Mehran University Research Journal of Engineering and Technology

https://doi.org/10.22581/muet1982.2202.15

2022, 41(3) 149-160

Comparative study of hatchback vehicles through modelling and computational tools

Faraz Akbar

Department of Automotive and Marine Engineering, NED University of Engineering and Technology, Karachi Pakistan

Correspondence: Faraz Akbar, Email: faraz.akbar@neduet.edu.pk

Received: 12 December 2019, Accepted: 22 December 2020, Published: 01 July 2022

KEYWORDS

Computer-Aided Design (CAD)

Vehicle Body

Centre of Gravity (CoG)

Aerodynamics

Computational Fluid Dynamics (CFD)

Diffuser Angle

ABSTRACT

The purpose of this research is to explore the world of Computer-Aided Modelling and then apply its tools to benefit the automotive sector. Three commercially available economical vehicles in Pakistan, namely (i) Daihatsu Mira X, (ii) Honda N-One and (iii) Suzuki Wagon R, have been virtually modelled in Computer-Aided Design (CAD) software by using CATIA-V5R19. This research focuses on providing all the steps that are needed to transform an existing real vehicle body to a virtual vehicle body. Detailed comparisons of all the three hatchback vehicles have performed which determined the most crucial factors of the body that affect the performance of the vehicle. Computational fluid dynamics (CFD) analysis is also performed using SolidWorks flow simulation tool. The base drag and lift coefficients of selected vehicles are computed and compared. Furthermore, the influence of different diffuser angles on the aerodynamics characteristics of vehicle is also investigated. The drag and lift coefficients are computed at different diffuser angles to find out the optimum angle of diffuser. With the increase in diffuser angle, it is found that the total lift coefficient decreases but there is not much effect on drag coefficient for all hatchback vehicles. In addition, this study revealed that Daihatsu Mira X is the most stable and fuel economic vehicle among all three considered vehicles because of its lower centre of gravity co-ordinates and smaller coefficient of drag values at all diffuser angles.

1. Introduction

The development of geometrical model is an important task in the manufacturing of the vehicle. The Computer-aided engineering (CAE) has revolutionized the automotive sector, through the CAD models. Through these models the design parameters can be improved and modified before the manufacturing of the vehicle and the usage of CAD tools lead to rapid prototyping that will result in the growth of the sector. CAD enables designers to layout and develops work on screen, print it out and save it for future editing, saving time on their drawings. Modern CAD tools offer wide range of functions that are utilized by the automotive industry in the product development. CAD systems provide additional details to the product that may include the material selection and specification, relevant data for the processes, and the structure of the product.

In the virtual development of the product, there are primarily three stages the first stage is the three dimension (3-D) CAD design, second stage is of the digital mock-up of the product components, and finally the third stage which is called the virtual mock-up. For the designing of different models of vehicle bodies' style, CAD is used in this research work to convert the traditional methods into the digitalized form. Three commercially available hatchback vehicles namely (*i*) Daihatsu Mira X, (*ii*) Honda N-One 2013 and (*iii*) Suzuki Wagon R 2012 are selected in this research work because of their price effectiveness and large number innovative features available to the customers in terms of space, flexibility, visibility, driving experience and post-sale support. In addition, safety, performance, aesthetic appearance and value are positively influencing the consumer's overall satisfaction for hatchback vehicles. Table 1 presents the different parameters of the considered three hatchback vehicles in terms of their bodywork, engine, chassis and performance. Using the modelling tools along with the computational analysis can help the designers for further optimization in the design features of automobile.

Table 1

Detailed comparison of various specifications of Daihatsu Mira X, Honda N-One 2013, and Suzuki Wagon R vehicles [1, 2]

Specifications	Vehicles		
	Daihatsu Mira X	Honda N-One 2013	Suzuki Wagon R 2012
Body			
Body type	4/5 seater hatchback	4/5 seater hatchback	4 seater hatchback
Number of doors	5	5	5
Dimensions			
Wheelbase	2455 mm	2520 mm	2425 mm
Track/tread (front)	1300 mm	1305 mm	1295 mm
Track/tread (rear)	1295 mm	1305 mm	1290 mm
Length	3395 mm	3295 mm	3395 mm
Width	1475 mm	1475 mm	1475 mm
Height	1500 mm	1610 mm	1660 mm
Ground clearance	150 mm	150 mm	155 mm
Length:wheelbase ratio	1.38	1.35	1.4
Fuel tank capacity	30 L	35 L	27 L
Engine			
Engine type	naturally aspirated petrol	naturally aspirated petrol	naturally aspirated petrol
Engine manufacturer	Daihatsu	Honda	Suzuki
Engine code	KF	S07A	R06A
Cylinders	Straight 3	Straight 3	Straight 3
Engine volume	0.7 L	0.7 L	0.7 L
Bore x Stroke	63 x 70.4 mm	64 x 68.2 mm	64 x 68.2 mm
Bore/stroke ratio	0.89	0.94	0.94
Engine properties	*DOHC 4 valves per cylinder	DOHC 4 valves per cylinder	DOHC 4 valves per cylinder
Max engine power	38kW @ 6800 rpm	43kW @ 7300 rpm	38kW @ 6000 rpm
Specific output	77.5 bhp/L	86.6 bhp/L	77.5 bhp/L
Max torque	60 Nm at 5200 rpm	65 Nm at 3500 rpm	62 Nm at 4000 rpm
Specific torque	91.19 Nm/L	98.78 Nm/L	94.22 Nm/L
Sump	wet sumped	wet sumped	wet sumped
Compression ratio	11.3:1	11.2:1	11:1
Fuel system	EFI	EFI	EFI
**BMEP	1145.9 kPa	1241.4 kPa	1184.1 kPa
Engine coolant	Water	Water	Water
Unitary volume	219.33 cc	219.33 cc	219.33 cc
Aspiration	Normal	Normal	Normal
Intercooler	None	None	None
Catalytic converter	Y	Y	Y
Performance			
Power-to-weight ratio	52.1 kW/g	50.6 kW/g	47.54 kW/g
Weight-to-power ratio	19.2 kg/kW	19.76 kg/kW	21.04 kg/kW
Chassis			
Engine position	Front	Front	Front
Engine layout	Transverse	Transverse	Transverse

*DOHC (double overhead camshaft)

**BMEP (brake mean effective pressure)

Computational fluid dynamics (CFD) is certainly very predominant in many fields of engineering research and application. CFD has come a long way in influencing the design of automotive components due to continuing advances in computer hardware and software as well as advances in the numerical techniques to solve the equations of fluid flow. The automotive industry's interest in CFD applications stems from its ability to improve automotive design and to reduce product cost and cycle time. CFD can be utilized more and more in day-to-day automotive design.

