INTERNATIONAL RESEARCH JOURNAL OF AUTOMOTIVE TECHNOLOGY (IRJAT)

http://www.mapletreejournals.com/index.php/IRJAT
Received 22 June 2018 ISSN 2581-5865
Accepted 29 July 2018 2018; 1(4);41-50
Published online 30 July 2018

CFD Analysis and Drag Reduction in Maruti Suzuki Swift

K.Balamanikanda Suthan[1*], S.Mugilan[2], S.Arvind Rishi[2], K.Akash[2],G. Hithesh Kumar[2]

[1]*Assitant Professor, Department of Mechanical Engineering, Velammal Institute of Technology, Chennai-601204.*

[2]*UG Students, Department of Mechanical Engg, Velammal Institute of Technology, Chennai-601204, India*

**Corresponding author E-Mail ID: suthanapatvit@gmail.com, Mobile: 9786280979*

ABSTRACT

Today's most hot selling cars on the market are hatchbacks. Thus people have more expectations at these mid-ranged budget cars. These expectations could be achieved by increasing the car performance such as fuel economy, aesthetics and speed. Fuel economy and speed could be easily achieved by decreasing the drag coefficient and drag forces. Drag plays one of the major roles in fuel economy and speed. These two, form and skin friction drags decides the speed and efficiency. Thus altering the car design would be helpful in achieving this. We've chosen Maruti Suzuki Swift 2005 model since it has been the hot selling used car in the Indian market. On further improving the design, it may achieve a special place in middle class people's heart. The model of the car is developed in Solid Works Software and analysed in ANSYS Fluent by Computational Fluid Dynamics (CFD Analysis). Modifications such as Spoilers, Vortex Generators and Draft tubes would reduce the drag coefficient in the car.

Keywords: Drag coefficient, Form Drag, Skin friction Drag, Vortex Generator, Spoilers

1. INTRODUCTION

The outcome of this journal is to reduce the drag coefficient in Maruti Suzuki Swift car, because the drag is one of the important factors that affect the performance of a car. Aerodynamics is the property of moving air and the interaction between the air and solid bodies moving through it. It is a subfield of fluid dynamics and gas dynamics. The two primary forces in aerodynamics are lift and drag. Road vehicles are analyzed for the aerodynamics which is called automotive aerodynamics which mainly relies on drag reduction, reduction of noise caused by the air flow and to prevent the unwanted lift forces which cause imbalance in moving vehicles. Aerodynamic evaluation of air flow over an object can be performed using analytical method or CFD approach. On one hand the analytical method of solving air flow over an object can be done only for simple flows over simple geometries like laminar flow over a flat plate. If air flow gets little complicated over a bluff body, the flow becomes turbulent .CFD has also become he one of the three basic methods to solve problems in fluid dynamics and heat transfer .In order to improve the drag/down force characteristics of the vehicle, the geometry was modified and a rear diffuser and vortex generator was added at the rear. By adding the vortex generator to swift car, the drag is much reduced. Thus the aerodynamics of the car is improved which will make the car to run faster and smoother in a fuel efficient way

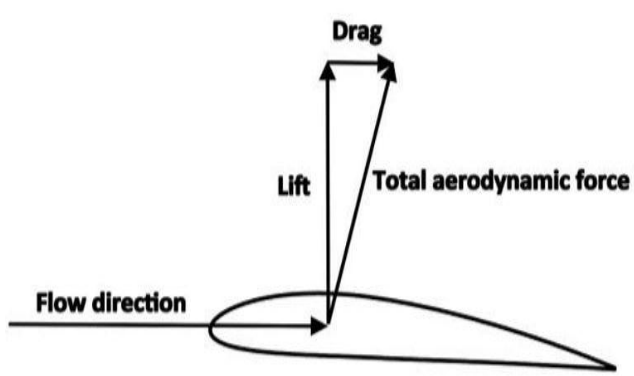
2. PARAMETERS AND TOOLS CONSIDERATION

2.1 Lift force

A fluid running past the surface of a body exerts a force on it. Lift is the component of the force that is perpendicular to the flow direction. It contrasts with the drag force, which is the component of the force parallel to the flow direction. Lift acts in an upward direction in order to counter the force of gravity, but it can act in any direction at right angles to the flow of the fluid. If the surrounding fluid is air, the force is called an automotive aerodynamic force. In any other liquid, it is called a hydrodynamic force.[2] Dynamic lift is distinguished from other different kinds of lift in fluids. Aerostatic lift or buoyancy, in which an internal fluid is lighter than the fluid which surrounds it, does not require movement and is used by balloons, boats, and submarines. Planning lift happens if only the lower portion of the body is immersed in a liquid flow. It is used by motorboats, surfboards, and water-skis.

