Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2019
(ICMERE2019) 11 – 13 December, 2019, Chittagong, Bangladesh

ICMERE2019-PI-249

# THE EFFECT OF AUTOMOBILE BODY MODIFICATION ON AERODYNAMICS

**Nirvik Barua**[1], **Farida Ahmed Koly**[1], **Sourav Paul**[1]

[1]Department of Mechanical Engineering, CUET, Chattogram, Bangladesh.

[1]nirvikbarua@gmail.com, [1]faridakoly@gmail.com, [1]sourav@cuet.ac.bd

***Abstract****- Aerodynamics play an important role in modern automobile designing. By changing aerodynamic parameters such as drag force and drag coefficient, performance of an automobile can be improved. In this study, aerodynamic properties of a standard model are analyzed by numerical calculations. The model is subjected to design modifications in order to improve aerodynamic performance. This study shows that among all modifications the most effective is using a front splitter because this modification results in the best balance between drag and lift force compared to the base model. It increases minimal drag force but reduces lift force by a big margin from the base model of the car.*

**Keywords:** Drag co-efficient, Lift co-efficient, Turbulence, Negative lift.

## 1. INTRODUCTION

Aerodynamics is the science which studies the interaction of gases flowing over bodies. Aerodynamics is primarily concerned with drag and lift forces which are caused by air passing over and around solid bodies. The principles of aerodynamics are applied in designing various things but primarily used for aircrafts and automobiles.[1]

Altering the shape of the body of a car can result in various improvements in aerodynamic parameters. In this paper, drag and lift forces of a Toyota Camry is studied with the use of different parts which are added on the standard model. Drag and lift forces are two most important parameters for evaluating aerodynamic performance.

The most significant aerodynamic force that applies to nearly everything that moves through the air is drag. Drag is generated in the direction the air is moving when it encounters a solid object. In most cases, such as in automobiles and aircraft, drag is undesirable because it takes power to overcome it. There are, however, some cases when drag is beneficial, such as with parachutes, for example. To describe the amount of drag on an object, we use a value called the drag coefficient ($c_d$). This number depends not only on the shape of the object but also on other factors, such as its speed and surface roughness, the density of the air and whether the flow is laminar (smooth) or turbulent. Forces that affect drag include the air pressure against the face of the object, the friction along the sides of the object and the relatively negative pressure, or suction, on the back of the object. For example, coefficient of drag ($c_d$) for a flat plate moving face-on through the air is about 1.3, a face-on cube is about 1, a sphere is about 0.5 and a teardrop shape is about 0.05. The drag coefficient for modern automobiles is 0.25 to 0.35, and for aircraft it is 0.01 to 0.03. Calculating coefficient of drag ($c_d$) can be complicated. For this reason, it is usually determined by computer simulations or wind tunnel experiments.[2]

Another force which is to be concerned is known as lift force. It is the upward force exerted on a body when air is passed under a body. In aircrafts, lift force is desirable as it enables it to fly but in automobiles it is mostly undesirable. Excessive lift force hinders lateral movement of vehicles which is dangerous for steering the vehicle at a higher speed. Lift force can be decreased by designing the vehicle in such a way that the pressure of the fluid flowing under the body is less than the pressure of the fluid passing over the body. This creates negative lift which is also known as downforce. Downforce is desirable for better handling of a vehicle but excessive downforce results in increase of drag force. Hence fuel consumption is increased as the engine needs to work more to produce more power to move at speed. So, there should be a balance between the drag force and the lift force.[3]

### 1.1 Literature Review

Numerous works have been carried out to improve aerodynamic capabilities of automobiles. The automobile sector is now focusing on modification of car models by various experimental methods. Computational fluid dynamics or CFD in short is being used for this kind of computations. A few of the works performed on this topic are given below to complete the present study.

Xiao‐Ming Xu et. al.[4] in their study on "The effect of automobile tail shape on aerodynamic performance" describes how the performance of an automobile is affected by differing of automobile tail shapes. They used wind tunnel tests as well as FLUENT software to carry out the calculations. They analyzed the basic model of the car and calculated a drag co-efficient of 0.7568 and lift co-efficient of 0.0545. After altering with the tail design of the model the drag co-efficient was lowered by 4.5% and the lift force was 41.6% hence improving the aerodynamic performance of the model significantly.

