VOL. 15, NO. 13, JULY 2020 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

ANALYSIS OF THE INFLUENCE OF THE REAR END SEDAN VEHICLE PROFILE ON THE AERODYNAMIC EFFICIENCY

V. Kausalyah[1], M. S. Shafiq[1] and S. Shasthri[2]
[1]Faculty of Mechanical Engineering, Universiti Teknologi MARA, Selangor, Malaysia
[2]School of Engineering and Physical Sciences, Heriott Watt University Malaysia, Putrajaya, Putrajaya, Malaysia
E-Mail: kausalyah@uitm.edu.my

ABSTRACT

When the fluid flows over the surface, the surface will resist its motion and this is called drag. Aerodynamic drag is the sum of pressure drag and viscous drag. This research will focus on how rear end parameters will affect the drag coefficient of the sedan car. Although drag coefficient will not give a big effect towards the rear end of the vehicle in comparison to the front end, this research focuses its initial stages in studying what is the effect of rear end drag if present .16 common sedan vehicle models from the different segments are measured at the rear end profile to determine the maximum and minimum range of the rear end parameters. Seven rear end design parameter will be used in the designing process while the front end data is taken from previous researches. Using the DoE Central Composite Design(CCD) sampling, 79 units of car models were designed using CATIA software. The ANSYS software will be used in this research to identify the rear end parameters that will give the lowest drag coefficient value. From the findings, the lowest Cd value was 0.22495.

Keywords: aerodynamic efficiency, drag coefficient, rear-end vehicle profile, sedan car.

INTRODUCTION

In recent years, every car company keeps trying to produce a vehicle with the most possible aerodynamic shape to reduce the drag coefficient. Car profile is one of the major factor that effects the aerodynamic efficiency. Moreover, vehicle profile optimization for low drag has become crucial part in designing a car because vehicle with an aerodynamic shape uses less fuel (Vinayagam *et al,* 2017). Besides, by reducing drag an aerodynamic profile can offer an inexpensive solution to improve fuel efficiency (Ghani, 2013). These research will focus on the influence of the rear end sedan vehicle profile on the aerodynamic efficiency. The frontal model of vehicles has been long studied and parametric development of a computational model was developed by Kausalyah *et al*, 2014b.

Every sedan car basically must contain three main parts where it is an engine part in the front, passenger compartment with 4 doors and luggage part or trunk part in the back.

The rear end area does not have a lot of different from the front area. The profile basically consists of wind shield angle, wind shield length, trunk angle, trunk length, bumper height, trunk height and height from ground to the bumper. Besides, during these research, parameter at the front and middle part will been keep constant. Even a little change in the design and shape can make a difference (Vinayagam *et al,* 2017).

Two basic aerodynamic forces are drag and lift. When a car moves forward, some air will pass through the surface body, the surface will resist its motion and automatically will produce drag and lift force (HEtawal *et al.*, 2014). Drag is the air force that exerts against a car as it moves while lift is the perpendicular force exerted by an air on the car.

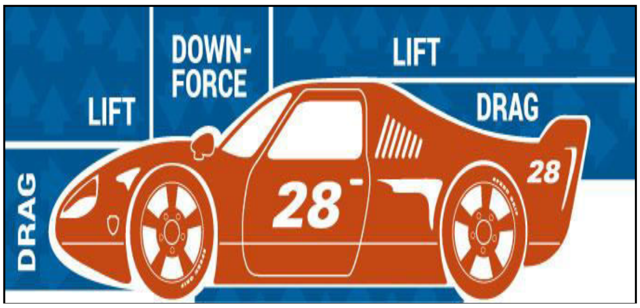


Figure-1. Aerodynamic forces on vehicle.

Air moves in a very similar way to liquid although air is not as dense as liquid, it still creates friction which brings about discomfort and trouble. The friction that exists when something moves through it and that's makes drag probably the most important aerodynamic factor that must be considered. Drag coefficient depends on a lot of factor such as object overall shape, speed and surface roughness. In this study, speed and surface roughness will become a constant variable.

