Middle-East Journal of Scientific Research
24 (Recent Innovations in Engineering, Technology, Management & Applications): 133-140, 2016
ISSN 1990-9233; © IDOSI Publications, 2016
DOI: 10.5829/idosi.mejsr.2016.24.RIETMA121

Design and Aerodynamic Analysis of a Car to Improve Performance

G. Siva and V. Loganathan

Department of Mechanical Engineering, Priyadarshini Engineering College Vaniyambadi, India

Abstract: Aerodynamics plays an important role while designing any automotives. Due to the aerodynamics the entire performance of the automotive will be changed. In this project a car model Toyota fortuner is considered and it is modeled using solid works modeling software. Attention is only given to the external design of the car, while the interior is not modeled. Furthermore the modeled car is considered as a 2D model for future analysis using ANSYS Fluent to determine the aerodynamic characteristics like pressure, down-force and drag. The 2D model is considered here because the time taken to analyze the 3D model will be more, so to reduce the analysis time we are considering the 2D model alone. However the result obtained in 2D model will be same as the 3D model. Drag plays an important role in car aerodynamics which is an external resistance of the car. Objective of this project is to reduce the drag of the car by modifying the car shape. The modified 2D car model is also analyzed in ANSYS fluent and the results for the existing 2D model and modified 2D car model is analyzed for various speeds. Due to this design modification the drag will be reduced without decreasing the car performance. By decreasing the drag entire car performance can be increased. Since there is a reduction in drag, the car speed can be increased with the reduction in fuel consumption.

Key words: 2D Carmodel · Aerodynamics · Drag · Performance

INTRODUCTION

The external aerodynamics plays an important role in the development process of modern automotives. The vehicle's performance, its stability and the vehicle's cooling system are all influenced by aerodynamic loads. Furthermore, the driver's comfort and the driver's visibility depend on the external flow field. The automotive industries apply wind tunnel experiments and computational fluid dynamic (CFD) simulations to study the aerodynamic loads on their vehicles.

With increasing of automobile speed, people begin to pay attention to dynamic performance of automobiles. Due to increasing of oil price, high demands bring forward to automobile design, especially to aerodynamic characteristics of automobiles and the aerodynamic characteristics directly affect driving characteristics, stability, operation, oil consumption and safety of automobiles.

[1] They carried out the analysis in sports utility vehicle. In an era improving the fuel economy of vehicle has become need of automobile industries to survive in the cut throat competition. As rapid and continuous increase in prizes of fuels, consumers are going for most fuel efficient vehicles. Recently stringent emission norms, fuel economy and recycling become important social concern. Fuel economy has become latest topic of discussion among not only the responsible scientists but also common citizens. The company those will cater the need of consumers will survive in the market. By aerodynamic styling of vehicle one can not only improve the fuel efficiency but also ensure better stability and good handling characteristics of vehicles at higher speed especially on highways. The paper describes assessment of drag force (F_d) and drag coefficient (C_d) by using computational fluid dynamics (CFD). The model of sports utility vehicle (SUV) on reduced scale 1:32 is drawn with aid of PROE Software [2]. This work proposes an effective numerical model based on the Computational Fluid Dynamics (CFD) approach to obtain the flow structure around a passenger car with Tail Plates. The experimental work of the test vehicle and grid system is constructed by ANSYS-14.0. FLUENT which is the CFD solver & employed in the present work. In this study, numerical iterations are completed, then after aerodynamic data and detailed complicated flow structure are visualized. In the present work, model of generic passenger car has been developed in solid works-10 and generated the wind

Corresponding Author: G. Siva, Department of Mechanical Engineering, Priyadarshini Engineering College Vaniyambadi, India.

