

Study of Dimple Effect on Aerodynamic Drag Characteristics of a Car

Karan Kumar Shaw¹, Yash Kesarwani², Priyam Chakravarty³

U.G. Student, Aerospace Engineering, SRM Institute of Science & technology, Kattankulathur, Tamil Nadu, India¹

U.G. Student, Automobile Engineering, SRM Institute of Science & technology, Kattankulathur, Tamil Nadu, India^{2,3}

ABSTRACT: On a constant hunt for ideas and techniques to improve fuel economy and performance of cars, vehicle aerodynamics is a vital research area which cannot be neglected, as it plays a pivotal role in stability, handling and fuel efficiency. This paper deals with Dimple Effect as seen in golf balls used as a technique to create rough surface along the contour of the vehicle body to create turbulent boundary layer which results in reducing drag force and improving downforce. We have considered studying the effects of the dimple phenomena in regular passenger or commercial vehicles by studying pressure contour, airstream flow and calculating drag force for different dimple depth ratio over a wide range of air speed. Fuel efficiency is a key factor for commercial vehicles and taxi cars as they are on the road all the time. With their boxy, less aerodynamic design it's very difficult to achieve a good mileage. This paper is all about evaluating the aerodynamic parameters of a car model using Computational Fluid Dynamics as a tool and propose dimple modifications to the critical regions that might help achieve a better fuel economy by beating the wind. We have employed this technique to a spoiler, rear-view wing mirror, hood and trunk of a sedan. In this paper we have seen a reduction in aerodynamic drag and the wake region behind the car hence improving downforce and stability.

KEYWORDS: Dimple, Aerodynamics, Drag, Car, Mesh, Computation fluid dynamics, Automobile, fuel efficiency.

I. INTRODUCTION

During the early years of the automobile industry, vehicle aerodynamics was mainly incorporated in designing racing cars. However, the major motivation to reduce drag in mass produced cars came in 1973 when a group of oil exporting countries formed cartel, drastically increasing the price of crude oil and simultaneously cutting production. This forced the automakers to invest in designing cars which are fuel efficient. The fuel consumption depends upon efficiency of engines, transmission system, shape of car, and the power required to overcome the resistance to motion. The vehicle aerodynamics makes us realize that it's not incredible that so many cars look so similar but what actually makes the cars look different from other. For the past decades, there is a basic tension between aerodynamic engineers and car designers. Each side believes that their missions are in harmony, that good design should not preclude good aerodynamics, and that aerodynamics should not limit design. In reality, the wind doesn't care whether our grille is distinctive, or whether it is not there at all because it will always want the car to look like a raindrop, and anything else is a compromise.

In India, the automotive sector accounts for 45 % of the country's manufacturing gross domestic product (GDP), 7.1% of the country's GDP and employs about 19 million people both directly and indirectly but currently it is facing downturn due to economic misfortunes, inadequate testing techniques, failure to match the demand of the market, overpricing of fuel, particularly in commercial vehicles which are not aerodynamically viable. A elucidate literature survey depicts that several attempts have been made in order to improve fuel economy by changing the designs which could reduce the drag forces acting on the vehicle. CFD simulation technology proved useful in understanding the phenomena of air interaction with body and to stimulate real-life environment to optimize its performance such as drag reduction.

This project was instigated as a solution to the challenges & demand as it explores the effect of surface roughness over the vehicles when it is simulated at different speeds. The dimple indentation of various depths was explored on spoiler, hood and shell of rear mirror which are subjective to increased drag to check if it could reduce the drag even by trivial amount. This could pave the way for successful improvement in fuel economy and minimizing emissions.

Certain factors that must be understood before proceeding with the research are:

1.1 WAKE REGION- The aerodynamic wake refers to the region of disturbance behind the body as it moves across the fluid. The classic example can be sited in the flow pattern of wave downstream of ship as they move forward and is commonly referred as wake structure. In case of car, the wake region is indicated by two major contra-rotating vortices

in the streamwise direction immediately behind car. It occurs due to viscosity, flow separation and followed by turbulence.

1.2 FLOW SEPARATION- In ideal case, the flow will remain attach when the pressure distribution on both upstream and downstream side of the body remains same, hence there will be no unbalanced forces acting on the body and thus no drag. Nevertheless, the boundary layer does get separated from the body owing to the low-pressure region downstream. This sudden pressure drop occurs due to sharp edges on the vehicle body. This pressure is much lower than pressure at upstream side of the body and this difference in pressure leads to drag force transpiring from the body. As evident from the figure, as the thickness of the boundary layer increases followed by the velocity reversal indicating boundary layer separation.

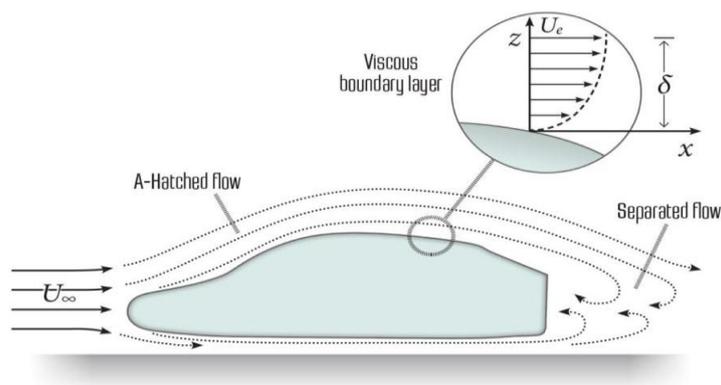


Fig.1 Flow Separation around the car

II. RELATED WORK

Shobhit Senger *et al.* (2014) [1] evaluated aerodynamics performance on various F1 and supercars. Their paper briefly describes the different forces acting on a car (drag force, lift force) which were being measured and visualized using Computational Fluid dynamics (CFD) and Wind Tunnel Testing (WTT). Based on the data obtained, the comparison has been done over a Hindustan ambassador, Lamborghini Aventador LP 700-4 and the F1 car. The statistical data and tests performed showed that F1 car is the most aerodynamically efficient. Its design is made in a way that it cuts the air with ease and steers the air flowing over it to the rear wings. This in turn, generates more amount of down force making the car stable at high speeds which is very useful for racing scenario.

