Journal of Advanced Research in Applied Sciences and Engineering Technology

Journal homepage:
https://semarakilmu.com.my/journals/index.php/applied_sciences_eng_tech/index
ISSN: 2462-1943

An Analysis of Urban Vehicle Body Aerodynamics Using Computational Fluid Dynamics for the Shell Eco-Marathon Challenge

Harris Fadzillah Zainal Abidin[1], Md Tasyrif Abdul Rahman[1,*], Abdul Hamid Adom[2], Mohd Ridzuan Mohd Jamir[1], Sufi Suraya Halim[1], Mohd Al Hafiz Mohd Nawi[1]

[1] Faculty of Mechanical Engineering & Technology, Universiti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia
[2] Faculty of Electrical Engineering and Technology, Universiti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia

ARTICLE INFO

Article history:
Received 15 November 2022
Received in revised form 10 March 2023
Accepted 20 March 2023
Available online 6 April 2023

Keywords:
Aerodynamics; urban vehicle; shell eco marathon; bodywork; computational fluid dynamics; fuel efficiency

ABSTRACT

The Shell Eco-Marathon challenge is an annual competition held to challenge students in innovating the most fuel efficiency vehicle for either a prototype or an urban concept vehicle. An urban concept vehicle is designed for fuel efficiency using electricity as source of power. Apart from the use of electricity as an alternative to internal combustion engines, the design of the vehicle is also crucial for efficiency. The car bodywork design needs to be aerodynamically designed to minimise drag and subsequently use less energy to move. The design must also incorporate structural integrity to protect the driver as well as providing airflow for sufficient ventilation both inside the passenger and the engine compartment. Five models for the rear and front designs were produced using CATIA and analysed using Computational Fluid Dynamics in ANSYS Fluent. The models underwent a virtual wind tunnel on three different air velocity speeds, 20 km/h, 30 km/h and 40 km/h to generate a force report of drag force and coefficient on each design. The front design is chosen based on the lowest drag coefficient and force while the rear design is selected based on a balanced downforce while achieving the lowest practical drag force. The results demonstrated that the air resistance faced by a car was highly influenced by both front and rear design of the body.

1. Introduction

The Shell Eco-Marathon challenge is an annual competition held worldwide to high school and university students as a platform for them to innovate a way to improve vehicle fuel efficiency. The efficiency is measured by testing the designed vehicle farthest distance travelled using the least amount of fuel. Shell Eco-marathon Urban Concept challenge focuses on designing a maximum energy efficiency car with city driving elements such as a luggage compartment included. Urban vehicles are vastly used in densely populated city where the vehicles are required to maneuver amidst heavy traffic. The current focus for urban areas vehicles focuses on clean energy and efficiency. The Urban Concept Vehicle was created to design, test and build an energy-efficient

* *Corresponding author.*
E-mail address: tasyrif@unimap.edu.my

https://doi.org/10.37934/araset.30.2.7591

concept city car and is typically tested through five subgroups: chassis-suspension, drive train, electrical, safety-braking-steering, and fairing-ergonomics.

To improve engine optimization, modifications towards the engine operative inlet were done to create a leaner air-fuel mixture for better combustion and efficiency [1]. Remote diagnostics are also implemented by applying Internet of Things (IoT) for better analysis in an urban setting [2-8].

Fuel consumption reduction methods do not necessarily focus on engine modifications [9]. While a lightweight body will reduce the mass and consequently reduce the energy required to move the vehicle, the body shape itself also plays a role in reducing fuel consumption through overcoming forces. The forces that affect fuel consumption on bodyworks by acting opposed to the car includes drag force and downforce.

1.1 Drag Force

An object that moves in a fluid, whether liquid or gas, will experience force around it which can be classified in a certain angle [10-11]. One of the forces is drag force which is the force that acts oppositely to the object that resists the movement of the object [12], as shown in Figure 1. In a car body, the drag force will reduce its speed by increasing the load applied on the body [13].

Fig. 1. Drag force on a car [14]

A high drag force can cause a car to lose up to 50% of the energy consumption to overcome the load which requires 13% of the fuel energy [15-17]. Therefore, the drag force needs to be as low as practically possible to increase the energy efficiency of the car. Drag force can be determined from the equation below:

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

where
F_D = Drag Force (N)
ρ = Density of the fluid (air) (kg/m^3)
V = Velocity of the object (m/s)
C_D = Drag Coefficient
A = Cross-sectional Area (m^2)

The drag coefficients shown above are the way to measure the effectiveness of aerodynamics. The drag coefficient usually associates with the frontal area of the car where drag coefficient is multiplied by the cross-sectional area to acquire the drag force of a car [17]. Therefore, a smaller frontal area with a low drag coefficient can help reduce the drag force of a car [18].

1.2 Downforce

Lift is a force acting on an object that pushes the object upwards which is created by differences of air pressure between high pressure that acts on the front part and low pressure on the rear end of the object which results in generating lift [11], as shown in Figure 2. Lift and drag force will both affect an object to move in a particular direction depending on the flow [10].

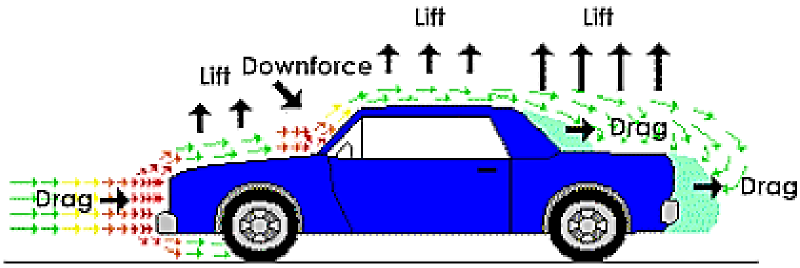


Fig. 2. Force acting on a car [19]

Lift force can be explained by the following equation

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

where
L_D = Lift Force (N)
ρ = Density of the fluid (air) (kg/m^3)
V = Velocity of the object (m/s)
C_L = Lift Coefficient
A = Cross-sectional area (m^2)

Lift is mostly applied in an aerial vehicle type during elevation. However, in automobiles, the lift is generated oppositely and is called downforce. Downforce is crucial for stability in a car when maneuvering on a road, especially during cornering [17].

Downforce will increase the car's weight if it is too high. An increase of downforce will lead to greater friction between the car wheel with the road which ultimately increases the drag force [20]. A higher drag force will lower the performance for energy efficiency. Therefore, for energy efficiency mileage purposes, the downforce should be low as to not generate too much load on the car but not too low until stability is compromised [21].

However, a very low downforce has an adverse effect on energy efficiency since it will reduce the grip between the car wheel and road [20,22]. A low grip requires more energy to move the car forward as it does not have an adequate amount of contact with the road [20]. Therefore, the downforce should not be too high nor too low to ensure the car stability and increase its energy efficiency [23-24].

1.3 Body Shape Concept

The body shape of a car has diversified a lot to due to factors such as aesthetics and mobility. The consideration for the body shape selection is based on vehicle's purpose.

1.3.1 Streamlined shape

A streamlined shape as shown in Figure 3 has a similar design to a teardrop shape by having a small round front with a long pointy rear end. It is a shape that presents very little resistance to airflow or water which increases the speed and mobility of an object.

