Design and Analysis of rear car spoiler for Drag reduction and fuel efficiency.

Chaitanya Bari
Student, Mechanical Engineering
Saraswati College of Engineering,
Kharghar, Navi Mumbai, India
chaitanyabari1@gmail.com

Kalpesh Bhujbal
Student, Mechanical Engineering
Saraswati College of Engineering,
Kharghar, Navi Mumbai, India
kalpesh18.bhujbal@gmail.com

Shubham Bhalerao
student, Mechanical Engineering
Saraswati College of Engineering,
Kharghar, Navi Mumbai, India
shubhambhalerao887@gmail.com

***Abstract* — The body form and size of the automotive should have allowable aerodynamic attribute. The aerodynamic aspects are going costlier however lowering aerodynamic drag the economy of quickest automotive is improved. Aerodynamic characteristics of race car are of great interest in drag reduction and increasing fuel potency. At the current stage changed automobile is additional widespread for everybody. At the current most of the sports automobile are seen with a spoiler. Even they need a additional firm chassis and a stiff suspension system for top speed. At this stage spoiler is additional benefitted.**

This document offers an outline of a simple and a reliable way to do simulations within the automotive field. Several automobile have a firmly sharp downward angle going from rear fringe of the roof all the way down to the trunk/tail of the automobile. It's useful for reducing flow separation decreasing drag and most significant is reducing fuel consumption. It additionally helps to stay the rear window accessible as a result of the air is flow swimmingly through rear window. In this thesis will concentrate on CFD-based drag prediction on the sedan kind automobile body when the spoiler is mounted at the rear fringe of the vehicle. An automobile rear spoiler is design and model in 3D modeled software system solidworks by completely different models of spoilers. And also the analysis is finished by ANSYS software system. We tend to ansys on completely different speeds i.e. (30m/s, 40m/s).we additionally ansys 2 completely different bodies i.e. without spoiler and with spoiler. This offer US the variation between the drag force and also the potency of the spoiler within the use of cars. In this thesis the CFD analysis to see the drag force, coefficient of drag, velocity, pressure

This thesis can present a numerical simulation of flow around race car with spoiler situated at the rear end using commercial ansys fluent software system. The thesis can concentrate on CFD-based raise and drag prediction on the automobile body when the spoiler is mounted at the rear fringe of the vehicle. A 3D model of a 4-door sedan automobile (which going to be designed with business software system SolidWorks) going to use as base model. Various spoilers are going to situated at rear of car and also the simulation are going to run so as to see the aerodynamic results of spoiler.

Keywords'- drag reduction, fuel potency, aerodynamic effects, spoilers, cfd.

1. INTRODUCTION

Active aerodynamics could be a combination of a varied stage adjustable rear spoiler. The spoiler lip is protected. Rear spoiler is partly extended, this is often make sure you to a directional level of stability. There area unit are forces working on automobile. First is air drag force appearing in an exceedingly direction of a vehicle motion. Second is the aerodynamic force working on vertically upward, Third is that the cross wind force acting in lateral direction. The body form and size of the automobile should have acceptable aerodynamic attribute. The aerodynamic aspects are going to costlier however lowering aerodynamic drag the economy of quickest automobile will be improved. Aerodynamic characteristics of a automobile area unit of serious interest in drag reduction and increasing fuel potency. At this stage changed automobile is additional widespread for everybody. At this most of the sports automotive area unit seen with a spoiler. Even they need a additional firm chassis and stiff mechanical system for prime speed. At this stage spoiler is additional benefitted.

Aerodynamic analysis of air flow above object will performed using precise technique or CFD approach. On one hand, Analytical technique of resolution air flow above an object will be done just for easy flows over easy geometries like streamline flow over a horizontal plate. If air flow gets advanced as in flow over show body, the flows became turbulent and it's not possible to unravel Navier- Stoke and continuity scientific equations. On the opposite hand, getting direct numerical resolution of Navier-stoke equation isn't nonetheless potential even with modern-day computers. So as to return up with cheap resolution, a time standard Navier-Stoke equation is getting used (Reynolds Averaged Navier-Stoke Equations – RANS equations) at the side of turbulent models resolve difficulty involving Reynolds Stress result from the time averaged method.