METHODOLOGY

Data Collection

Various rear end shapes of sedan cars from the B to D segments were studied. In order to obtain the required parameter and dimension of the rear end of the sedan car, measuring tape and rope was used to measure the length of the rear end meanwhile the windshield and hood angle was obtained by using protector. There are seven required parameters that will be obtained from the rear end of the car. These parameters are wind shield length (WSL), trunk length (TL), bumper height (BH), trunk height (TH), height from ground to the bumper (BHG), wind shield angle (WSA) and trunk angle (TA).

Table-1. Vehicle Rear-end parameters.

x1	WSL	wind shield length
x2	TL	trunk length
x3	BH	bumper height
x4	TH	trunk height
x5	BHG	height from ground to the bumper
x6	WSA	wind shield angle
x7	TA	trunk angle

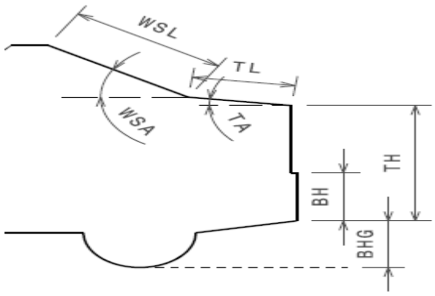


Figure-2. Vehicle Rear-end parameters labelled

The rear end shape of the sedan vehicle will be categorized into three groups which are B segment, C segment and D segment. Figure-2 below shows rear end shapes for the sedan vehicle from the B to D segments.

Figure-3. B to D segment cars.

Design of Experiments

By using the design of experiments method, a set of 100 situational experiments were generated through the MATLAB software using the Central Composite Design sampling technique. The range of parameters were coded into min, centre and max values. Central composite design (CCD) method was selected as its coding values are easily designed through software's.

The coded value of -1 is presented as the minimum range of the vehicle meanwhile coded value of 1 represent the maximum range for the sedan vehicle. Moreover, the 0 coded value represent the medium range for the vehicle. Based on the table 2.2.1 below, the x1 indicates to the wind shield length of the sedan car which show that the minimum value for the x1 is 520mm from 16 different types of cars meanwhile the maximum and median value for the x1 is 740mm and 630mm respectively.

Table-Fehler! Kein Text mit angegebener Formatvorlage im Dokument.**.** Vehicle Rear-end parametric ranges.

Parameter	Minimum (-1)	Median (0)	Maximum (1)
x1 (mm)	520	630	740
x2 (mm)	220	400	580
x3 (mm)	220	300	380
x4 (mm)	650	720	790
x5 (mm)	200	290	380
x6 (degree)	21°	31°	41°
x7(degree)	5°	75°	10°

Drawing and Simulation

Using the CATIA V5R20, 79 models for sedan vehicles with various ranges for rear-end profile was designed. The last 21 models are a repetitive experimental set of model 79 with median values thus the designs were not made. The CATIA models were built based on the 7 rear-end parameters based on CCD sampling, and the front profile was fixed at a constant value based of previous research. An optimum front end profile was identified from previous literature (Kausalyah *et al.,* 2014a). The models are converted into IGS file extension and imported into ANSYS CFD to run the CFD analyses.

The profile modelling process is simple and can be repeated in dimension and length. It can be a huge advantage to designing a lot of model. To ensure that the error in drag coefficient is very minimal, the vehicle design profile in the software must be as close as the actual size and dimension of sedan vehicle.

There are seven rear end parameters that will change respectively along with the design as the car model only consists of the exterior design of the car and the other part of the car is neglected since the objective is to see the air flow passing through the rear end of car and get the drag coefficient value. To choose the best design, the value of each coefficient of drag will be reviewed.

Figure-3 below shows the design for one of the models which has the lowest drag coefficient among the 79 model.

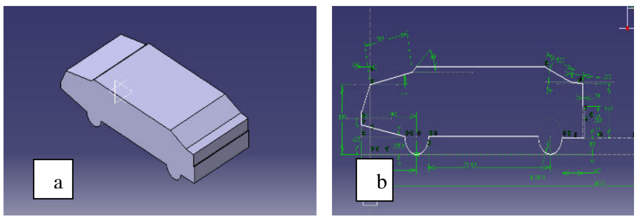


Figure-4. a) 3D CATIA model b) Full vehicle dimension.

Simulation and Analysis

The CATIA model of the sedan vehicle is imported to the ANSYS for simulation process. In order to

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

create the air flow around the vehicle, a fluid enclosure is created which is functioning as the fluid volume in order for it to act as the fluid domain for the simulation. The enclosure has a rectangular shape which represent a virtual wind tunnel for external aerodynamic. Figure-4 below displays the created enclosure in the ANSYS software.

