DESIGN OF AUTOMOBILE'S BODY SHAPE AND STUDY ON EFFECT OF AERODYNAMIC AIDS USING CFD ANALYSIS

Akshay S[1], Ashik Vincent[2], Athul Anand R[3], George Kurian[4], Dr. Shajan Kuriakose[5]

[1,2,3,4] B-Tech Degree student, Mechanical Engineering, Mar Athanasius College of Engineering, Kothamangalam, Kerala, India
[5]Professor and Head of Department, Department of Mechanical Engineering, Mar Athanasius College of Engineering, Kerala, India

---

Abstract – *Aerodynamic characteristics of a car are of significant interest with regards to improved performance, fuel efficiency, and stability. The drag force resists the motion of the vehicle and increased drag results in increased fuel consumption. Lift force tends to lift the vehicle off the ground and increased lift results in decreased stability. Therefore, the target is to reduce drag and decrease lift or increase downforce. This can be achieved by incorporating different aerodynamic aids such as air dams with front splitters, side skirts, vortex generators, spoilers, rear wings which modify the values of coefficients of drag and lift. However, each setup yields different combinations of these values and it is important to identify their individual contributions so that a suitable aid can be selected by a customer according to his requirement. This study aims to analyze different performance characteristics like lift and drag, with and without different aerodynamic aids like air dam with front splitters, rear wing and their combination. This work will present a numerical solution of air flow around the outer body shape of Audi R8's 3D model with the help of Computational Fluid Dynamics (CFD) based predictions using ANSYS Fluent software which will help in identifying the individual effects on aerodynamic characteristics of various aerodynamic aids.*

Key Words: Drag, Lift, Computational Fluid Dynamics(CFD), Aerodynamics, Air dam with front splitter, Rear Wing

1. INTRODUCTION

Each aerodynamic aid modifies the drag and lift forces according to their geometry and the change is different for different aids. The main purpose of an air dam and front splitter combination is to aid in the optimization of the flow of air over the rest of the car and reduce drag while creating downforce by reducing the average air pressure under the car.

A splitter is a horizontal shelf mounted to the bottom of an air dam, with the top side of the splitter sealed to the body. A splitter produces downforce from the difference in air pressure on the top and bottom surfaces of the splitter area. Because airflow over the top is blocked by the body or air dam, the local airspeed is low and the air pressure on top is high. Because air can flow under the splitter freely, the local airspeed under it is high and the air pressure on the bottom side is low.

The design goal of a rear wing is to increase downforce. This is achieved by using an inverted aerofoil profile which causes the air to flow faster underneath the wing creating a low pressure on the bottom surface of the wing while the slower moving air above the wing exerts high pressure on the top surface of the wing however drag also increases with this setup.

But it is being seen nowadays that most vehicle owners incorporate different aerodynamic aids on to their vehicles without understanding the effect each aid produces on the motion of the vehicle. Each setup yields different combinations of coefficient of drag and lift values and it is important to identify their individual contributions so that a suitable aid can be selected by a customer according to his requirement. This work will help in educating a customer about the individual effect of each aerodynamic aid and compare the relative magnitude of change in C_d and C_l values of the vehicle which were brought about by incorporating these aids.

2. METHODOLOGY

We have run the simulation in ANSYS Fluent 16.0 which is a robust computational fluid dynamics tool after importing the geometry and meshing.

2.1 Modelling

Audi R8 model was selected as the base model for our study due to it being a popular and high performance, fast car. Modelling of the body shape of Audi R8 was performed using the SolidWorks software. SolidWorks was chosen on account of the ease of modelling and the user-friendly options that it offered. The blueprints were superimposed on the front, top and right plane which were then used as reference. Splines were sketched on different planes tracing the curves of the model as shown in the blueprints. Model was completed using different surface operations.