tunnel and applied the boundary conditions in ANSYS workbench 14.0 platform then after testing and simulation has been performed for the evaluation of drag coefficient for passenger car. In another case, the aerodynamics of the most suitable design of tail plate is introduced and analyzed for the evaluation of drag coefficient for passenger car. The addition of tail plates results in a reduction of the drag-coefficient 3.87% and lift coefficient 16.62% in head-on wind. Rounding the edges partially reduces drag in head-on wind but does not bring about the significant improvements in the aerodynamic efficiency of the passenger car with tail plates, it can be obtained. Hence, the drag force can be reduced by using add on devices on vehicle and fuel economy, stability of a passenger car can be improved [3]. The study focuses on the slipstreaming effect on an aerodynamic sedan vehicle, trailing a container truck on highways. The simulation was carried out using ANSYS FLUENT CFD software. The effect of distance of separation of the vehicles to slipstreaming has been studied and critical distance is found out. The relative drag during slipstreaming at different velocities at the critical distance has also been addressed. f reckless driving in sensitive areas [4]. This study focuses on the mileage increase. The usage of automobiles is in predictable in day to day life which is common among people for travelling in Indian economic condition. More Indian families prefer family cars like Tavera, Innova, Qualis, etc… for long drives. But, these family cars provide less mileage when compared to sedan class and small cars. In order to increase the mileage of car implemented aerodynamic device which is known as cut section of wing on ceiling of a car. When the car is in motion at the average speed of about 120km/hr the aerodynamic device generates sensible lift force which tries to pull the car from ground and the weight of car acts downwards which pushes the car towards ground. Each force cancels each other and the net force will be the subtraction of lift force generated by aerodynamic device and this would reduce Gross Vehicle Weight. By the known phenomena Miles per Gallon of a vehicle which can be improved by reducing the GVW [5]. The main cause of pressure drag is the separation of air flow at the top surface of car. So this invention aims to delay flow separation by keeping Vortex Generators. The experimental investigations were performed on BLWT, while computational analysis was carried out using Standard computational software. Pressure measurements were made for the model when the wind was flowing parallel to the length of the car, with and without Vortex generators. As per the computational and experimental analysis it's observed and proved that, the drag co-efficient of car model was reduced by keeping Vortex Generators.

Objective of the Work: The objective of this project is to improve the aerodynamic characteristic of a car. The aerodynamic characteristics can be improved by altering the front / back shape or by adding spoilers. The primary objective of this project is to decrease the drag, where the secondary objectives are;

- Increasing the speed
- Decreasing the fuel consumption
- Decrease in damage over the structure

Design and Specification of Existing Car Model

Toyota Fortuner: The Toyota Fortuner is also known as the Toyota SW4, is a mid-sized SUV. It was a successful SUV model in India. Originally it was assembled only in Thailand, but later in Indonesia and other countries. The Fortuner is built on the Toyota Hilux pickup truck platform, as a shown in below that Fig. 1. It features three rows of seats and is available in rear-wheel drive or four-wheel drive.

The Fortuner is part of Toyota's IMV project in Thailand, which also includes the Toyota Hilux and the KijangInnova (in Indonesia). Developed in large part by Toyota's Thai operations, the Fortuner has piggybacked the success of the Hilux and is now built in a number of countries including India, Argentina and Indonesia; although outside Thailand its success has been mixed.

However, in the Sydney Morning Herald, Gillard

Fig. 1: Toyota Fortuner

Who worked for the Toyota Technical Centre (TTC-AU) stated that the organization has been working on the Fortuner since 2006. In fact, this Sport Utility Vehicle (SUV) is designed in Thailand by Thai and Japanese engineers. However the facelift version of the IMVs vehicle including the Fortuner was designed in Australia by Toyota Australia which is also responsible for developing the next generation of the Fortuner. However, this second generation was under development.

The Fortuner is not offered in Japan, Europe, North America, Australasia, or China. For the medium pick-up based SUV segment in those markets, Toyota offers the Hilux Surf (Japan), 4Runner (North America) and Land Cruiser Prado (Europe, Australasia and China). However, in some Central American countries (Panama for example), Toyota offers the Fortuner alongside the 4Runner and Land Cruiser Prado.