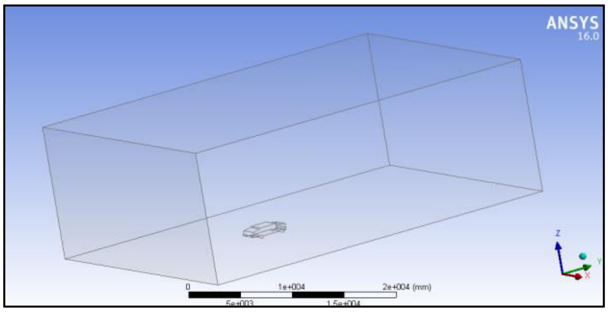


Figure-5. Enclosure dimension.

Based on the literature reading and previous research (Hetawal *et al*., 2014), the enclosure should be designed to follow the standard size. As the car is placed inside the enclosure, the distance of the car from the inlet velocity, outlet pressure and the wall is crucial. The distance of the car from inlet velocity is about three times the car's length meanwhile the outlet pressure has a range of five to ten times the length of a car from the outlet. Apart from that, the side and top wall is same as the length of the inlet which is three times the car's length and lastly for the bottom of the car, a tiny distance is required which is 100mm for the set up. Finally, the surface of the enclosure will be renamed and the inlet velocity, outlet pressure and the walls are designated as the boundary conditions. Figure5 displays the enclosure setup in ANSYS.

Sketching | Modeling

Details View

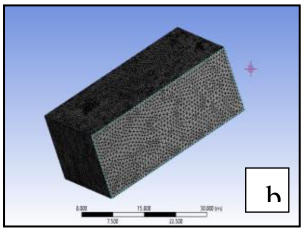
Details of Enclosure1	
Enclosure	Enclosure1
Shape	Box
Number of Planes	0
Cushion	Non-Uniform
FD1, Cushion +X value (>0)	12600 mm
FD2, Cushion +Y value (>0)	21600 mm
FD3, Cushion +Z value (>0)	12600 mm
FD4, Cushion -X value (>0)	12600 mm
FD5, Cushion -Y value (>0)	12600 mm
FD6, Cushion -Z value (>0)	100 mm
Target Bodies	All Bodies

Figure-6. Enclosure setup in ANSYS.

The mesh generation is performed next. Using the mesh generation, the element of the vehicle and the area inside the domain is generate. To obtain a more accurate value, the sizing function is very important. Curvature are used because the is curvature able to determine the edge and face sizes based on Curvature Normal Angle. A relevance centre is set to fine and high smoothing which will gave a very detail meshing. Figure-6 below shows the meshing set up and generated mesh of the model.

Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Slow
Span Angle Center	Fine

a



b

Figure-7. a) Sizing Parameter b) Mesh Generation.

The first step when starting the solver is to choose the Double Precision options. The analysis only run for viscous model. Pressure based steady state was used in the analysis. For Viscous model, the realizable k-epsilon with non-equilibrium wall function was selected which is the most common setup for turbulence model. The k refers to the kinetic energy transported and the epsilon is present the turbulent dissipative energy. It is a two equations model which means it includes two extra transport equation to represent the turbulent properties of the flow. This allows a two equations model to account for history effect like convection and diffusion of turbulent energy.

Next, at the boundary condition 13.89m/s was setup as an inlet velocity. The value was chosen because it was the safe speed based on the previous research (Kausalyah *et al*., 2014). The pressure outflow at the outlet area should be 0kpa. The reference values are used in the computation of derived physical quantities and non-dimensional coefficient like the Drag coefficient. The first parameter that is defined is the Frontal Area of each geometry which can be easily calculated using the Projected Surface Area tool. Hybrid initialization is setup for solution initialization. Hybrid initialization is a collection of recipes and boundary interpolation method. It solves Laplace's equation to determine the velocity and pressure field. Then, 300 number of iteration was setup. The calculation will stop if the solution converges.