One of the major application of CFD in automotive is the aerodynamics study of a vehicle that is based on its designing. Proper use of CFD early, helps to significantly reduce prototyping needs and consequently, reduce cost and cycle time [3, 4]. For this, SolidWorks has been considered the optimal design following up on the diverse auto models. SolidWorks component strategies are generally considered. The accompanying cross sections will be naturally be made by the SolidWorks-CFD programming [5].

Starting around the 1970s, the research was moved from 2-D to 3-D through the research work of Versprille [6] who gave the basics of 3-D curve and surface modelling. In the early 1980s the emergence of commercial CAD systems such as CATIA and others began their contribution in the aerospace and automobile industry and CAD has proved to be highly effective in evaluative and analytical design development, and in manufacturing [7]. By the 1990s, computer was capable of the computational requirements for the 3-D CAD. CAD is a tool that helps to improve the productivity of the design, and the quality of the design as well as saves time and resources that are necessary for the initial testing of the product. CAD has become an especially important technology within the extent of/the range of computer-helped technologies, with benefits such as lower product development costs and a greatly reduced design cycle. Current computer-aided design software packages range from 2-D vector-based drafting systems to 3-D solid and surface modellers [8]. Modern CAD packages can also often allow rotations in three dimensions, allowing viewing of a designed object from any desired angle, even from the inside looking out. Some CAD software are capable of performing the mathematical modelling [9-11].

CAD has a great influence in automotive sector, a big reason of this is that every global automotive industry is trying to reduce time and cost for the development of new vehicle and CAD has made this target possible to the industries quite efficiently, by replacing the prototype formation and experiments by CAD modelling [12]. Towards the CAD applications in automotive industry, two different methods of computer-based vehicle styling which are texture, mapping and direct computer modelling are examined in a study and a comparison and evaluation of the methods within a typical framework for automobile concept design is represented [13]. The challenges of conceptual full-vehicle model development are discussed and the application of a centralized product master model by an exemplarily illustration of a 3-D CAD approach is done [14]. Vehicle architects, design engineers, project managers, administrators for CAD, stylists as well as styling engineers have contributed to a survey at automotive manufacturer and supplier, and the results are found to be quite supportive and concerned towards the CAD in contrast to the conventional methods [15].

Road vehicle aerodynamics has been treated by Barnard [16], where he gave a very readable account, and then further explain by Hucho [17], who demonstrated it with graphical representations. The main developments with vehicle aerodynamics probably occurred during the early 198Os, and the use of low-drag vehicles has now become common. The development of low-drag vehicle shapes is now more rapid because of greater past experience and better computational techniques [18-21]. CFD simulations make cells which is the division of fluid volume in a limited number of squares. To isolate the fluid into number of squares or to make the limited number of cells a limited volume around the chose to demonstrate is made which is known as the computational area. For the exactness everything relies upon the structure and the span of the cells [22, 23]. Fig. 1 is demonstrating the computational area [24].

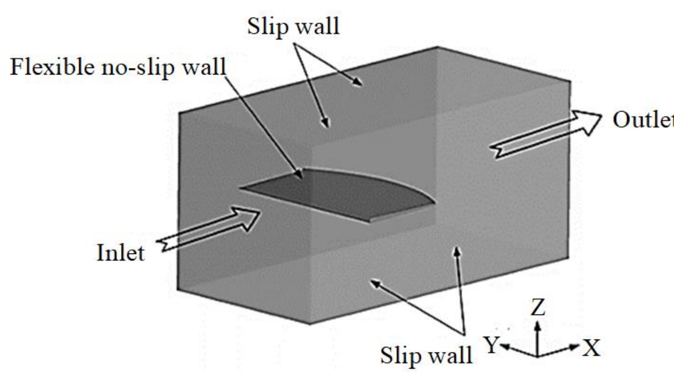


Fig. 1. Computational domain in SolidWorks [24]

The drag and lift coefficients are the aerodynamic properties which greatly affect the performance of a vehicle. These coefficients are calculated by using Eqs. 1 and 2.

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

Where C_D is the coefficient of drag, ρ is the density of the fluid, A is the frontal projected area, V is the velocity of the vehicle taken as 30 m/s, and F_D is the drag force acting on the vehicle. Coefficient of lift can be obtained as follows.

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

Where C_L is the coefficient of lift, V is the velocity of the vehicle taken as 30 m/s and F_L is the lift force acting on the vehicle.

The automotive industry has been taking the initiative in introducing CAE at various stages of manufacturing and it can be seen that the state of the art engineering i.e., CFD by reviewing its applications in automotive engineering [25]. CFD played its vital role in determining the vehicle body shaping to understand and for the clear explanation of the theory and the subject area of the topic. A moving object which is exposed to air always experience three main forces and moments presented well by Genta [26]. CFD applications are examined by Benjamin and Hickman [27] within the automotive industry and in particular, demonstrate its implementation which is explained in detail where and how CFD can feature within the design/development process. By using 3-D CFD simulations, a study is made by Ahmed Al-Saadi [28] and investigated ways to reduce the aerodynamic drag coefficient and to increase the stability of full-size road vehicles. Firdaus Azmi [29] analysed the aerodynamics characteristics of a multi-purpose vehicle (MPV) generic design using ANSYS Workbench. Coefficient of drag and lift recorded for the car modelled is found to be 0.28 and 0.05 respectively [29]. In another research, Thabet [30] concluded that CFD simulation can be carried out to investigate the flow characteristics over a model car and calculated the aerodynamic coefficients from the CFD simulation and compared them with the available experimental data [30]. A review attempted by Shashi Kant [31] and compared the effect of drag forces on sedan car on applying different types of spoilers, vortex generators and compare the sedan's car drag force with that of hatchback type car.

It can be established from the literature review that a more profound investigation on the most crucial factors of the vehicle body that affect the performance of the vehicle is demanded to produce a more convincing CAD and computational models of vehicles' aerodynamics. Therefore, this paper presents firstly the motion study, estimation of Centre of Gravity (CoG), weight estimation and aerodynamics of three considered vehicle bodies through CAD tool CATIA-V5R19. Secondly, this research performed a detailed comparative analysis of the aerodynamic properties of the three hatchback vehicles, namely (i) Daihatsu Mira X, (ii) Honda N-One 2013 and (iii) Suzuki Wagon R 2012, through commercially available computational software SolidWorks flow simulation tool. Finally, the aerodynamic properties such as the drag and lift coefficients are obtained at different diffuser angles to study their behaviour at a fixed velocity.