Xingjun Hu et. al.[5] in their study of “Influence of Different Diffuser Angle on Sedan’s Aerodynamic Characteristics” analyzed and described the influence of rear diffuser to improve aerodynamic performance of a vehicle. In this study they took the existing model of sedan and examined the aerodynamic forced acted on it using computational simulation. FLUENT was used in this case. They analyzed different angles and found out at an angle of 6 degree the model established the lowest co-efficient of drag and lift forces. It decreased the wake region by a noticeable margin from the standard model.

Ziyu Guo et. al.[6] in their study of “Optimization of the aerodynamic drag reduction of a passenger hatchback car” established the optimization process by employing computer aided styling, computational fluid dynamics, grid deformation, an optimization was conducted for the aerodynamic shape of a hatchback car. First, they made a CAD model of the vehicle and carried out numerical calculations using the software. Then the model was optimized by altering approach angle, front car face radian, hood angle, frond windshield angle which resulted in a substantial improvement in reduction of aerodynamic drag. Front drag was reduced by 5.64% and rear drag by 7.21%. Overall the drag of the whole car was reduced by 10.34%.

Jian Feng Wang et. al.[7] in their study on “Aerodynamic research of a racing car based on wind tunnel test and computational fluid dynamics” presented the effective method to study the aerodynamic characteristics of a racing car based on wind tunnel test and computational fluid dynamics simulation. The model was established by themselves and accuracy was judged by comparing both methods.

Jia Cheng Li et. al.[8] on their study about “CFD Based research on control strategy of Active Grille Shutter on automobile”, a grille of a vehicle was modeled in a way that it can be closed or shut at different angles actively. The aerodynamic forces acting on it were calculated using CFD. The results showed that the aerodynamic drag coefficient would be lowered with the decrease of the grille shutter. Therefore, the grille shutter can reduce fuel consumption of an automobile to a certain degree.

Ying chao Zhang et. al.[9] in their study about “Aerodynamics of Open Wheel Racing Car in Pitching Position” carried out CFD simulations and measured the aerodynamic influence of wings at different pitching angles. As the pitch angle increases, the drag co-efficient decreases gradually. It is because of the angle of attack on the front wing and the rear wing changes. The negative lift is mainly provided by the front wing, rear wing and floor. Increasing of pitching angle, the distance between the diffuser and ground is smaller, the flow gets blocked, causing the diffuser effect to gradually weaken and bottom pressure increase causing the lift to increase. The wheels and body always contribute to lift and increase as the pitching angle increases.

## 2. METHODOLOGY

In this analysis a car model was made with CAD software. Several modifications were made to the original model. Then the models were simulated and studied using CFD software.

### 2.1 Model Selection

For modeling, SOLIDWORKS is used. Simplified model of a Toyota Camry was created using this software. In Figure 1, we can see the basic model of the car. The model does not have any vents or side mirrors for simplification and for a basic overview of simulation.

The model was then subjected to a few modifications. First of all is addition of side skirts and diffuser which can be found on Figure 2 and Figure 3. Adding these should result in decrease of overall lift force of the car.

In Figure 4, the model is subjected to modification consisting of a spoiler at rear of the car which may help reduce the lift at the back of the car.

In the lower front side of the car a splitter is added to generate downforce during higher speed which can be observed in Figure 5.

Finally, in Figure 6, all of the parts mentioned above are added together in the basic model of the car. All the modified models are then used for simulation.