RESULTS AND DISCUSSIONS

Data Collection

There are seven parameters that will change respectively along with each computational design. These parameters include the wind shield length(x1), trunk length(x2), bumper height(x3), trunk height(x4) height from ground to the bumper(x5), wind shield angle (x6) and trunk angle(x7). Besides, the parameters of the front car will be keep constant to ensure that the drag force will only be influence by the rear of the car. The table below shows the data of the 16 different type of sedan vehicle.

These data is then classified into its different segments and the maximum and minimum values are obtained to proceed with DoE.

Table-3. Data from different types of Sedan vehicles.

Car	X_1 (mm)	X_2 (mm)	X_3 (mm)	X_4 (mm)	X_5 (mm)	X_6 (°)	X_7 (°)
Proton Preve	703	348	333	774	362	30	8
Kia Sentra	640	492	380	790	240	40	10
Proton Saga Flx	590	380	380	720	330	39	8
Proton Waja	601	500	340	715	320	33	5
BMW 3 Series	692	530	322	760	260	30	8
Proton Gen 2	720	240	330	750	330	28	8
Toyota Vios	580	400	250	790	290	31	8
Proton Saga (old)	580	580	300	650	380	41	6
Audi A4	620	430	241	720	200	25	8
Honda Accord	680	350	360	740	202	29	8
BMW 6 Series	740	220	250	760	310	22	6
Merchendes E class	550	410	273	720	301	21	6
Merchendes C class	520	570	220	750	240	30	7
Volkswagen Passat	740	430	310	730	310	20	6
Mitsubishi Evo 9	530	510	320	700	280	36	5
Mitsubishi Lancer	690	410	400	770	200	34	6

CATIA Models and ANSYS

The design and data from the CATIA and ANSYS FLUENT analysis is presented in Table-4. From the 79 simulation runs, the best 10 rear end design are selected to be studied.

Table-4. The 10 best selected models.

DESIGN MATRIX																Drag Coefficient
CODED VALUES									ACTUAL VALUE							
Model no.	x1	x2	x3	x4	x5	x6	x7		x1	x2	x3	x4	x5	x6	x7	
9	-1	-1	1	-1	-1	-1	-1		520	220	380	650	200	21	5	0.22495
51	1	1	-1	-1	1	-1	-1		740	580	220	650	380	21	5	0.23048
17	-1	1	-1	-1	-1	-1	-1		520	580	220	650	200	21	5	0.23667
65	-1	0	0	0	0	0	0		520	400	300	720	290	31	7.5	0.23673
15	-1	-1	1	1	1	-1	-1		520	220	380	790	380	21	5	0.23792
23	-1	1	-1	1	1	-1	-1		520	580	220	790	380	21	5	0.23797
35	1	-1	-1	-1	1	-1	1		740	220	220	650	380	21	10	0.23843
13	-1	-1	1	1	-1	-1	1		520	220	380	790	200	21	10	0.23880
6	-1	-1	-1	1	-1	1	1		520	220	220	790	200	41	10	0.24044
75	0	0	0	0	0	-1	0		630	400	300	720	290	21	7.5	0.24006

Through design of experiment setup (DoE), these 10 model were chosen due to the minimum value of drag coefficient, Cd. The table above indicates the top 10 models with the lowest drag coefficient values placed in ascending order. The best selected model is model 9, followed by model 51 and 17. It was shown that model 9 give the lowest drag coefficient value which is 0.22495. This prove that the model 9 is the best model within the

ARPN Journal of Engineering and Applied Sciences

best 10 model. Next, the model 51 present 0.23048 drag coefficient value thus has placed it to be the second model. Apart from that, model 17 has been ranked at the third ranking as the drag coefficient is 0.23667.

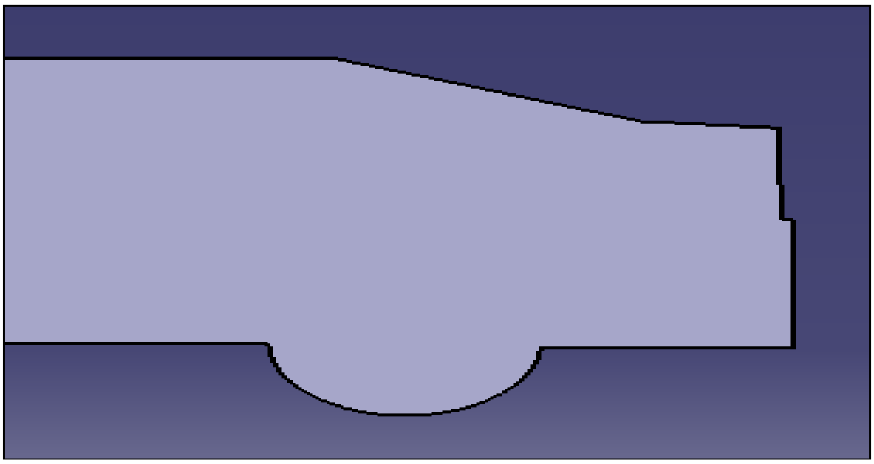


Figure-8. Model 9 rear-end profile.