2. Research Methodology

In this research work CAD models are developed for three different hatchback vehicles and then performed aerodynamic analysis of these vehicles through CFD tool. The complete research methodology is presented in Fig. 2. Firstly, a detailed literature review is carried out in this regard then the models of the vehicles are made by using different Computer-Aided Designing tools which involves the formation of blueprints and the surface modelling. In the next step, CFD analysis of three considered hatchback vehicles is carried-out which involves the construction of computational domain, meshing and the application of boundary conditions. Various simulation results have been achieved in this research such as study of moving parts of a vehicle, coordinates of the centre of gravity, weight estimation of the vehicle body by implementing steel material and the comparison of drag and lift coefficients under different diffuser angles. The set of modelling and simulations were performed for three selected hatchback vehicle models inspired by their designs, with proper care taken for their aesthetic appearance, sizing with respect to new regulations and driver's space requirements. In addition, all the sleeted three car models have close vehicle dimensions, styling and engine specifications. Furthermore, Daihatsu Mira X, Honda N-One 2013 and Suzuki Wagon R 2012 hatchback vehicles brands are considered to be affordable in comparison to other expensive brands like Ford, Fiat and Chevrolet in hatchback segment. All the selected vehicles are essentially basic levels cars and have the ability to carry maximum 4 to 5 passengers.

3. Computer-Aided Modelling and Aerodynamics of Vehicles

Modelling on CAD software has become a need for every industry. However, different software works a little differently but the basic logic and concepts are same. To make the CAD of a vehicle body it is required that the blueprints of that car are available. These blueprints are then used for the Surface Modelling of the vehicle body. CAD model of vehicle bodies (a) Daihatsu

Mira X, (b) Honda N-One 2013 and (c) Suzuki Wagon R 2012 are developed as shown in the Fig. 3.

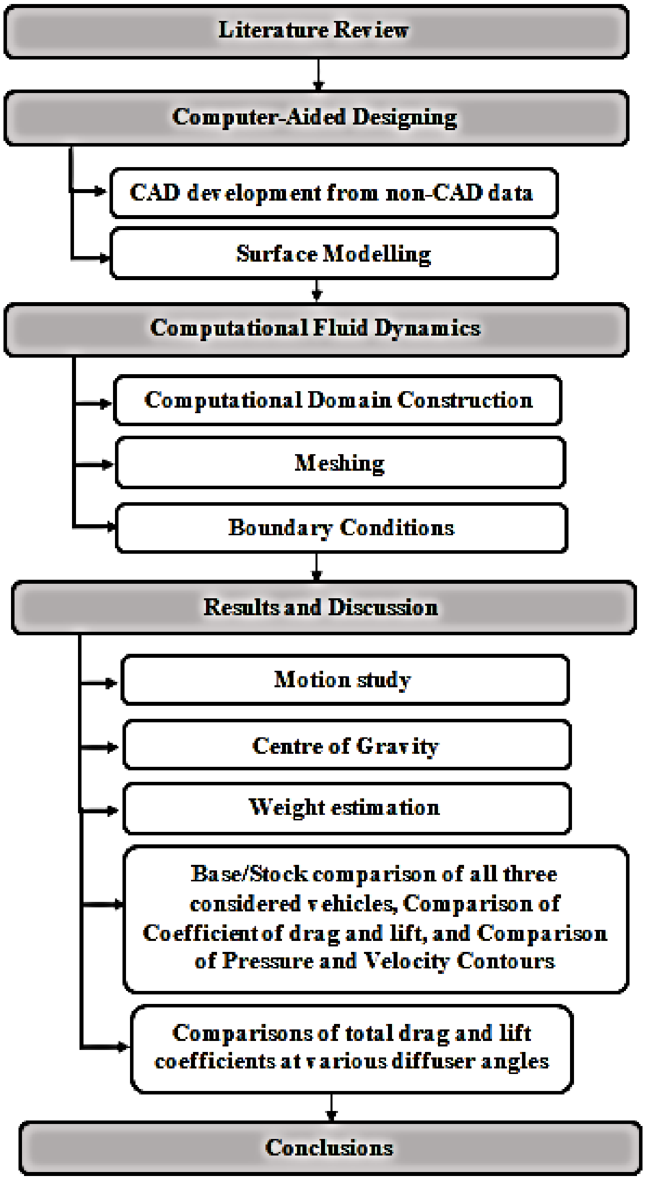


Fig. 2. Research methodology

These CAD models of the vehicle body can be used to extract individual parts and further analysis on each part can be done by exporting it to the computational software. In this manner individual parts are optimized if needed which results in less computational power required because of less nodes and elements being involved in the computational method.

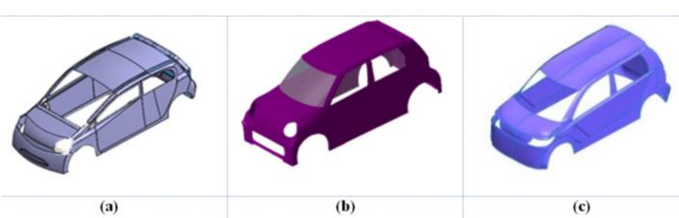


Fig. 3. CAD model of vehicle bodies (a) Daihatsu Mira X, (b) Honda N-One 2013 (c) Suzuki Wagon R 2012

3.1 CAD Development from non-CAD Data

By using modern 3-D CAD software packages 2-D pictures, studies, drawings or sketches in 3-D models can be transformed easily. This technique relates the engineering based construction and digital mock-up development to the styling procedure. CATIA-V5 and SolidWorks come with an additional feature to generate 3-D surfaces from 2-D sketches, through which a direct implementation of studies can be done for automotive 3-D CAD model. If the 3-D hardware is provided, clay models style studies or scaled detail models, surface generation process can directly be started by importing the scan or measurement data into the CAD software. This feature is also called as concept design. This feature is usually used for surface modelling of mechanical products and has a very high rendering quality. The process begins with a rough sketch of the mechanical part, colour and texture is then applied to it, the 3-D model from the 2-D sketch is created and then advanced photo-realistic rendering and animation is used to further evaluate and present. The method that is used here is the conversion of 2-D sketch into the 3-D surfaces using workbench of free style in CATIA-V5R19. Each step is discussed below in detail.

3.1.1 Generation of blueprints

Vehicles considered for this research work are Daihatsu Mira, Honda N-One and Suzuki Wagon R. These three vehicles are very common in Pakistan and all are hatchback with engines ranging between 660–700 cc. The blueprints of these cars are not available on the internet hence they were generated by using the following method. First the pictures of all the views are taken from 180º at a fixed distance as shown in Fig. 4. These views were then edited and a sketch was obtained with the proper dimensions and curves. These blueprints are then used for the surface modelling of the vehicle body as shown in the Figs. 5, 6 and 7.