Gavin Dias *et al.* (2016) [2] studied and optimized aerodynamic characteristics of the Swift Dzire and proposed different methods of reducing drag. The first modification was converting the sedan's three box design was changed into a fastback design and comparing with original model, it is observed that fastback model the airflow remained attached and separation is delayed. The second modification was the addition of the spoiler in shape of inverted airfoil, on analysis it is observed that wake region is smaller than original model which greatly increases the traction of the vehicle and reduces the drag. They concluded that C_d value is reduced by 0.04 by changing into fastback model and 0.025 by attaching spoiler at the speed of 40 m/s.

Prof. Dr. Bruno Venturini *et al.* (2016) [3] under Universidade Federal Espirito Santo, investigated different critical points on the structure that increases drag and suitable modifications were done to improve aerodynamic efficiency. Different CFD simulations were performed to check if the modifications achieved the overall expected results. Total of 6 different concepts were made and their corresponding coefficients of lift and drag were calculated using STAR-CCM+ software by creating virtual wind tunnel conditions. Loss of aerodynamic characteristics are observed in Concept 4 having wheel cover protruding out. However, they fail to achieve the C_d of less than 0.25.

Harun Choudhary *et al.* (2017) [4] performed the study of dimple characteristics on golf balls. Their purpose of the paper was to examine the aerodynamic effect of on golf balls with 11 different variations in dimple depths. The drag coefficient for the golf balls were found to vary with varying dimple geometry. The statistical data also established a positive linear correlation between relative roughness and drag coefficient. This idea paved our way to change the surface roughness of our automobile to check if we can reduce drag or not.

III. METHODOLOGY

3.1 CAD MODELLING: The models used for the study of the topic were 3D modelled in SOLIDWORKS 2018 software. The dimensions for the sedan model was taken from a BMW 3 Series blueprint, the spoiler profile was taken from a spoiler blueprint and mirror dimensions were estimated out from a conventional passenger car. We used simple features like extrude, extrude cut, fillet etc. to model out the stock bodies. Later dimples were created in the required surfaces of the stock bodies and dimensions are as mentioned below using features like line, 3-point center arc, revolve cut, curve driven pattern and fill pattern. The models were then transferred to ANSYS in .IGES file format for aerodynamic analysis.

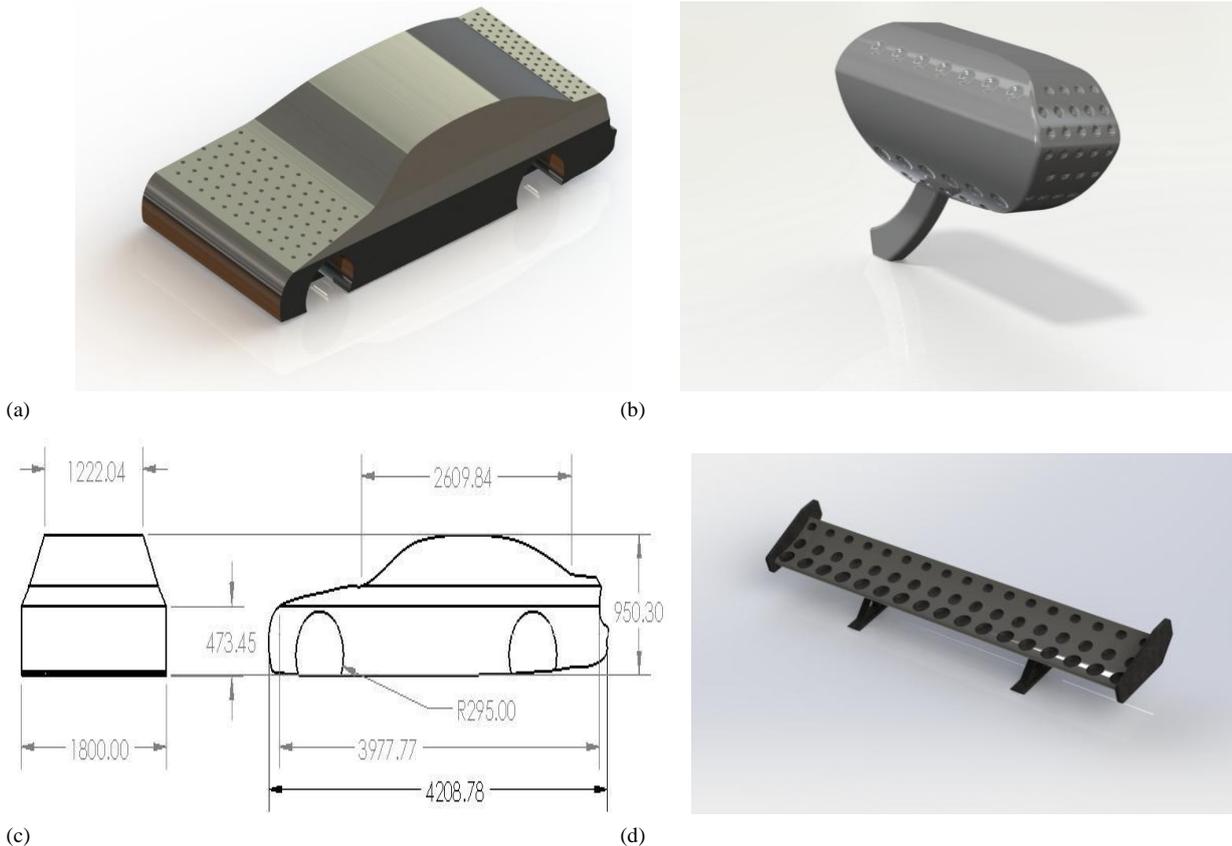


Fig. 2 CAD Model of (a)Car, (b) Side Mirror (d)Spoiler and (c) 2D Drawing of a Car

