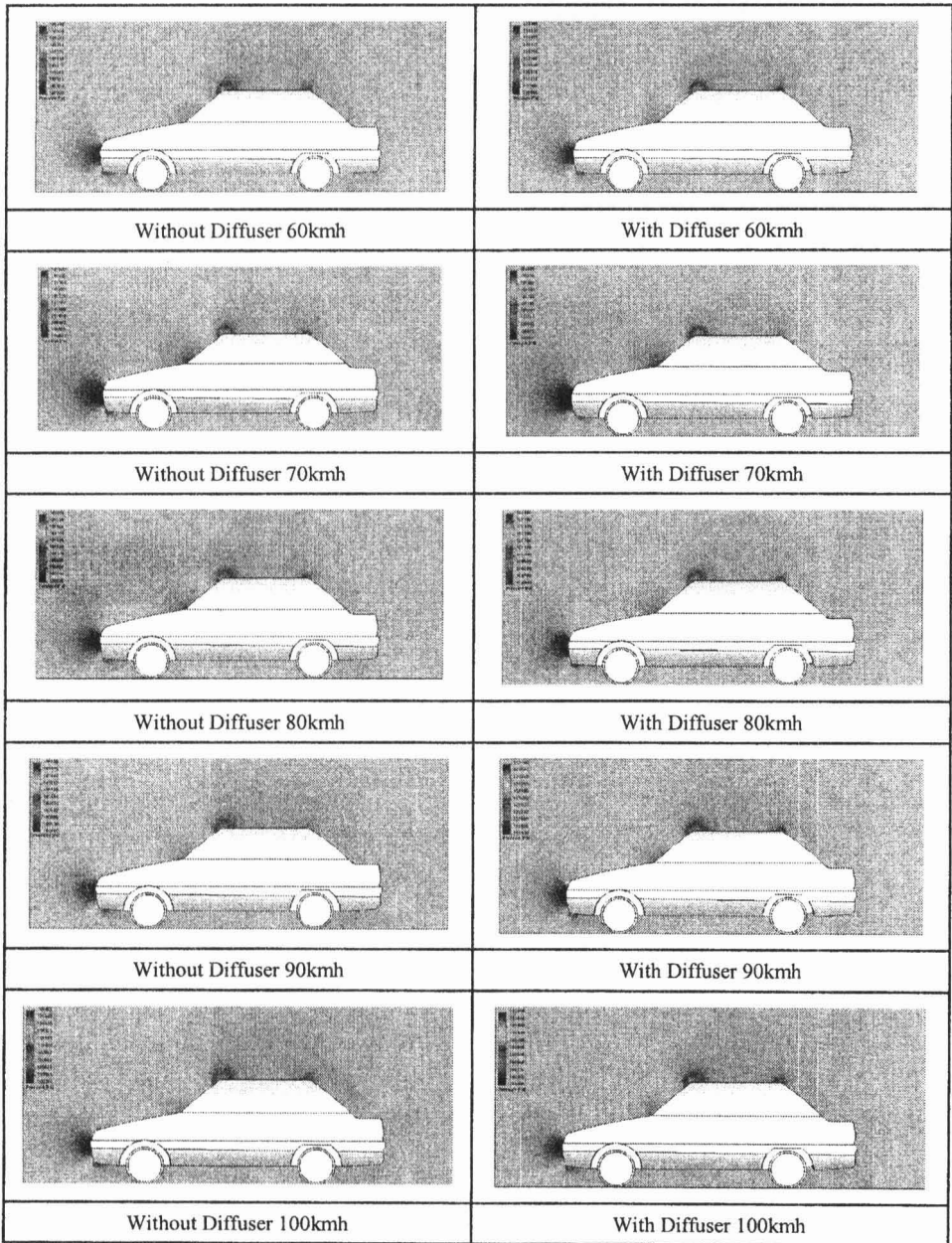|	
Without Diffuser 60kmh	With Diffuser 60kmh
Without Diffuser 70kmh	With Diffuser 70kmh
Without Diffuser 80kmh	With Diffuser 80kmh
Without Diffuser 90kmh	With Diffuser 90kmh
Without Diffuser 100kmh	With Diffuser 100kmh

Figure 9: Pressure Plot on the Vehicle Center Line at Various Speeds

After looking at the velocity plot a consideration of the pressure distribution is also required and most of the drag on a vehicle is in the form of pressure drag as shown in Figure 9. The model without diffuser shows a constant built up of low pressure region below the bumper. This is seen from the green contour increasing as the model is seen from 60kmh to 100kmh. Oppositely, for the model with a diffuser the low pressure region is not seen at 60kmh. It is only seen on 80kmh until 100kmh. The density of the low pressure region is also lower than the model with diffuser at any speed.

