International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

Numerical Study of Effect of Drag Reduction Add-on Device on a Generic Car

A. Sankaran
Department of Aeronautical Engineering,
Nehru Institute of Engineering &Technology,
Coimbatore,India.

L. Balaji
Department of Mechanical Engineering,
Prince Dr. K.Vasudevan College of Engineering and Technology,
Chennai, India

Surendra Kumar Yadav
Department of Mechanical Engineering,
Prince Shri Venkateswara Padmawathy Engineering College,
Chennai, India

Sovan Samanta
Department of Mechanical Engineering,
Prince Dr. K.Vasudevan College of Engineering and Technology,
Chennai, India

Prashant Sinha
Ground Instructor,
FSTC pvt Limited,
Gurgaon, India

***Abstract*—This paper present the effects of drag reduction add-on device on the drag of a generic car model. In the present paper, modeling of the generic car with and without add-on device have been done using Catia V5 and numerical simulation have been performed using ANSYS Fluent 17.0 to evaluate the coefficient of drag and coefficient of lift for the generic car with and without add-on device. The introduction of add-on device at the generic car model results in reduction of drag coefficient at all the speed whereas about 29.04% drag coefficient reduction was achieved at maximum velocity of 33.33 m/s and the lift coefficient was decreased by 11.11%. Reduction of the drag and lift coefficient improve the fuel economy and the stability of the generic car.**

Keywords—Drag coefficient; lift coefficient; add-on device; stability; generic car

I. INTRODUCTION

Aerodynamics is the study of the interaction of air with moving bodies; it forms a major and specialized part of fluid dynamics. Whereas, the aerodynamics dealing with road vehicles is known as automotive aerodynamics. This part of fluid dynamics deals with the main objectives like the reduction of drag, reduction of undesired lift forces and improving the stability of the vehicle at higher speed, reduction of noise and thus producing an attractive profile for the vehicle. Also, there is a necessity to increase the down-force for better cornering abilities of the vehicle. A clear understanding of the profile of vehicle with respect to the aerodynamics can produce innovative thoughts so as to achieve the reduction in drag and therefore increase the fuel efficiency. The evolving technologies have led to the downfall of automobile industries, due to the fuel crisis. At present, it is not possible for everyone to replace their conventional car with an electric car, as they are costlier for the majority of our population. Also, the increased air quality index and air pollution are alarming. So, the need of the hour is to lower the fuel consumption applying vehicle aerodynamics. It is possible to modify the flow around the car body and wake behind it using some add-on devices. There are many existing add-on devices like vortex generators and spoilers. Here the idea is to have an attachable device at the rear end of the car body.

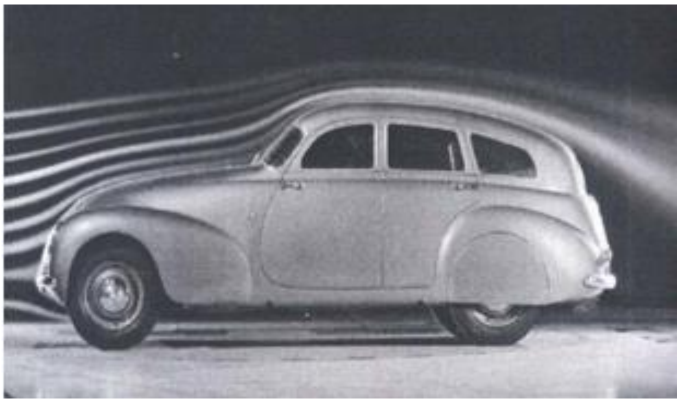

Fig.1. Flow around Austin A70

The streamlines around a vehicle show the flow pattern around the vehicle as shown in "Fig. 1,"when the vehicle moves at a certain velocity, there is a formation of a thin layer called the boundary layer, within this layer the viscous effects of the fluid are dominant and outside this layer the in-viscid effects are dominant. As the flow moves towards the rear part of the car body, the boundary layers separate and fluid flow is detached. The boundary layers lose kinetic energy and swirl around to form vortices and result in drag and wake formation.