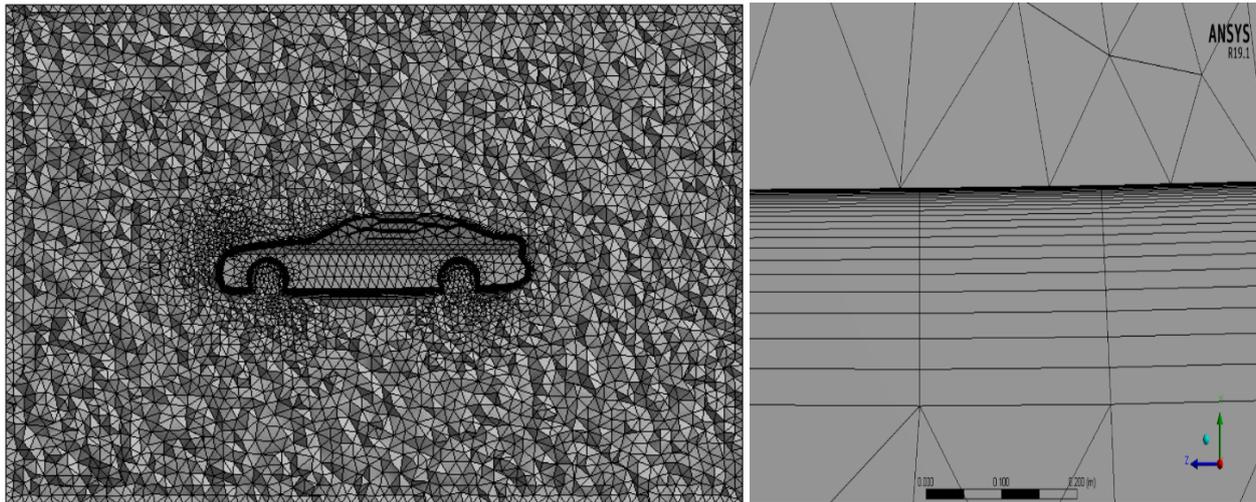
3.2 MESHING: A quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. We should strive to maintain an optimal balance between the computational cost and the level of fineness achieved. We preferred CFD physics as they offer finer mesh in default settings and the minimum edge length is defined by the grid independence theory. Based on selected model equation, the Y^+ values have certain ranges (30-100). Using reference length as 4209.63 mm which was model car length and applying target Y^+ value as 30, we calculated the first layer height to be $3.72e-4$ m. This value is implemented into meshing using Inflation with 25 layers having growth rate at 1.2. Then, we intricated Tetrahedron method because it has better bricking into the structure and takes less CPU time. The meshing quality was found to be within preferred condition.

Table 1. Meshing Parameters

Physics Preference	CFD	Inflation Option	First Layer Thickness
Solver Preference	Fluent	First Layer Height	$3.72e-4$ m
Element Order	Linear	Maximum Layers	25
Resolution	7	Growth Rate	1.2
Minimum Edge Length	$8.94598e-3$ m	Mapped Mesh	Yes
Span Angle Center	Fine	Method	Tetrahedrons
Smoothing	High	Algorithm	Patch Conforming
No. of CPU	Program Controlled	Element Order	Use Global Setting

Table 2. Meshing Parameters

Meshing Quality	Minimum	Maximum	Average	Preferable Condition
Skewness	4.523e-4	0.895516	0.249	Max value should be less than 0.95 and av should be below 0.33
Orthogonal Quality	7.438e-3	0.99414	0.739	For max value range is 0(bad) - 1(good)



(a)

(b)

Fig. 3 Images from Meshing Setup showing (a)Prism/Inflation layer and (b) Zoomed View

3.3 SIMULATION: The paper focuses on the Dimple effect which we observe in Golf Balls. A moving object has a high-pressure area on its front side. Air flows smoothly over the contours of the front side and eventually separates from the object toward the back side. A moving object also leaves behind a turbulent wake region where the air flow is fluctuating or agitated, resulting in lower pressure behind it. The size of the wake affects the amount of drag on the object. By applying dimples on a car, we can create a thin turbulent boundary layer of air that clings to the car’s surface. This allows the smoothly flowing air to follow the car’s surface a little farther around the back side of the car, thereby decreasing the size of the wake. Modelling a CFD simulation is the most important task when analysing the aerodynamic performance of the car. Any mistake made during modelling can impact the outcome results. The analysis was divided in three parts:

Pre-Processor- In Design Modeler first step was to create an Enclosure (Wind Tunnel), using the feature inside geometry a block was created of the size (1m x 1m x 1m). After that, using the Boolean feature the car surface was subtracted from the box (Wind Tunnel) by selecting main body as block and target body as Car’s body. The front face (z-axis) was named as Inlet and opposite to that as Outlet and the remaining face was named as Wall.

ANSYS uses Newton-Raphson method for predicting the results at each iterations and check whether the result is converged or not. Converging the solution to steady state ensures that there won’t be any change in results when we run the same simulation with additional 50 iterations. Here, we attained convergence through meshing independence study. This refers to the smallest of the elements required in the model to ensure that the results of the analysis were not affected by changing the size of the mesh. The criterion set to meet are the Residual RMS Error values should be at least 10^{-4} and overall imbalance in the domain will be less than 1% for all variables. We ran initial simulation with initial mesh and monitored whether our points of interest reached steady solution or not. We kept on increasing the mesh refinement globally with the criterion, until the result was finally converged and no more changes in values obtained with further refinement.