Table 1: Design Parameter

S.No	Dimensions	Size in mm.
1	Distance between Axle (Side)	2750 mm
2	Distance between Axle (Front)	1540 mm
3	Length	4705 mm
4	Width	1840 mm
5	Height	1850 mm

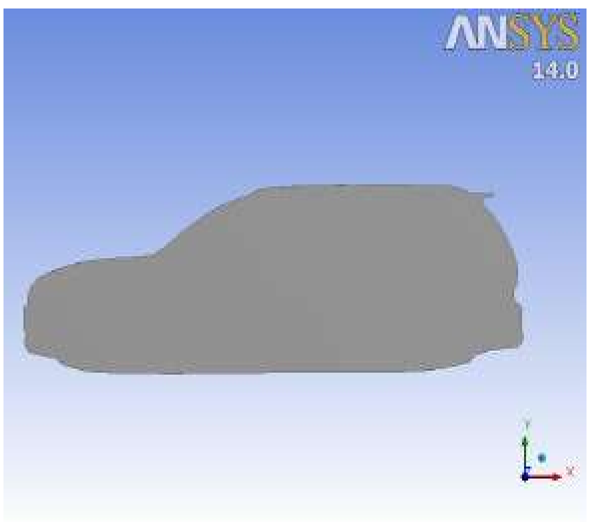

Fig. 2: Which represents the 2D sketch of Toyota Fortuner

Solver Settings: The problem of SUV numerical analysis requires the solver settings to be completed before starting the simulations. The solver setting includes type of solver (3D or 2D), the viscous model, boundary condition and solution controls. The inlet of the wind tunnel is indicated by the term 'Velocity inlet', while the outlet of the wind tunnel is termed as 'Pressure outlet'. The fluid properties were calculated taking into account the temperature and density of the average ambience condition of the area near the lab of University of Michigan. The solver settings and boundary condition for both the benchmark simulations are shown below Table [2-3-4].

Table 2: Solver setting

CFD Simulation	2D
Solver	
Solver	Pressure-Based
Space	2D
Formulation	Implicit
Time	Steady
Velocity Formulation	Absolute
Gradient Option	Cell-Based
Porous Formulation	Superficial Velocity

Table 3: Viscous model and Turbulence model settings

Viscous Model	
Turbulence Model	k-ε (2 eqn)
k-epsilon Model	Standard
Near-Wall Treatment	Standard Wall Functions
Operating Conditions	Ambient
Total Kinetic Energy Prandtl Number	1
Total Dissipation Rate Prandtl Number	1.3

Table 4: Boundary condition

	Boundary Conditions	
Velocity Inlet	Magnitude (Measured normal to Boundary)	30 m/s (constant)
	Turbulence Specification Method	K and Epsilon
	Turbulent Kinetic Energy (m^2/s^2)	1
	Turbulent Dissipation Rate (m^2/s^3)	1
Pressure Outlet	Gauge Pressure magnitude	0 Pascal
	Gauge Pressure direction	normal to boundary
	Turbulence Specification Method	Intensity and Viscosity Ratio
	Backflow Turbulent Kinetic Energy (m^2/s^2)	1
	Backflow Turbulent Dissipation Rate (m^2/s^3)	1
Wall Zones	No Slip	
Fluid Properties	Fluid Type	Air
	Density	$\rho = 1.225$ (kg/m^3)
	Kinematic viscosity	$v = 1.7894\times10^{-5}$ ($kg/(m\bullet s)$)

RESULTS AND DISCUSSION

The car model has to be optimized for some portion to obtain a stability and reduced drag. For the standard hatchback car model it's always recommended to optimize the angle of tailgate or radius where roof meets tailgate or C-Post. In here the angle of tailgate and C-Post are optimized to get a better solution.