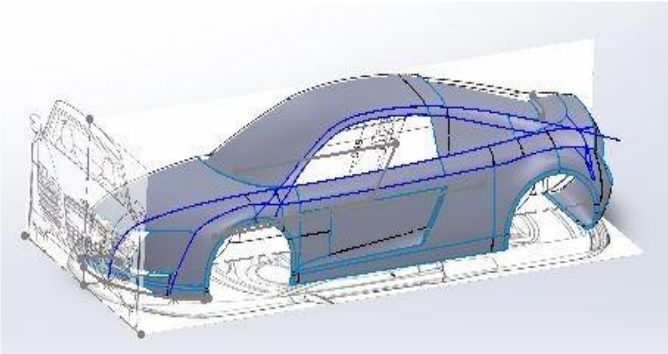

Fig-1: Surface generation in Solidworks

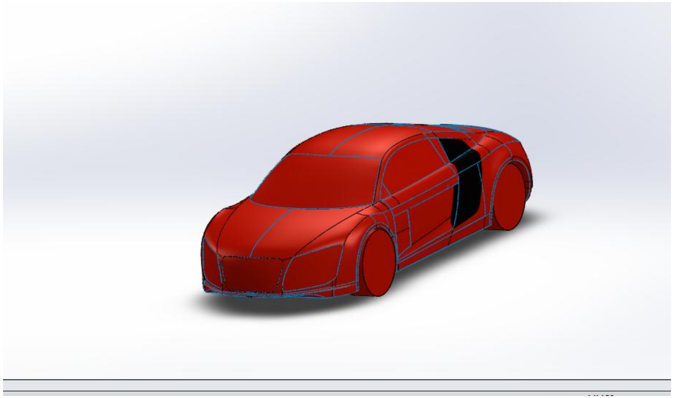

Fig-2: Completed CAD model of Audi R8 in Solidworks

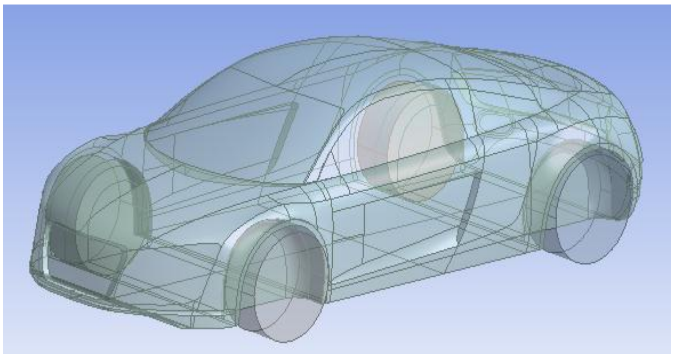

Fig-3: Base model imported to ANSYS

Air dam, splitter and rear wing were modelled using the base model as reference to obtain the remaining test cases. The effectiveness of air dam and splitter combination can be increased by increasing the height of the air dam and the length of the front splitter. However, in order to ensure that the car remained fairly usable on the road, certain restrictions were imposed. Ground clearance was kept at 8 cm. The length of the front splitter was arbitrarily chosen as 10cm. This ensures that it is long enough to serve its purpose but not so long that it scrapes and digs into the ground. The front splitter thickness was arbitrarily selected as 1.5cm as this is the average thickness of most commercially available carbon fibre and plastic front splitters. For rear wing, a general inverted aerofoil profile was drawn.