It is apparent that parameters in the rear-end vehicles does affect the value of drag coefficient as seen in the table. Figure-7 presents model 9 which has the lowest drag co-efficiency. Figure-9 displays the rear-end profiles for the 10 best models selected. This study does not take into consideration any add on device on the rear end such as spoiler, wing and vertex generator to reduce coefficient of drag.

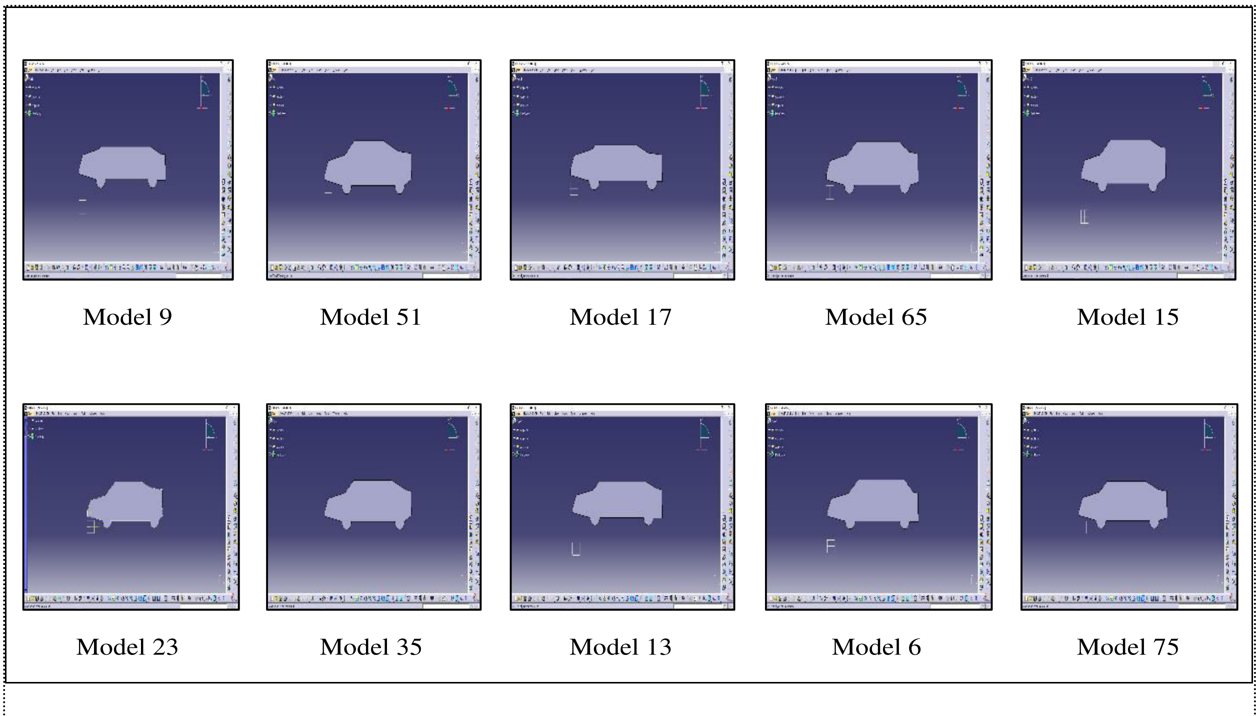


Figure-9. Vehicle profile for 10 best models.