In present work of the k-e turbulence equation model with non-equilibrium wall perform is chosen to research flow over generic coach model. The model is incredibly strong, having cheap process work time, and wide utilized by the automobile industries.

2. METHODOLOGY AND APPROACH OF WORK

In this initial of all generic model of car is prepared within the SOLIDWORKS software system, this generic model is imported in the ansys software system to try and do the ansys of coefficient of the drag within the wind tunnel that is generated within the style module of ANSYS software system. Once the meshing is generated on the above the car.

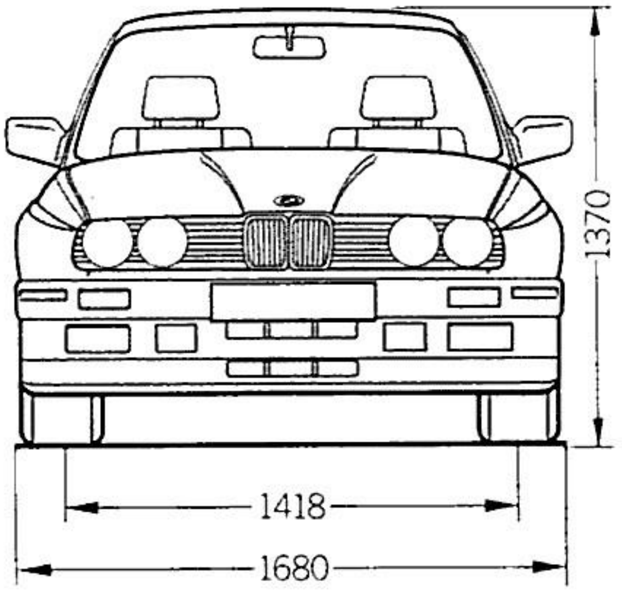


Figure-1 Dimensions of the generic vehicle model [Front view]

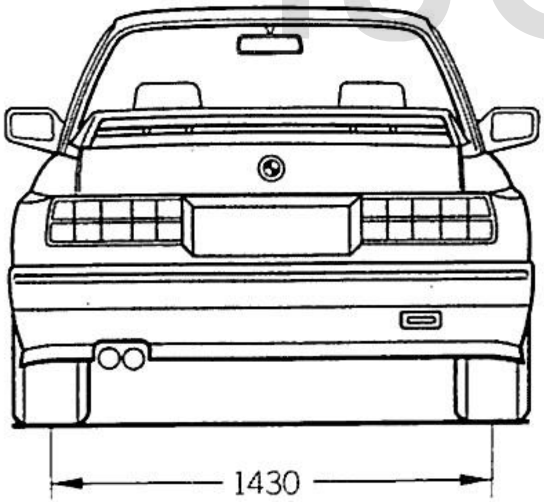


Figure-2 Dimensions of the generic vehicle model [Back view]

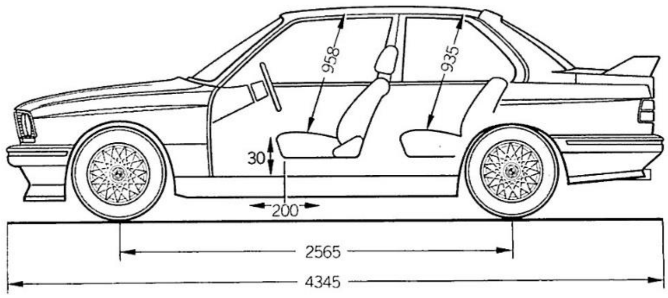


Figure-3 Dimensions of the generic vehicle model [Side view]

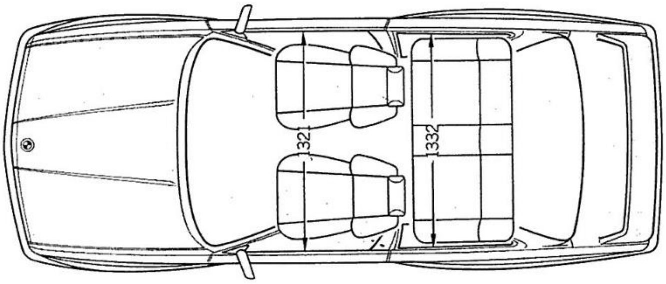


Figure-4 Dimensions of the generic vehicle model [Top view]

3. DIFFERENT CAD MODELS

The models of each vehicle and two totally different spoilers are 3D printed exploitation the software package known as SolidWorks to CAD format for numerical analysis. Then after, this model been analysed for the coefficient of drag & forces beneath the Ansys Fluent style module & values of coefficient of drag.