V. Naveen kumar et al. [1] stated that the spoiler can increase tires capability to produce cornering force which can stabilize vehicles at high speed. The reduction of lift that acted on the rear end, a rear spoiler can be attached on it to create more pressure and eliminate the vacuum space created. Spoilers are mostly used on sedan-type cars. They act like barriers to air flow, in order to build up adequate air pressure in bottom surface of the spoiler. A spoiler is an automotive aerodynamic device whose intended design function is to 'spoil' unfavorable air movement across a body of a vehicle in motion, usually described as turbulence or drag. Rear spoilers are provided to increase the negative lift of the vehicle. An investigation is performed by S.M. Rakibul Hassan and Toukir Islam [2], study the effect of change of rear end spoiler inclination angle over co-efficient of drag & lift. Flow separation is responsible for the major portion of the aerodynamic drag of cars. The aerodynamic drag coefficient of the car model used here is found to be 0.3233. The main design consideration to reduce the drag of any bluff body should be- keep the flow attached to the body as much as possible. That means maintaining streamline shape, reducing surface roughness, fewer joints of the body or avoiding sharp fillets, controlling lift force, air or exhaust gas flow towards the low-pressure zone at the rear portion of the car etc. It is clear from C.V. Karthick Bala Murugan et al. [3] research that at a certain height of spoiler and wind collision angle, the change in Cd is negligible as the speed of the racing car increases. However, the downward force acting on the racing car with the spoiler at the rear end increases significantly lower as the speed of the racing car increases. Moreover, the lower spoiler height tends to gives both higher Cd and Cl. T. D. Ipilakyaa et al. [4] explains how more down force than having less drag can be more important for passenger cars because driving safely is always number one priority. Also, with a suitable and efficient down force device as this, the self-weight of the vehicle can conveniently be reduced without consequence and this will compensate for the increased drag in terms of fuel consumption. Shyam P. Kodali and Srinivas Bezavada [5] explained that the disturbance created in the streamline flow due to the presence of a rear spoiler, there is reduction in the flow separation at the trunk resulting in increase of turbulence. In conclusion, their study reveals that rear spoilers have considerable effect on lift, i.e. vehicle stability and moderate effect on drag i.e. fuel consumption. Rubel Chandra Dasa and Mahmud Riyada [6] have concluded that at a particular spoiler height the spoiler that possess smaller angle of wind collision gives higher drag force. This is due to the fact that with smaller angle of wind collision, the spoiler would create smaller recirculation zone behind the rear end of the running vehicle. This implies to higher pressure behind spoiler but lower pressure behind the rear end of the vehicle. Rear spoilers redirect the airflow behind the vehicle & increase the negative lift of the vehicle. See-Yuan Cheng et al. [7] investigated that there is a critical value of spoiler height where the hatchback model would experience a dramatic increase in its aerodynamic forces associated with the change in the flow topology above its slanted rear surface. Since down force is proportional to the tractive force, it is imperative for the ride's stability and safety. Furthermore, high down force performance is vital during cornering when the vehicle needs sufficient traction for it to pass the curve without slipping. Sneh Hetawal et al. [8] reveal that the drag force is the unwanted thing which normally acts against the drive force of the car. The down force is useful to maintain the race car in ground. The velocity of the air is reduced in the rear surface. This layer above the surface is called boundary layer. The boundary layer initially will be laminar and transforms into turbulent as it moves further over the larger distance from car surface. When there is sudden change in the surface this would lead to flow separation. Flow separation normally takes place when the upper layer of the fluid can no longer pull the lower layer of fluid within the boundary layer. Krzysztof Kurec et al. [9] explains the orientation of the spoiler affect not only the separation on the wing but also on the trunk which contributes to the changes in the pressure distribution. It is important to note that the presence of the rear wing is very important in this case, however the change of the load on itself is not the biggest factor.

When a car moves forward, a low pressure zone is formed in the vicinity of the car specifically behind the car. This low pressure zone pulls the car back-word opposite to its moving direction and creates drag force. This low pressure zone is created due to the flow separation and consequent are the formation of vortices behind the car. In order to reduce this pulling-back effect a unique idea is to add one cylindrical shape body behind the car named as add-on device. Which actually, direct the flow towards low pressure zone. This reduces the effect of low pressure and hence reduced the drag force acting on the car.

II. METHODLOGY OF STUDY

In the present research paper, a model of a generic car is modeled using the Catia v5 and this generic model is imported into the ANSYS Fluent 17.0 and meshing was generated on the surface of the model with and without add-on device. Aerodynamic study of airflow over an object can be carried out either using analytical method or CFD approach. One side, simple flow over simple geometries like laminar flow over flat plate can be solved analytically but the complex flow like over bluff body; the flow is turbulent and can't solve Navier-Stokes and continuity equations analytically. Another side, it is impossible to get direct solution for Navier-stokes equation with latest computer available. In order to overcome with this problem, a time averaged Navier-Stokes equation is used i.e. Reynolds Averaged Navier-Stokes Equations (RANS) along with turbulent models to solve the issue involved in Reynolds stress due to the time averaging process. In the present work the k-ε turbulence model with non-equilibrium wall function is selected to examine the flow over the generic car model.