2.2. Drag Force

Drag is a mechanical force which acts as a pulling force in the opposite direction of the movement. It happens when a solid body is trying to move through a fluid especially air. In case of aerodynamic drag, when the solid object is in movement, a drag is generated due to the difference in velocity between the solid object and the fluid. The cause behind this can be discussed in two ways. The first point of view for the cause of drag is called skin friction. As the solid object moves through the fluid, all the parts of the object touches and goes through the fluid. If the surface of the solid is rougher, there will be a huge amount of drag. On the other hand, when a smooth surface and a fluid of low viscosity comes in contact, a very low drag force is generated. Another reason could be mentioned to be the aerodynamic shape of the solid object (example: an Aircraft or ship). This type of drag occurs due to the pressure difference happening on different parts of the solid body. If the solid body is in movement through the fluid and the solid and fluid are of different velocity, the local velocity and pressure of the fluid acting on the solid will be different in different touching surfaces due to the shape of the solid body. As you know pressure is a function of force, varying pressure will cause a net force acting on the solid which is to be in the opposite direction of the movement. In aircraft, this varying pressure principle is used to create lift, basically for which a plane flies. The shape of the aircraft is done in a way that fluid pressure under the plane is larger than that of over the aircraft. It is controlled by controlling the Angle of Attack, which can cause a huge amount of drag if it is larger than usual angle.



2.3 DRAG REDUCING TOOLS

2.3.1 REAR DIFFUSER

A Rear diffuser, in an automotive term, is a shaped section of the car under body which improves the car's aerodynamic properties by enhancing the transition between the high-velocity airflow underneath the car and the much slower free stream airflow of the ambient atmosphere.It works by providing a space for the under body airflow to decelerate and expand (in area, density remains constant at the speeds that cars travel) so that it does not cause excessive flow separation and drag, by providing pressure recovery. The down force is also generated by the diffuser since it accelerates the flow in front of it. It works by accelerating the velocity of the airflow under the car. The diffuser affects the pressure under the car so that it can expand back to ambient in the diffuser, as the car moves through the air.[3] It uses Bernoulli's principle, "If pressure decrease, velocity increases". Since the pressure below the car is lower than the sides and top of the car, downforce is produced if implemented correctly. The diffuser "drives" the underbody, which produces the downforce acting on the car. Front diffusers also exists, however they generate downforce purely from momentum exchange with the air, as there is nothing ahead of them to drive. A poorly designed front diffuser can create a low pressure region towards the front of the car which slows the air behind it and reduces the effectiveness of the rest of the underbody. Front diffusers usually route air away from the car so that it doesn't affect the rest of the underbody. The air can be vented through channels. As the velocity increases, the pressure is reduced at the throat area and the subsequent reduction in pressure happens for the under floor as the diffuser sucks the car to the ground.[5] The velocity of the air decrease as it moves along the diffuser, which in turn increases the pressure. This faster air helps in evacuating the diffuser more quickly, which helps drop the pressure at the underbody

Fig 1 Rear Diffuser

Fig 2 Vortex generator

2.3.1 VORTEX GENERATORS

A vortex generator (VG) is an aerodynamic device, consisting of a small vane attached to a lifting surface (or air foil, such as an aircraft wing) or a rotor blade of a wind turbine. Vortex Generators may also be attached to some part of an aerodynamic vehicle such as an aircraft fuselage or a car. The purpose of adding VGs is to supply the momentum from higher region where has large momentum to lower region where has small momentum by stream wise vortices generated from VGs located just before the separation point.Vortex generators are positioned obliquely so that they have an angle of attack with respect to the local airflow in order to create a tip vortex which draws energetic, rapidly moving outside air into the slow-moving boundary layer in contact with the surface of the car. A turbulent boundary layer is less likely to separate than a laminar one, and is therefore desirable to ensure effectiveness of trailing-edge control surfaces. Vortex generators are used to trigger the transition.Shifting the separation point, downstream enables the expanded airflow to persist proportionately longer, the flow velocity at the separation point to become slower, and consequently the static pressure to become higher. The static pressure at the separation point governs over all pressures in the entire flow separation region. It works for reducing drag by increasing the back pressure. Shifting the separation point downstream, therefore, provides dual advantages in drag reduction: one is to narrow the separation region in which low pressure constitutes the cause of drag; another is to raise the pressure of the flow separation region. However, the vortex generators that are installed for generating stream wise vortices bring drag by itself. The actual effectiveness of installing VGs is therefore deduced by subtracting the amount of drag by itself from the amount of drag reduction that is yielded by shifting the separation point downstream. Larger-sized VGs increase both the effect of delaying the flow separation and the drag by itself. The effect of delaying the flow separation point, however, saturates at a certain level, which suggests that there must be an optimum size for VGs.