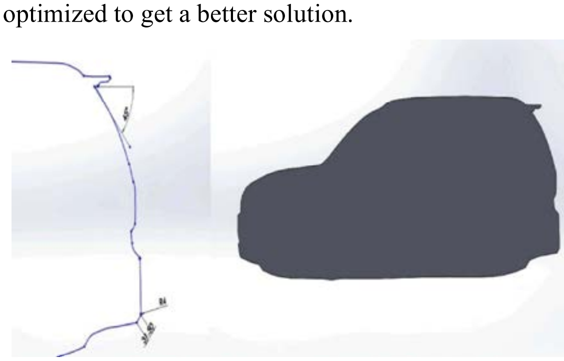

Fig. 1: Which represents the toyota fortuner Existing model at angle 45°

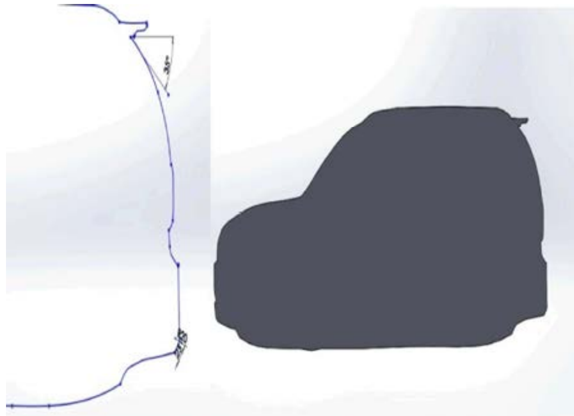

Fig. 2: Which represents the toyota fortuner that Angle optimized for 35° with C-post

Table 5: Optimized angle

Sl.No	Optimized angle	C_L	C_D
1.	45° (Existing)	-.95893	0.569493
2.	35°	-2.7798	0.48463

Pressure Distribution: The pressure distribution over the model is shown in below figures. From the figure it's clear that the pressure distribution over the optimized model has been reduced.