A. Modeling and Domain Setup

The basic model of the car is designed in the Catia V5 with and without add-on device is used for CFD analysis. The table I, shows the dimension of the basic car and the dimension of the add-on device. The car model with and without add-on device is imported to ANSYS Fluent 17.0 and the domain of the study is created using design modeler as shown in "Fig. 2,"and "Fig. 3,".

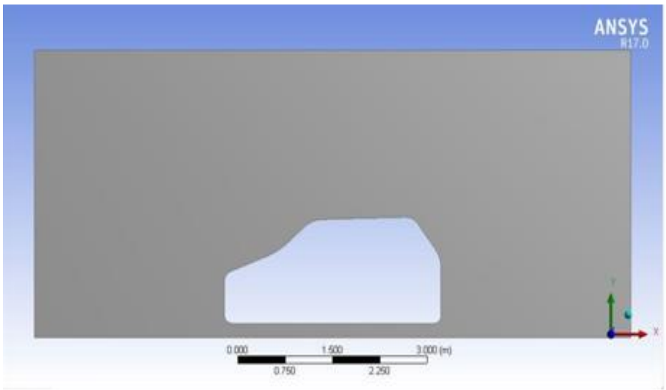

Fig.2. Domain creations of car without add-on device in design modeler

Fig.3. Domain creations of car with add-on device in design modeler

TABLE I.

S.NO	GEOMETRY DETAILS	VALUE
1.	Length of car body	3430 mm
2.	Width of car body	1490 mm
3.	Height of car body	1475 mm
4.	Length of enclosure	6430 mm
5.	Height of enclosure	4000 mm
6.	Diameter of the hemi spherical add-on device	0.468 m
7.	Conversion to 2d	Mid surf operation
8.	Creation of domain	Boolean subtract

a. Dimensions of the basic car and add-on device

B. Meshing

Meshing of the model is done using ANSYS Fluent 17.0 meshing tool. The mesh around the car model with and without add-on device is shown in “Fig. 4,”and “Fig. 5,”The mesh parameters are described in table II.

TABLE II.

S.NO	MESH DETAILS	INPUT VALUE
1.	Mesh relevance	40
2.	Mesh method	Un-structural
3.	Mesh inflation	3 Layers
4.	Name selections	Inlet, Outlet, Car body
5.	Number of nodes	2860
6.	Number of elements	5100

b. description of the mesh

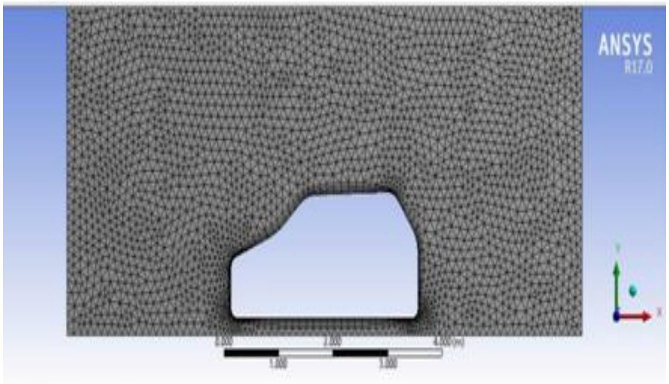

Fig.4. Meshed model of car without add-on device

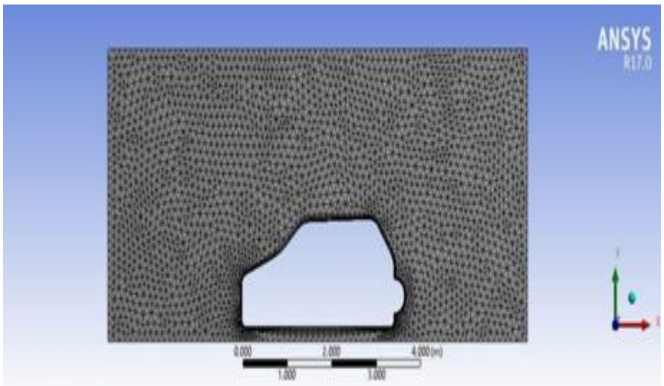

Fig.5. Meshed model of car with add-on device

C. Boundary Conditions

The domain is divided into three parts inlet, outlet and car body. In Ansys fluent setup, the inlet is taken as velocity inlet and the outlet as pressure outlet. The common boundary conditions for all the test cases are listed in table III.

