International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 12 | Dec 2018 www.irjet.net p-ISSN: 2395-0072

Fluid Dynamics Simulation of a Car Spoiler for Drag Reduction and to Increase Downforce

AKSHAY BODAKE[1], SANKET CHARKE[2]

[1]*Department of Mechanical Engineering, Sinhgad Academy of Engineering, Pune, Maharashtra, Pune, India*
[2]*Department of Mechanical Engineering, G. S. Moze College of Engineering, Pune, Maharashtra, India*

Abstract - *This paper contains the simulation analysis of car spoiler by using the ANSYS Fluent software. For designing of spoiler we have used NACA 4 digit airfoils. We have done analysis of different 4 digit NACA series airfoils at different angle of attack and from that we took most efficient airfoil for design of spoiler. Then we took the different heights of spoiler and did the analysis and chose the best height for spoiler.*

Key Words - *Aerodynamic Drag, Coefficient of drag, lift, Flow of fluid, CFD simulation, wind tunnel test.*

1. INTRODUCTION

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. Spoilers on the front of a vehicle are often called air dams. Spoilers in racing are used in combination with other features on the body or chassis of race cars to change the handling characteristics that are affected by the air of the environment. The goal of many spoilers used in passenger vehicles is to reduce drag and increase fuel efficiency.[8] Passenger vehicles can be equipped with front and rear spoilers. Front spoilers, found beneath the bumper, are mainly used to decrease the amount of air going underneath the vehicle to reduce the drag coefficient and lift.

An active spoiler is one which dynamically adjusts while the vehicle is in operation based on conditions presented, changing the spoiling effect, intensity or other performance attribute. Found most often on sports cars and other passenger cars, the most common form is a rear spoiler which retracts and hides partially or entirely into the rear of the vehicle, then extends upwards when the vehicle exceeds a specific speed.

2. AERODYNAMICS

Aerodynamics, is the study of the motion of air, particularly its interaction with a solid object, such as an airplane wing, car spoiler. Understanding the motion of air around an object (often called a flow field) enables the calculation of forces and moments acting on the object. In many aerodynamics problems, the forces of interest are the fundamental forces of flight: lift, drag, thrust, and weight. Of these, lift and drag are aerodynamic forces.

Automobile started using aerodynamic body shapes in the early part of their history. As engines became more powerful and cars became faster, automobile engineers realized that wind resistance significantly hindered their speed.

2.1 Automotive Aerodynamics

Automotive aerodynamics is studied using both computer modelling and wind tunnel testing. For the most accurate results from a wind tunnel test, the tunnel is sometimes equipped with a rolling road. This is a movable floor for the working section, which moves at the same speed as the air flow. This prevents a boundary layer from forming on the floor of the working section and affecting the results.

Automotive aerodynamic is much different from the aircraft aerodynamic such as road vehicle shapes is bluff, vehicle operates very close to the ground, Operating speed lower, the ground vehicle has fewer degrees of freedom and its motion is less affected by aerodynamic forces.

Total Aerodynamic drag = C_D multiplied by the frontal area. The width and height of curvy cars lead to gross overestimations of frontal area.

2.2 Aerodynamic Principles

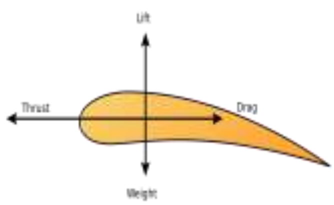

Fig 1. *Basic Terminology*

2.2.1. Drag

Drag is a force that tries to slow down car. It makes it hard for an car to move. It is harder to walk or run through water than through air. That is because water causes more drag than air. The shape of an object also changes the amount of drag. Most round surfaces have less drag than flat ones. Narrow surfaces usually have less drag than wide ones. The more air that hits a surface, the more drag it makes.