Solution was completed in ANSYS Fluent (Fluid flow Fluent) general we selected Pressure based type solver, absolute velocity and steady time. We selected physics as viscous (Laminar) model and medium material as air having density 1.225 kg/m^3 at temperature 300 kelvin. After this the boundary condition was created setting Inlet (Velocity Inlet) at different speed for this paper, we have selected (60 Kmph, 100 Kmph, 120 Kmph and 150 Kmph) and Outlet as Pressure Outlet with zero-gauge pressure. As the boundary condition was generated, then taking the reference value from Inlet. We created the Drag force report in the option report definition. In Monitor option we decreased the value of residual to $1e^{-06}$ for better accuracy of the solution. Now moving to initialization method, we selected Standard



Initialization computing from inlet having initial values (0 Pa pressure, x- axis velocity 0 m/s, y-axis velocity 0 m/s and z-axis velocity -27.778 m/s for case 1 & so on) and selected Initialize. For Calculation we selected timescale factor as default (1 sec) and setting the number of iterations as 1000 with 1 profile and 1 reporting interval. Then to Results, in report option we generated the projected area of the Car in z-axis direction. Thus, completing the step solution.

Post Processing was completed in CFD-Post. We created an iso-surface at the centre of car parallel to walls. Pressure contour was created on the solid body and Velocity contour was created on the iso-surface. 3-D streamline was generated from the inlet as velocity component to observe the particle movement. Using the function calculator, we generated the Force on the Solid body (Car).All the forces were generated on function calculator.

Table 3. Forces Direction in Co-ordinate System

Global X-axis as	<i>Lateral Force</i>
Global Y-axis as	<i>Lift Force</i>
Global Z-axis as	<i>Drag Force</i>

To Calculate the Drag coefficient, we used the formula:

Equation (i)

$$c_d = \frac{2F_d}{\rho u^2 A}$$

where:

F_d is the Drag Force (the force component in the direction of flow of velocity)

ρ is density of fluid (Air)

U is the flow speed of the object (Car)

A is the frontal area (Area projected in the direction of flow of velocity).

Similarly, the coefficient of lift calculated using same formula, only difference is that the force will be lift force (F_L).

IV. EXPERIMENTAL RESULTS

4.1 AERODYNAMIC STUDY ON CAR:

In order the study the Dimple Effect, the simulation was performed both on normal car and its equivalent with varying dimple depths. The various depth considered for the sake of study are 6 mm, 10 mm & 14 mm keeping fixed diameter of dimple as 20 mm. In order to understand the fluid interactions with the car,the coefficient of forces acting on the car were calculated using equation (i) and values were tabulated against diverse speeds as mentioned in Table 4.

Table 4. Coefficient of Forces Acting on the Car

Coefficient	C_d	C_l	C_d	C_l	C_d	C_l	C_d	C_l
Speed	60 kmph		100 kmph		120 kmph		150 kmph	
Case 1 (Without Dimple)	0.5054	0.139	0.5009	0.1084	0.4949	0.1017	0.497	0.110
Case 2 (depth=6mm and dia. 20mm)	0.485	0.187	0.476	0.1673	0.4864	0.2404	0.4753	0.165
Case 3 (depth=10mm and dia. 20mm)	0.489	0.243	0.4702	0.2041	0.4808	0.237	0.4768	0.207
Case 4 (depth=14mm and dia. 20mm)	0.4579	0.119	0.4651	0.1299	0.4548	0.1413	0.4671	0.143

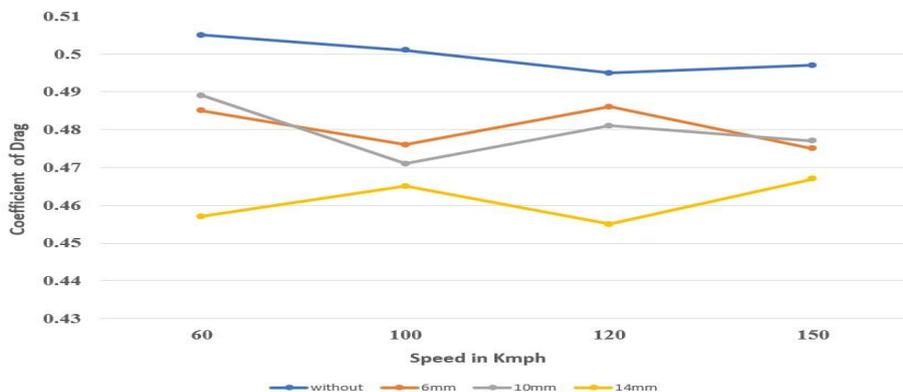


Fig. 4 Graph plotted for Coefficient of Drag for four cases at different speeds

It is clearly noticeable that significant changes were observed particularly in case 4 when it is compared with that of undimpled car.

➤ Case 1: Normal car (without dimple)