3. SOFTWARES USED

3.1 SOLIDWORKS

SolidWorks is a solid modelling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs on Microsoft Windows. SolidWorks is published by Dassault Systemes.[6] According to the publisher, over two million engineers and designers at more than 165,000 companies were using SolidWorks as of 2013. Also according to the company, fiscal year 2011–12 revenue for SolidWorks totalled $483 million. SolidWorks is a solid modeller, and utilizes a parametric feature-based approach which was initially developed by PTC (Creo/Pro-Engineer) to create models and assemblies. The software is written on Parasolid-kernel Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, etc. Numeric parameters can be associated with each other through the use of relations, which allows them to capture design intent.

3.1 CREO

Creo is a family or suite of Computer-aided design (CAD) apps supporting product design for discrete manufacturers and is developed by PTC. The suite consists of apps, each delivering a distinct set of capabilities for a user role within product development.Creo runs on Microsoft Windows and provides apps for 3D CAD parametric feature solid modeling, 3D direct modelling, 2D orthographic views, Finite Element Analysis and simulation, schematic design, technical illustrations, and viewing and visualization.Creo Elements/Pro and Creo Parametric compete directly with CATIA, Siemens NX/Solidedge, and SolidWorks. The Creo suite of apps replace and supersede PTC’s products formerly known Pro/ENGINEER, CoCreate, and Product View. Creo has many different software package solutions and features.

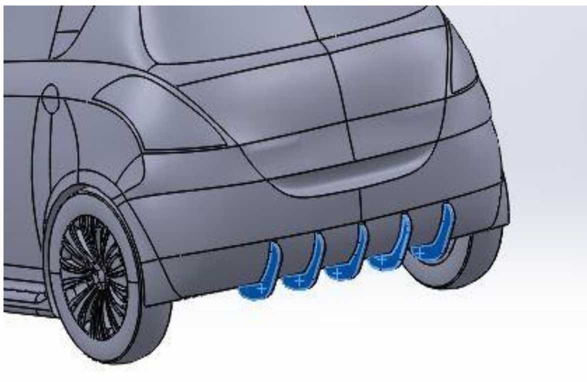
3.2 ANSYS

Ansys develops and markets finite element analysis software used to simulate engineering problems. The software creates simulated computer models of structures, electronics, or machine components to simulate strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. Ansys is used to determine how a product will function with different specifications, without building test products or conducting crash tests. For example, Ansys software may simulate how a bridge will hold up after years of traffic, how to best process salmon in a cannery to reduce waste, or how to design a slide that uses less material without sacrificing safety. Most Ansys simulations are performed using the Ansys Workbench software, which is one of the company's main products. Typically Ansys users break down larger structures into small components that are each modeled and tested individually. A user may start by defining the dimensions of an object, and then adding weight, pressure, temperature and other physical properties. Finally, the Ansys software simulates and analyzes movement, fatigue, fractures, fluid flow, temperature distribution, electromagnetic efficiency and other effects over time.