TABLE III.

S. No.	PARAMETERS	VALUES
1.	Reynolds number	104 and above
2.	Density of fluid	1.225 Kg/m^3
3.	Pressure outlet	0 Pascal
4.	Courant number	2.5 & 7.5

5.	Velocity inlet	11.11 m/s 13.88 m/s 16.66 m/s 19.44 m/s 22.22 m/s 25 m/s 27.77 m/s 30 m/s 33.33 m/s
6.	Y-plus values	1 - 40

c. Boundary condition

D. *Solver Selection and Solution Setup*

The accurate solution can be achieved by using proper solver to perform simulation. Generally, pressure based solver is used for in-viscid flow whereas density based solver is used for viscous flow. The basic difference between the two solvers is how energy and momentum equation are coupled together. For example, pressure based solver solve energy and momentum equation in a decomposed manner using simple algorithms whereas density based solver solve energy and momentum equations in a coupled manner.

In the present research, pressure based solver is used because pressure based solver recently has been modified and extended to solve for viscous flow. Also, this project required surface pressure to calculate the lift and drag force acting on the models can be calculated accurately using pressure based solution. The solution setup details are listed in table IV.

TABLE IV.

S.NO	SOLUTION DETAILS	INPUT VALUE
1.	Solution launcher	Double precision
2.	Solution type	Serial solver
3.	General setup	Pressure based – Steady state analysis
4.	Model setup	Energy On
5.	Nature of flow	Inviscid
6.	Boundary conditions	Velocity inlet Pressure Outlet
7.	Solution method	Pressure-velocity coupling
8.	Solution scheme	SIMPLE
9.	Gradient discretisation	Least Square Cell Based
10.	Pressure discretisation	Second Order
11.	Momentum discretisation	Second order upwind
12.	Energy discretization	Second order upwind
13.	Solution monitors	Coefficient of drag Coefficient of lift
14.	Solution initialization	Standard initialization
15.	Number of iterations	case