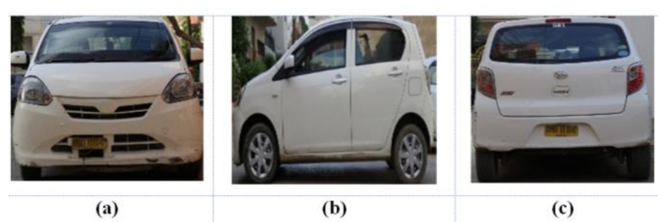


Fig. 4. Photographic technique for Daihatsu Mira X (a) Front view (b) Side view (c) Back view

3.1.2 Development of wireframe

The surface modelling is a tool that is used in developing complex surfaces by using 3-D curves. In CATIA-V5 the shape module and then sketch tracer is used in which the views are imported and then the views are sketched by using 3-D curves. These curves are modified and fitted to all dimensions. The wireframe of the vehicle is then obtained.

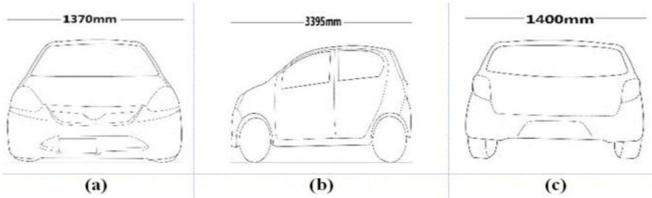


Fig. 5. Blueprints of Daihatsu Mira X (a) Front view (b) Side view (c) Back view

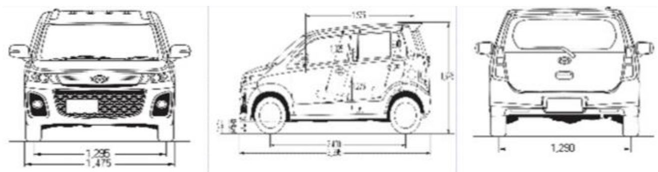


Fig. 6. Blueprints of Suzuki Wagon R 2012 (a) Front view (b) Side view (c) Back view

Fig. 7. Blueprints of Honda N-One 2013 (a) Front view (b) Side view (c) Back view

3.1.3 Filling surfaces

After the wireframe is ready it is checked for the errors and then the surfaces are added. Free style fill command is used after which a dialogue box appears. There are three types of filling options in CATIA-V5 that includes Power, Auto and Analytical. Each element is selected one by one and then it is filled using any of the three options. The difference between power, auto and analytical is that in power only one filling surface is created, in auto the best order and patches are chosen and in analytical one or more surfaces are formed depending on the number of elements. After filling the wireframe, a surface is generated, which is ready to be rendered. Without filling the curves and generating the surfaces, the model is not completely ready, and it cannot be used for any further computational study.

3.1.4 Rendering in CATIA-V5

Rendering is the tool used in CATIA-V5 to apply pre-defined materials to the selected parts. This is also used to edit the density of the material. It defines the material specification that would be used in the entire product development process. It gives a realistic image to the product. The materials are provided in a material library and it consists of a wide variety of materials. The material specification plays an important role in defining the physical properties, 3-D presentation which may include the geometry and pattern of the product and in 2-D representation of patterns in drafting.

3.2. Computational Fluid Dynamics Analysis

There are certain steps involved in the process to conduct CFD analysis for the vehicle. The computational fluid domain was constructed around the model in the SolidWorks and the inlet velocity of 30 m/s was specified. The pressure outlet was set at ambient pressure. All of the walls of the domain were considered as slip wall while the vehicle geometry is modelled as solid wall. The governing equations of fluid flow are the equation of mass conservation, momentum conservation and energy conservation [32].

3.2.1 Computational domain construction

In order to restrict the analysis to a certain region of interest, for this purpose an enclosure is created around the 3-D body of the vehicle. This will define the boundary under which the flow simulation of the vehicle will take place. This is called the computational domain, as illustrated in Fig. 8. Table 2 shows the dimensions of the computational domain.

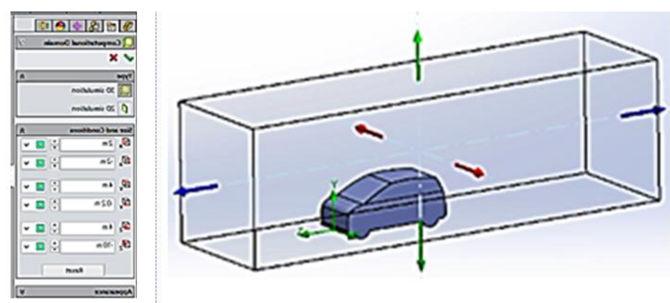


Fig. 8. Fluid domain for 3-D model of Daihatsu Mira X

Table 2

Dimensions of computational domain

+X axis	2 m	-X axis	-2 m
+Y axis	4 m	-Y axis	-0.2 m
+Z axis	4 m	-Z axis	-10 m

3.2.2 Meshing

Meshing is a crucial step in design analysis and the process starts with the creation of a geometric model. Then, the program subdivides the model into small pieces of simple shapes called elements connected at common points called nodes. The process of subdividing the model into small pieces is called meshing. As shown in Fig. 9 meshing is done by default in the SolidWorks flow simulation. The models are meshed to the finest quality i.e., on a scale of 1-8, level 8 was selected.

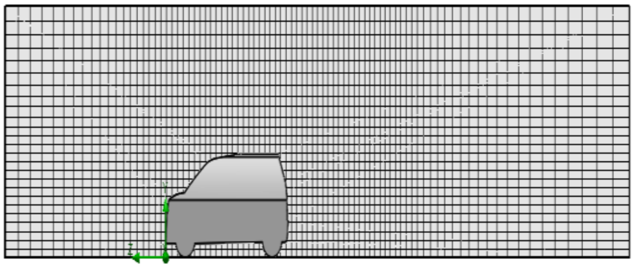


Fig. 9. Mesh generated at fine level in SolidWorks

3.2.3 Boundary Conditions

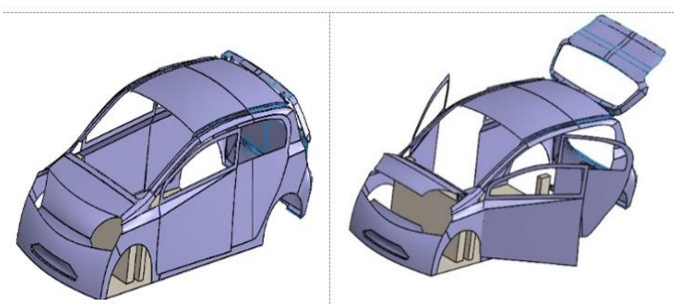
After the construction of the fluid flow domain of the vehicle model the essential boundary conditions were applied to the model. The inlet air velocity was given as 30 m/s and the reference axis to the model was Z-axis. The software will use these boundary conditions to give approximate value of the results while satisfying exactly and approximately the essential and natural boundary conditions respectively. After the application of boundary conditions, goals were setup. These include the values that are desired as output from the CFD analysis.