3. RESULTS AND DISCUSSIONS

3.1 Modifications

In the below model, the rear diffusers are installed and tested for result



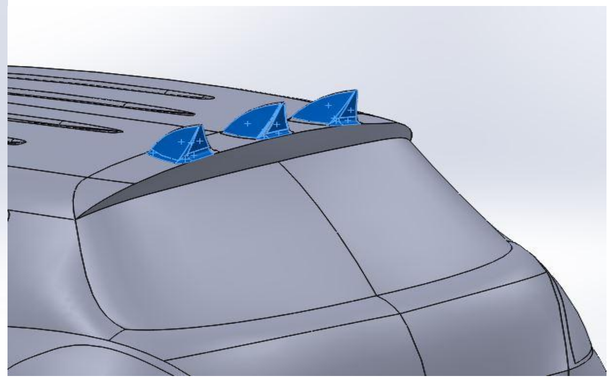


Fig 3 Modified Rear Diffusers ***Fig 4 Modified Vortex Generators***

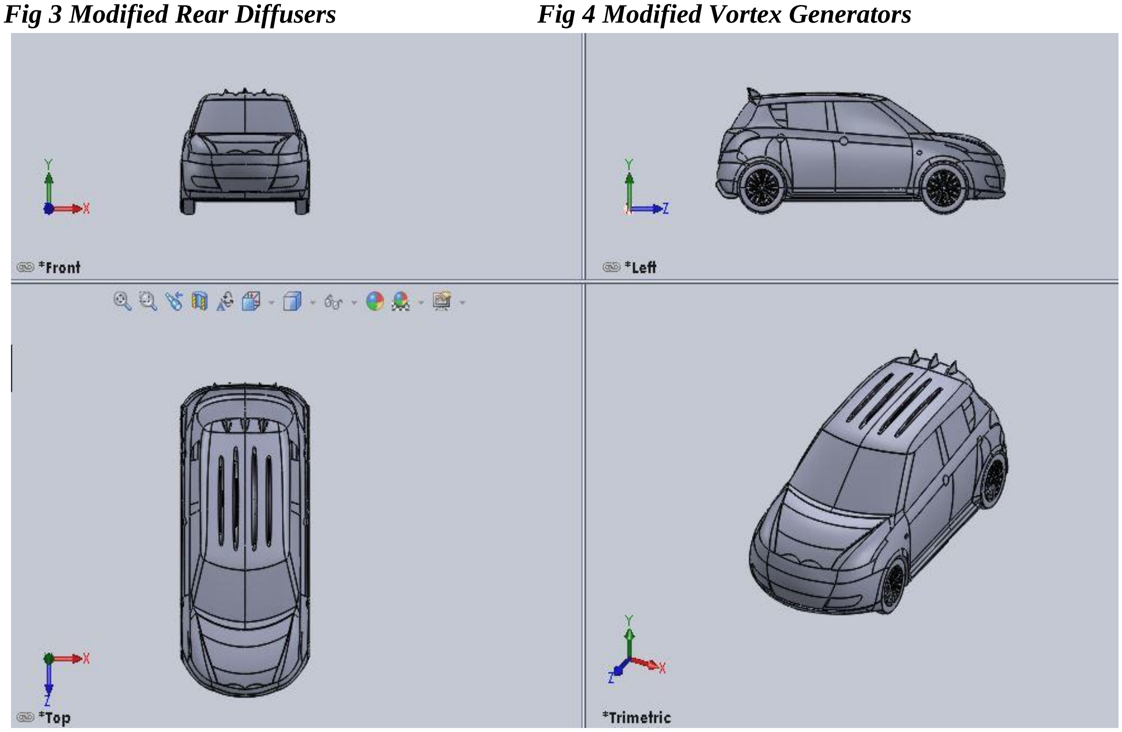


Fig 5 Multi View of Modified Model

3.2 Boundary Conditions

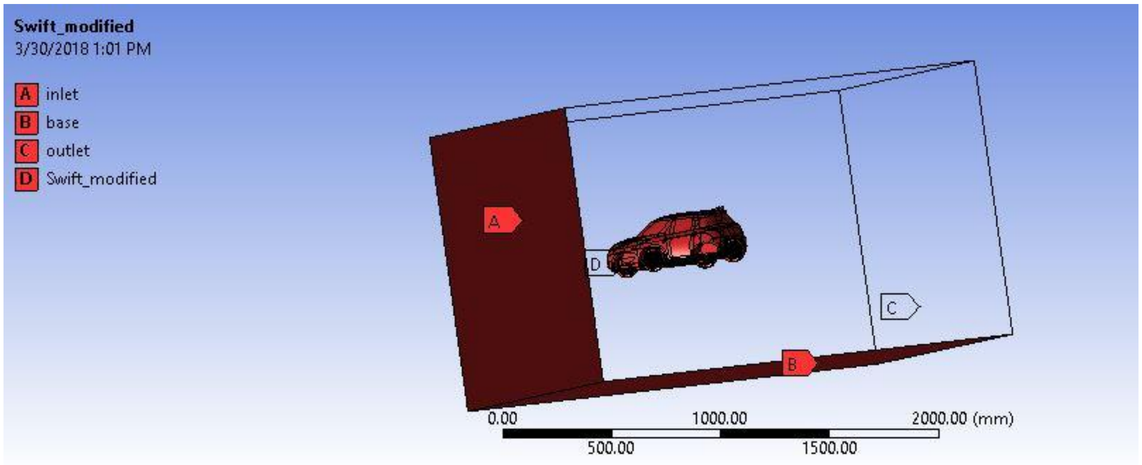


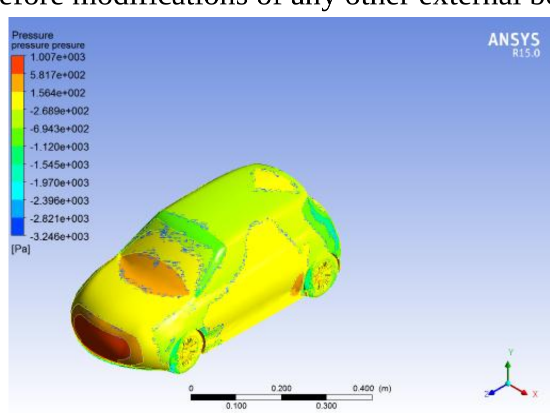
Fig 6 Boundary conditions

Table 1 Boundary conditions

S.No	Parameter	Boundary conditions	Values
1	Velocity inlet	Magnitude measured normal to boundary Turbulence intensity Turbulence specification method Turbulence viscosity ratio	40 m/s Intensity and density 1.00% 20