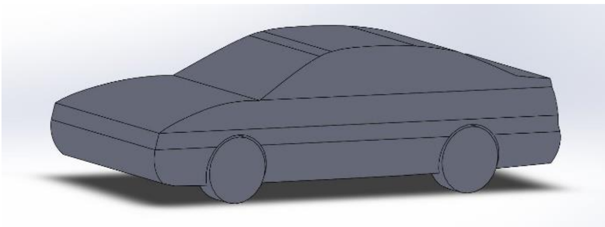

Figure 1: Basic model

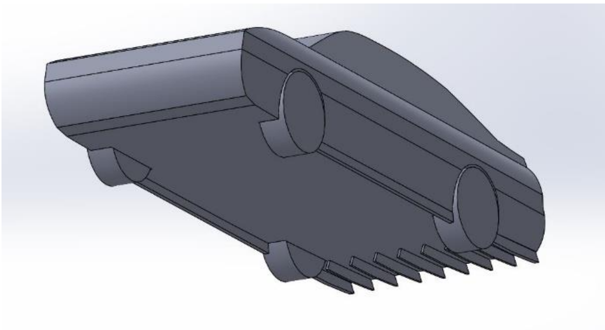

Figure 2: Side skirts and diffuser

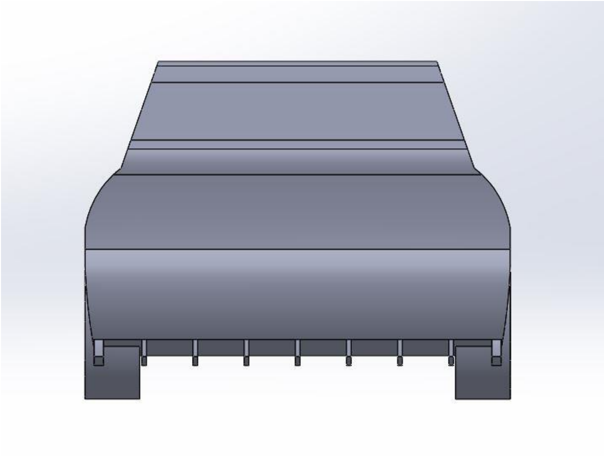

Figure 3: Diffuser

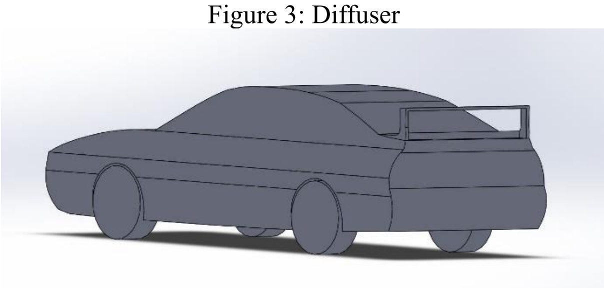

Figure 4: Spoiler

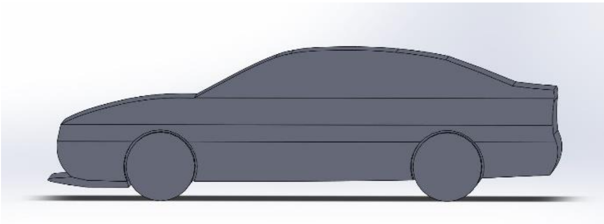

Figure 5: Splitter

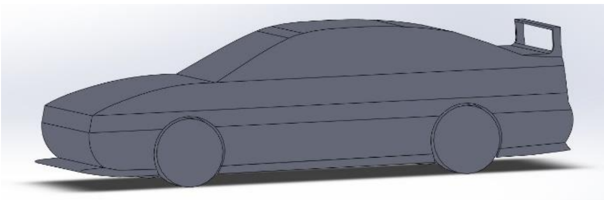

Figure 6: Combination of all modifications

### 2.2 Numerical Calculation

Computational fluid dynamics (CFD) discusses about visualizing how a fluid flows with the help of computational software and applied physics and mathematics. Using CFD the effect of the object when fluid is flown past it. Navier-Stokes equation is the base of CFD. The relation between temperature, pressure velocity and density of a fluid in motion are described by this equation.

When fluid flows over a car the nature of the flow is incompressible. Considering the separation of fluid is because of the shape of the car, the following is the control equation,

$$\partial\rho/\partial t + \partial/\partial_{xi}\,(\rho_{ui}) = S \qquad (3)$$

Here, $\rho$ is density, $ui$ is the velocity in i direction, t is time. S indicates the continuous phase quality which is 0 in case of incompressible flow. So, the equation becomes,

$$\partial\rho/\partial t + \partial/\partial_{xi}\,(\rho_{ui}) = 0 \qquad (4)$$

It is known as the energy equation.
The momentum equation is,

$$\partial/\partial t\,(\rho_{ui}) + \partial/\partial_{xj}\,(\rho_{ui\,uj}) = -\,\partial p/\partial x_i + \partial\tau_{ij}/\partial x_j + \rho g_i + F_i \qquad (5)$$

Here, $\rho gi$ indicates gravity. Fi is the external force in i direction which is 0 in this occasion. $\tau ij$ is the stress tensor. In this analysis, energy equation and momentum equation are the base of simulation.[10]

The simulation is completed in ANSYS Fluent simulation software. We used the Fluent module to analyze the drag force and lift force and also the drag co-efficient and lift co-efficient. ANSYS Student 2019 R1 is the version of the software which is used.