Figure-10 illustrates the parametric effect of the rear-end profiles on the models. Chart which is the 3D line chart and also the histogram which indicates the 7 rear end parameter and 5 different models. Based on the graph, parameter x1, x6 and x7 have contributed to the highest influences of the model being repeatedly selected in 5 of the best models. At x1(wind shield length), the 520mm of length of the windshield has been used by 4 type of the model whereas only model 51 that used 740mm of windshield length. Meanwhile for parameter x6 (wind shield angle) and x7 (trunk angle) which is the wind shield and trunk angle also has the majority of the model used the same angle but the only model that used different angle is model 65. For x6, 21°has been chosen as the majority meanwhile for x7 the angle that been selected by the four model is 5°.

ARPN Journal of Engineering and Applied Sciences

Figure-11 displays the drag coefficient of 0.22495 for model 9 with the iteration value of 300 whereby convergence has taken place. At the early stage of the simulation, the drag coefficient value is shooting up and at some point it starts to decrease until it is converged and display a constant value.

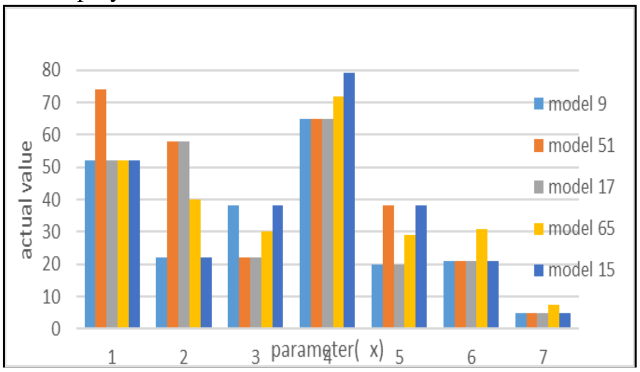


Figure -10. Bar chart for the best 5 models.

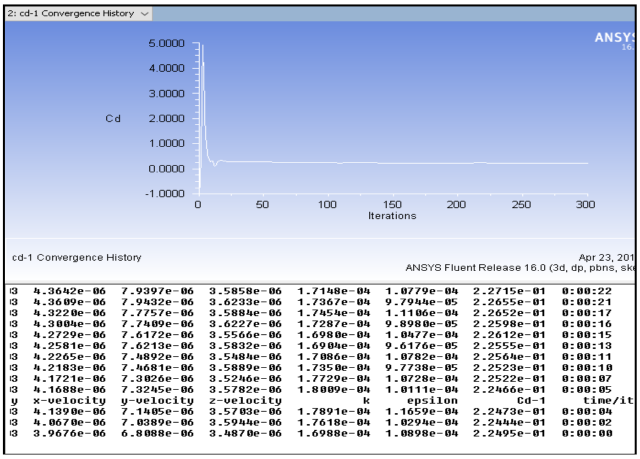


Figure -11. Drag coefficient for model 9.

The maximum Cd value obtained from the 79 simulation runs is 0.28899 of model 60. The minimum value is 0.22495 of model 9. The percentage difference between the max and min values of Cd is 22.16% which is a substantial amount to indicate that the vehicle rear-end profile does affect the drag coefficiency of a vehicle in motion. Figure 10 displays the model 60 with the highest Cd.

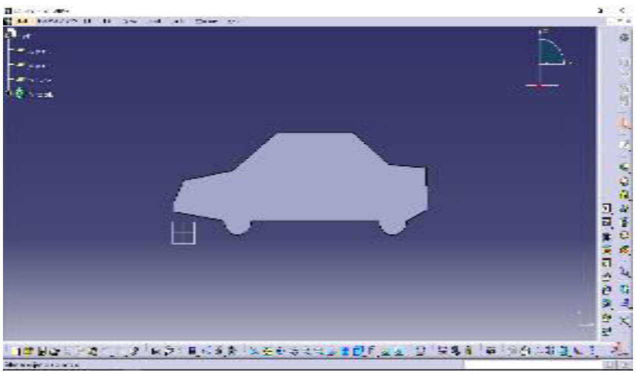


Figure-12. Model 60 with the highest Cd.

Pressure Contour

Figures 13, 14 and 15 shows the pressure contour which illustrate the position of the highest pressure occurred of model 9, model 51 and model 17. The maximum value of the pressure at model 9 is 105.024 Pa and the minimum pressure is -318.911 Pa. Next for model 51 where 102.459 Pa and -370.13 Pa is account for the maximum and minimum values of pressure. Lastly, for the third model, which is model 17, the maximum and minimum value of pressure is 102.084 Pa and -393.735Pa respectively. The pressure contour shows the highest impact region where friction occurs the highest. The higher pressure point for model 9 is 9.637e+001, follows by model 51 with 8.814e+001 and 8.706e+001 for the higher pressure point for model 17. For model 9, model 51 and model 17the higher pressure point appear on the stagnation point of the bumper on the front end.