2	Pressure outlet	Gauge pressure magnitude Gauge pressure direction Turbulence specification method Backflow turbulence intensity Backflow turbulent viscosity ratio	0 Pa Normal to boundary Intensity & viscosity 10% 10
3	Fluid properties	Fluid type Density Kinematic viscosity	Air ρ =1.175 (kg/m^3) V = 1.7894 × 10^{-5} (kg/(m·s))

3.2.1 Before Modification

The following pictures shows the analysis of Maruti Swift in ANSYS software which is done before modifications of any other external body modifications.



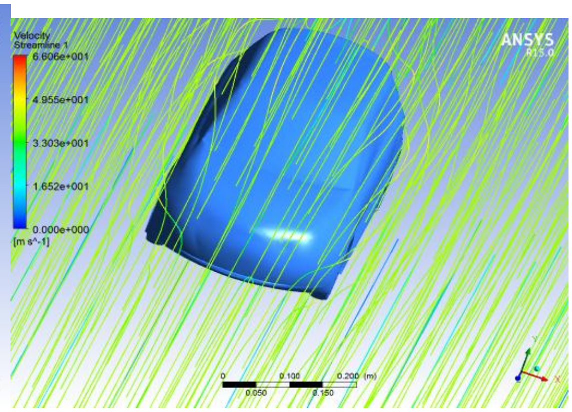


Fig 7 Contours of pressure coefficient *Fig 8- Path lines coloured by particle ID*