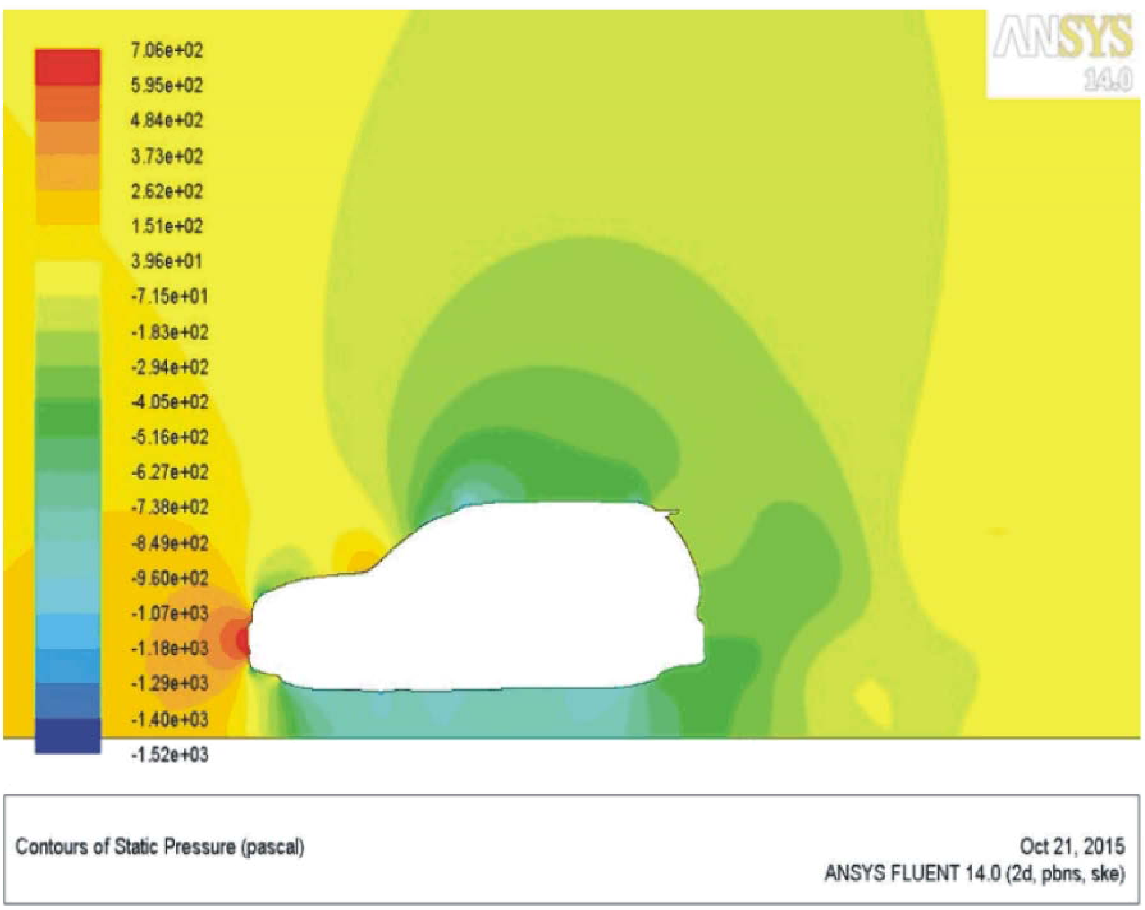

Fig. 3: Which represents the Pressure distribution over existing model at 45°

Fig. 4: Which shows the Pressure distribution over Optimized model at angle 35°

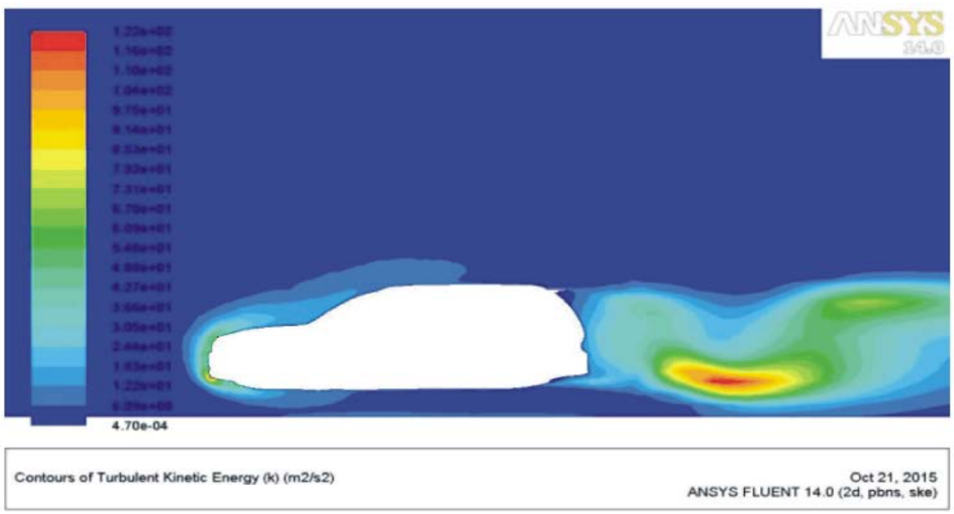

Fig. 5: Which represents the Turbulence for existing model

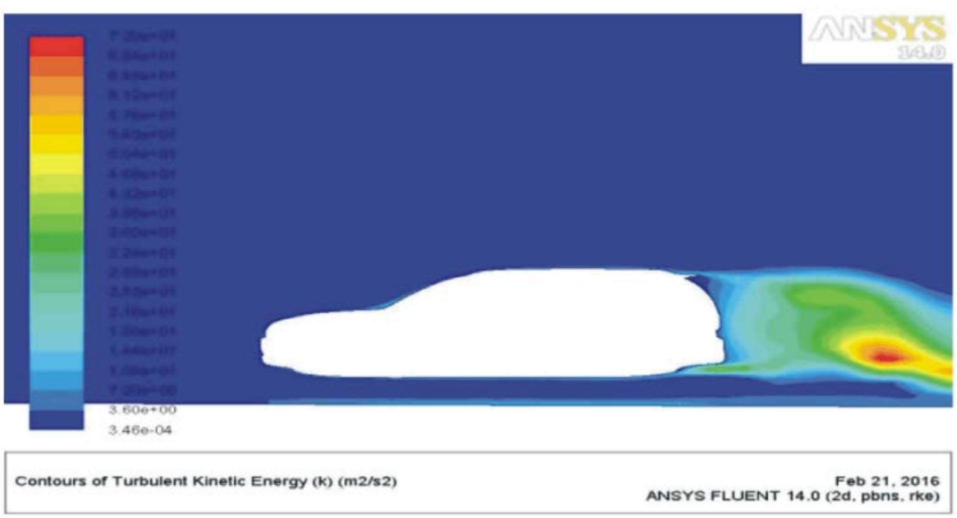

Fig. 6: Which represents the Turbulence for Optimized model

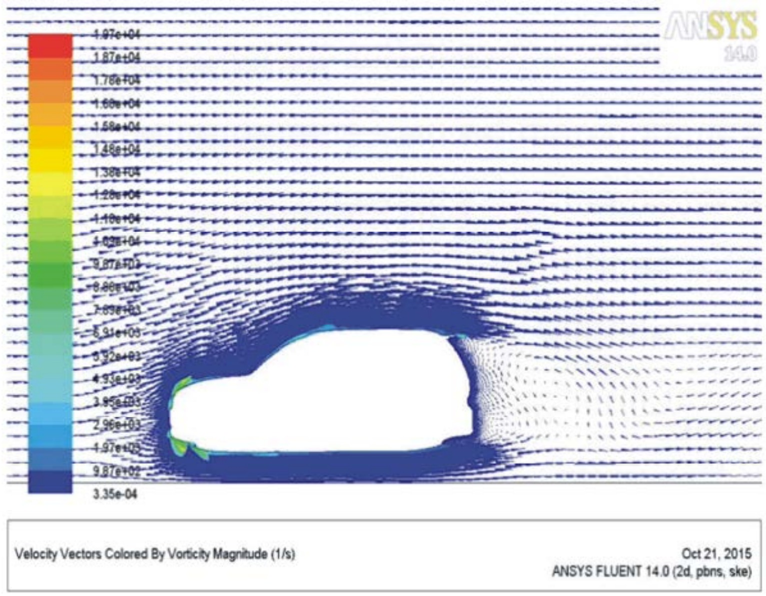

Fig. 7: Which represents the Vorticity over existing model

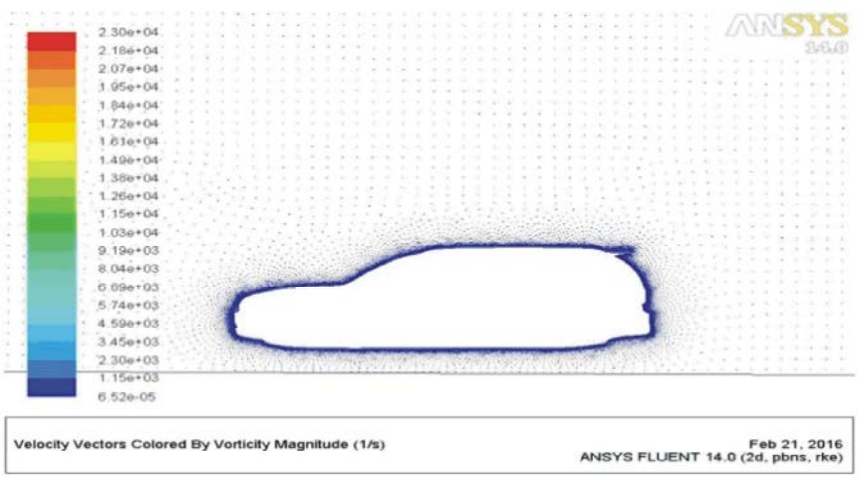

Fig. 8: Which shows the Vorticity over Optimized model Velocity distribution

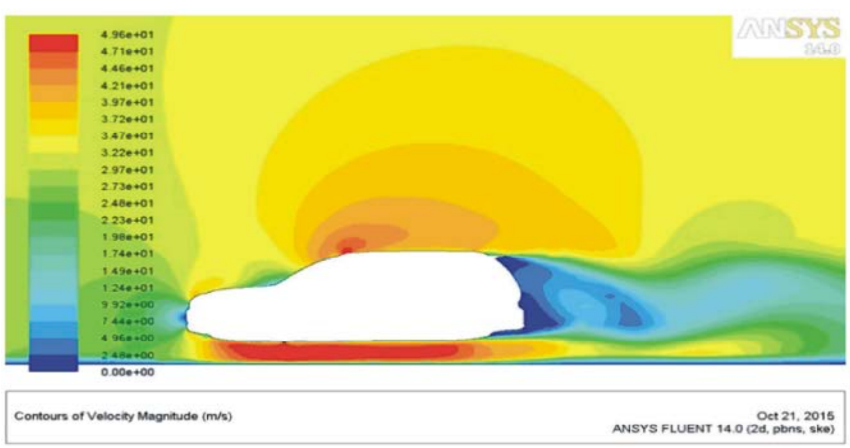

Fig. 9: Which represents the Velocity distribution over existing model

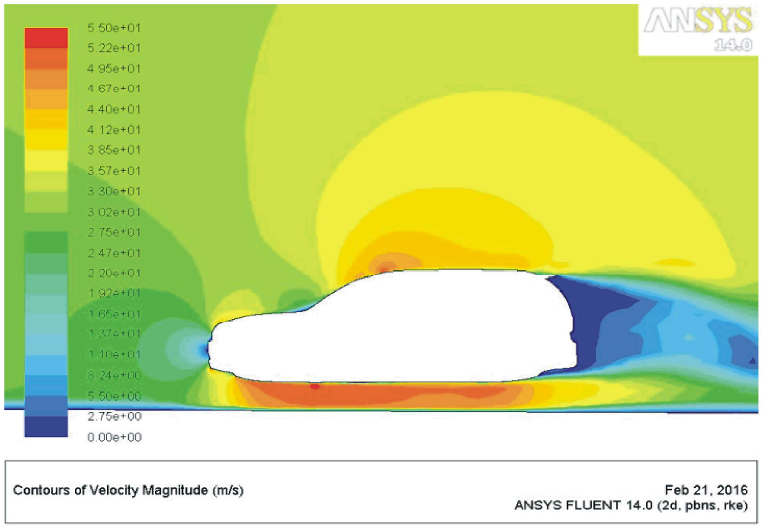

Fig. 10: Which shows the Velocity distribution over Optimized model

Table 6: Comparison of Aerodynamic characteristics

Model	Pressure		Turbulence	
	Min.	Max.	Min.	Max.
1	-1.52×10^3 Pa	7.06×10^2 Pa	4.7×10^{-4} m^2/s^2	122 m^2/s^2
2	-1.79×10^3 Pa	6.54×10^2 Pa	3.4×10^{-4} m^2/s^2	72 m^2/s^2

Table 7: Comparison of Aerodynamic characteristics

Model	Vorticity		Velocity	
	Min.	Max.	Min.	Max.
1	3.35×10^{-4} s^{-1}	1.97×10^4 s^{-1}	0	49.6 m/s
2	6.52×10^{-5} s^{-1}	2.3×10^4 s^{-1}	0	55 m/s