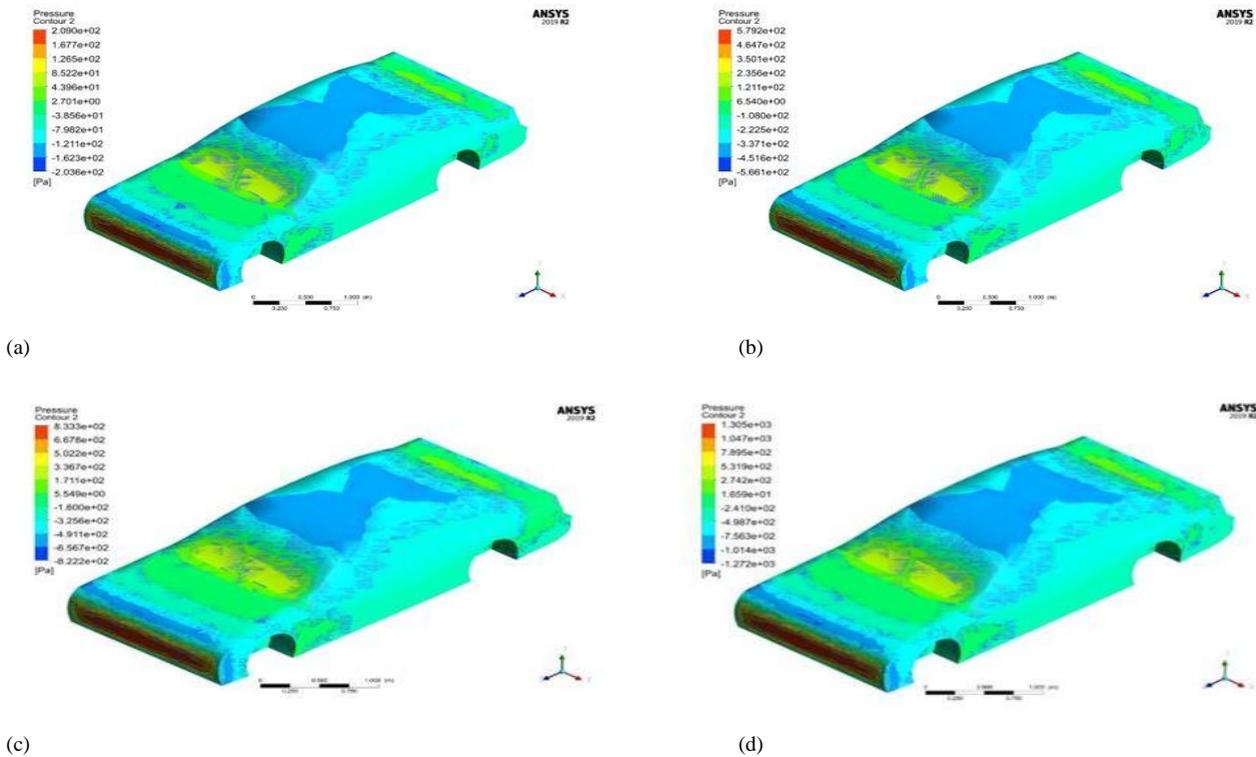


Fig. 5 Pressure Contours of Case 1 at speeds (a)60 kmph, (b)100 kmph, (c)120 kmph and (d)150 kmph

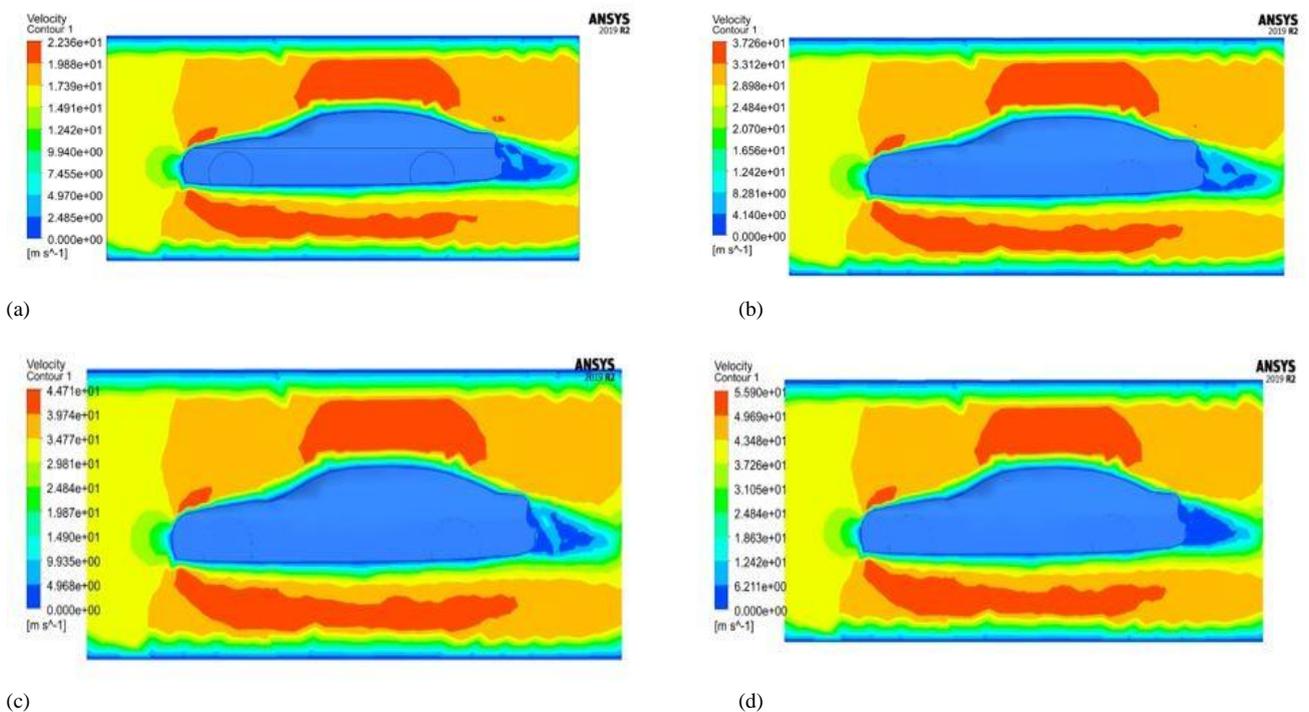


Fig. 6 Velocity Contours of Case 1 at speeds (a)60 kmph, (b)100 kmph, (c)120 kmph and (d)150 kmph

➤ Case 4: Dimpled Car (Depth = 14 mm and Diameter = 20 mm)