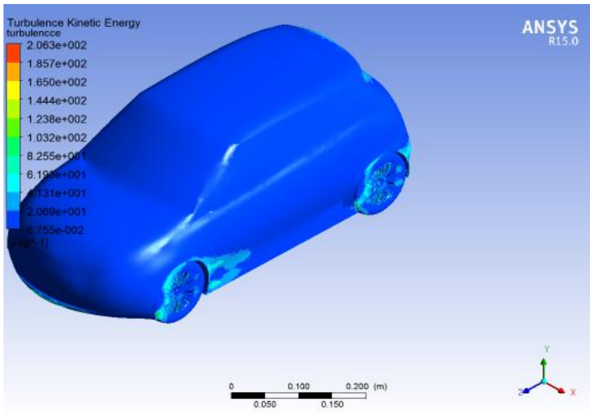


Fig 9 Turbulence

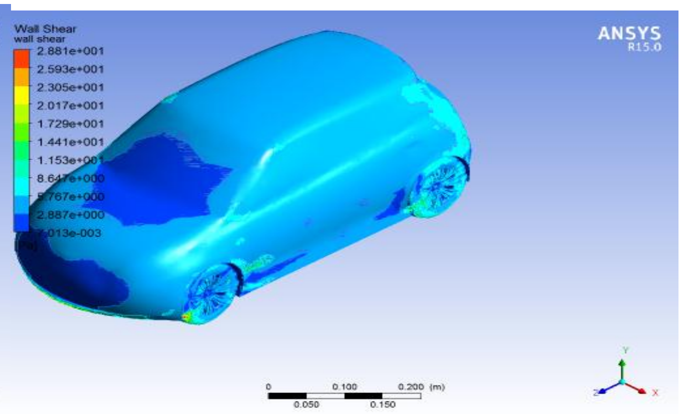


Fig 10 Contours of wall shear

3.2.2 After Modifications

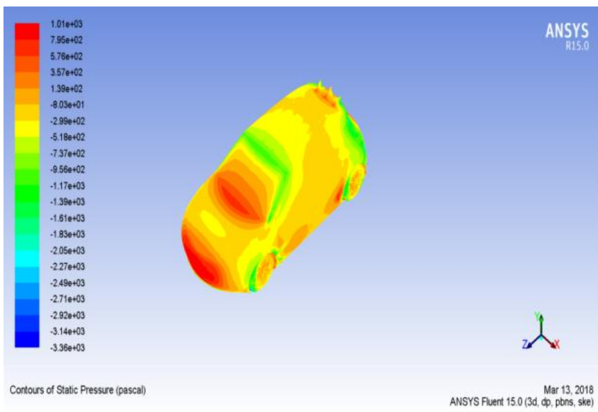


Fig 11 Static pressure of Modified Model

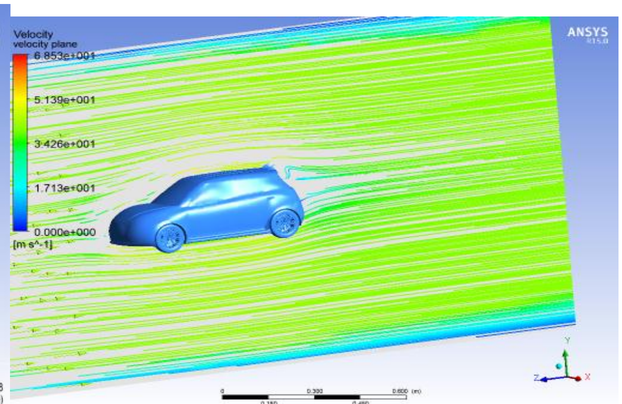


Fig 12 Path lines coloured by particle ID

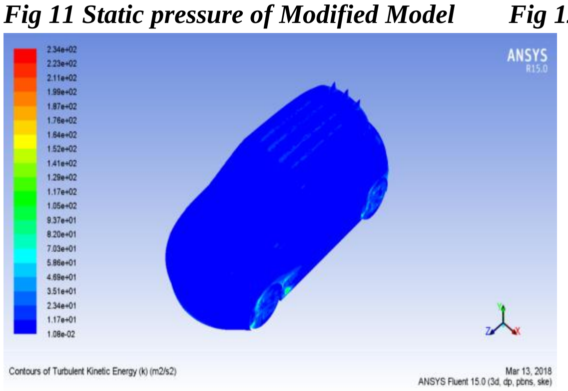


Fig 13 Contours of Turbulence

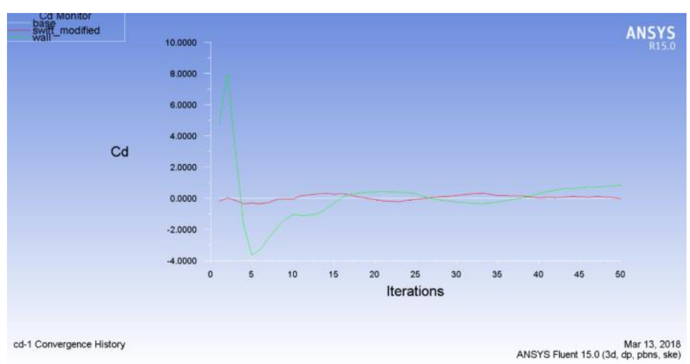


Fig 14 C_d value for modified object

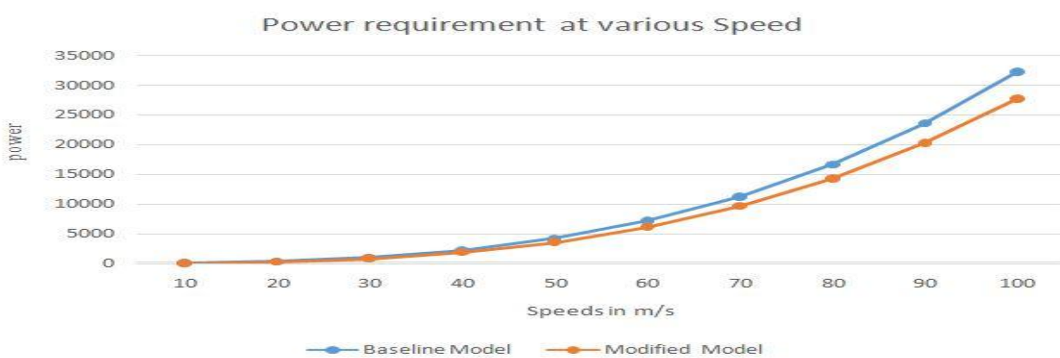
Table 2 Results

Model	Drag coefficient, Cd	Lift coefficient, CL
Baseline	0.408	0.211
Modified	0.342	0.183

The Cd value is reduced from0.408 to 0.342 (16.17%)[8] and the CL value is reduced from0.211 to 0.183 (13.2%) at a speed of 40m/s. Though this number may appear to be substantially low, this reduction in drag and lift coefficient are considerable from aerodynamic point of view as it reduce the power consumption, improve the acceleration and handling behavior of the car.

Table 3 Comparison

Kmph	Speed m/s (v)	Baseline Model	Modified Model
10	2.78	82.94514	78.77505
20	5.56	358.8031	324.025
30	8.33	1014.251	896.4015
40	11.11	2243.41	1963.055
50	13.89	4234.676	3686.116
60	16.67	7178.007	6229.093
70	19.44	11246.39	9740.851
80	22.22	16658.6	14409.76
90	25	23592.07	20388.52
100	27.78	32236.75	27840.64



4. CONCLUSION