Drag force is proportional to the velocity for a laminar flow and the squared velocity for a turbulent flow.

$$D = \frac{1}{2}\rho V^2 S C_D$$

Where D = Drag Force

ρ = Density of free air

V = Velocity of free air

S = Cross sectional area or drag area

C_D = Coefficient of Drag

The above formula shows that drag is directly proportional to the square of the velocity. The drag coefficient depends on the shape of the object and on the Reynolds number:

$$\mathrm{Re} = \frac{\rho u L}{\mu} = \frac{uL}{\nu}$$

Where Re = Reynolds number

u = Relative speed of the object

L = Cross sectional area, and

v = Kinematic viscosity of the fluid

The resistance experienced by a spoiler in car is a function of the Reynolds Number. Normally, the Reynolds Number is the decisive factor in the air-flow in determining whether the inertial effect or the viscous effect wins. If the Reynolds Number is large, the viscosity effect is small. For the for us practical values the inertia or density forces dominate, and the parasite drag increases with the square of the velocity. A low Reynolds Number gives laminar flow while a high Reynolds Number gives turbulent flow. For both a laminar and a turbulent boundary layer increasing Reynolds Number gives lower skin friction drag. However, because of the higher energy loss in the boundary layer, a turbulent layer always has higher skin friction drag.

2.2.2. Lift/Downforce

Lift is the push that lets something move up. It is the force that is the opposite of weight. Everything that moves in air must have lift. For an car to move upward, it must have more lift than weight. A hot air balloon has lift because the hot air inside is lighter than the air around it. Hot air rises and carries the balloon with it. A helicopter's lift comes from the rotor blades at the top of the helicopter. Their motion through the air moves the helicopter upward. Lift for an airplane comes from its wings.

3. METHODOLOGY

The geometry and physical bounds of the problem can be defined using computer aided design (CAD).

The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform, structured or unstructured, consisting of a combination of hexahedral, tetrahedral, prismatic, pyramidal or polyhedral elements.

The physical modeling is defined.

Boundary conditions are defined. This involves specifying the fluid behaviour and properties at all bounding surfaces of the fluid domain. For transient problems, the initial conditions are also defined.

The simulation will start and the equations will solve iteratively as a steady-state or transient.

3.1 Wind Tunnel Testing

Fig 2. *Wind Tunnel Test Set-up*

Wind tunnel testing has the big advantage that once the vehicle model is produced and rigged in the wind tunnel test section, it can quickly provide highly accurate data. If similar changes in conditions are done on a computer model, the whole simulation has to be run over again for each case. On the other hand, wind tunnel testing can be both highly costly and time consuming. The wind tunnel itself is a huge investment, and the production of prototypes can be very expensive. Small changes in design will take much more time to implement on a physical prototype than on a computer model.

3.2 Computational Fluid Dynamics

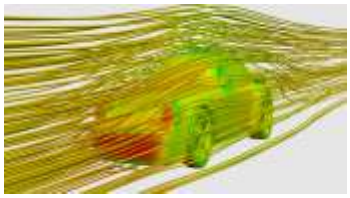

Fig 3. *CFD analysis of car*

According to Oleg Zikanov [3] CFD can be defined as: "CFD (Computational fluid dynamics) is a set of numerical methods applied to obtain approximate solution of problems of fluid dynamics and heat transfer."

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, weather simulation, natural science and environmental engineering, industrial system design and analysis, biological engineering and fluid flows, and engine and combustion analysis.

4. MODELLING

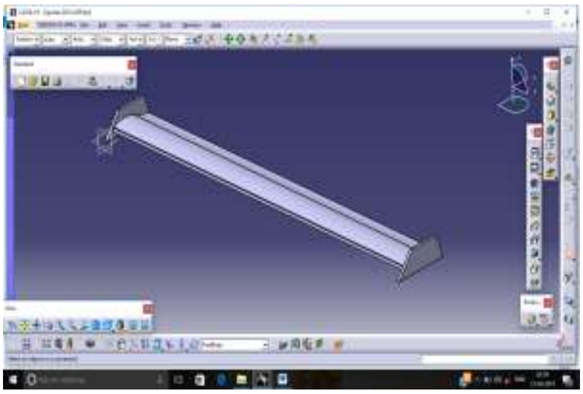

Fig 4. *Design of Spoiler in SOLIDWORK*

After all the measurements of the spoiler the next step was to create the CAD model suitable for analysis.The modelling was done in CAD environment of SOLIDWORKS.