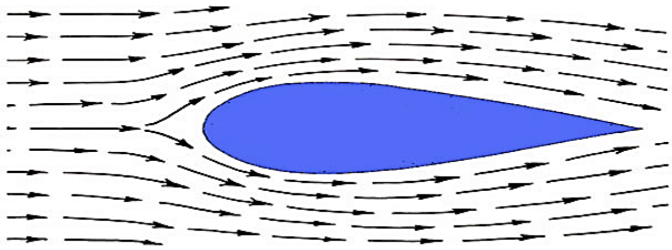


Fig. 3. Streamlined shaped [25]

The streamlined shape helps to reduce the drag force efficiently as it has a very low drag coefficient, C_D of 0.04 [19]. This is due to the streamline's practically small frontal area that results in a low drag force and coefficient [26]. Drag coefficients can be calculated using the following equation:

$$C_D = \frac{D}{\left(\rho A \frac{V^2}{2}\right)} \tag{3}$$

where
C_D = Drag Coefficient
D = Drag of the object (N)
ρ = Density of the fluid (air) (kg/m^3)
A = Reference area (m^2)
V = Velocity of the object (m/s)

Reference area, A is closely related to the drag coefficient, C_D where a low reference area can lower a drag coefficient of an object which explains the low drag force of a streamlined shape [26]. However, even though streamlined shape can help lower the drag force of a car, it is too lengthy. Regulations have stated that the vehicle should not exceed 3500 mm hence disqualifying of applicating this shape unless the length can be shortened [27-28].

1.3.2 Hatchback shape

A hatchback car contains a cargo area in the back that is practically smaller than a sedan car. It has a cargo door at the rear of the car which opens upwards. Hatchback cars are usually designed to be small and short in size as shown in Figure 4. The small size is possible due to the cargo area being reduced and usually shares the space with the passenger area.

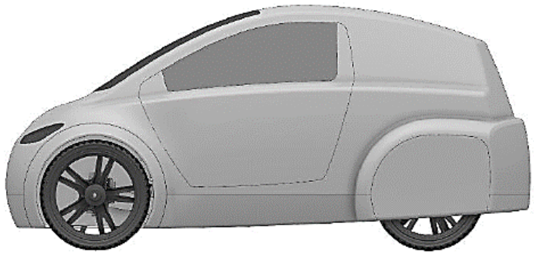
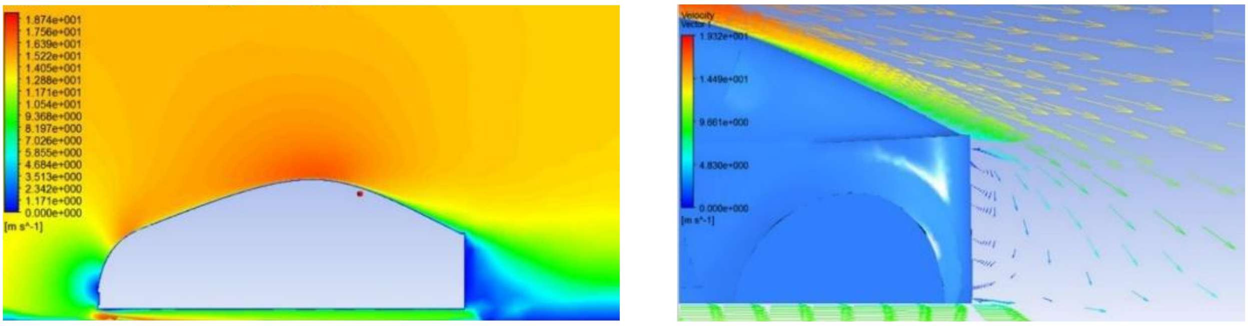


Fig. 4. Hatchback car concept [23]

Hatchback design is a viable choice since it has the advantages of the streamlined design while overcoming the length problem. A shape similar to a teardrop with a hatchback rear end design can reduce the drag force on a car [28]. This is due to the streamline long length inability to eliminate concentrated airflow that occurred along the body. When the rear end is cut off into a hatchback concept, it can stop the concentrated airflow along the body at the hatchback end as shown in Figure 5.



(a) (b)

Fig. 5. Simulation on hatchback car airflow (a) Body airflow (b) Rear end flow [16]

A wide range of studies, utilizing simulation, have been carried out to investigate the body shape effects on air resistance. As evidenced from the literature, a hatchback design is the most suited choice as the fundamental shape for an urban vehicle. Due to this reason, this paper focuses on the drag coefficient and downforce occurring on the simulated body.

2. Methodology

The methodology is separated into two sections which are i) Design and ii) Analysis. The design method is conducted using CATIA where five designs for each front and rear car end are modelled. The analysis procedure is conducted using ANSYS Fluent to measure the drag force, coefficient and lift force for each design. The design with the lowest drag force and balanced lift force will be chosen. The best design will be analyzed for its overall airflow applied around the car.

2.1 CATIA Modelling

The first step is to create a model of each design using CATIA software. The modelling is intended to better understand about the flow characteristics of the vehicle. The materials used for the model is carbon fiber. Wheels were not modelled in the scope of the bodywork since the main objective was to create the basic bodywork of the vehicle. A simplified model also increases computational efficiency for the simulation [29].

2.1.1 Front design

To achieve a front design with the lowest drag coefficient, all the designs will be developed with the same rear design. This is done to identify how each design influence the airflow when the rear design is fixed.

From Figure 6, a total of five frontal designs were generated from CATIA. Each design was made with alterations to their corner shapes, from curvy to sharp edges. The shapes that were considered include smooth streamlined shape, top half streamline shape with a sharp end, and eliminating the bottom part. The different designs were made to test for the best aerodynamical shape with low drag forces during high velocity.

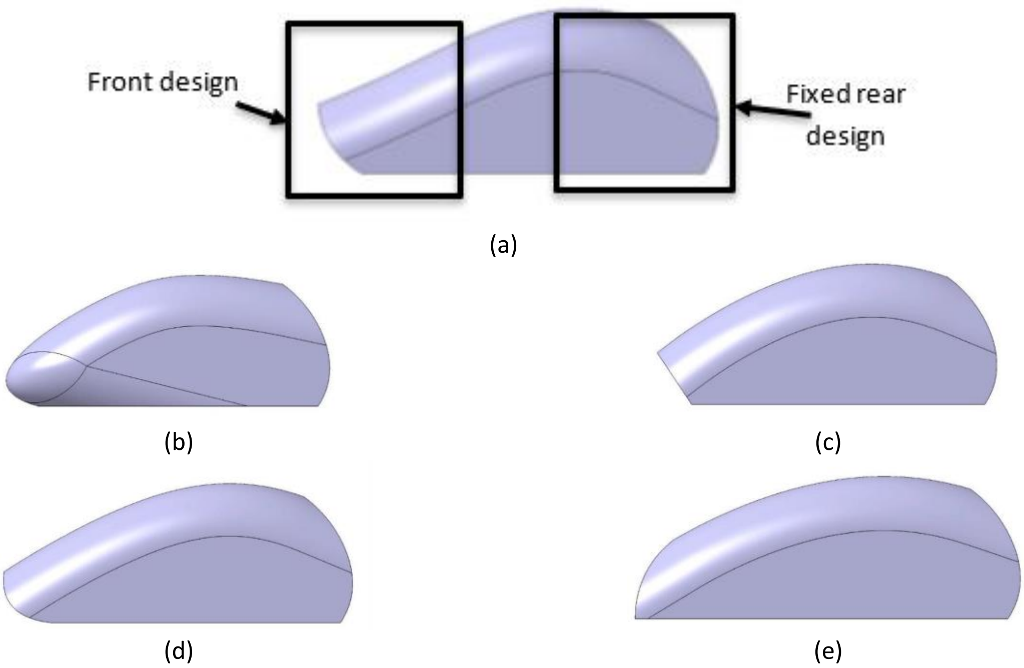


Fig. 6. Front designs (a) Design 1 (b) Design 2 (c) Design 3 (d) Design 4 (e) Design 5

2.1.2 Rear design

Figure 7 shows the rear designs modelled with a base front design, with the best frontal design selected to be the base. Each design was made with a different focus, from size reduction to airflow concentration shifting and elimination,

While the front design works to assist the flow above the vehicle and along the body, the flow cannot follow a sharp downward turn in the rear. Having a long pointy rear will put the airflow back together, reducing drag but since vehicles should be kept short, a shorter and rounder rear works for reducing drag.