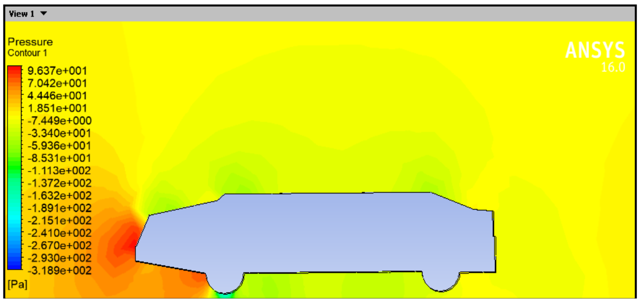


Figure-13. Pressure contour for model 9.

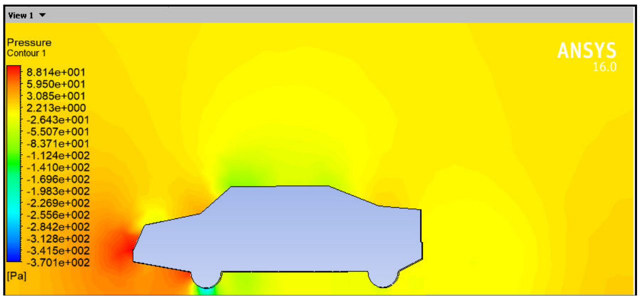


Figure-14. Pressure contour for model 51.

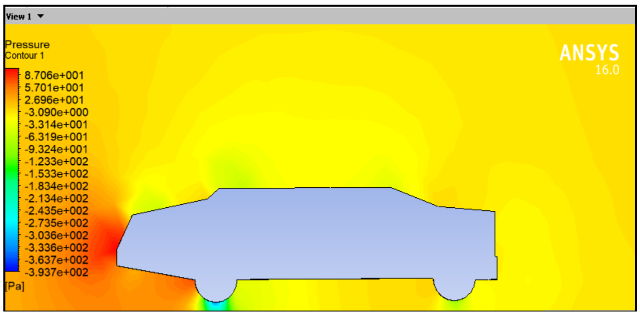


Figure-15. Pressure contour for model 17.

Velocity Contour

Figures 16, 17 and 18 shows the maximum and minimum velocity contour for model 9, model 51 and model 17. The velocity field around the car is projected and calculated on a path line along the car. Path line is used in order to visualize the airflow around the vehicle. The blue colour indicates the point where velocity is minimum and close to zero, meanwhile the turquoise and green vectors shows the region were the magnitude of the velocity is higher.

The highest region of velocity on the contour is defined as the contact region where air moves at highest speed. Maximum value of the velocity contour for model 9 is 24.3501 m/s. Next, from the contour it is shown that the maximum value of the velocity for model 51 is 25.4451 m/s follow by 25.4488 m/s for model 17.

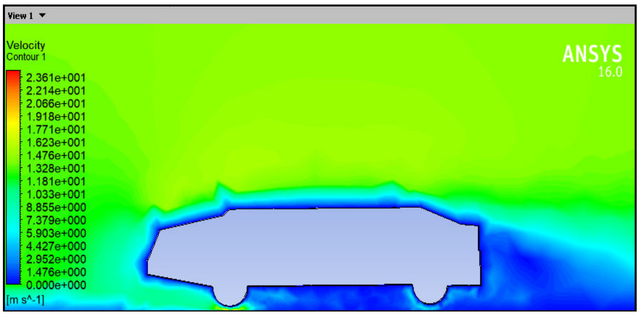


Figure-16. Velocity contour for model 9.

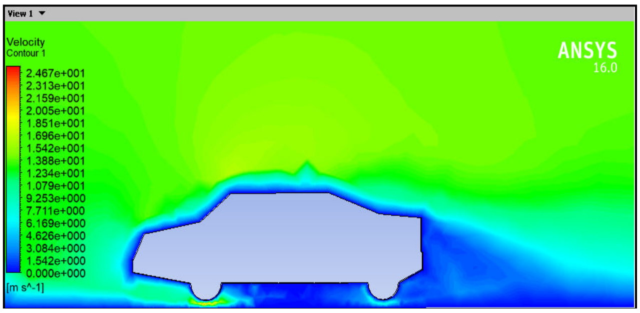


Figure-17. Velocity contour for model 51.