4. Results and Discussion

4.1 Motion Study

It is concerned with the moveable parts of the body of vehicle. Figs. 10 (a) and (b) presented below the CAD model with their parts that attach to the frame or body by hinges and are rotate about it.

In order to make supports of the parts to attach latches on which they can rotate about, a simple floor was created for vehicle on which certain pillars were created acting as chassis pillars. The hinges were created on these pillars to which the body parts (doors, hood, etc.) were attached as shown in Fig. 11.



(a)	(b)

Fig. 10. Motion Study of Daihatsu Mira X (a) Vehicle body with hinges (b) Rotation of individual parts about the hinges

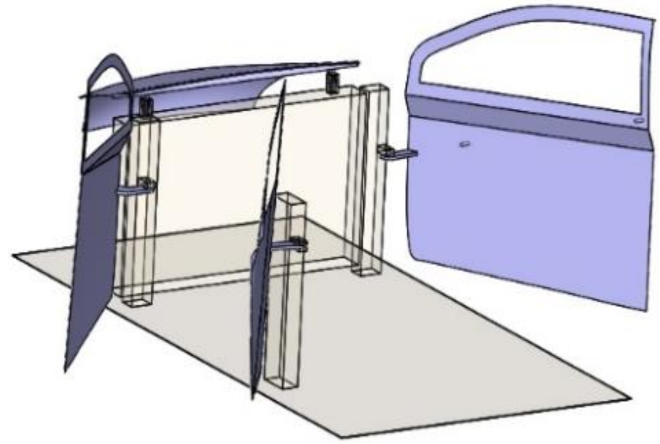


Fig. 11. Show the attachment of hinges with the pillars

By accessing the move component option, access on number of different useful commands can be made. Here it can also have observed that a certain constraint can be applied to the body to restrict movement in parts after certain distance. This is used for appropriate door angles as shown in the Fig. 12. In the same manner the angles of all individual parts in motion can be studied taking in account the clearance so that the parts do not interfere with one another. Further optimization can be done using this method which provides us with the appropriate results for each part.

4.2 Centre of Gravity

By using the command of measure inertia in CATIA, it will find the coordinates of centre of gravity. Feature is shown in the Fig. 13. It is provided by the software in form of x, y, z co-ordinates. By selecting any element its centre of gravity can be found. In the similar way centre of gravity of each model was found. The centre of gravity of the three modelled vehicles is found to be as presented in the Table 3.

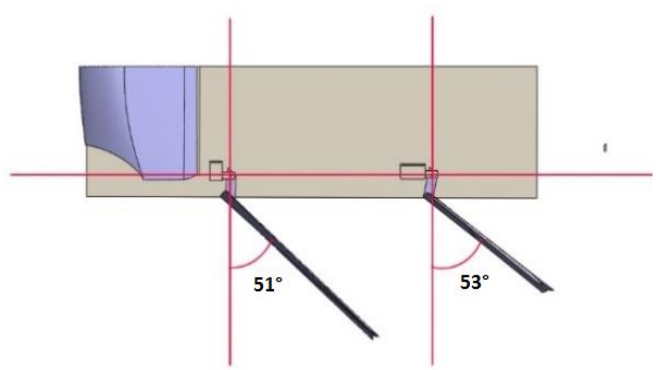


Fig. 12. Show setting of appropriate door angles

4.3 Weight Estimation

By using the command of measure inertia in CATIA, mass of the models can also be obtained. By using the command of rendering, material is added to the each of the model. The vehicle body is made-up of steel, it is considered in each model with a thickness of 0.6 mm. Comparison is done by taking into account the mass of the three vehicle bodies. Table 4 demonstrates the mass of each model.

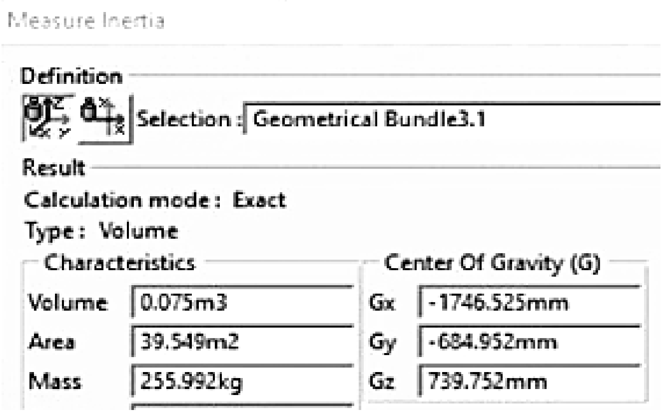


Fig. 13. Centre of gravity of Daihatsu Mira X

Table 3

Centre of gravity of three considered vehicles

S. No.	Car Models	Centre of Gravity (mm)
1.	Daihatsu Mira X	(-1746.525, -684.952, 739.752)
2.	Honda N-One 2013	(1645.23, 341.1, 914.82)
3.	Suzuki Wagon R 2012	(1726.46, 444.1, 845.228)

4.4 Base/Stock Comparison of All Three Considered Vehicles

Frontal projected area as calculated from Software for all three vehicles as 1.95 m^2, 1.59 m^2, and 2.05 m^2 for Daihatsu Mira X, Honda N-One 2013, and Suzuki Wagon R 2012 respectively as illustrated in Fig. 14.

Table 4

Mass of each vehicle

S. No.	Vehicle Models	Mass (Kg)
1.	Daihatsu Mira X	255.992
2.	Honda N-One 2013	290.996
3.	Suzuki Wagon R 2012	270.455

4.4.1 Comparison of total coefficient of drag

As shown in the Fig. 15, it can be observed that the Daihatsu Mira has the least overall Drag coefficient while Suzuki Wagon R has the highest total drag coefficient in stock configuration. It is understood that more the drag coefficient broader will be the wake region. This implies that Daihatsu Mira having the least drag coefficient will have a narrow wake region than the other two vehicles.