The spoiler to be completely customised. Initially a basic model of spoiler was generated considering the various components to be assembled over it. After the complete generation of the model structural analysis was carried out to determine the weak points in the design of the spoiler. Then the design of the spoiler was accordingly modified and improved.

5. RESULTS

CASE 1

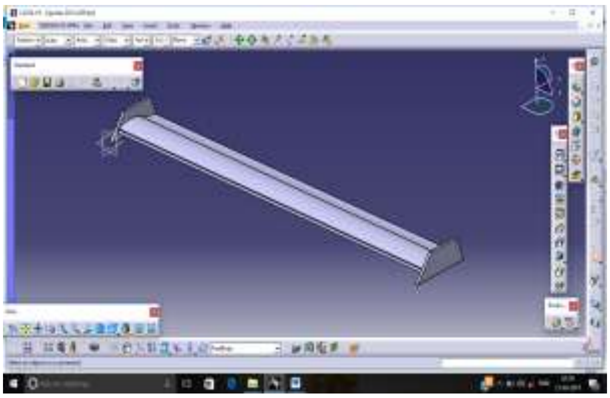

Fig 5. *Design Of Spoiler For Case 1*

This is the cad model of pedestal spoiler. It is made of aerofoil shape which is generally used for design of aeroplane wing. In this case the spoiler frontal face is upside than the rear tail. which means the angle of attack of this shape is negative. This shape gives the result which is negative for automobile aerodynamics. The analysis of above spoiler gives following results.

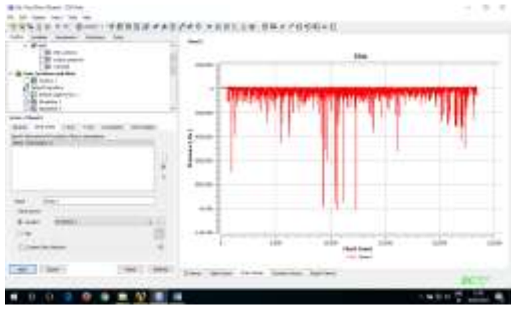

Fig 6. *Pressure Plot*

As we can see in the above pressure plot partial vacuum is created on the below side of the spoiler but on topside also low pressure area is introduced which is undesirable. This chart is plot pressure on the y axis and chart count the x axis which gives maximum vacuum upto $-1*10^6$

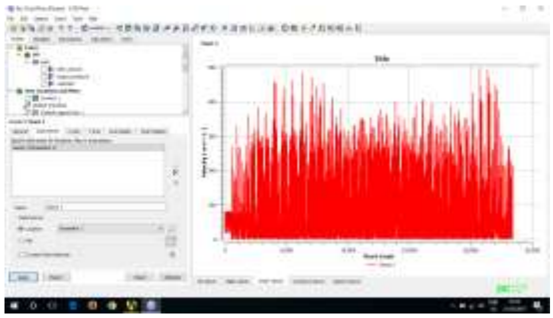

Fig 7. *Velocity Plot*

The figure shows the velocity plot in which velocity is plotted on y axis and chart count on the x axis .we can see the distribution of the velocity over the range in this maximum air flow at constant speed which ranges 0 to 200 m/s .some of the stream fluctuates which causes changes in velocity which we can in the above chart.

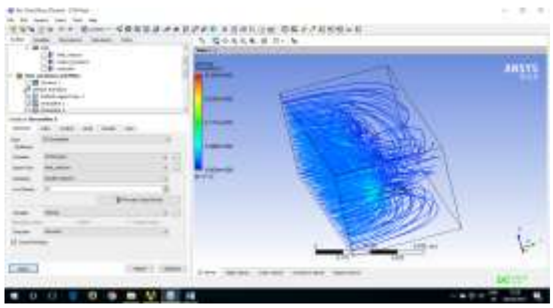

Fig 8. *Partical Path Track*

This is velocity 3D streamline path. In which the line shows the particle flow path on which particle flow form inlet side to outlet side. On the left side of the figure there is a velocity standard bar in which we can see the ranges of the velocity of the particle. The figure shows blue color on the outer side and the light blue color near the spoiler it means velocity is higher near the spoiler. The flow of particle passes over the spoiler body then get mixed afte rpassing it. It creates turbulence on the rear side of the spoiler which undesirable case for the automobile aerodyanamics.