d. Solution setup

III. RESULTS AND DISCUSSION

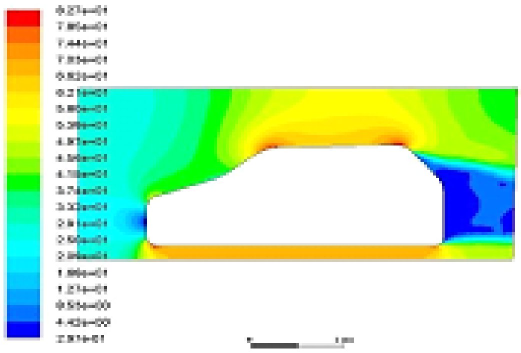

Fig.6. Velocity contour at 33.33 m/s of generic car without add-on device

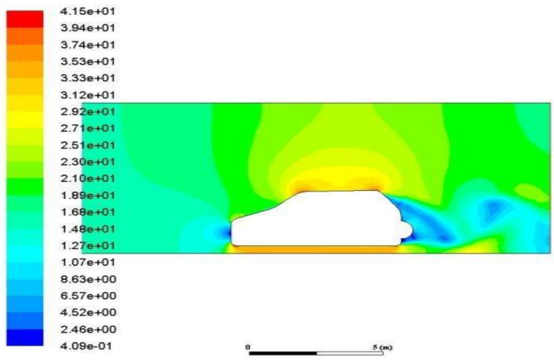

Fig.7. Velocity contour at 33.33 m/s of generic car with add-on device

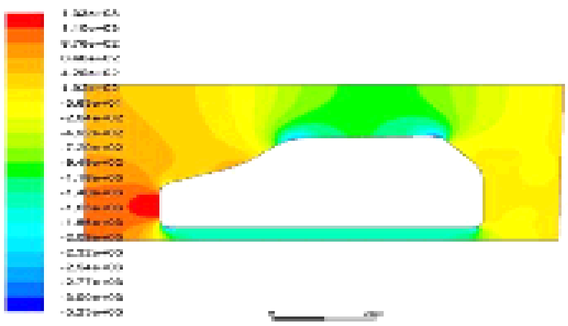

Fig.8. Pressure contour at 33.33 m/s over generic car without add-on device

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

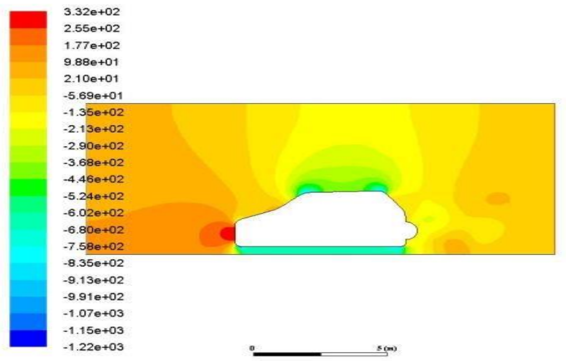

Fig.9. Pressure contour at 33.33 m/s over generic car with add-on device

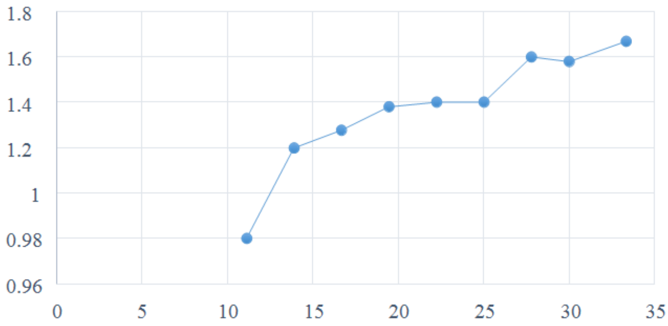

Fig.10. Speed vs drag coefficient curve for generic car without add on device

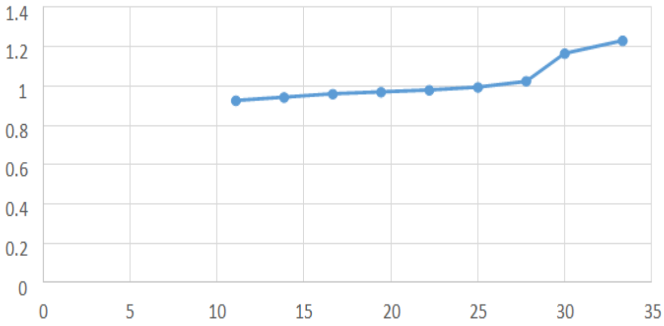

Fig.11. Speed vs drag coefficient curve for generic car with add on device

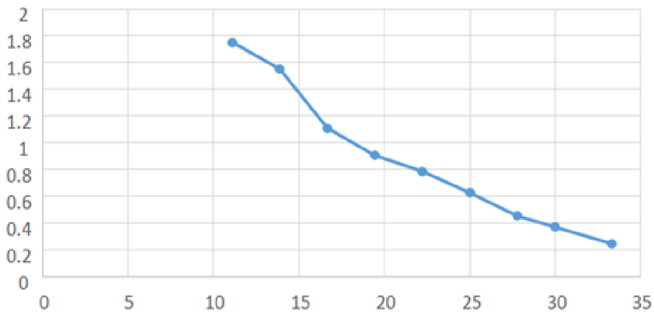

Fig.12. Speed vs lift coefficient curve for generic car without add on device.

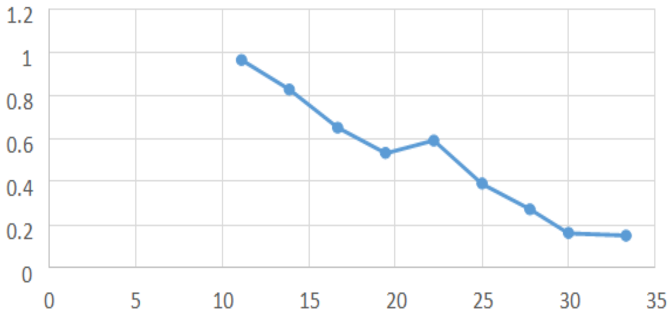

Fig.13. Speed vs lift coefficient curve for generic car with add on device

"Fig. 6," and "Fig. 7," shows the velocity contour for the generic car without and with add-on device respectively, it clearly demonstrates that the circulation zone of the air behind the generic car with add-on device is reduced significantly compared to a generic car with add-on the device and hence a significant reduction in drag coefficient for the generic car with add-on device compare to without add-on device has been achieved. From "Fig. 8," and "Fig. 9," it can be clearly observed that the low-pressure zone of the air on the upper surface of the generic car is reduced for the generic car with an add-on device compare to the generic car without an add-on device. It shows that the lift and hence lift coefficient is reduced for the generic car with an add-on device compare to generic car without an add-on device.