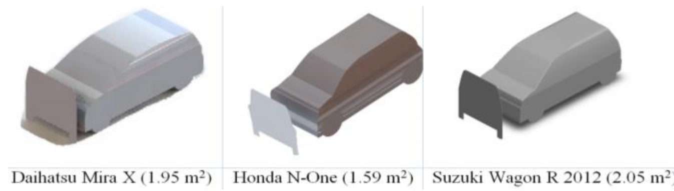


Fig. 14. Frontal areas calculated in SolidWorks

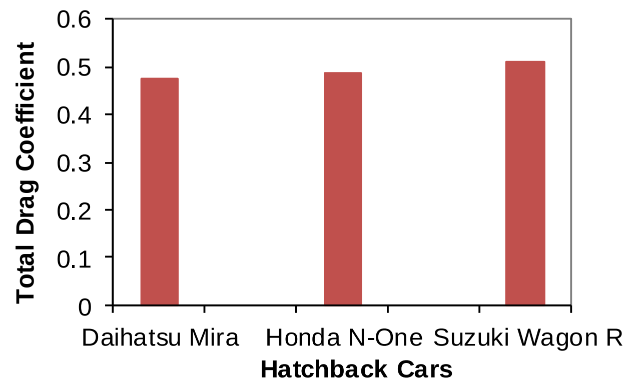


Fig. 15. Comparison of total drag coefficient of three hatchback vehicles

4.4.2 Comparison of total coefficient of lift

As shown in the Fig. 16, the Daihatsu Mira has the highest value of total lift coefficient as shown in the figure owing to its sleek aerodynamic shape while the Suzuki Wagon R has the least value because of its boxy and poor aerodynamic shape. The Mira streamlined body creates less disturbance for the air flow and air leaves more smoothly from the aft section of vehicle which is directly associated with its high overall lift coefficient. Honda N-One's Lift coefficient lies in between the other two vehicles because this vehicle has flat fore section and less streamlined body than Mira but on contrary more aerodynamically optimized than Suzuki Wagon R.

4.4.3 Comparison of velocity contours

As shown in Fig. 17, the velocity contours are showing the magnitudes of flow velocity around vehicle and also representing the thickness of wake of the three studied vehicles. It has been observed from the contour plot that the Wagon R has the strongest wake region and it propagates farther downstream comparatively. Wagon R has a region of highly separated flow behind the vehicle owing to its poor aerodynamic shape. The other two vehicles have wake region of lesser strength. The wake has a strong influence on the drag force of the vehicle i.e., large wake resulting in large drag forces on vehicle bodies.

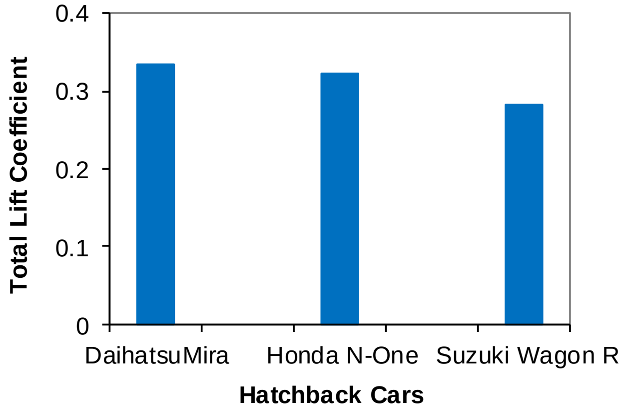


Fig. 16. Comparison of total lift coefficient of three hatchback vehicles

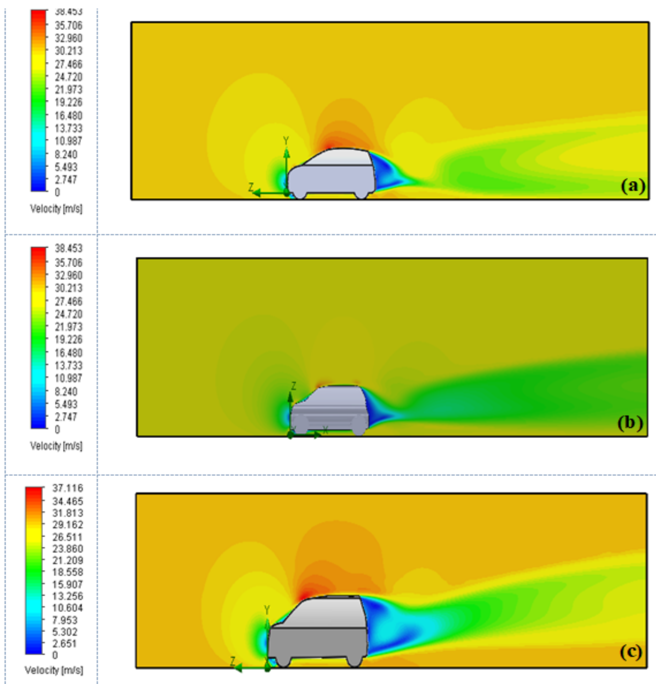


Fig. 17. Velocity contours of vehicles at 0° diffuser angle for (a) Daihatsu Mira (b) Honda N-one (c) Suzuki Wagon R

4.4.4 Comparison of pressure contours

As shown in Fig. 18, after comparing the contour plots of pressure, it is evident that the Suzuki Wagon R has the highest stagnation pressure on the front among the three vehicles which directly contributes to its higher drag force.

4.5 Comparison of All Three Vehicles at Various Diffuser Angles

Fig. 19 presents the different diffuser angles on which the analysis is performed for all three considered vehicles. From the Fig. 20, it has been observed that there is a slight variation in drag coefficient of all three hatchback vehicles after the addition of diffuser. The drag coefficient is slightly reduced at high diffuser angles.

As shown in the Fig. 21, the total lift coefficient of the hatchback vehicles first increases then decreases gradually after 5° of underbody diffuser angle. So, the addition of diffuser effectively improves the aerodynamic characteristics of vehicle. The study reveals that a relatively short diffuser has not a relevant aerodynamic benefit for a bluff body on wheels.