On the basis of the car model, 2d and 3d simulations were performed for both car geometries to visualize the airflow and pressure distribution. The mentioned CFD analyses are achieved to see critical places in geometry which are resulting in bad aerodynamics. Leading to the obtained 2d simulation and leading with modifications of an existing 2d model in terms of the redesigned side contour of the car, the existing 3d car model is redesigned. Redesign is in terms of increasing angle between the hood and the front windshield of the car, and adding the rear wing. With the obtained 2d and 3d results, it is concluded that the mentioned changes in the geometry of the redesigned car are resulting in better airflow around the car, and producing more down – force using the rear wing. Bigger amount of down – force is resulting in better stability of the car and the increasing traction. CFD analysis was successfully carried out on the production vehicle. In the process of redesigning, exterior styling with improved aerodynamics of existing car plying on Indian roads, a detailed computational analysis has been done. Once the validity of the simulation was achieved, the next step was to make modifications in the geometry of the original model which could positively affect performance characteristics (lift and drag).

The results obtained showed that by modifying the Swift by adding diffuser, Vortex Generators, and modification in bonnet. The C_d value is reduced from 0.408 to 0.342 (16.17%) and the C_L value is reduced from 0.211 to 0.183 (13.2%) at a speed of 40m/s. Though this number may appear to be substantially low, this reduction in drag and lift coefficient are considerable from aerodynamic point of view as it reduce the power consumption, improve the acceleration and handling behaviour of the car.