"Fig. 10," and "Fig. 11,"shows the curve of speed vs drag coefficient for generic car without and with add-on device respectively. From this, it can be observed that the drag coefficient of generic car with add-on device is reduced at all the speed and about 29.04% drag coefficient reduction was achieved at the speed of 33.33 m/s compared to drag coefficient of generic car without add-on device. Whereas, from "Fig. 12," and "Fig. 13," it can be clearly observed that the lift coefficient for generic car is decreased at all the speed and about 11.11% reduction was achieved at 33.33 m/s. This reduction in the lift and drag coefficient is mainly due to the reduction in low pressure zone and hence reduced formation of the vortices.

IV. CONCLUSION

Numerical simulations of the flow field around the generic car models with and without add-on device have been done and compared the results. In this analysis, the coefficient of drag of the car model with add-on device at the maximum velocity 33.33 m/s has been reduced by 29.04% compared to the car model without add-on device whereas the lift coefficient is decreased by 11.11%. Therefore, the add–on device is an effective tool to reduce the coefficient of the drag of the generic car model and hence improve the fuel economy and stability of the generic car model.

ACKNOWLEDGMENT

We thank the writers whose research papers we referred and developed the basics for our research work, which we were able to utilize and deliver good results according to the requirement of Indian society. My mentor and friend, Dr. A

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 9 Issue 10, October-2020

Sankaran, I am grateful for his motivation and helpful feedback.

REFERENCES

[1] V. Naveen Kumar, K. Lalit Narayan and L.N.V.Narasimha Rao, “Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler using CFD” International Journal of Advances in Scientific Research 2015, 1(08): 331-339.

[2] S.M. Rakibul Hassan, Toukir Islam, “Numerical Study on Aerodynamic Drag Reduction of Racing Cars”, science direct published on ICME 2013, Procedia Engineering 90 (2014) 308 – 313.

[3] C.V.Karthick Bala Murugan, P.A.Nigal Ashik and P.Raju,”CFD Analysis and Optimization of a Car Spoiler”, International Journal of Mechanical Engineering and Research, ISSN 0973-4562 Vol. 5 No.1 2015 http://www.ripublication.com/ijmer.htm

[4] T. D. Ipilakyaa, L. T. Tuleun and M. O. Kekung, “Computational fluid dynamics modelling of an aerodynamic rear spoiler on cars” Nigerian Journal of Technology (NIJOTECH) Vol. 37, No. 4, October 2018, pp. 975 – 980 Copyright© Faculty of Engineering, University of Nigeria, Nsukka, Print ISSN: 0331-8443, Electronic ISSN: 2467-8821 http://dx.doi.org/10.4314/njt.v37i4.17

[5] Shyam P. Kodali and Srinivas Bezavada, “Numerical simulation of air flow over a passenger car and the Influence of rear spoiler using CFD”*International Journal of Advanced Transport Phenomena Vol. 01, No. 01, December 2012,* https://www.researchgate.net/publication/336589740

[6] Rubel Chandra Dasa and Mahmud Riyada, “ CFD Analysis of Passenger Vehicleat Various Angle of Rear End Spoiler”, 10th International Conference on Marine Technology, MARTEC 2016, Procedia Engineering 194 (2017) 160 – 165. www.sciencedirect.com

[7] See-Yuan Cheng , Kwang-Yhee Chin , Shuhaimi Mansor , Abd Basid Abd Rahman, “Experimental study of yaw angle effect on the aerodynamic characteristics of a road vehicle fitted with a rear spoiler” Journal of Wind Engineering & Industrial Aerodynamics 184 (2019) 305–312.www.elsevier.com/locate/jweia

[8] Sneh Hetawal, Mandar Gophane, Ajay B.K, Yagnavalkya Mukkamala, “Aerodynamic Study of Formula SAE Car” 12th Global Congress On Manufacturing And Management, Gcmm 2014, Procedia Engineering 97 (2014) 1198 – 1207.

[9] Krzysztof Kurec , Michal Remer , Tobiasz Mayer ,Sylwester Tudruj , Janusz Piechna, “Flow control for a car-mounted rear wing” International Journal of Mechanical Sciences (2018), https://doi.org/10.1016/j.ijmecsci.2018.12.034