CASE 2

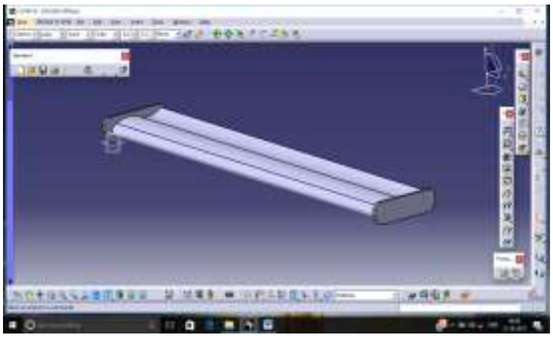

Fig 9. *Design Of Spoiler*

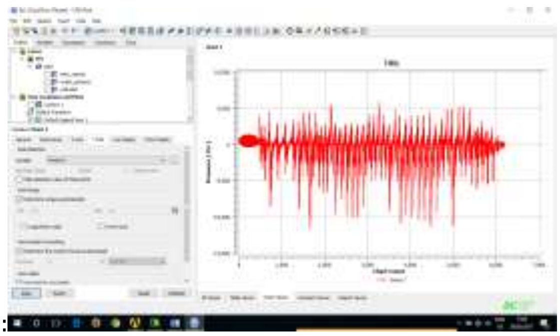

Fig 10. *Pressure Plot*

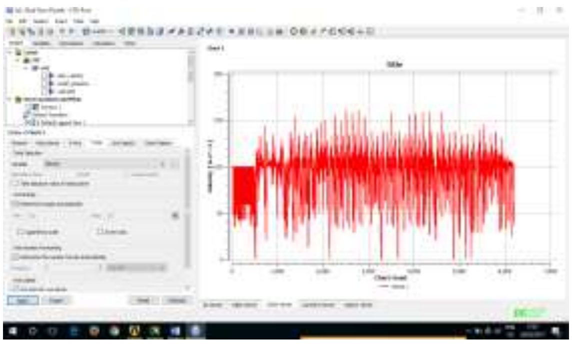

Fig 11. *Velocity Plot*

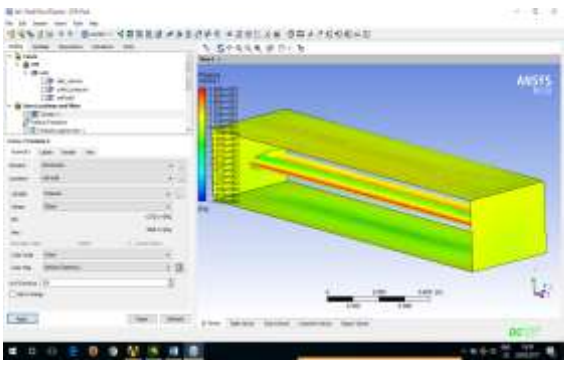

Fig 12. *Pressure Distribution*

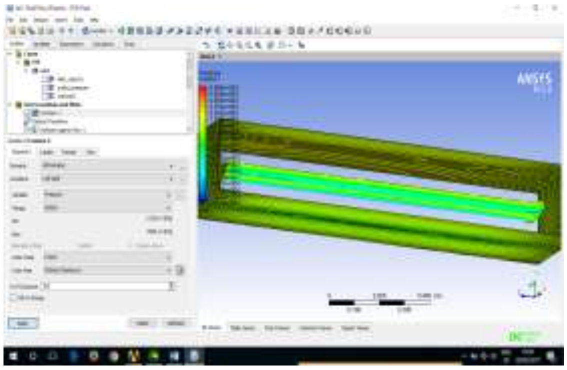

Fig 13. *Pressure Distribution*

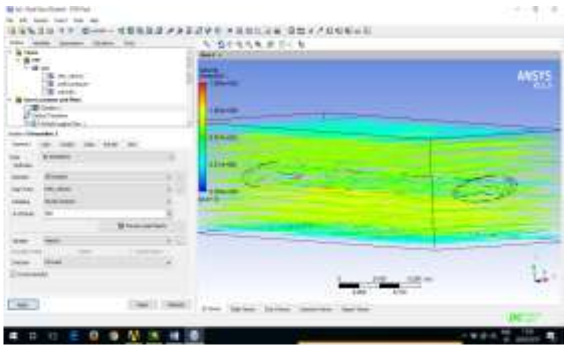

Fig 14. *Partical Path Track*

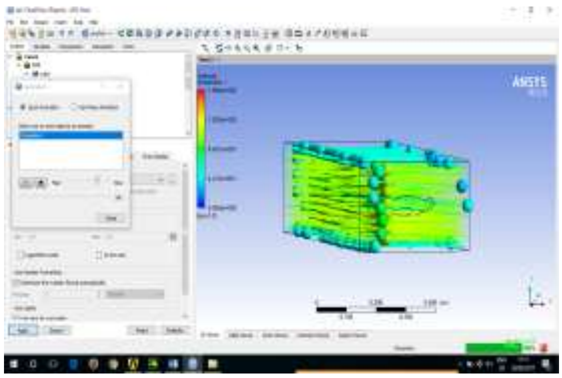

Fig 15. *Partical Path Track*

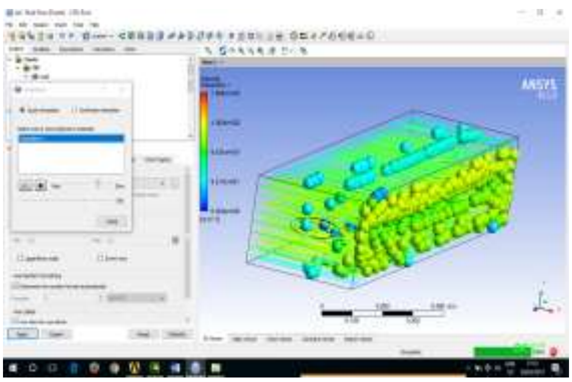

Fig 16. *Partical Path Track*

CASE 3

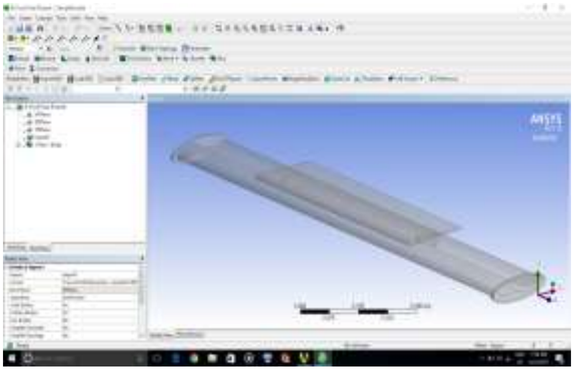

Fig 17. *Design of Solar Spoiler*

Fig 18. *Pressure Plot For Solar Spoiler*

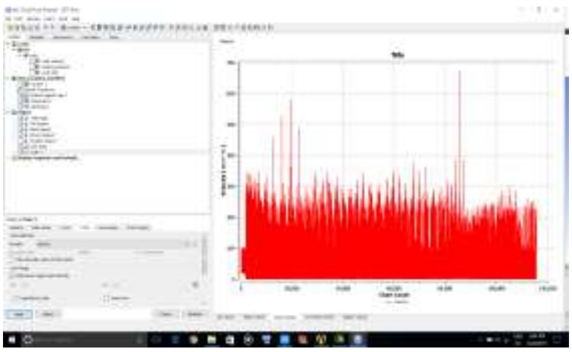

Fig 19. *Velocity Plot For Solar Spoiler*

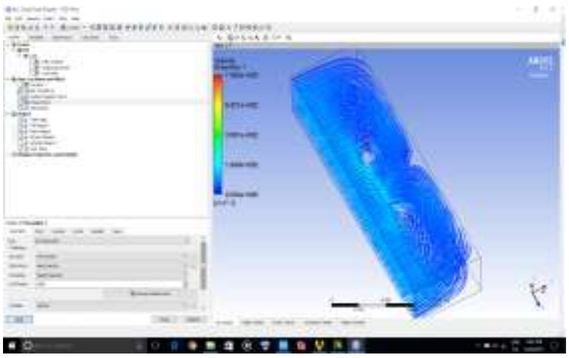

Fig 20. *Partical Path Track*

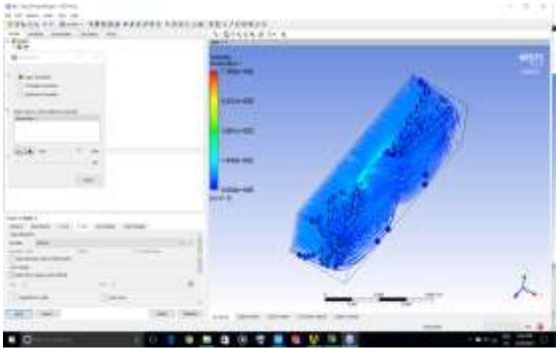