Conditions:

- The fluid was considered incompressible.
- The k-epsilon model was considered.
- Fluid type: Air
- Temperature: 298 K
- Wind velocity: 5.3 m/s
- Density of air: 1.225 kg/m3
- Pressure: 101325 Pa

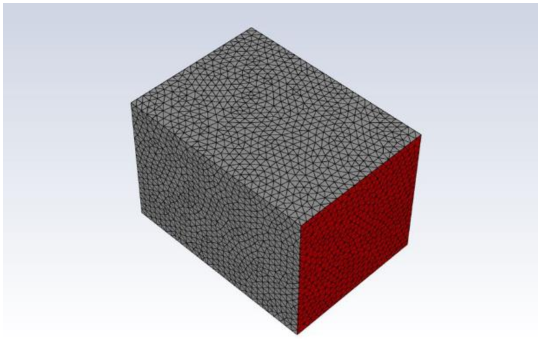

Figure 7: Boundary conditions applied

## 3. RESULT ANALYSIS

From simulation, the drag force and lift force of all the models are compared in Figure 8 and Figure 9.

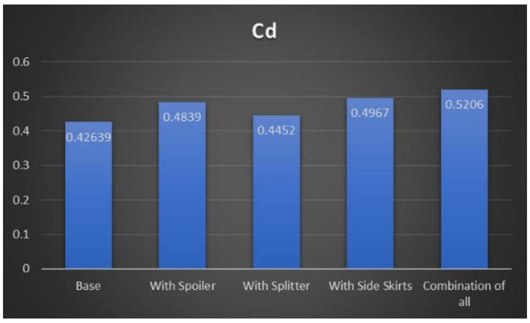


Figure 8: Comparison of drag forces

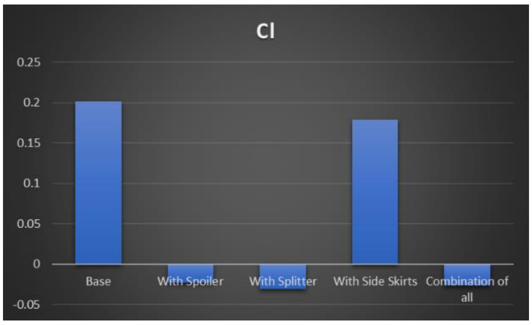

Figure 9: Comparison of lift forces

From Figure 8, it can be seen that drag co-efficient of the base model is the lowest. So, none of the modifications were suitable for reducing drag force. To find out whether the modifications are beneficial or not, the lift co-efficient should also be checked.

Figure 9 shows the different values of lift co-efficient of the modifications as well as the base model. It can be seen that all of the modifications resulted in reduction of the lift co-efficient.

Analyzing the modifications individually, the main reason for the change of the co-efficients can be observed.

For using side skirts and diffuser, the drag co-efficient increased slightly but the lift co-efficient decreased quite a lot. This is because side skirts and diffuser work together to channel the flowing wind under the body of the car. The velocity of the wind under the body of the car is greater than that of the wind which is flowing over the body. As a result, the upper body of the body has a higher wind pressure than the underbody. When travelling at a speed, the high-pressure air from top of the body tries to go to the lower pressure region under the car by flowing from side. Adding side skirts stops the high-pressure air from going to the lower part of the car. From Figure 10 and Figure 11, the velocity pathline of the base and modified model can be seen.