REFERENCES

1. Inchul Kim, Hualei Chen and Roger C. Shulze, "A Rear Spoiler of a New Type that Reduces the Aerodynamic Forces on a Mini-Van", 2006-01-1631.
2. Kumaragurubaran.J Raj Kamal M.D Kaliappan S (2015) Investigation of jet noise reduction using fan flow deflectors using on CFD ", International Journal of Applied Engineering Research, Vol. 10 No.33, PP- 26003- 26010.
3. Kaliappan S, Revanth Raam AP , Charan B, Asswin S , Mohammed Ibrahim SM , Dr.T.Mothilal , M.D.Rajkamal , "Modal and kinematic analysis of connecting rod for different mateials" , International Journal of Pure and Applied Mathematics (IJPAM) , Volume 119 No. 12 2018, 14599-14608.
4. Maniraj T, Sathishkumar S. Design of Rear wing for high performance cars and Simulation using Computational Fluid Dynamics, Anna University, Automobile Engineering. International Journal For Trends In Engineering & Technology, 5(2), 2015.
5. Raj Kamal M.D Socrates.S Kaliappan S (2015) "Aerodynamic Effects on Formula One Car Using CFD" , International Journal of Applied Engineering Research (IJAR) Vol. 10 No.33,, PP.28164-28172.
6. Sambit Majumder and Somnath Saha, A Method of Drag Reduction of a Vehicle by Computational Investigation and Geometric Modification. International Journal of Applied Engineering Research, 9(6), 2014, 687-699.
7. S. Kaliappan, M.D.Rajkamal and D.Balamurali "Numerical analysis of centrifugal pump impellor for performance impovement " , International Journal of Chemical Sciences (IJCS) , , Volume –14, Issue – 02, May– 2016, PP – 1148-1156 .
8. Sharma RB, Ram Bansal. CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction. IOSR Journal of Mechanical and Civil Engineering(IOSR-JMCE), 7(5), 2013, 2835.
9. M D Raj Kamal, S.Kaliappan, S.Socrates, G.Jagadeesh Babu ((2017) "CFD Analysis of Single Cylinder IC Engine Inlet Swirl Valve", International Journal of Latest Engineering Research and Applications (IJLERA) ISSN: 2455-7137, Volume – 02, Issue – 08, August – 2017, PP – 34-46.
10. Oleg Zikanov. Essential Computational Fluid Dynamics, John wiley & sons, Inc. Hoboken, New Jersey, 2010.
11. S.Kaliappan, J.Lokesh, P.Mahaneesh, M.Siva, " Mechanical Design and Analysis of AGV for Cost Reduction of Material Handling in Automobile Industries" , International research journal of automotive technology (IRJAT) Volume 01-Issue 1, January 2018, PP.1- 7.
12. Yuvan J, Guan Heng Yeoh and chaoqun Liu TV. Computational fluid Dynamics: A practical Approach, Butter worth-Heinemann: 1st edition, Burlington, M.A, 2007.

13. P.Krishna Teja, G. Moorthy, S.Kaliappan, "Finite element analysis of propeller shaft for automobile and naval applications" , International research journal of automotive technology (IRJAT) Volume 01-Issue 1, January 2018, PP.8-12.
14. Gustavsson T and Melin T. Application of Vortex generators to a blunt body, Technical Report KTH, Department of Aeronautical and Vehicle Engineering, Royal Institute of Technology, Stockholm, Sweden, 2006.
15. Mothilal T, and Pitchandi K, Computational fluid dynamic Analysis on the effect of particle density and body diameter in a tangential inlet cyclone heat exchanger', Thermal science, DOI 10.2298/TSCI151105055T, ISSN 0354-9836 vol 21 ,No 6B , PP 2883 - 2895
16. Mr. M.D.Rajkamal , M. Mani Bharathi , Shams Hari Prasad M, Santhosh Sivan.M, Karthikeyan.S .H.Bahruteen Ali Ahamadu, S.KALIAPPAN , Dr.T.Mothilal,"Thermal analysis of shell and tube exchangers", International Journal of Pure and Applied Mathematics (IJPAM) , Volume 119 No. 12 2018, 14299-14306.
17. Mothilal T, Velukumar V, Pitchandi K and Selvin Immanuel M, Effect of cyclone height on holdup mass and heat transfer rate in solid cyclone heat exchanger- CFD Approach, ARPN Journal of Engineering and Applied Sciences, volume 11, no. 2, pp. 1269-1275, ISSN 1819 – 6608.
18. Panu Sainio, Kimmo Killstrom and Matti Juhala (2007), "From Aalto University Conducted a Research on Aerodynamics Possibilities for Heavy Vehicles".
19. S.Kaliappan, M.D.Raj Kamal, Dr.S.Mohanamurugan, Dr. P.K.Nagarajan (2018) "Analysis of connecting od using finite element analysis",Taga journal of graphc design, Vol. 14 - 2018, PP-1147-1152.
20. E Selvakumar, M K Murthi and A Muthuvel (2013), "Conducted Research on Aerodynamic Exterior Body Design of Bus".
21. G Buresti, G V Lungo and G Lombardi (2007), "Carried out a Research on Methods for the Drag Reduction of Bluff
22. Sivaram S Kaliappan S Raj Kamal M.D Suresh kumar R. BalamaniKanda Suthan K " Performance Analysis of Gasifier on CFD ",International Journal of Applied Engineering Research, Vol. 10 No.33, PP- 25880- 25889.
23. Dr T.Mothilal, P.Harish Krishna, G.Jagadeesh Babu, Ashwin Suresh, K.Baskar , S.Kaliappan , M.D.Rajkamal, " CFD Analysis of Different Blades in Vertical Axis Wind Turbine ", International Journal of Pure and Applied Mathematics (IJPAM) , Volume 119 No. 12 2018, 13545-13551.