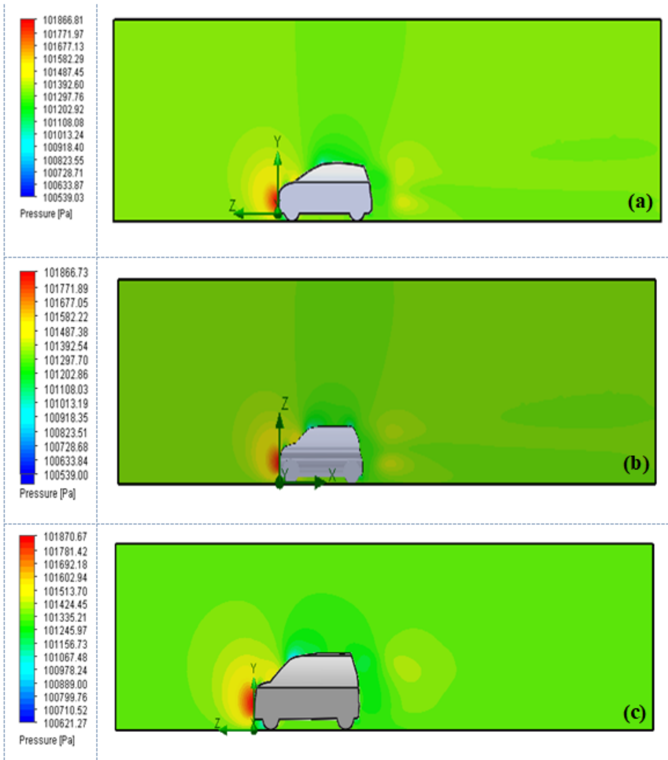


Fig. 18. Pressure contours of vehicles at 0° diffuser angle for (a) Daihatsu Mira (b) Honda N-one (c) Suzuki Wagon R

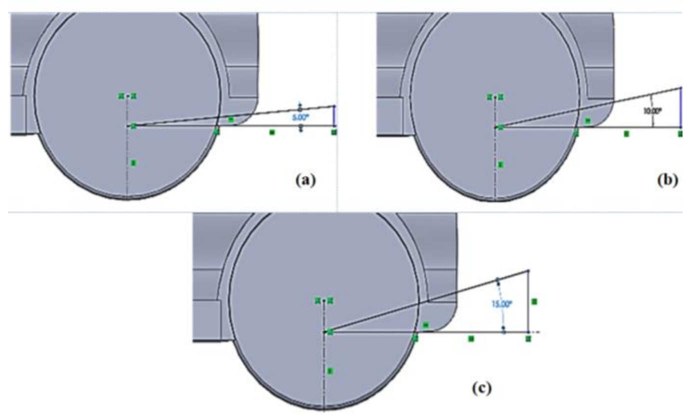


Fig. 19. Various diffuser angles on which analyses are performed for (a) 5° (b) 10° (c) 15°

The development of complex computer-aided models and aerodynamics simulations of vehicle body through CFD tool can rely on already established models and results due to the involvement of costs and complexities. For the validation of computational model and CFD procedure in this research work, the obtained results of the total coefficients of drag and lift, as well as, the variation of the total drag and lift coefficients against various diffuser angles were compared with already published data [33-37] and results were found in good agreement. Overall differences between the simulation and already published data results were less than 2.0%. The differences between the simulation and already published data analysis were expected due to the simplified assumptions of CAD modelling at a certain body and possibly lack of meshing in CFD simulation analysis. The error is persistent, and this is due to the nature of simulations which over predict the actual data. The data from simulations are ideal and do not consider the real-life turbulence and heat from the air from the time of the day in the CFD analysis of three considered vehicles.

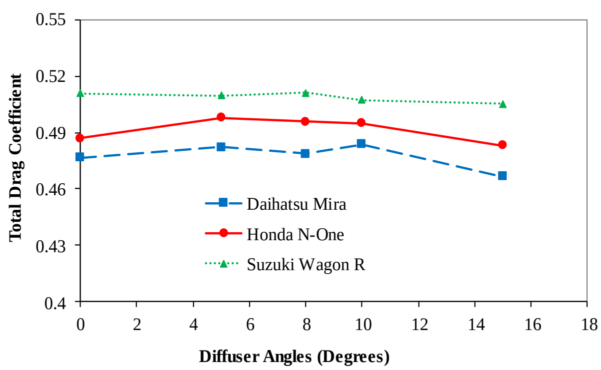


Fig. 20. Relation between drag coefficient and diffuser angles

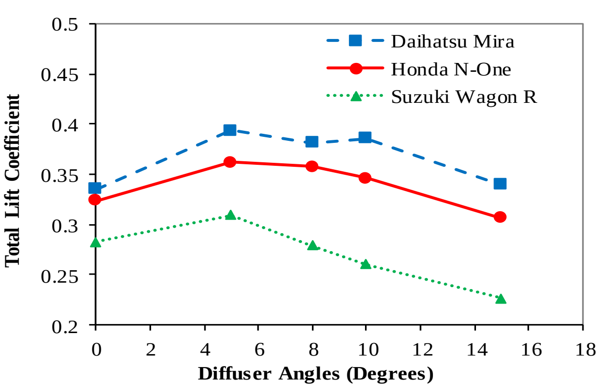


Fig. 21. Relation between lift coefficient and diffuser angles

5. Conclusion

Comparative analyses of three different hatchback vehicles were successfully performed through modelling and computational tools. It is found that when the individual parts are made through CAD, it is beneficial for the designers to study the motion and clearances required for each part to not interfere with one another. The appropriate door angles have been calculated through SolidWorks. The centre of gravity of the vehicle has been found by using CATIA-V5 R19 which shows that Honda N-One has lesser roll-over stability than the other two as its centre of gravity lies more in the z-axis while Daihatsu Mira X has the most stability. Furthermore, the weight of Honda N-One is found to be more than the other two vehicles. The aerodynamic characteristics of these hatchback vehicles were studied with and without underbody diffuser using CFD analysis. From CFD simulations results, it has revealed that in the stock configuration the Daihatsu Mira X has the least drag coefficient among the three vehicles which is one of the reasons of its good fuel economy. Suzuki Wagon R has the highest drag coefficient among the three vehicles owing to its large frontal area and shape which is one of the reasons for its comparatively poor fuel economy. The addition of underbody diffuser is beneficial from aerodynamic perspective since it reduces the overall lift coefficient at high diffuser angles. Moreover, the overall drag coefficient of hatchback vehicles is also slightly reduced at high diffuser angles. All of the three vehicles show reasonable reduction in total lift coefficient and slight reduction in total drag coefficient at 15° diffuser angle.

6. Acknowledgment

The author sincerely thankful to Dr. Muhammad Aslam Sheikh, retired as a ‘Reader’ from the Department of Mechanical, Aerospace and Civil Engineering (MACE), The University of Manchester, UK, for his critical discussion on obtained modelling and computational results and reading during manuscript preparation.

7. References

[1] "World car specifications, car specifications listed by manufacturer", https://www.carfolio.com (accessed on 15 Dec 2019).

[2] "Car-Specifications, CAR FROM JAPAN", https://carfromjapan.com (accessed on 14 Nov 2019).