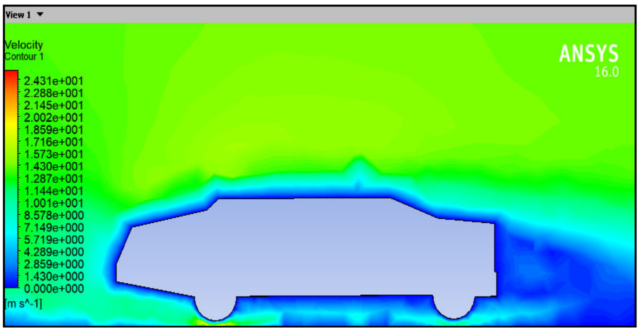


Figure-1. Velocity contour for model 17.

From the pressure and velocity contour for model 9, model 51 and model 17, a relationship between the pressure and velocity models can be seen. The velocity is inversely proportional to the pressure. The higher the velocity acting on the surface of the vehicle, the lower the pressure. The contour is important to show the most pressurized area and the region which has the highest velocity.

CONCLUSIONS

In conclusion, the objective of the study is achieved which is to study the effect of the drag coefficient on the rear end of the sedan vehicle profiles. It can be seen that; drag force does create an effect within at the back of the car although theoretically the front end that made contact with the air first. The study has been done by using the ANSYS Fluent software analysed the external geometry of the vehicle and the result of maximum and minimum value of coefficient of drag observed for 79 simulation runs are 0.29115 and 0.28899.

For future recommendation, adding the part such as the spoiler, wings and also the vortex generator can influence the effect towards the drag coefficient of the vehicle as basically these parts will help to reduce the drag coefficient as turbulence of air is created when it makes contact with the part that will decrease the drag.

ACKNOWLEDGEMENTS

This work was supported in part by the LESTARI Grant of Universiti Teknologi MARA with the Grant 0035-2016. The authors are most grateful to the university for its contribution.

REFERENCES

P. Vinayagam, M. Rajadurai, K. Balakrishnan and G. M. Priya. 2017. Design modification on Indian Road Vehicles to Reduce Aerodynamic Drag. (no. 8): 850-854.

O. A. Ghani. 2013. Design Optimization of Aerodynamic Drag At the Rear of Generic Passenger Cars Using Nurbs Representation. (no. May): 83.

V. Kausalyah, S. Shasthri, K. A. Abdullah, M. M. Idres, Q. H. Shah and S. V. Wong. 2014b. Development of Economical Vehicle Model For Pedestrian Friendly Front-end Profile Study. Int. J. Sinul Model. 13(4): 419-432.

S. M. Rakibul Hassan, T. Islam, M. Ali and M. Q. Islam. 2014. Numerical study on aerodynamic drag reduction of racing cars. Procedia Eng. 90: 308-313.

S. Hetawal, M. Gophane, B. K. Ajay, and Y. Mukkamala. 2014. Aerodynamic study of formula SAE car. Procedia Eng. 97: 1198-1207.

V. Kausalyah, S. Shasthri, K. A. Abdullah, M. M. Idres, Q. H. Shah and S. V. Wong. 2014a. Optimisation of vehicle front-end geometry for adult and pediatric pedestrian protection. Int. J. Crashworthiness. 19(2): 153-160.

C. Chomchai, S. Sonjaipanich, and S. Cheewaisrakul. 2013. Patient expectations for health supervision advice in continuity clinic: Experience from a teaching hospital in Thailand. J. Med. Assoc. Thail. 96(1): 26-32.

ARPN Journal of Engineering and Applied Sciences

X. Hu, R. Zhang, J. Ye, X. Yan and Z. Zhao. 2011. Influence of different diffuser angle on Sedan's aerodynamic characteristics. Phys. Procedia. 22: 239-245.

R. Baxter, N. Hastings, a. Law and E. J. . Glass. 2008. [No Title]. Anim. Genet. 39(5): 561-563.