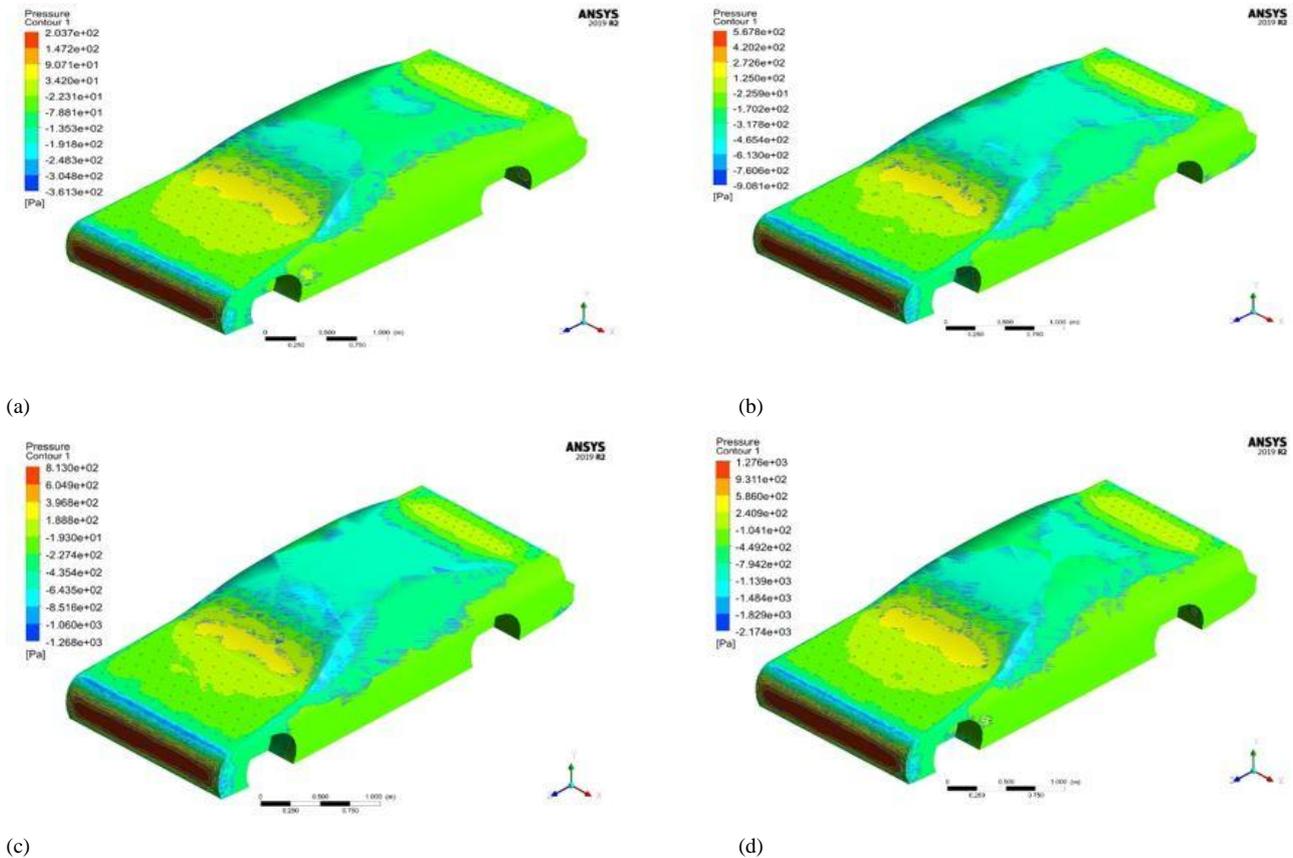


Fig. 7 Pressure Contours of Case 4 at speeds (a)60 kmph, (b)100 kmph, (c)120 kmph and (d)150 kmph

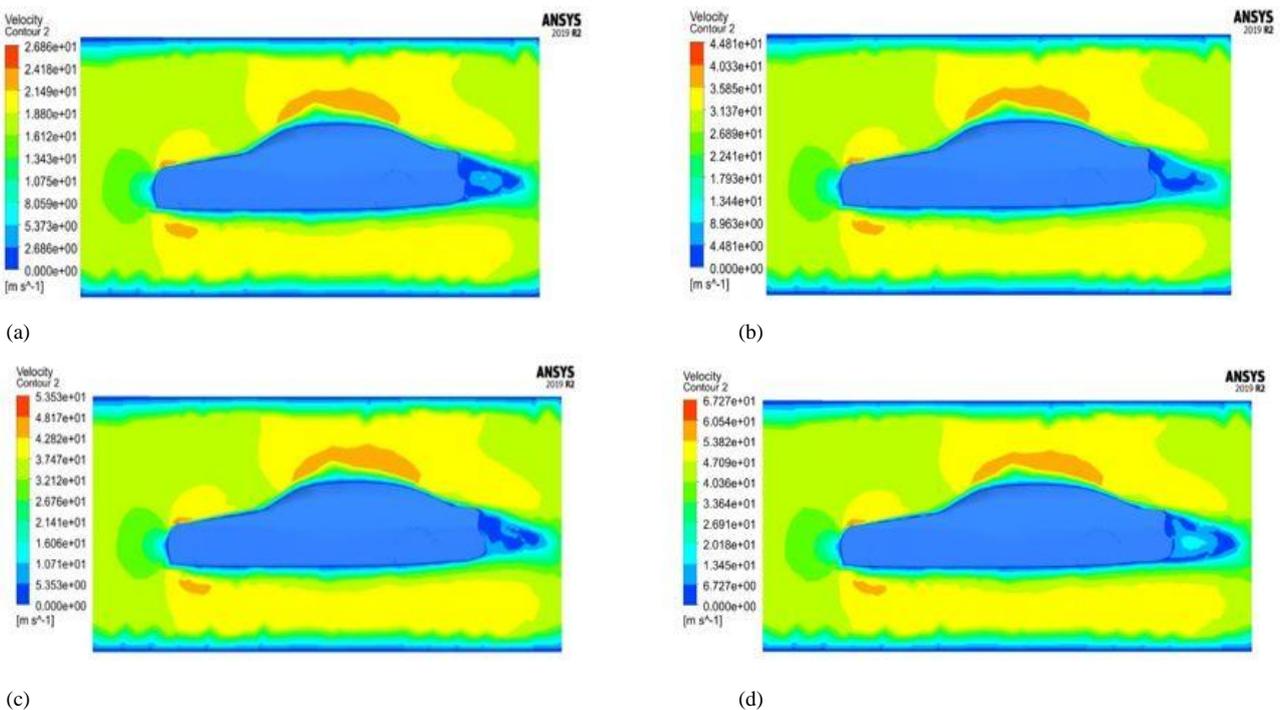


Fig. 8 Velocity Contours of Case 4 at speeds (a)60 kmph, (b)100 kmph, (c)120 kmph and (d)150 kmph