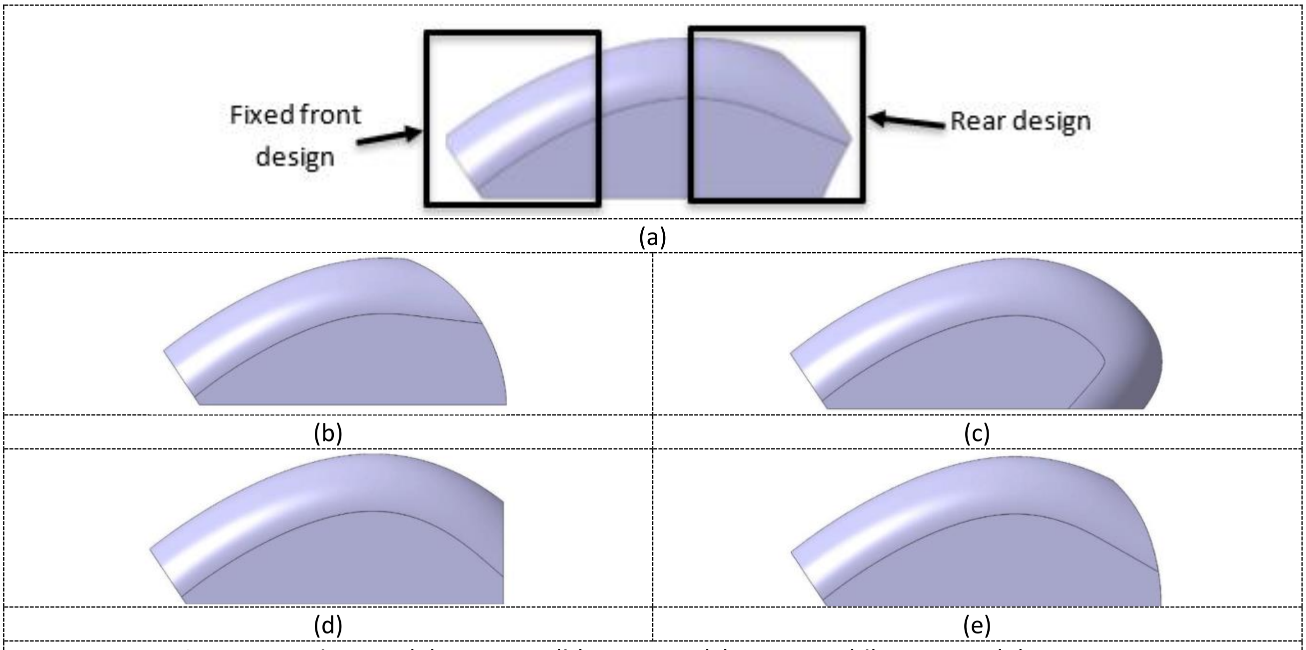


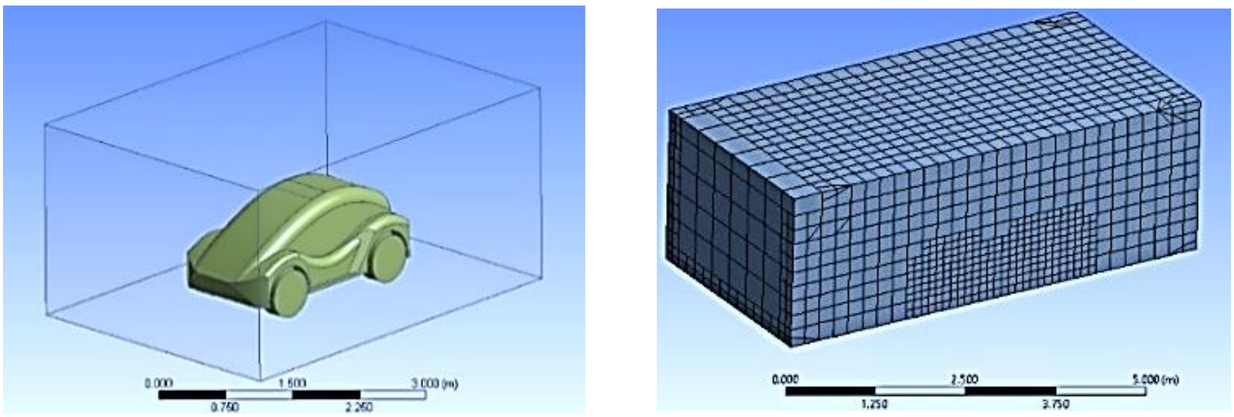
Fig. 7. Rear designs (a) Design 1 (b) Design 2 (c) Design 3 (d) Design 4 (e) Design 5

2.2 Computational Fluid Dynamics Simulation Analysis

The analysis was conducted using ANSYS Workbench and Fluent. This program is used to simulate the model in a virtual environment to provide reliable and efficient results more quickly and cost-effectively.

The model was imported in .STEP file type and generated into the workspace. An enclosure of 1 m space was created as in Figure 8(a) to represent a wind tunnel for conducting the analysis. The software would recognize the enclosure and apply the appropriate boundary conditions automatically. Extra spaces were provided inside the enclosure at the front and rear end of the vehicle to capture the airflow characteristics in greater detail.

Mesh was generated for the model to divide the whole component into a finite number of small elements. As shown in Figure 8(b), the meshing is done to the whole enclosure to limit the mesh sizing within that control volume so that the mesh quality can be changed only within the region where the high-resolution mesh is needed instead of having to fine mesh the entire domain; this reduces the time required to run the simulation [30].



(a) (b)

Fig. 8. Model analysis (a) Model enclosure (b) Generated mesh

For the boundary condition, the inlet velocity is set as 20 km/h (5.56 m/s), 30 km/h (8.333 m/s), 40 km/h (11.11 m/s). Each of the analysis will be done separately. Hence, each design will have three separate analyses for 3 different air velocity. While the inlet temperature of the air is set to 306.15 K (35 °C).

A force report is created by calculating drag, lift and downforce. Drag force report is created with the force vector acting oppositely to the model. Lift force has vectors acting towards the top of the model while downforce acts oppositely towards the bottom of the model instead. The force and coefficient values will then be generated. The whole activities are shown through the Figure 9 below.