Finally, the result from FEM is obtained and a general graph is seen in Figure 10. Two lines were found and zooming into the drag coefficients found that the value decreased with the model

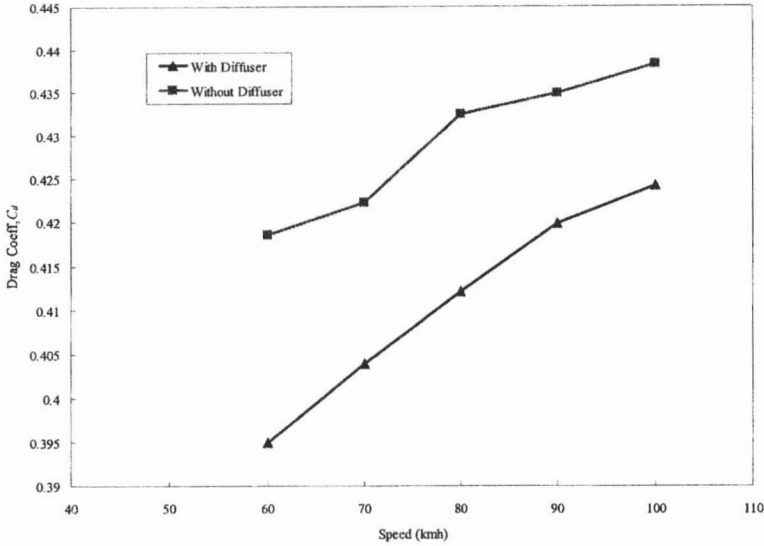with a diffuser. An overall reduction of 4.25% is seen in the drag coefficient.

Figure 10: Drag Coefficient Versus Speed (kmh)

Conclusion

The conclusion of the paper can be gathered from the flowing points:

a) The application of a diffuser in a car body design has the potential to reduce the drag of the vehicle. This is due to the device shape that causes more flow to be separated near the ground rather than at the back of the car.

b) The use of the diffuser has an average drag coefficient of 0.411 as compared to 0.43 for the normal car. This is a form of energy saving of 4.25% and gives a better aerodynamic characteristics to the car.

c) Further measurements of pressure and velocity will prove the theoretical finding from CFD.

d) Combination of CFD and FEM gives a good approximation for the drag and drastically reduces cost for the development stage of the HEV car.

References

Masaru Koike, Tsunehisa Nagayoshi & Naoki Hamamoto. (2004). Research on aerodynamic drag reduction by vortex generators. *Mitsubushi Motors Technical Review*.

Colin, P. B. & Wael, A. M. (2005). *Comparison of aerodynamic measurements on a full-scale NASCAR in open and closed jet test section*.

Van Steenberg, C.K.(2004). *Vortices in favorable pressure gradients*. MSc Thesis.

Cengel, Y. A. & Cimbala, J. M. (2006). *Fluid mechanics: Fundamentals and application*. McGraw-Hill.

Mark Coombs & Spencer Drayton (2003). *Proton service and repair manual*. USA: Haynes Ptd. Ltd.

Wolf-Heinrich. (1998). *Aerodynamics of road vehicles*. (4th ed.). USA: SAE International.

Amir Shidique. (2007). *Simulation and analysis of hybrid electric vehicle (HEV) by addition of a front spoiler*. Thesis. Universiti Malaysia Pahang.

Kevin, P.G. & Geoffrey, M. L. G. (2005). *An investigation of the sensitivity of rear wing orientation for saloon race cars*. USA: SAE International.

Zhigang Yang & Bahram Khaligi (2005). *CFD simulation for flow over pickup trucks*. USA: SAE International.

ROSLI ABU BAKAR, DEVARAJAN RAMASAMY, FAZLI ISMAIL & AHMAD SYAHRIZAN SULAIMAN, Automotive Focus Group, University Malaysia Pahang, Locked Bag 12, 25000 Kuantan, Pahang, Malaysia. deva@ump.edu.my