From the figures above, it is observed that, as the flow approaches the front of the car with low velocity and then accelerates above the car. The acceleration of the flow under the car generates a low pressure and thus increasing the down force. Red area in pressure contour indicates high pressure region and blue area as low pressure. The frontal area of car shows high pressure area, therefore, a considerable amount of drag force is observed. As we know the less pressure difference between upstream and downstream on the car means less drag force. The fast-moving air does not enter in the region. Due to this, the flow velocity will be slower in this region and similarly the pressure will be higher, thereby decreasing the overall pressure difference and hence the pressure drag.

4.2 AERODYNAMIC STUDY ON SPOILER AND SIDE MIRROR:

Side mirrors and spoiler do impact aerodynamics and fuel economy of vehicles. Through modern automobile manufacturers expedient to replacing the mirror with high definition cameras. However, large amount of commercial vehicle producers in India are still opting for side mirrors. Exterior side view mirrors increase total aerodynamic drag by an average of 2 to 4 percent which doesn't affect much in aerodynamics but is a major concern in high speed cars. The spoiler is also not common except for the performance cars. Here, we implement surface roughness as dimples in an attempt to reduce drag.

We designed the cad model of the side mirror with dimensions in reference to conventional sedan in SOLIDWORKS having frontal area of about $8.2437e^{-3} m^2$ and spoiler having frontal area of $0.8994 m^2$. We induced dimples of varying depth ratio in different regions of the model. The fluid interactions were studied in ANSYS 19.1 Fluent. The boundary box was of uniform dimension of 0.5 m on all sides. The meshing method used is quadratic tetrahedron with resolution of 7 in CFD physics preference. For the sake of simplicity, the inlet velocity was kept constant at 100 kmph (27.778 m/s) while the pressure at outlet 0 Pa. Standard Initialization was done with second order upwind momentum & turbulent kinetic energy. The simulation was run at 300 iterations at time interval of 1. Coefficient of lift and drag were calculation from *equation (i)* using the forces obtained in mesh calculator in Results. Comparison was made between normal mirror with smooth surface and equivalent mirror with dimples on surface. The coefficient for the respective model is shown the table.

Table 5. Coefficient of Forces Acting on Side Mirror and Spoiler at 100 kmph

Model	Coefficient of Drag	Coefficient of Lift
Normal Side Mirror	0.6233	0.3059
Dimpled Side Mirror	0.6076	0.3378
Percentage of Drag Reduced	2.52 %	
Normal Spoiler	0.5734	2.4189
Dimpled Spoiler	0.5452	2.7251
Percentage of Drag Reduced	4.93 %	

From the Table 5, we can observe that there is reduction in coefficient of drag which indicate lesser drag force. So we can say that little modifications bit in car parts play a major role in reduction of drag.

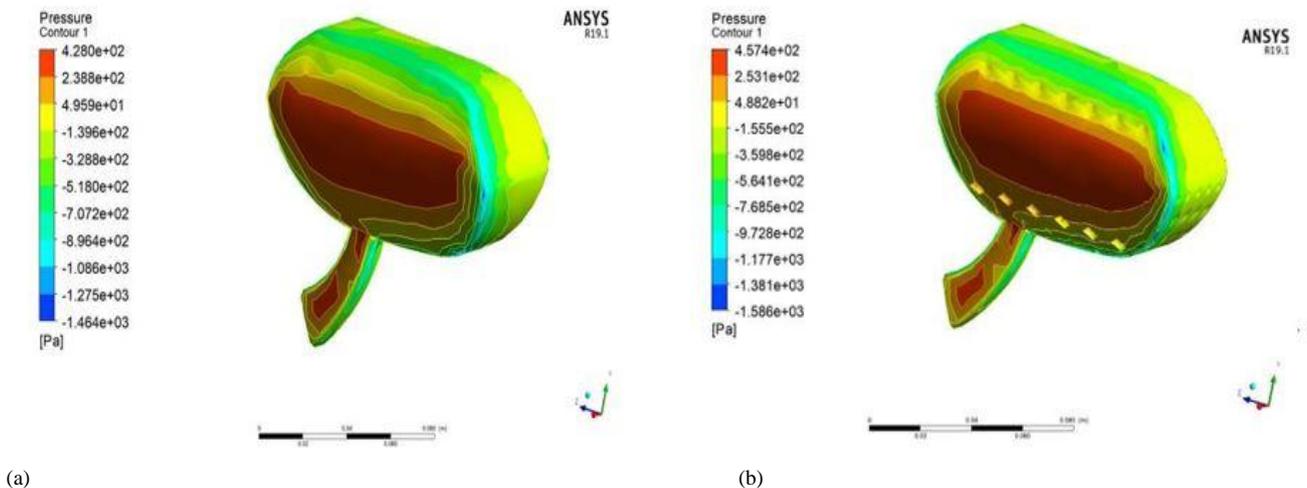


Fig. 9 Pressure Contours on Side Mirror at 100 kmph having surface (a)Smooth and (b)Dimpled