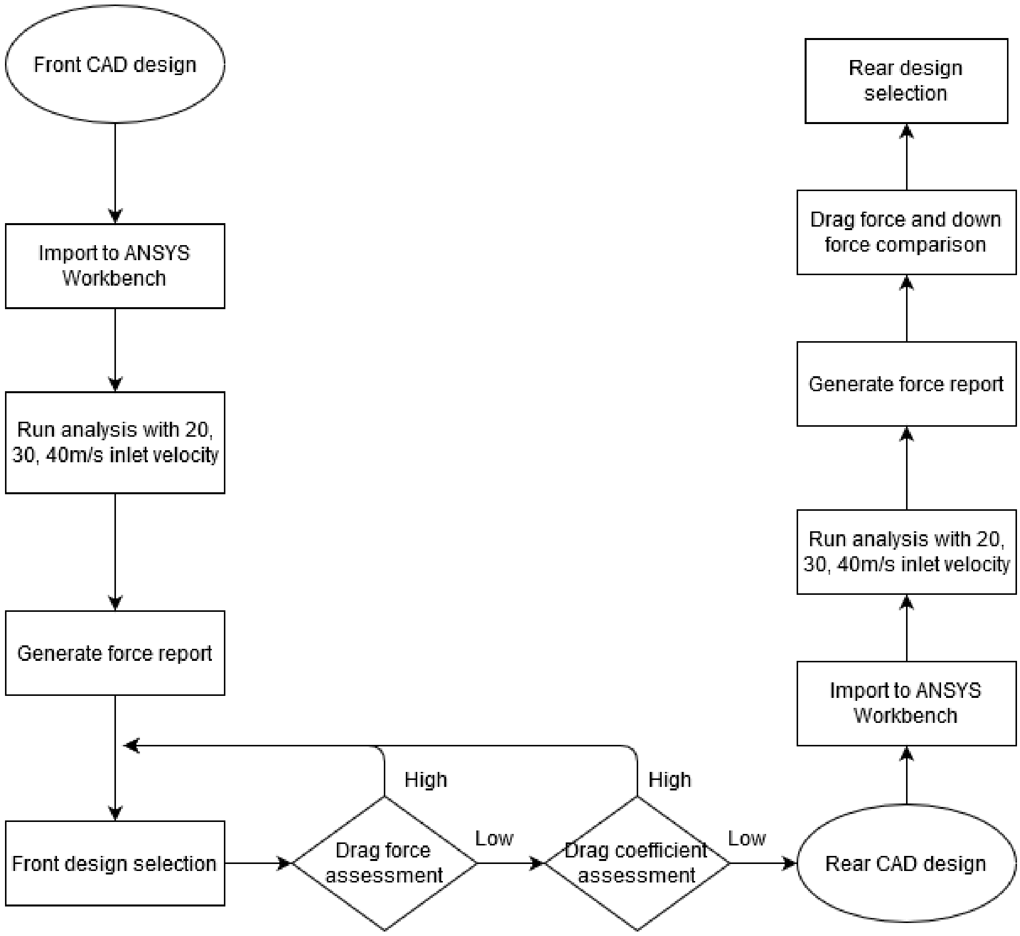


Fig. 9. Flowchart

3. Results and Discussions

The generated force report obtained for each design from the analysis procedure in methodology are compared and discussed. The results are based on the boundary condition that has been set while conducting the analysis. Comparison is done for each design based on the pressure generated, drag force and coefficient to validate the best design.

3.1 Frontal Design

The frontal designs that were made using CATIA and underwent ANSYS Fluent simulation analysis are compared and discussed. The component having the largest influence on the car design will be the frontal area as the front profile influences the rest on the design.

3.1.1 Velocity streamline

Figure 10 below shows all the designs have an air accumulation on top of the car resulting in a downforce that pushes the car downwards, increasing the load of the car. Design 1 from Figure 10(a) is the only design that has a windshield profile that is curved inwards, albeit slightly. Although a large concentrated accumulated air occurred, the velocity magnitude of the accumulated air is the lowest among all the designs. The high accumulation may still lead to high downforce pressure despite having a low velocity magnitude.

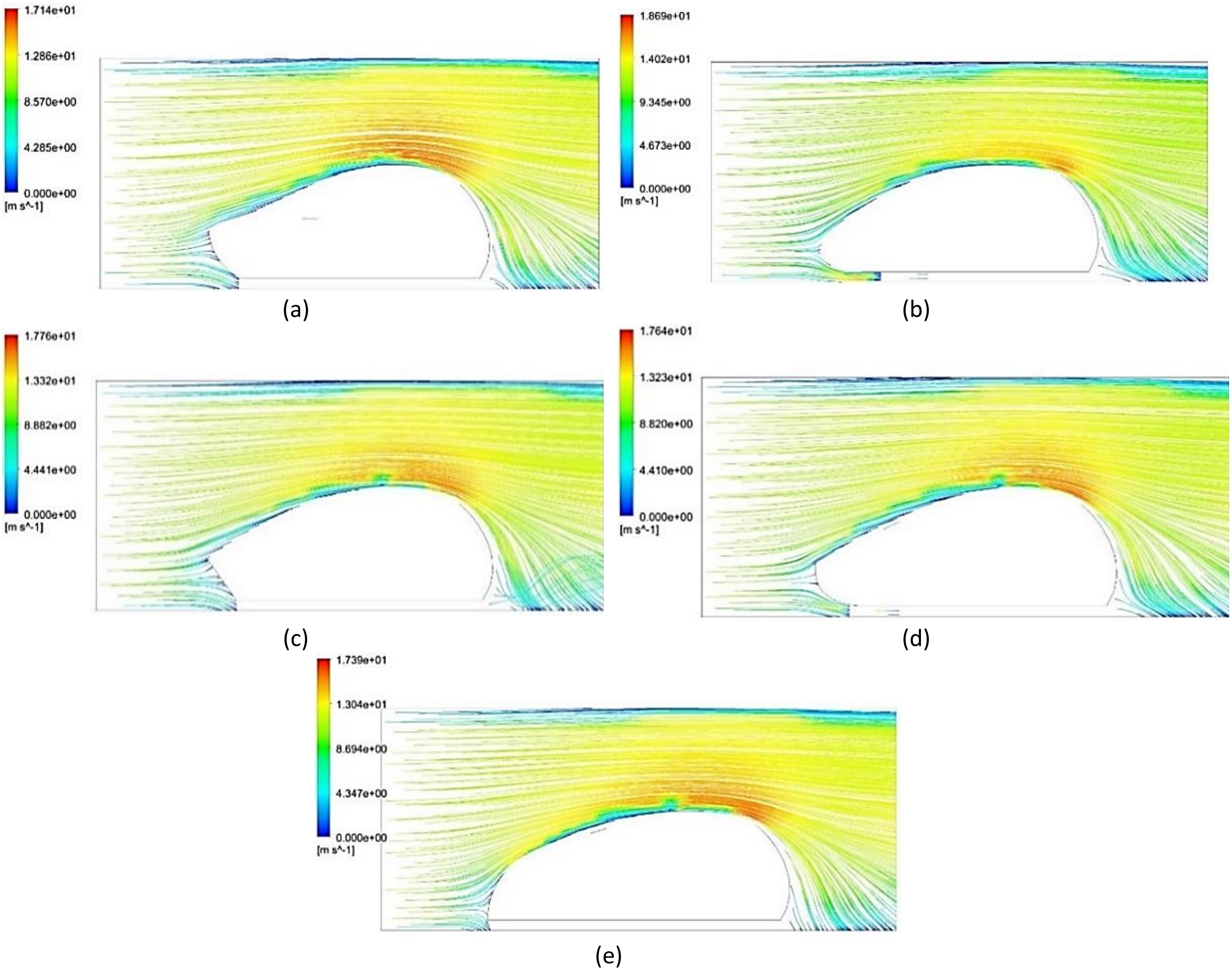


Fig. 10. Front designs velocity streamline (a) Design 1 (b) Design 2 (c) Design 3 (d) Design 4 (e) Design 5

A rounded leading shape like Design 2 in Figure 10(b) assists in preventing flow separation and turbulence due to the aerodynamic shape directing the oncoming wind to a better flow. The design resulted in less wind accumulation. Although the pressure of accumulated air is not concentrated in a small area, the magnitude velocity of said air is the highest among all designs.

Design 3 in Figure 10(c) has a similar design to Design 1 except that the curve joining to the bottom is replaced with a sharp angle instead. A sharp edge performs well when the force is coming from one direction, combining it with a smooth top for keeping the airflow laminar. It resulted in achieving the lowest velocity magnitude of accumulated air on top of the car along with Design 4. Design 4 has a similar result to Design 3 despite having a longer and rounded front.

Design 5 has a semi teardrop shape which should have a low drag force by supposedly minimizing the detachment of flow which leads to drag from turbulence. The airflow has become more condensed and less dispersed throughout the body. While the velocity of accumulated air is gradually built up, a high concentration occurs at where the rear end is supposed to begin.

3.1.2 Pressure contour

From Figure 11 shown below, all the designs experienced high frontal pressure. The frontal pressure is developed from the compression of wind going against the car during speeding. The wind air molecules moving near the vehicle body boundary tend to move slower due to the boundary effect and generate resistance against the vehicle.