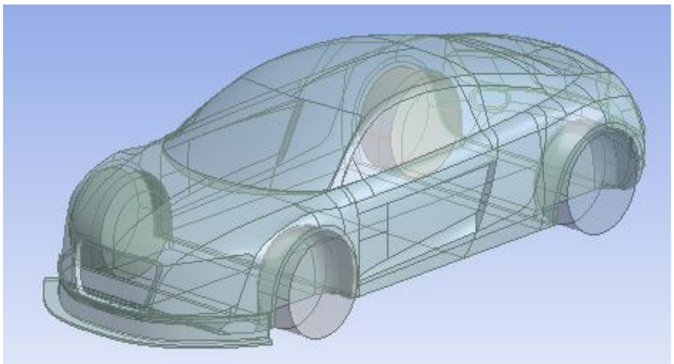

Fig-4: Model with air dam and splitter

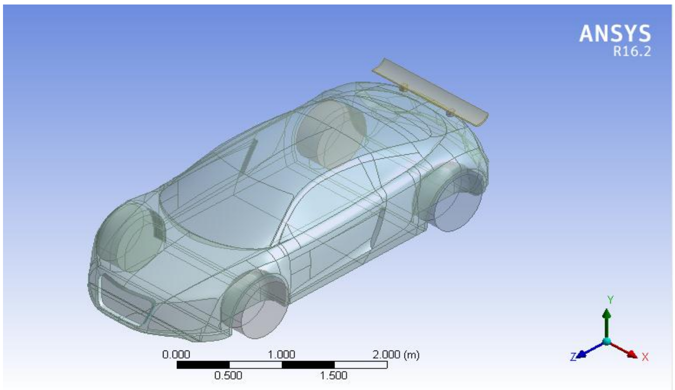

Fig-5: Model with rear wing

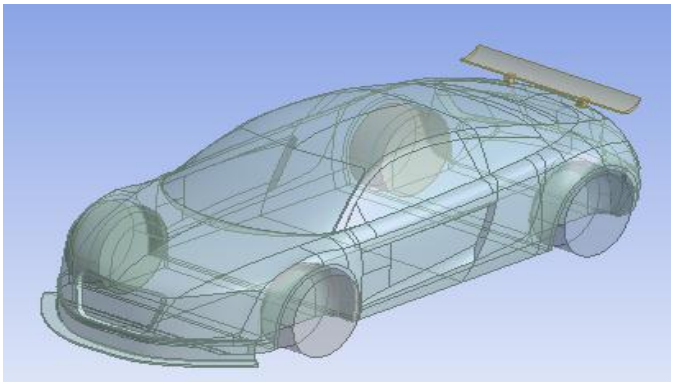

Fig-6: Model with air dam, splitter and rear wing combined

2.2 Computational Procedure

Enclosure and the car body inside it were split into half and coarse mesh was generated keeping in mind the computational time required to solve such a complex body. 'Proximity and Curvature' settings were switched on which helped the mesh to capture the curves of the car effectively. Suitable boundary conditions and solver settings like magnitude and direction of velocity, viscous model, solution methods and controls were selected based on general practices and guidelines for automotive aerodynamics.

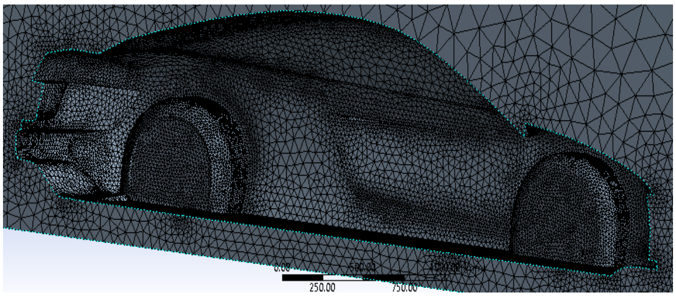

Fig-7: Meshing

3. RESULTS AND DISCUSSION

Table-1: Simulation results

S No	Case	Coefficient of drag	Coefficient of lift
1	Base Model	0.308	0.148
2	Air dam and Splitter	0.287	0.008
3	Rear Wing	0.414	-0.265
4	Air dam, Splitter and Rear wing	0.368	-0.348

Simulation was done for each of the 4 cases and the relative magnitude of change in C_d and C_l values after incorporating each aerodynamic aid was obtained.