Fig 21. *Partical Path Track*

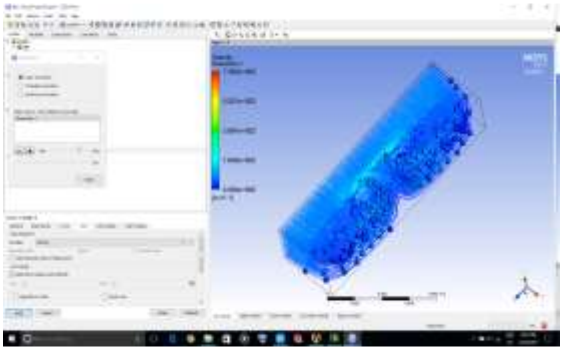

Fig 22. *Partical Path Track*

CASE 4

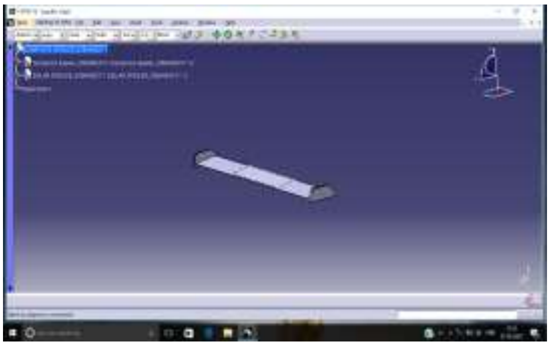

Fig 23. *Design of Solar Spoiler*

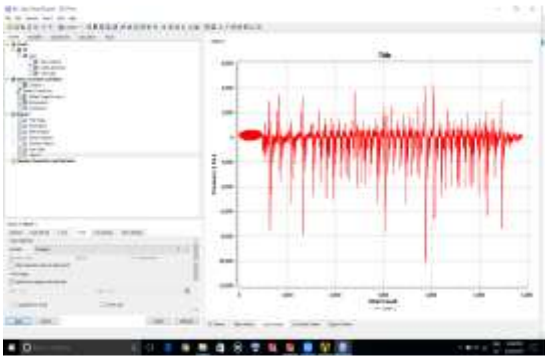

Fig 24. *Pressure Plot*

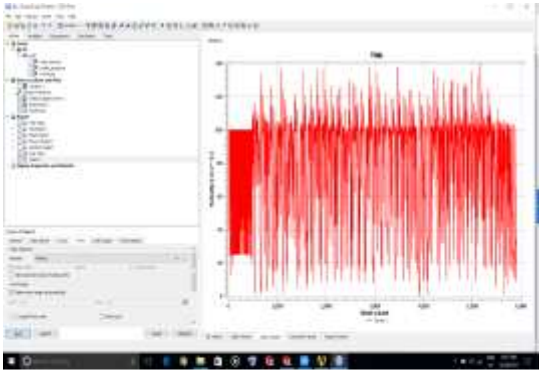

Fig 25. *Velocity Plot*

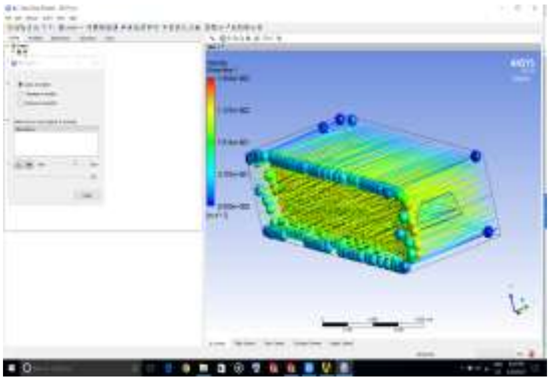

Fig 26*. Particle Path Track*

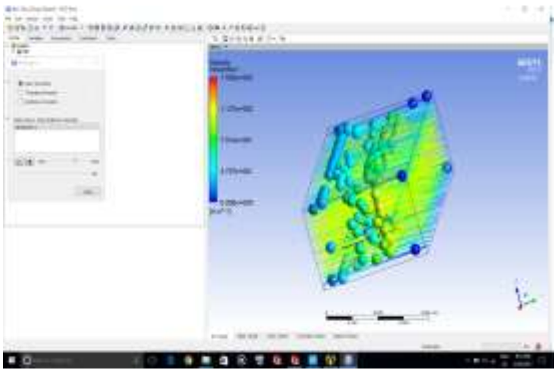

Fig 27. *Particle Path Track*

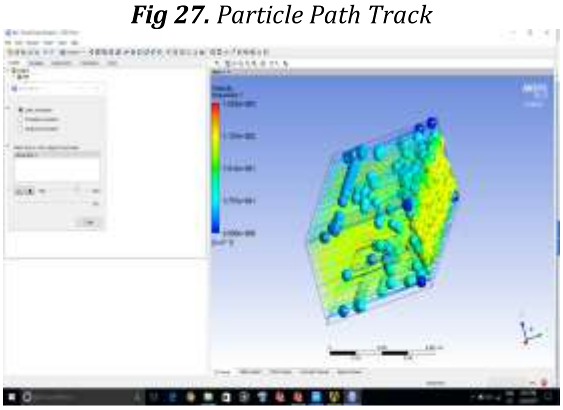

Fig 28. *Partical Path Track*

Fig 29. *Coefficient Of Drag Plot*

6. CONCLUSIONS

By using different methods aerodynamics of the car body can be improved by a greater extent, which in turn increases the performance of the vehicle. Most aerodynamic design applied with streamlining, rear-fairing, twin venturi diffusers and multi-element spoilers. It has downforce coefficient of CL= -3.0 and a drag coefficient CD= 0.75 which were determined to be the most efficient values for a race car.