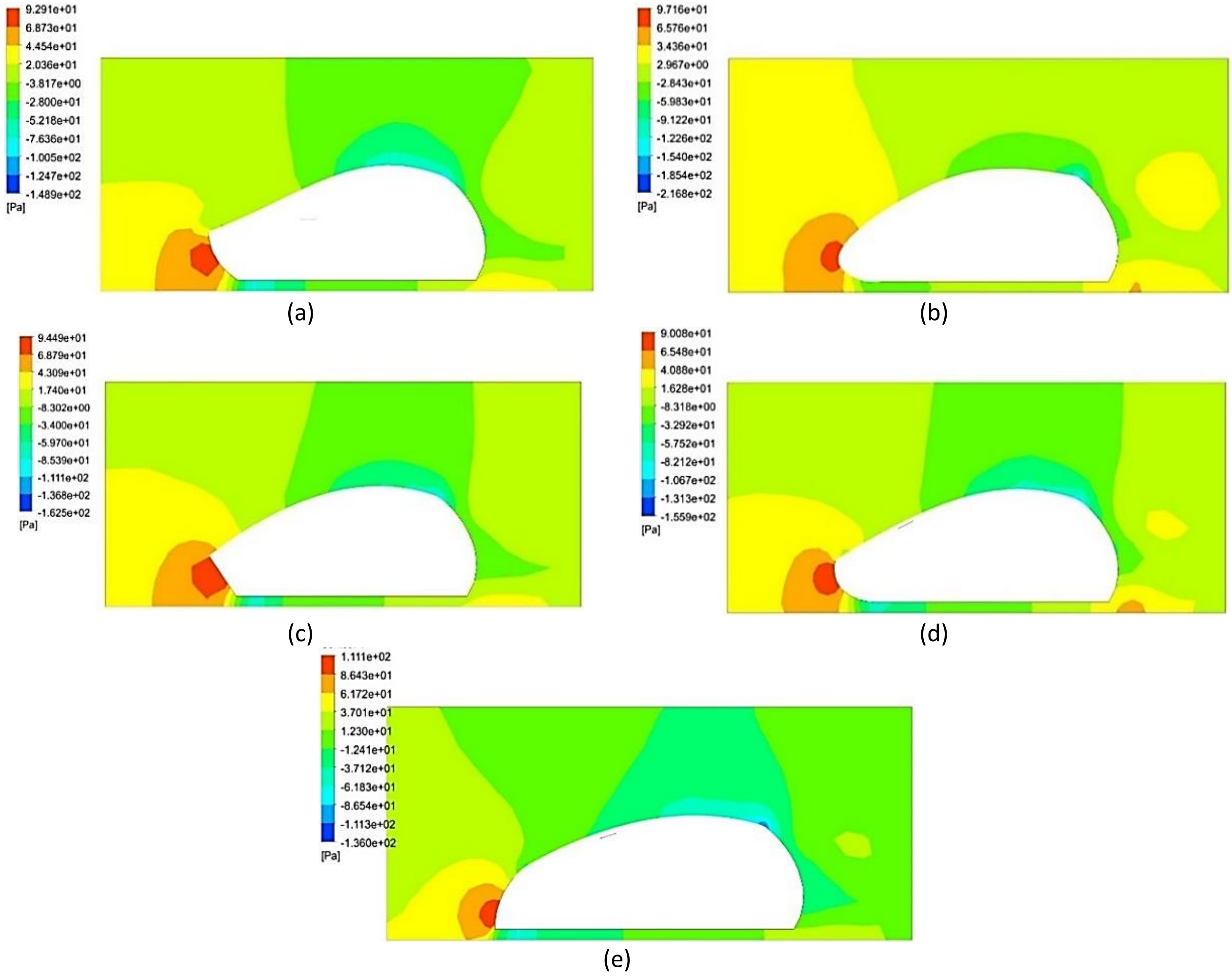


Fig. 11. Front designs pressure contour (a) Design 1 (b) Design 2 (c) Design 3 (d) Design 4 (e) Design 5

Design 1 in Figure 11(a) has pressure generated on the lower half of the body, facing upwards. While it may reduce the downforce load, a high lift force will cause instability during high speed. In a rounded design like Design 2 in Figure 11(b), the pressure envelops the curve with a concentration on the rounded edge. Since this design experienced the highest velocity magnitude, it experienced a high pressure of 85 Pa throughout the front nose.

Designs with sharp corners, specifically Design 3 from Figure 11(c), experienced a larger frontal pressure area due to the flat shape facing the oncoming air. Having a rounded corner design will smoothen the airflow which will reduce wind resistance towards the body to reduce airflow turbulence. However, Design 3 managed to achieve the second-lowest pressure force acting on the body with 56.1 Pa.

Figure 11(d) shows the Design 4 where the rounded design has managed to achieve the lowest pressure magnitude of 55.4 Pa. Since the rounded front is positioned higher compared to Design 2, this design has also experienced the pressure force from below, generating a lift force. Design 5 of Figure 11(e) has a small, concentrated pressure contour but has the highest magnitude of 88.1 Pa. This may lead to a failure or fracture in that body area in the long run.

3.1.3 Drag force and coefficient

From Figure 12, the drag force acting on a vehicle varies in a parabolic manner with velocity, directly and exponentially increasing with speed. Design 3 has the lowest drag force among all the designs analyzed by generating 3.73 N, 7.85 N and 13.33 N for each velocity, respectively.

The highest drag force was generated in Design 4 with 4 N, 8.44 N and 14.33 N. This proves that having a rounded front does not necessarily increase the aerodynamic efficiency of design despite having the lowest velocity and pressure magnitude.

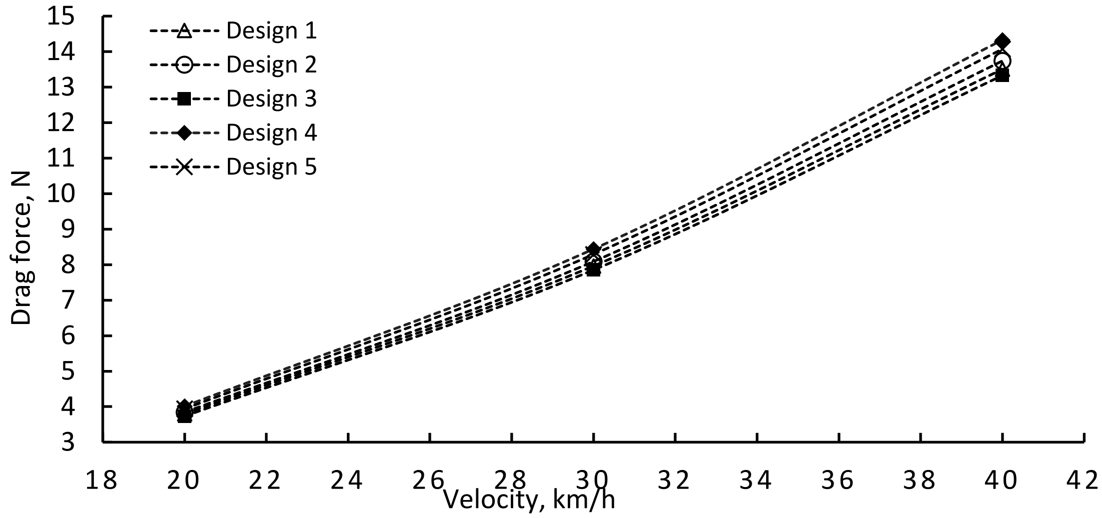


Fig. 12. Front designs drag force on different velocity

According to Eq. (2), drag coefficient is directly proportional to the drag force, so from Table 1, the lowest coefficient is Design 3 with 0.213. The coefficient has a considerable variation with the change in the body shape. While aerofoil shapes tend to have a rounded front to enhance the stall characteristics, if the main purpose is to reduce drag and not generate lift, a slender symmetrical shape may produce the least aerodynamic drag. Therefore, Design 3 is the best design where both drag coefficient and drag force are the lowest among all the designs.