[3] G. H. Yeoh, C. Liu, J. Tu, and V. Timchenko, "Advances in computational fluid dynamics and its applications", Modelling and Simulation in Engineering, vol. 1, pp. 3-4, 2011.

[4] M. N. Dhaubhadel, "Review: CFD applications in the automotive industry", Journal of Fluids Engineering, vol. 118, pp. 647-653, 1996.

[5] "Dassault-Systèmes, SolidWorks help - element types",

http://help.solidworks.com/2012/English/SolidWorks/cworks/Mesh_Types.htm?id=5600bd43ae334ceb9f0521de4f3b09e5 (accessed on 19 Nov 2019).

[6] K. J. Versprille, "Computer-aided design applications of the rational b-spline approximation form", Ph.D Thesis, Syracuse University USA, 1975.

[7] M. Tovey, "Drawing and CAD in industrial design", Design Studies, vol. 10, pp. 24-39, 1989.

[8] D. R. Ziethen, "CATIA V5: Macro programming with visual basic script", McGraw Hill, 1st Edition, 2013.

[9] G. F. K. Tecklenburg, "Body design, overview, targeting a good balance between all vehicle functionalities", Wiley Online Library, 2014.

[10] M. Hirz, T. Göber, and M. Lang, "Catia V5 Basic training; lecture script", Graz University of Technology, Germany, 2006.

[11] M. Hirz, "Advanced 3D-CAD Design Methods in Education and Research", Systemics, Cybernetics And Informatics, vol. 7, pp. 30-34, 2009.

[12] Y. Kojima, "Mechanical CAE in Automotive Design", R&D Review of Toyota CRDL, vol. 35, pp. 1-10, 2000.

[13] M. Tovey and J. Owen, "Sketching and direct CAD modelling in automotive design", Design Studies, vol. 6, pp. 569-588, 2000.

[14] S. Stadler, K. Thum, M. Hirz, and P. Rossbach, "Conceptual Full-Vehicle Development supported by Integrated Computer-Aided Design Methods", Computer-Aided Design and Applications, vol. 1, pp. 159-172, 2013.

[15] M. Hirz, J. Gulanova, and P. Rossbacher, "Future trends in CAD - from the perspective of automotive industry", Computer-Aided Design and Applications, vol. 14, pp. 734-741, 2017.

[16] R. H. Barnard, "Road vehicle aerodynamic design: An introduction", 1st edition, Longman Publisher, 1996.

[17] W.-H. Hucho, "Aerodynamics of road vehicles", Butterworth Ltd. England, 1987.

[18] R. Stone and J. K. Ball, "Automotive Engineering Fundamentals" SAE International, 2004.

[19] A. P. Mears and R. G. Dominy, "Racing Car Wheel Aerodynamics – Comparisons between Experimental and CFD Derived Flow-Field Data", SAE Technical Paper, vol. 1, pp. 1-14, 2004.

[20] E. Guilmineau, "Computational study of flow around a simplified car body", Journal of Wind Engineering and Industrial Aerodynamics, vol. 96, no. 6-7, pp. 1207-1217, 2008.

[21] J. Levin and R. Rigdal, "Aerodyanamic analysis of drag reduction devices for SAAB 9-3 by CFD", Master Thesis, Chalmers University of Technology Sweden, 2011.

[22] J. Donea and A. Huerta, "Finite Element Methods for Flow Problems", John Wiley and Sons, 2003.

[23] J. Katz, "Race Car Aerodynamics: Designing for Speed (Engineering and Performance)", 2nd edition, Bently Publishers, 1995.

[24] S. Hassan, "Aerodynamics Simulation of vehicle Body by using CFD Technology", Master Thesis, Teesside University UK, April 2014.

[25] M. N. Dhaubhadel, "Review: CFD Applications in the Automotive Industry", ASME Journal of Fluids Engineering, vol. 118, no. 4, pp. 647-653, 1996.

[26] G. Genta and A. Genta, "Road Vehicle Derodynamics", World Scientific Publishing Co. Pte. Ltd., 2008.

[27] S. F. Benjamin and C. Hickman, "The application of computational fluid dynamics within the automotive industry", Autotech 89 Seminar Papers. vol. C399/18, IMechE edition, 2014.

[28] A. Al-Saadi, A. Hassanpour, and T. Mahmud, "Simulations of Aerodynamic Behaviour of a Super Utility Vehicle Using Computational Fluid Dynamics", Advances in Automobile Engineering, vol. 5, pp. 1-5, 2016.

[29] F. Azmi, M. A. B. Marzuki, and M. A. A. Bakar, "Vehicle aerodynamics analysis of a multi purpose vehicle using CFD," ARPN Journal of Engineering and Applied Sciences, vol. 12, pp. 7, 2017.

[30] S. Thabet and T. H. Thabit, "CFD Simulation of the air flow around a car model (Ahmed Body)", International Journal of Scientific and Research Publications, vol. 8, pp. 517-525, 2018.

[31] S. Kant, D. D. Srivastava, R. Shanker, R. K. Singh, and A. Sachan, "A Review on CFD analysis of drag reduction of a generic sedan and hatchback", International Research Journal of Engineering and Technology, vol. 4, pp. 973-982, 2017.

[32] H. K. Versteeg and W. Malalasekera, "An Introduction to Computational Fluid Dynamics, The Finite Volume Method", 2nd edition, Pearson Education Ltd. UK, 2007.

[33] D. E. Aljure, O. Lehmkuhl, I. Rodríguez, and A. Oliva, "Flow and turbulent structures around simplified car models", Computers and Fluids, vol. 96, pp. 122-135, 2014.

[34] D. Ramasamy, K. Mohanesan, K. Kadirgama, M. M. Noor, and M. M. Rahman, "Hybrid electric vehicle car body drag analysis using computational fluid dynamics", International Journal of Automotive and Mechanical Engineering, vol. 14, pp. 4496-4507, 2017.

[35] B. Prajwal, D. Unune, A. Aherwar, and Jaikishan, "Modelling, simulation and validation of results with different car models using wind tunnel and Star-CCM+", Journal of the Serbian Society for Computational Mechanics, vol. 9, pp. 46-56, 2015.

[36] A. Huminic and G. Huminic, "Aerodynamic study of a generic car model with wheels and underbody diffuser", International Journal of Automotive Technology, vol. 18, pp. 397-404, 2017.

[37] C. Artur, P. Wojciech, S. Mateusz, and W. Dariusz, "Investigation on Aerodynamics of Super–Effective Car for Drag Reduction", Mechanics and Mechanical Engineering, Lodz University of Technology, vol. 20, pp. 295–308, 2016.