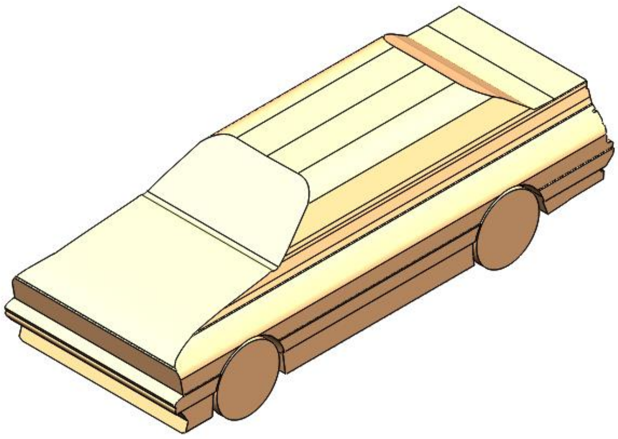


Figure-5 Sedan type of car without spoiler

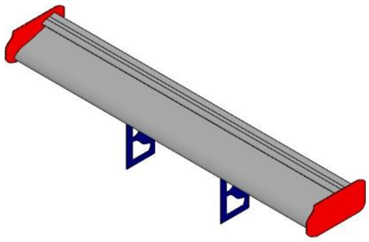


Figure-6 Spoiler

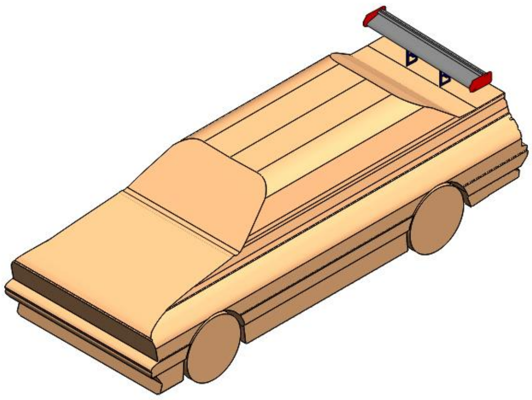


Figure-7 Sedan type of car with spoiler

4. MESHING

The triangular form surface mesh was used because of its proximity to ever-changing curves and bends. These components simply go with the advanced bodies employed in automobile and part bodies. With the default settings for mesh generation, ANSYS Meshing has generated the meshes as

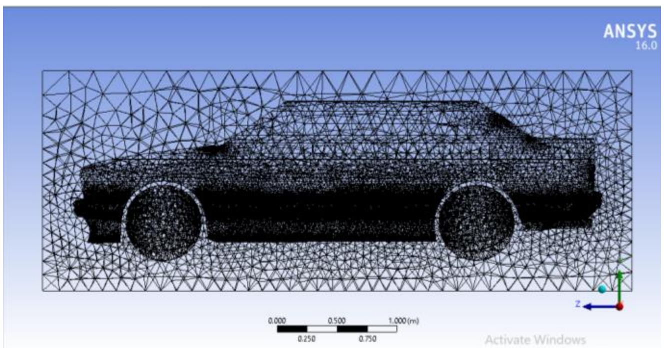


Figure-8 meshing of car without spoiler

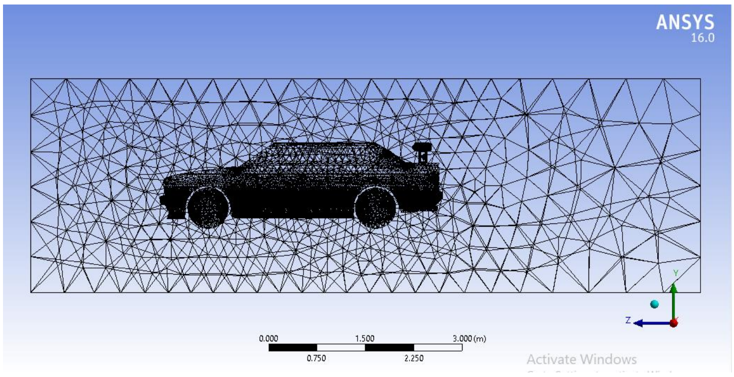


Figure-9 Meshing of car with spoiler

5. SIMULATION RESULTS

A 3D steady state, incompressible answer of the Navier-Stoke equations was performed using ANSYS FLUENT style computer code. Turbulence modeling was through with the realizable k-ε model using non- equilibrium wall functions. The process results for the subsequent case was conferred and discussed:

Case 1- Vehicle without rear spoiler

Every results for various cases were achieved with a similar meshing resolution, a similar $k - \varepsilon$ turbulence models, and conjointly a similar boundary limits. The free stream speed was set to 30m/s. For the primary fifty iterations, a primary series upwind discretization was accustomed accelerating the convergence then when fifty iterations 2nd series upwind scheme has been applied and iterations have continuing till it reached to the convergence criteria.

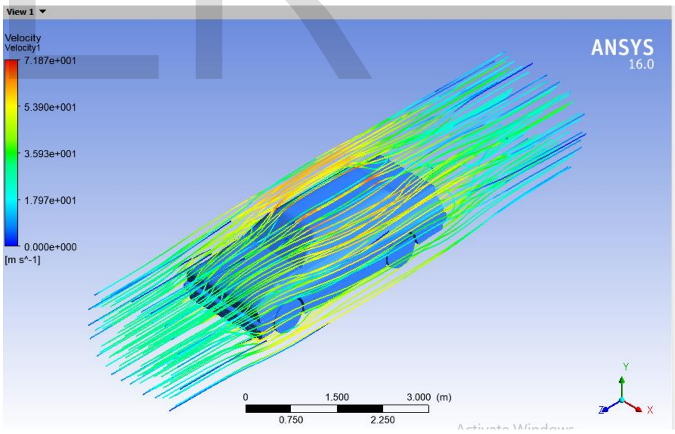
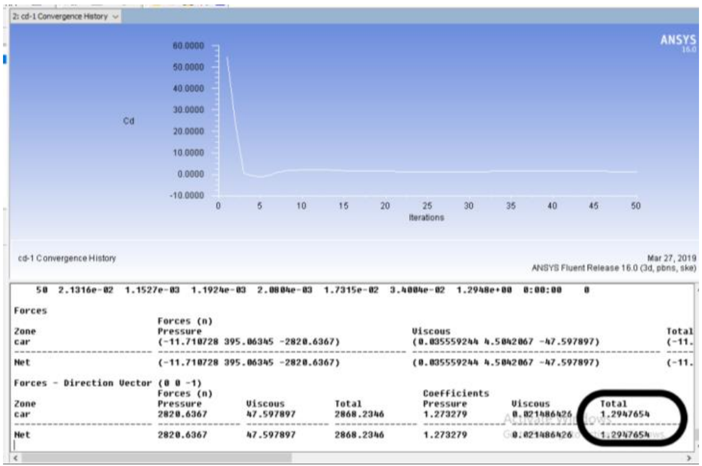
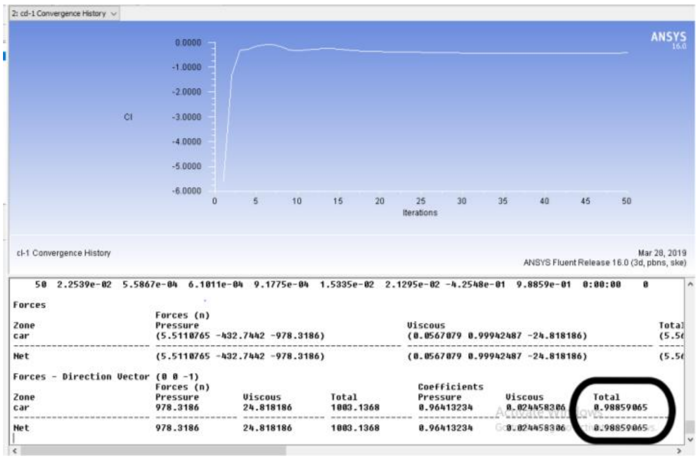


Figure-10 Drag coefficient (CD) convergence history of case 1



Graph-1 Drag coefficient (CD) iteration convergence history of case 1



Graph-2 Drag coefficient (CD) iteration convergence history of case 2

Case 2- Vehicle model with rear spoiler

Every results for various cases were achieved with a similar meshing resolution, a similar $k - \varepsilon$ turbulence models, and conjointly a similar boundary limits. The free stream speed was set to 30m/s. For the primary fifty iterations, a primary series upwind discretization was accustomed accelerating the convergence then when fifty iterations 2nd series upwind scheme has been applied and iterations have continuing till it reached to the convergence criteria.