Table 1
Front designs drag coefficient

Design	Drag Coefficient, C_D
1	0.221
2	0.251
3	0.213
4	0.272
5	0.235

3.2 Rear Design

The analysis was extended further to improve the aerodynamics on the effect of rear end on the vehicle.

3.2.1 Velocity streamline

While the front design works to assist the flow above the vehicle and along the body, the flow cannot follow a sharp downward turn in the rear. By having a long pointy rear will put the airflow back together, reducing drag by allowing the air molecules to converge back into the vacuum smoothly along the body. Since vehicles should be kept short, a shorter and rounder rear works for reducing drag.

When a pointy rear end has a sharp and sudden slant to converge to the bottom as in Design 1 from Figure 13(a), a vacuum is generated due to the hole developed in the air from the movement of the vehicle. This is because the air cannot occupy the space displaced by the vehicle at the same rate. At high speed, all these factors can affect the performance and acceleration of the vehicle.

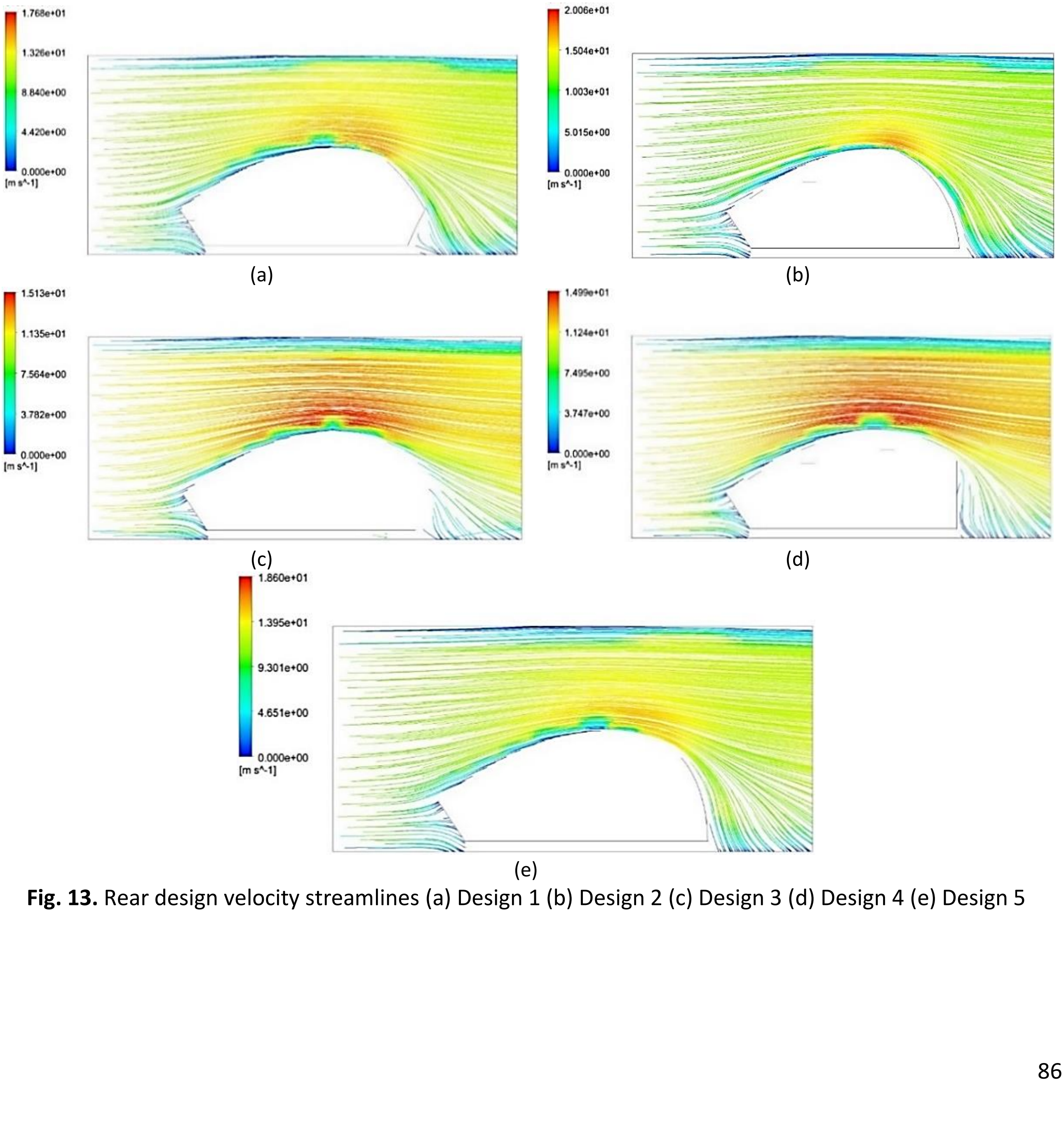


Fig. 13. Rear design velocity streamlines (a) Design 1 (b) Design 2 (c) Design 3 (d) Design 4 (e) Design 5

Design 2 and 5 (Figure 13(a) and (e)) shares a similar slanted curve profile with Design 2 has a higher inclination and larger radius. Design 2 developed a small, concentrated velocity magnitude area where the curve begins while Design 5 has a more dispersed magnitude due to a longer length leading to a smoother transition. However, Design 2 has developed the highest velocity of the accumulated air.

Rear design which quickly converges or cut off, such as Design 3 and 4 in Figure 13(c)(d), forces the airflow into turbulence and generates a great deal of drag. Similar to Design 1, a vacuum is generated from the since the airflow along the body is suddenly interrupted and does not have a proper flow to channel air into that space. A high velocity magnitude is developed onto the car roof slightly before where the rear end is about to converge.

3.2.2 Pressure contour

Despite having a fixed front design for this analysis, the rear designs still influenced the pressure developed in the frontal area, as shown in Figure 14. The pressure for Design 1, 2 and 5 had spread out and enveloped the front nose, this is due to the low air accumulation in these designs.

The high air accumulation on top of the car in Design 3 and 4 had resulted in a low pressure generated towards it and at the below of the car frontal area. The pressure concentrated at the frontal area has become focused on a smaller area. Despite not having the highest pressure magnitude, it is not practical to have pressure focused on a small area.