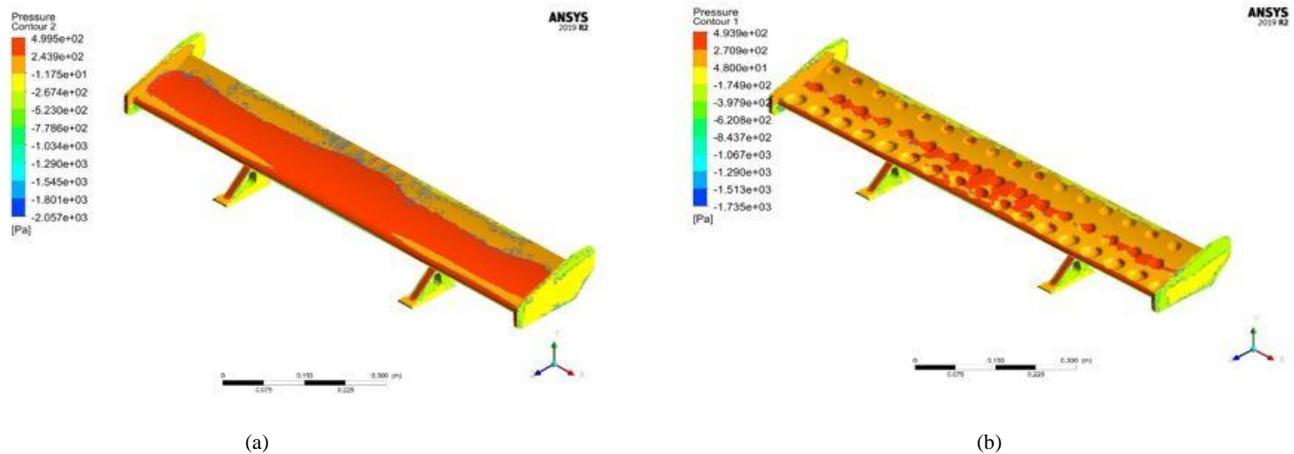


Fig. 10 Pressure Contours on spoiler at 100 kmph having surface (a)Smooth and (b)Dimpled

V. CONCLUSION

The purpose of the project was to investigate the Dimple Effect on the surface of the car. The area of interest was the fountainhead of maximum drag on the vehicle. Different components of the car were modified in computer aided designing software. Surface roughness induced on the surface with range of depths depending upon the area of interest. The meshing methodology used was tetrahedron along with inflation layer close to the surface. The components were simulated in fluid domain and its interaction with the surface was studied. The computational fluid dynamics provided visual images and numerical data on aforementioned simulation were in the favour of the research. The numerical study shows that the introduction of dimple on hood and trunk, the coefficient of drag was reduced. The most significant change was obtained in Case 4 where the dimple depth was 14mm at 60 kmph, the coefficient of drag was reduced by 9.4 %. The basic conclusion was drawn that different dimple depths reduces C_d at different percentages when it is dealt with different speeds. We have also experiment at other components like spoilers and side mirror at fixed speed of 100 kmph. We observed the reduction in drag with dimpled surface, however substantially low. In spoilers, the C_d was reduced by 4.93 % while in side mirror, the C_d was reduced by 2.52 %. In table 4, we observed an anomaly, where a sudden decrease in the coefficient of lift was seen in case 4 when compared to previous cases.

This computational data showed that the presence of dimples changes the forces acting on the vehicles. The reduction in directly proportional to the reduction in fuel consumption. This improvement in design might not bring considerable change in fuel economy of commercial vehicles however, it can be significant in high speed cars. So, by keeping these as references for comparison, design improvement can be done on Indian cars to make them more aerodynamically viable. This is because more importance was laid on quantity rather than quality and customers never get satisfied even after investing huge amount of money. Moreover, due to their higher fuel consumption, they have higher emissions in air. Our design will definitely result in the car being significantly more fuel efficient as well as faster and more stable. However, there is always a scope for improvement & region on the car that are yet to be optimized for performance.

REFERENCES

- [1] Shobhit Senger and S.D. Rahul Bharadwaj, "Aerodynamics Design of F1 and Normal Cars and Their Effect on Performance", International Review of Applied Engineering Research, Volume 4, pp. 363-370, April 2014.
- [2] Gavin Dias, Nisha R. Tiwari, Joju John Varghese, Graham Koyeerath, "Aerodynamic Analysis of a Car for Reducing Drag Force", Journal of Mechanical and Civil Engineering (IOSR-JMCE), Volume 13, Issue 3, pp 114-118, May 2016.
- [3] Breno Ventorim De Tassis, "Vehicular aerodynamics and performance optimization in TARF-LCV", Universidade Federal do Espirito Santo, VITORIA, 2016.
- [4] Harun Chowdhury, Bavin Loganathan, Yujie Wag, Israt Mustary and Firoz Alam, "A Study of Dimple Characteristics on Golf Ball Drag", 11th Conference of the International Engineering Association, ISEA 2016.



- [5] Praveen Padagannavar and Manohara Bheemanna, “Automotive Computational Fluid Dynamics Simulation of a Car using ANSYS CFX”, International Journal of Mechanical Engineering and Technology, Volume 7, Issue 2, pp. 91-104, April 2016.
- [6] ChanchanRajsinh B. and ThundilKarippa Raj R, “Numerical Investigation of External Flow around the Armed Reference Body using computational Fluid Dynamics”, research Journal of Recent Science, Volume 9, September 2012.
- [7] Wolf-Henrich Hucho, “Aerodynamics of Road Vehicles”, 4th Revised Edition, Society of Automotive Engineers, U.S, ISBN: 978-0768000290, 1998.
- [8] M. Cable, “An Evaluation of Turbulence Models for the Numerical Study of Forced and Natural Convective Flow in Atria”, 2009.
- [9] Ambepasad. S. Kushwaha, “Comparative Study of Rectangular, Trapezoidal and Parabolic Shaped Finned Heat Sink”, e-ISSN:2278-1684, volume 5. Issue 6, March 2013.
- [10] Katz.J and Robert Bentley, “Race car aerodynamics: Designing for speed”, 1995.
- [11] Rakesh Jaiswal, Anupam Raj Jha, Anush Karki, Debayan Das, Pawan Jaiswal, Saurav Rajgadia, Ankit Basnet and Rabindra Nath Barman, “Structural and Thermal Analysis of Disc Brake Using Solidworks and Ansys”, International Journal of Mechanical Engineering and Technology, pp 114-122,2015.