3.1 Case 1- Base Model

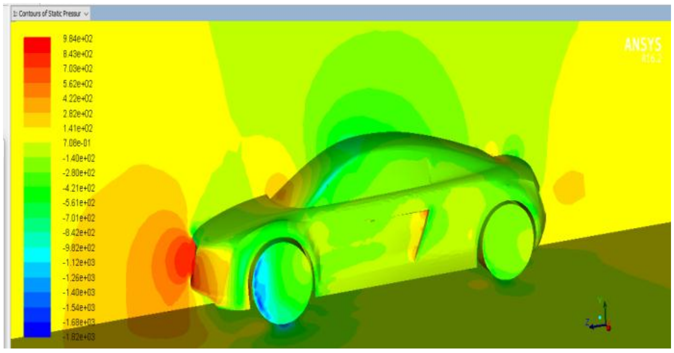

Fig-8: Pressure contours

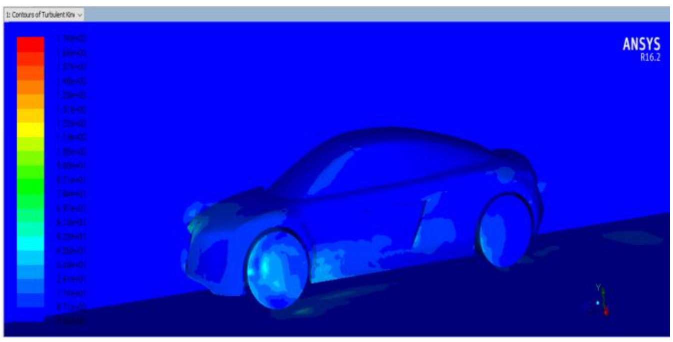

Fig-9: Turbulence contours

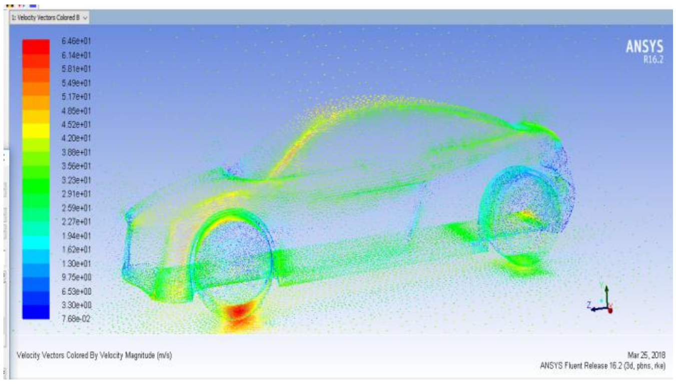

Fig-10: Velocity vectors

We take first case as a validation analysis. The C_d value specified by the manufacturer Audi for the model R8 is 0.36. The C_d and C_l values obtained from the simulation were 0.308 and 0.148 respectively. The values obtained are slightly different from the original value which is mainly due to minor inaccuracies in modelling and limitations of CFD compared to actual wind tunnel testing however, since the results obtained are within agreement to the actual value, our base model has proven to be a fairly accurate representation and the results obtained in CFD analysis are valid and correct within limited error.

3.2 Case 2- Model with air dam and splitter

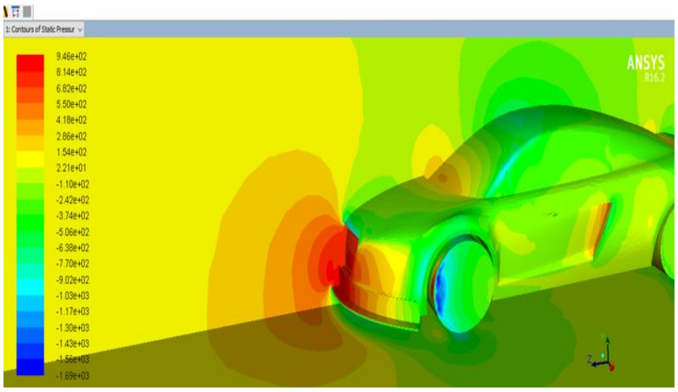

Fig-11: Pressure contours

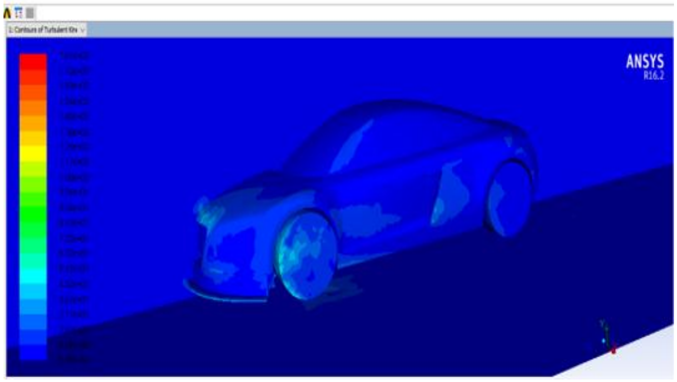

Fig-12: Turbulence contours

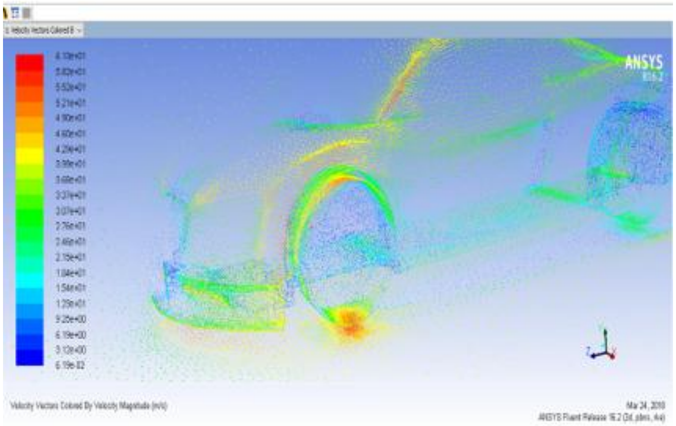

Fig-13: Velocity vectors

The C_d and C_l values obtained from the simulation were 0.284 and 0.008 respectively. The air flow is brought to stagnation above the splitter creating area of high pressure. The splitter directs air away from this stagnation point and accelerates air under car which causes high pressure above and low pressure below creates downforce by way of Bernoulli effect. With this setup, we observe that drag was reduced, and downforce is increased but not by a significant amount.