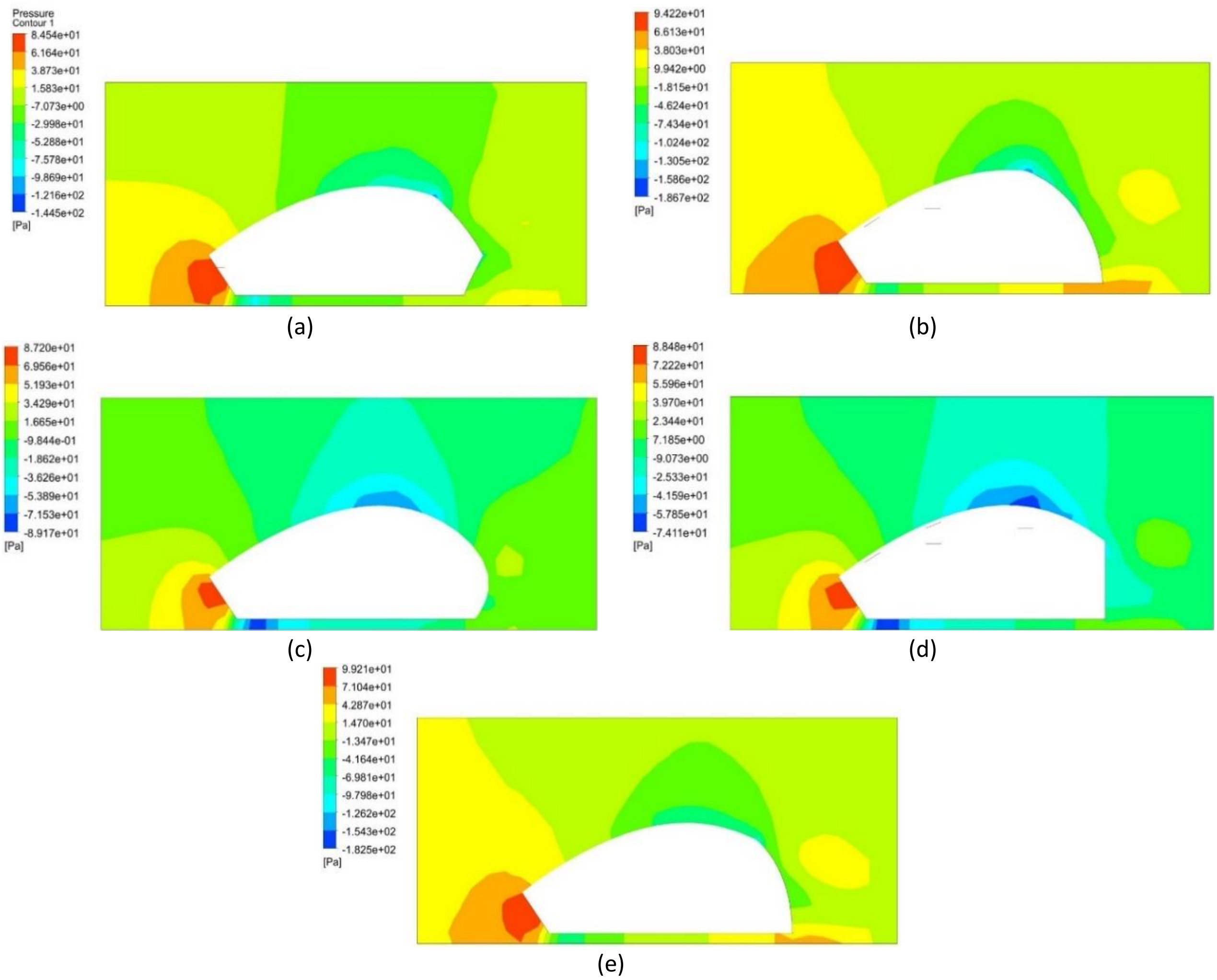
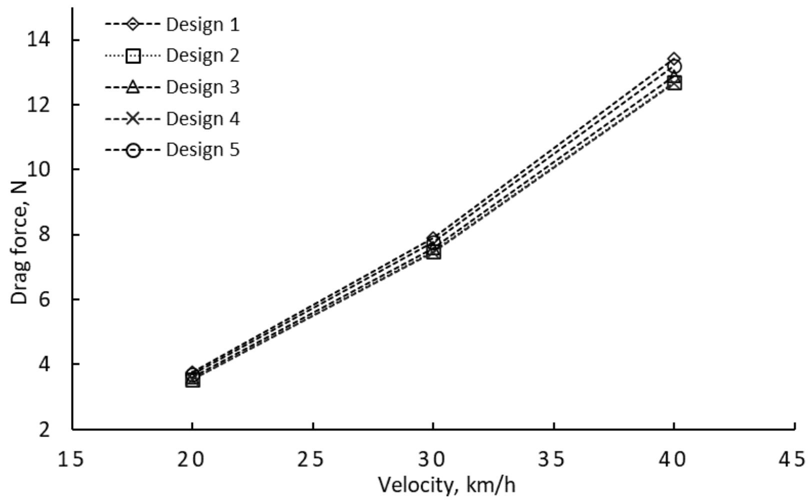


Fig. 14. Rear design pressure contour (a) Design 1 (b) Design 2 (c) Design 3 (d) Design 4 (e) Design 5

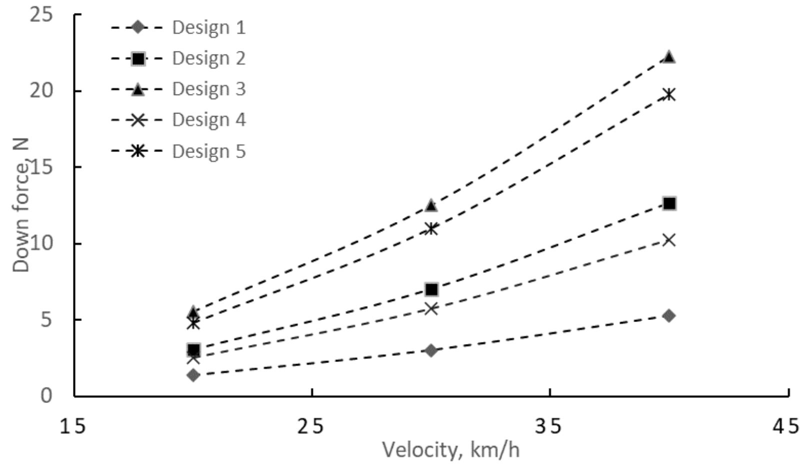
3.2.3 Drag and down force vs velocity

The drag force acting on a vehicle varies in a parabolic manner with velocity, directly and exponentially increasing with speed as can be observed from the Figure 15(a). Design 4 has the lowest drag force among the designs while Design 1 developed the highest. While a smoother design may have less drag, it does not necessarily mean it will be more aerodynamically efficient, depending on the purpose of the vehicle.

The downforce is directly proportional to the velocity as seen in Figure 15(b). The gap differences of downforce between the designs had grown exponentially as the velocity increases. Downforce is necessary in maintaining high speeds through the corners and forces the car to the track. It does not make the car have more mass to accelerate or manoeuvre. A speeding car can have an air build up underneath it, creating a lift that affects the traction. Downforce makes the car act heavier and with the adequate amount, can be used to counter the air build up.



(a)



(b)

Fig. 15. Forces on rear design for different velocities (a) Drag force (b) Down force

The best selection should be made for the optimum or balanced amount of downforce. While Design 4 has the lowest drag force, the downforce developed is too low and might not be the best choice to be implemented. The most balanced choice is Design 2 since it has the second lowest drag force while having an optimum amount of downforce. Hence, the selected rear design is Design 2.

Since the front profile influences the rest of the body, the front design selection criteria are the lowest drag force and coefficient. The rear design is selected based on balance and stability while achieving the lowest practical drag force.

4. Conclusions

A study to improve the aerodynamic bodywork of urban concept car has been presented choosing the lowest drag coefficient and downforce. The aerodynamic improvement is done as an alternate approach for fuel consumption reduction since drag contributes almost 50% of energy consumption to overcome the load which requires 13% of the fuel energy. Reducing the aerodynamic drag offers an inexpensive solution to improve fuel efficiency. The bodywork improvement is done by redesigning it using computer-aided drawings and analysed through simulations.

The bodyworks analysis is based on Computational Fluid Dynamics to simulate a car and analyse the aerodynamic properties by using a virtual wind tunnel. The aerodynamics design will be made according to the simulation result that is run in different velocities. The car aerodynamic properties such as drag force are to be calculated as low as possible according to the design provided to reduce load acting on the car which resulting in energy efficiency.

The selection criteria for front and rear profile differs since both components plays different roles in aerodynamic. The front profile is prominent since it will face the oncoming air resistance while also influencing the rest of the body, the selection will be made based on the lowest drag force and coefficient. The rear design, however, prioritizes on balance and utilize downforce in achieving it. Although a low drag force is important, traction and stability are essential for the rear design selection.