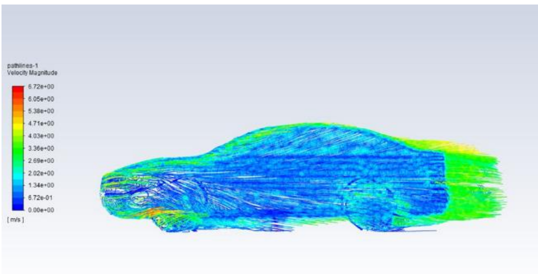
Figure 10: Velocity pathline of the base model

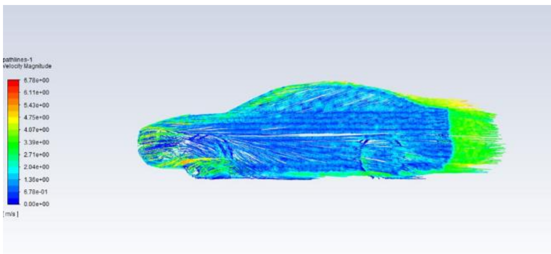
Figure 11: Velocity pathline of the model consisting side skirts and diffuser

While using spoiler, it can be seen that spoiler increases the drag co-efficient from the base model. In this case, the increase was less than the previous one. But the lift co-efficient of the car was decreased drastically to the point where it became negative. Which means the car now produces negative lift. It is very helpful for improving the stability of the car when cornering at a high speed. The back of the car usually creates turbulence which results in a very low-pressure area at the back of the car and thus increasing drag force. By adding a spoiler this turbulence can be minimized. So, pressure drag due to turbulence is less. But adding spoiler also can reduce lift and can also result in negative lift. Which is caused because of reducing the flow separation at the back of the car. This can also result in increase of drag which is the reason for increase of drag co-efficient in this case. In Figure 12 the velocity pathline of the modified model with spoiler can be observed.

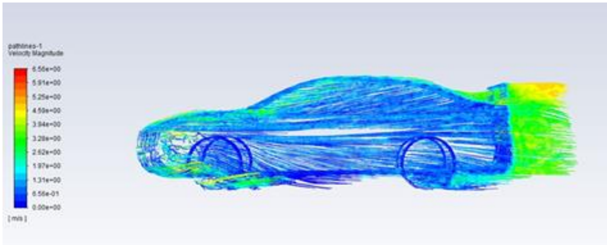
Figure 12: Velocity Pathline of the model consisting of Spoiler

Using splitter also results in increase of drag co-efficient but here the increase is very minimal compared to the base model. Splitter also produces negative lift like spoiler. But in this case, it is more negative than using a spoiler.

Adding a splitter results in increase of negative lift of the car as we can see here. A splitter produces negative lift by increasing the high-pressure region at the front of the car and sends the air under the body of the car where it moves at a high velocity. Figure 13 and Figure 14 shows the high-pressure region formed in the base and modified model respectively.

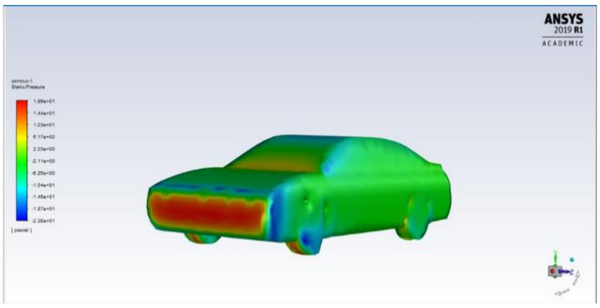

Figure 13: High pressure region of the base model

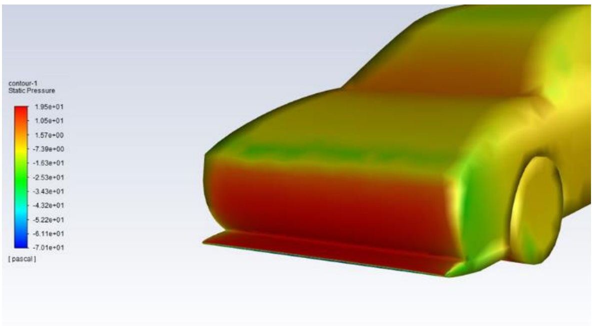

Figure 14: High pressure region of the model consisting a splitter

When all of the mentioned parts added together it results in increase of drag co-efficient from the base model. The increase is higher than all other parts individually. Which is not good at all. The lift co-efficient here is also negative like the previous two modifications but this configuration is not beneficial because the increase of the drag co-efficient is too big. Figure 15 shows the velocity pathline of this model.