Turbulence Distribution: Turbulent flow is a flow regime characterized by chaotic property changes. This includes low momentum diffusion, high momentum convection and rapid variation of pressure and flow velocity in space and time. The turbulence distribution over the model is as shown in below.

Vorticity: The vorticity is a pseudo vector field that describes the local spinning motion of a continuum near some point, as would be seen by an observer located at that point and traveling along with the flow. The vortices over the model must be low, so that it won't get induced due to the induced drag. The induced drag is a main parameter over the model.

The velocity is a parameter which is inversely proportional to the pressure. The velocity over the object must be high so that the model can escape from the drag creation. The model gets induced to the drag when the velocity over the model is low. The figures shown in below are the velocity contours for the model-1 to model-2.

CONCLUSION

Due to the aerodynamics the entire performance of the automotive will be changed. Toyota Fortuner model has been modeled in SOLIDWORKS by considering its external geometry alone. Then the model has been taken to ANSYS workbench as a 2D sketch to perform external flow analysis over the model. The external flow for the car has been carried out by assuming that the car is moving at a speed of 100 Km/h (30 m/s). From the analysis the aerodynamic characteristics has been studied in clear. The results show that the car may rollout at some extreme conditions. So to overcome this problem the model has been optimized and its modeled in solidworks and its analyzed to compared the results and the results obtained shows that the optimized model has a better aerodynamic performance than the existing model.

REFERENCES

1. Dinesh Dhande and Manoj Bauskar, 2013. "Aerodynamic Analysis Of Sport Utility Vehicle By Using Computational Fluid Dynamics Approach" in International Journal of Engineering Research & Technology During.
2. Sharma1, Ram Bansal, 2013. "CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction", Journal of Mechanical and Civil Engineering.
3. Adarsh, *et al*., 2014. "Numerical investigation of drag on a trailing aerodynamic sedan vehicle", International Journal of Mechanical and Production Engineering.

4. Udayagiri, Lingaiah *et al*., 2015. "Design and CFD Analysis of Aerodynamic of a Car with Various Aerodynamic", International journal & magazine of engineering, technology, management and research Devices.
5. Sivaraj G., *et al*., 2015. "The computational analysis of sedan car with vortex generator", International Journal of Advance Research in Science and Engineering.