3.3 Case 3- Model with rear wing

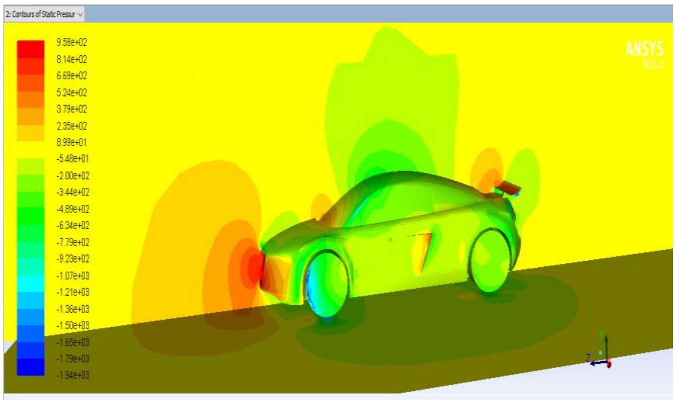

Fig-14: Pressure contours

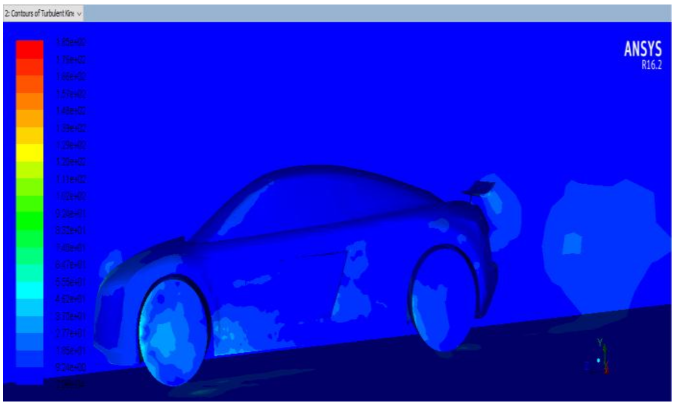

Fig-15: Turbulence contours

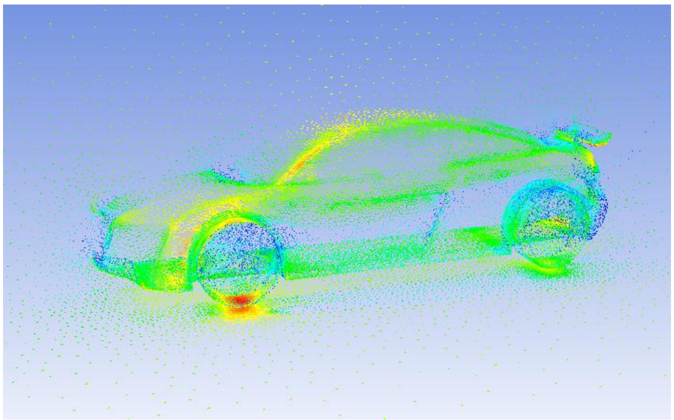

Fig-16: Velocity vectors

The C_d and C_l values obtained from simulation were 0.414 and -0.265 respectively. The design goal of a rear wing is to increase downforce. This is achieved by using inverted aerofoiled profile which causes the air to flow faster under the wing creating a low pressure on the bottom surface of the wing while the slower moving air above the wing exerts high pressure on the top surface of the wing. With this setup, we observe that downforce is increased greatly but drag increases.