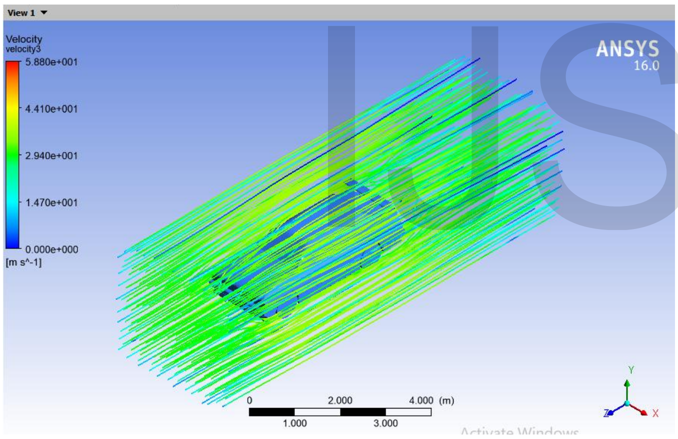


Figure-11 Drag coefficient (CD) convergence history of case 2

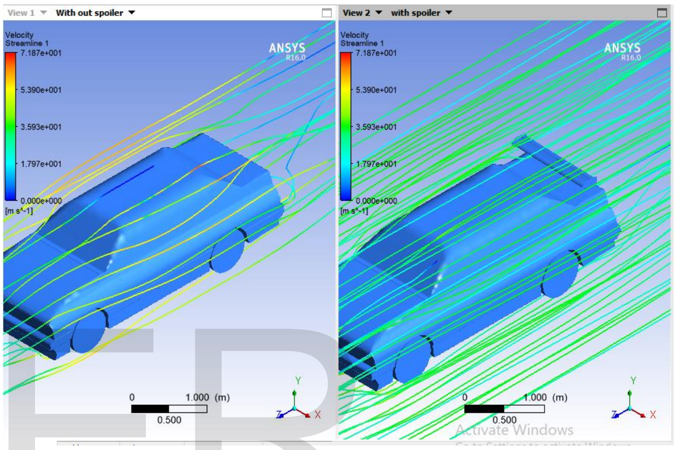


Figure-12 Drag coefficient (CD) comparison between case 1 and case 2

6. CONCLUSION

Modeling of sedan kind of automobile is completed using Solidworks. Analysis of effect of the rear spoiler in the automobile is completed using ANSYS FLUENT style software system. The Analysis and Modeling of automobile without the rear spoiler and therefore the CD (Coefficient of drag=1.294) was solved. The Analysis and Modeling of automobile with a rear spoiler and also the CD (Coefficient of drag=0.988) was calculated.

REFERENCES

[1] G.Ganesh, V.Vasudevan, Analysis of Effects of Rear Spoiler in Automobile Using Ansys International Journal of Scientific & Engineering Research, Volume 6, Issue 6, [June-2015] 762-766.

[2] Padmagari Naresh,N.Sreenivas, Design and Analysis of a Car Rear Spoiler for Drag Reduction, International Journal of Research Volume 04, Issue 13, [October 2017] 2374-2379.

[3] Mustafa Cakir, CFD study on aerodynamic effects of a rear wing/ spoiler on a passenger vehicle, Santa Clara University Scholar Commons
Mechanical Engineering Masters Theses (2012) 1–62.

[4] A.Anish, Suthen.P.G, Viju.M.K, Modelling And Analysis Of A Car For Reducing Aerodynamic Forces, International Journal Of Enginnering Trend And Tecnology, volume 47,number 1,[May-2017] 1-16

[5] Kelbessa Kenea Deressa, Kiran Kumar Sureddy, Design And Analysis Of A New Rear Spoiler For Su Vehicle Mahindra Bolero Using Cfd, International Research Journal of Engineering and Technology, Volume 03,Issue 06,[June-2016] 914-924.