In conclusion, the study highlighted the CFD analysis in improving the aerodynamic properties of a car body. This work was carried out since fuel consumption efficiency needs to be increased and the drag developed on the body proves to have a large part on it. The body improvements are focused on the front and rear profile since it faces air resistance the most. Correspondingly, future work will involve implementing side profile improvements and assessing their effects for the body aerodynamics.

Acknowledgement
The authors acknowledge technical support from the Faculty of Mechanical Engineering Technology at Universiti Malaysia Perlis (UniMAP). The authors are also grateful for financial support provided by the UniMAP via grant RESMATE 9001-00622.

References

[1] Aziz, S., N. A. M. Amin, M. T. A. Rahman, A. Rahman, A. R. A. Syayuthi, M. S. A. Majid, and S. Suhaimi. "Design and Analysis of an Operative Inlet." In *IOP Conference Series: Materials Science and Engineering*, vol. 429, no. 1, p. 012075. IOP Publishing, 2018. https://doi.org/10.1088/1757-899X/429/1/012075

[2] Tan, Z. A., M. T. A. Rahman, A. Rahman, A. F. A. Hamid, N. A. M. Amin, H. A. Munir, and M. M. M. Zabidi. "Analysis on LoRa RSSI in Urban, Suburban, and Rural Area for Handover Signal Strength-Based Algorithm." In *IOP Conference series: materials science and engineering*, vol. 705, no. 1, p. 012012. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/705/1/012012

[3] Din, M. A. C., M. T. A. Rahman, H. A. Munir, A. Rahman, and A. F. A. Hamid. "Development of CAN Bus Converter for On Board Diagnostic (OBD-II) System." In *IOP Conference Series: Materials Science and Engineering*, vol. 705, no. 1, p. 012011. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/705/1/012011
[4] Ibrahim, Abu Bakar, Nurul Husna Abdul Kahar, Hafizul Fahri Hanafi, Ashardi Abas, and Shamsul Arrieya Ariffin. "The Development of Cascode Low Noise Amplifier with Double Feedback Technique Architecture for Wireless Communication." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 87, no. 2 (2021): 82-89. https://doi.org/10.37934/arfmts.87.2.8289
[5] Abd Halim, M. A., M. T. A. Rahman, N. Abdul Rahim, A. Rahman, A. F. A. Hamid, and N. A. M. Amin. "Analysis on current flow style for vehicle alternator fault prediction." In *IOP Conference Series: Materials Science and Engineering*, vol. 670, no. 1, p. 012042. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/670/1/012042
[6] Ab Hamid, Azarul Fahmin, Md Tasyrif Abdul Rahman, Anas Rahman, and Zairul Ameer Tan. "Handover Delay Analysis on Cellular Communication for Connected Car Application in Urban Area." In *IOP Conference Series: Materials Science and Engineering*, vol. 551, no. 1, p. 012022. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/551/1/012022
[7] Hamid, A. F. A., M. T. A. Rahman, A. Rahman, and M. M. M. Zabidi. "Path Loss with Doppler Shift Effect and Handover Delay Analysis on Cellular Communication for Connected Car Application in Urban Area." In *IOP Conference Series: Materials Science and Engineering*, vol. 557, no. 1, p. 012043. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/557/1/012043
[8] Hamid, Azarul Fahmin Ab, Md Tasyrif Abdul Rahman, Anas Rahman, and M. M. M. Zabidi. "Path loss analysis considering doppler shift effect on cellular communication for connected car application at rural area." In *IOP Conference Series: Materials Science and Engineering*, vol. 557, no. 1, p. 012042. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/551/1/012023
[9] National Research Council. *Assessment of fuel economy technologies for light-duty vehicles*. National Academies Press, 2011. https://doi.org/10.17226/12924
[10] Hucho, Wolf, and Gino Sovran. "Aerodynamics of road vehicles." *Annual review of fluid mechanics* 25, no. 1 (1993): 485-537. https://doi.org/10.1146/annurev.fl.25.010193.002413
[11] J. Moffat. "Aerodynamic Vehicle Design and Analysis," no. November, pp. 0–25, 2013. https://doi.org/10.13140/RG.2.2.31696.10244
[12] Murty, Repalli Satya Venkat Narayan, Pranav Pabsetti, Jai Bhoje, and Harish Rajan. "CFD Simulation on Aerodynamic Performance of Hyperloop Vehicle." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 102, no. 1 (2023): 126-139. https://doi.org/10.37934/arfmts.102.1.126139
[13] C. Jeppesen and D. F. Dosil, "Aerodynamic Analysis and Design of a Car for the Shell Eco Marathon," Master Thesis, Aalborg University, 2016.
[14] H. Bengtson, "Use of a Drag Coefficient to Calculate Drag Force due to Fluid Flow past an Immersed Solid," 2018.
[15] Cieśliński, Artur, Wojciech Prym, Mateusz Stajuda, and Dariusz Witkowski. "Investigation on Aerodynamics of Super–Effective Car for Drag Reduction." (2016).
[16] Ambarita, Himsar, and Munawir R. Siregar. "Numerical Study on Aerodynamic Drag Reduction of an Urban-Concept Car for Energy-Efficient Competition." In *MATEC Web of Conferences*, vol. 220, p. 02001. EDP Sciences, 2018. https://doi.org/10.1051/matecconf/201822002001
[17] Khairi, Danial Mohd, Mohd Azman Abas, Mohd Farid Muhamad Said, and Wan Saiful-Islam Wan Salim. "Fuel consumption mathematical models for road vehicle–A review." *Progress in Energy and Environment* 16 (2021): 59-71.
[18] M. S. Shahrir, "Design And Analysis Of Bodyworks For Unimap Automotive Racing Car," no. June, 2018.
[19] Hobeika, Teddy, and Johan Zaya. "Road Vehicle Aerodynamic Advanced." *Chalmers University* (2011)..
[20] Unlimited Product, "Aerodynamic tips," 2007.
[21] Dobrev, Ivan, Fawaz Massouh, Amélie Danlos, Michael Todorov, and Plamen Punov. "Experimental and numerical study of the flow field around a small car." In *MATEC Web of Conferences*, vol. 133, p. 02004. EDP Sciences, 2017. https://doi.org/10.1051/matecconf/201713302004
[22] Priyadarshini, J. Sita, A. V. S. Abhinav, B. Sharath Chandra, and R. S. Swathi. "Use of aerodynamic lift in increasing the fuel efficiency of heavy vehicles." *IOSR-JMCE* 12, no. 4 (2015): 39-42. https://doi.org/10.9790/1684-12413942
[23] Nguyen, Myselie. "Aerodynamic Development of DNV GL Fuel Fighter-Towards Shell Eco Marathon 2015." Master's thesis, NTNU, 2015.
[24] Cogan, Donavan. "The aerodynamic design and development of an urban concept vehicle through CFD analysis." PhD diss., Cape Peninsula University of Technology, 2016.
[25] Othman, Muhammad Firdaus. "Design and Analysis of Urban Car for Shell Eco Marathons Asia." PhD diss., UMP, 2012.
[26] Craig Vetter, "The truth and motorcycle design," 2008.

[27] Dobronsky, Sayan. *Aerodynamic Improvement of the BYU Supermileage Vehicle*. Brigham Young University, 2015.
[28] Shell, "Shell Eco-Marathon 2020 Official Rules," 2019.
[29] Bahambary, Khashayar Rahnamay, and Brian Fleck. "A study of inflow parameters on the performance of a wind turbine in an atmospheric boundary layer." *Journal of Advanced Research in Numerical Heat Transfer* 11, no. 1 (2022): 5-11.
[30] Gan, Edwin Chern Junn, Mikhail Fong, and Yee Luon Ng. "CFD Analysis of Slipstreaming and Side Drafting Techniques Concerning Aerodynamic Drag in NASCAR Racing." *CFD Letters* 12, no. 7 (2020): 1-16. https://doi.org/10.37934/cfdl.12.7.116