In conclusion, we can say that NACA Airfoil at 10°Angle of attack, with 5 mm Height of Spoiler at rear side of car's trunk will definitely become effective for increasing Negative lift force and reducing drag force.

REFERENCE

1. M. Rouméas, P. Gilliéron & A. Kourta **"Drag Reduction by Flow Separation Control on a Car After Body"**, Int. J. Numer. Methods Fluids 60, 2008 . ISSN 0271-2091

2. Klaus Gersten, E. Krause, H. Jr. Oertel, C. Mayes "Boundary-Layer Theory", Herrmann Schlichting, 8th Edition, Springer 2004

3. Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu, "Computational Fluid

4. Dynamics: A Practical Approach", Butterworth-Heinemann; 1st edition, Burlington, MA, November 2007.

5. International Journal of Mechanical Engineering and Research, ISSN 0973-4562 Vol. 5 No.1 (2015) © Research India Publications; http://www.ripublication.com/ijmer.htm

6. "CFD Analysis and Optimization of a Car Spoiler", C.V.Karthick Bala Murugan1, P.A.Nigal Ashik, P.Raju,Assistant Professor.

7. Fundamentals of Computational Fluid Dynamics, Hardward Lomax and Thomas H. Pulliam, NASA Ames

Research Center, David W. Zingg, University of Toronto Institute For Aerospace Stuies, December 6, 1999.
8. "Why a Spoiler for Your Car?: Fuel Economy, Styling, Value Enhancement". Cardata.com. Archived from the original on 2011-10-30. Retrieved 2011-09-28.

AUTHOR'S BIOGRAPHIES

Mr. Akshay Bodake received his graduate degree B.E. from Sinhgad Academy of Engineering, Pune. Currently he aimed to pursue Post graduation in automotive field for German university.

Mr. Sanket Charke has completed his B.E. in Mechanical engineering from G.S. Moze Collage of Engineering, Pune. Presently he is working as Senior Engineer in Tech-Mahindra company.