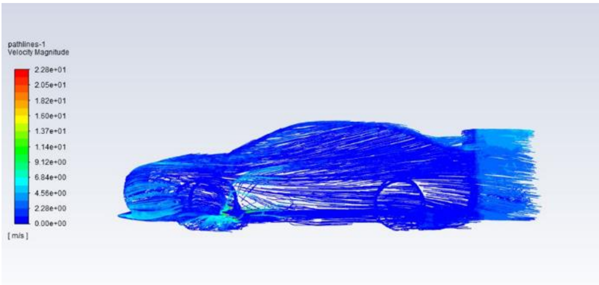

Figure 15: Velocity pathline of the model consisting combination of all parts.

## 4. CONCLUSION

(l) When all the modified models are compared to the base model it is seen that the drag co-efficient increases on every modification but lift co-efficient decreases by a lot on all the modifications.

(2) Although none of the modifications resulted in reduction of drag force, some modification only increased drag by a few percentages to yield a huge reduction in lift force which is very practical when travelling at higher speeds. Addition of front splitter resulted in the most significant improvement. It generated quite a lot of negative lift at the front of the car by sacrifices only a little amount of drag. To put that in numbers it is seen that the increase of drag co-efficient is 4.43% and for the sacrifice of this drag force the lift co-efficient is decreased by a huge margin of 115.37% which is very useful for the stability of the car while cornering at a high speed. Also, as the Toyota Camry is a front wheel drive vehicle, the extra downforce at the front of the car will help the front wheels to get better traction when turning at a high speed.

## 5. ACKOWLEDGEMENT

Firstly, I am very grateful to my honorable project supervisor Farida Ahmed Koly, Lecturer, Department of Mechanical Engineering, CUET for providing me valuable guidance, inspiration and also necessary information for the study to be successful. I would also like to thank to all the faculty members of Department of Mechanical Engineering, CUET and my classmates for assisting me whenever help was necessary.

## 6. REFERENCES


1) https://www.livescience.com/47930-what-is-aerodynamics.html (Accessed: 22 March 2019).
2) https://www.real-world-physics-problems.com/drag-force.html (Accessed: 22 March 2019).
3) https://www.grc.nasa.gov/www/k-12/airplane/lift1.html (Accessed: 22 March 2019).
4) Xu, X.M., Fu, J.Q., Zhang, X.Y., Hu, D.H., Li, R.Z. and He, R., 2018. The effect of automobile tail shape on aerodynamic performance. Heat Transfer—Asian Research, 47(2), pp.420-436.
5) Hu, X., Zhang, R., Ye, J., Yan, X. and Zhao, Z., 2011. Influence of different diffuser angle on Sedan's aerodynamic characteristics. Physics Procedia, 22, pp.239-245.
6) Guo, Z., Zhang, Y. and Ding, W., 2018. Optimization of the aerodynamic drag reduction of a passenger hatchback car. Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, p.0954410018786619.
7) Wang, J., Li, H., Liu, Y., Liu, T. and Gao, H., 2018. Aerodynamic research of a racing car based on wind tunnel test and computational fluid dynamics. In MATEC Web of Conferences (Vol. 153, p. 04011). EDP Sciences.
8) Li, Jia Cheng & Deng, Ya Dong & Wang, Yi Ping & Su, Chuqi & Liu, Xun. (2018). CFD-Based Research on Control Strategy of the Opening of Active Grille Shutter on Automobile. Case Studies in Thermal Engineering. 12. 10.1016/j.csite.2018.05.009.
9) Zhang, Y., Yang, C., Wang, Q., Zhan, D. and Zhang, Z., 2018. Aerodynamics of open wheel racing car in pitching position (No. 2018-01-0729). SAE Technical Paper.
10) https://whatis.techtarget.com/definition/computational-fluid-dynamics-CFD (Accessed: 22 March 2019).


## 7. NOMENCLATURE

| Symbol | Meaning | Unit |
|---|---|---|
| $T$ | Temperature | (K) |
| $C_d$ | Drag co-efficient | Dimensionless |
| $C_l$ | Lift co-efficient | Dimensionless |