3.4 Case 4- Model with air dam, splitter and rear wing combined

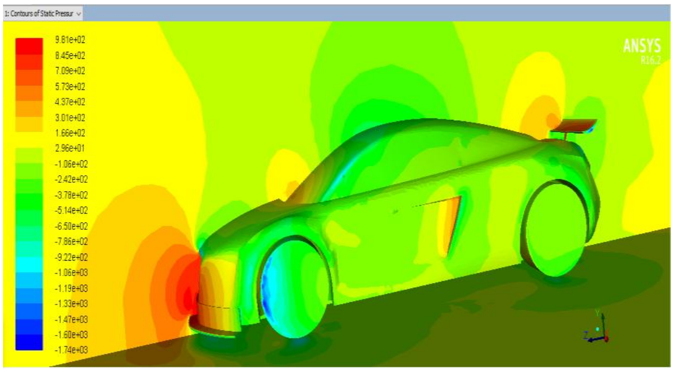

Fig-17: Pressure contours

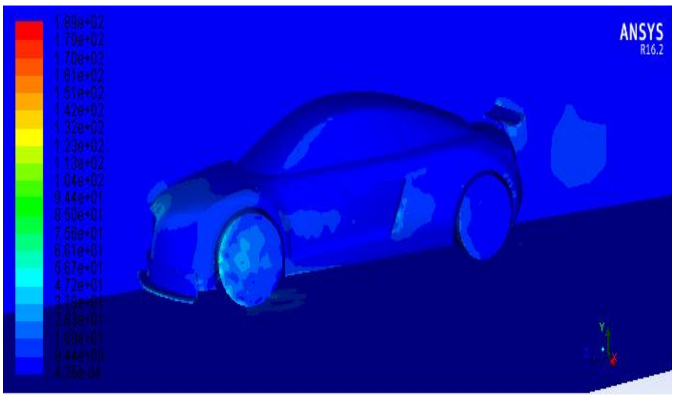

Fig-18: Turbulence contours

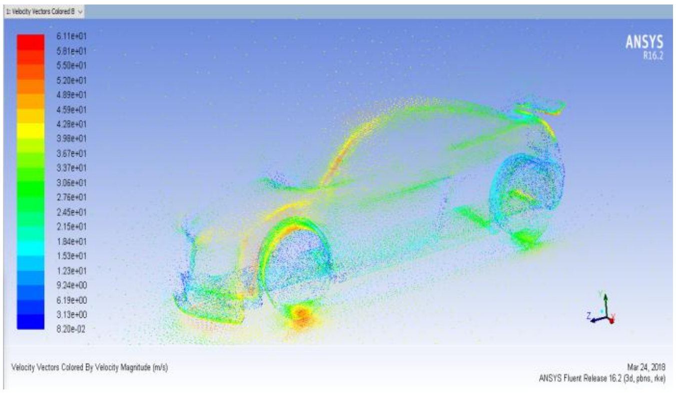

Fig 19: Velocity vectors

The C_d and C_l values obtained from simulation were 0.368 and -0.348 respectively. With this setup, we observe that downforce is considerably increased and drag is increased compared to base model but reduced compared to case when only rear wing is used.

4. CONCLUSION

The use of aerodynamic aids proves to be an effective means of varying the drag and lift forces on the vehicle body. Each setup yields distinct combinations of drag and lift coefficients. The setup with air dam and splitter proved to be the most effective in reducing the drag coefficient of the body. Therefore, this setup will give the highest fuel economy. The setup which incorporated air dam, splitter and rear wing proved to be the most effective in reducing the lift coefficient of the body. Therefore, this setup will ensure the highest degree of stability for the vehicle body by providing larger down-force, maximizing traction between the tire and the road surface.

The incorporation of aerodynamic aids to the vehicle body by the customer is usually done with the motive of augmenting the aesthetic beauty of the vehicle. However, the effect of these aids on aerodynamic performance are not taken into account. The understanding of the individual effects of each setup can be used by the customer to make a well-informed choice of which aerodynamic aids to incorporate so as to optimize vehicle performance according to his/her preference.

REFERENCES

[1] Lanfrit, Marco, 'Best Practice Guidelines for Handling Automotive External Aerodynamics with FLUENT', Fluent Deutschland.

[2] Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu, 'Computational Fluid Dynamics – A Practical Approach', Butterworth.

[3] S.M. Rakibul Hassan*, Toukir Islam, Mohammad Ali, Md. Quamrul Islam, "Numerical Study on Aerodynamic Drag Reduction of Racing Cars", 10th International Conference on Mechanical Engineering, ICME 2013

[4] Sneh Hetawal*, Mandar Gophane, Ajay B.K., Yagnavalkya Mukkamala, "Aerodynamic Study of Formula SAE Car", 12th Global Congress on Manufacturing And Management, GCMM 2014

[5] Rubel Chandra Das∗, Mahmud Riyad, "CFD Analysis of Passenger Vehicle at Various Angle of Rear End Spoiler", 10th International Conference on Marine Technology, MARTEC 2016