

# AERODYNAMIC DRAG REDUCTION IN CAR

<sup>1</sup>Oliver Petric Stephen, <sup>2</sup>Swati Prajapati, <sup>3</sup>Noman Kasmani

<sup>1</sup>M.Tech, <sup>2</sup>Assistant Professor, <sup>3</sup>Assistant Professor  
Department of Automobile,

Parul institute of Engineering and Technology, Parul University, Vadodara, Gujarat, India.

**Abstract:** Presently a-days the decrease of drag coefficient is turning into a significant test for all the Car makers. The outside styling and efficiently effective plan for decrease of motor burden which reflects in the decrease of fuel utilization are the two basic variables for a fruitful activity in the serious world. The streamlined impacts assume a job in diminishing the coefficient of drag of the vehicle, which may benefits for expanding the eco-friendliness. The outside structure of Car body is useful in lessening the drag coefficient which brings about diminishing the heap on the vehicle and prompts low fuel utilization. Despite the fact that the different strategies are found by specialists for improving vehicle execution and its soundness still we are needing further improvement. The fundamental reason for existing is to propose an idea vehicle model which will confront little drag power and furthermore to diminish the comparing drag coefficient. The object is served by breaking down various states of the vehicle and consolidating important changes to it to accomplish the ideal result. The Two model vehicle body has been demonstrated and examined by utilizing CFD to lessen the drag power. These are to be specific Baseline display and Modify model. Benchmark Model is existing vehicle display and Modify model is alteration of existing vehicle. Model 2 is to adjust and examine by utilizing CFD to decrease the drag power, which brings about expanded execution.

Index Terms - ansys, drag, drag power, streamlined, vehicle body.

## I. Introduction

In the contemporary world, security, execution, and solace of street vehicles are basic parameters. At the equivalent time, low fuel utilization is of pivotal significance for creating nations like India since it underpins the economy of the nation. In India, the majority of the vehicles are oil based and the normal fuel utilization of oil is 4.1 million barrel for each day and consistently fuel utilization is expanding at 5%. Henceforth specialists are attempting to lessen the fuel utilization of the street vehicle, even a limited quantity of decrease will be an extraordinary accomplishment. Because of the worldwide environmental difference in today the car business puts fundamentally in decreasing the fuel utilization of their items. The organizations are, from the outset hand, pushed by the administrations and enactments to decrease the discharges of their vehicles. There are numerous ways to deal with diminish the discharges and every one of them are of significance for the car business. However, due to the crisis in fuel price and environmental impacts from exhaust emission, the researchers have focused their concentration to design the cars aerodynamically in order to reduce the fuel consumption as well as lowering the pollution.

## II. Vehicle Dynamics

Optimal design is a part of liquid elements worried about examining the movement of air, especially when it cooperates with a moving item. Optimal design is additionally a subfield gas elements, with much hypothesis imparted to liquid dynamics. Aerodynamics is frequently utilized synonymously with gas elements, with the distinction being that gas elements applies to all gases. Understanding the movement of air around an item empowers the count of powers and minutes following up on the article

## III. Powers And Moment On Vehicle

At the point when the vehicle is moving at an impressive speed, the air disregarding it forces different powers and minute on the vehicle. The vertical power following up on the body makes the vehicle get lifted in air as applied the positive way, though it can bring about over the top wheel down power in the event that it is applied negative way. Specialists attempt to hold this incentive to a necessary cutoff to maintain a strategic distance from abundance down power or lift. Aerodynamic drag power is the power following up on the vehicle body opposing its forward motion. This power is a significant factor to be considered while planning the outside body of the vehicle, since it covers about 65% of the absolute power following up on the total body. Crosswinds produce a side power on a vehicle that demonstrations at the center of the wheelbase, and when the crosswinds don't act at the center of the wheelbase a yawing minute is created

## IV. Problem Statement

This paper centers understanding the stream around the car vehicle model. The streamlined delay the vehicle outer body can be utilized by streamlining the vehicle geometry or by utilizing additional items which can be joined to vehicle's outside body. The vehicles execution relies upon the resistive powers following up on the vehicle's outside body surface because of the progression of air around the vehicle body. There are two sorts of powers that follow up on the vehicle the streamlined drag streamlined Lift. Investigation is done on the current model and the adjusted model utilizing CFD Fluent 14 and the outcomes acquired for the two models are looked at and the best model is recommended. The vehicle is displayed utilizing Solid works.

### 1.1 Objective

- To model an existing car body, as a baseline model using solid works CAD modelling software.
- To perform the flow analysis on the baseline model using CFD tool fluent.
- To design new model of car such that drag force is reduced.
- Perform flow analysis on the new models.
- Reduction of Fuel Consumption

- More Favourable Comfort Characteristics (Mud Deposition On Body, Noise, Ventilating And Cooling Of Passenger Compartment)
- Improvement of Driving Characteristics (Stability, Handling, Traffic Safety)

### 1.2 Data and Sources of Data

A R Barnard et al. (2014). And Hucho and Sovran (1993) emphasized various aspects of drag reduction techniques by modifying the underbody shape and redirected the exhaust gas to the rear side of the vehicle. Hence drag reduction up to 22% was achieved by underbody modification techniques, and redirection of exhaust gas reduces up to 9% drag. Shankar G *et al.* (2018). Carried out both numerical and experimental studies on sedan car model using active airflow modification technique for reducing aerodynamic drag coefficient and lift coefficient

### 1.3 Types of Rear End Car Body

Cars are classified according to their design, comfort, engine, capacity, aerodynamic flow, etc. Depending on the aerodynamic flow over a rear body, the cars are categorized into the hatchback, sedan, square back, and fastback. A car with a rear end slope angle ranging from  $50^{\circ}$  to  $90^{\circ}$  is called a square back car. This type of car's rear side has almost constant low negative pressure wake region. When rear slope angle is reduced to  $22^{\circ}$  or less, the body profile is known as fastback car, which has lower drag coefficient because of airflow over the roof and rear downward slope surface air stream remains attached to the body like streamline body. When rear slope angle ranges from  $50^{\circ}$  to  $22^{\circ}$  then it is known as hatchback car. This type of car's rear side has more low negative pressure wake region compared to fastback and sedan car. Notchback or sedan car has a stepped rear end body. In such cars, the rear window is inclined downward to meet horizontal rearward extending boot. Among all the types of rear end car body, hatchback model was preferred by most of the people due to its lower cost. While moving, airflow on the hatchback car creates more wake region which arises a large amount of drag at its rear end when compared to other models. In computational approach, the car models are designed by the Solid works 2018. Then Ansys Flow Simulation is applied for simulation purpose. Different result viewing options related to Flow Simulation is applied to analyze the models and also to find the best one among them.

## I. RESEARCH METHODOLOGY

Most importantly we make a 3d model in solidworks software. Two perspectives on model are demonstrated as follows. Here we use solidworks programming for demonstrating vehicle body.

### 2.1 Model Development

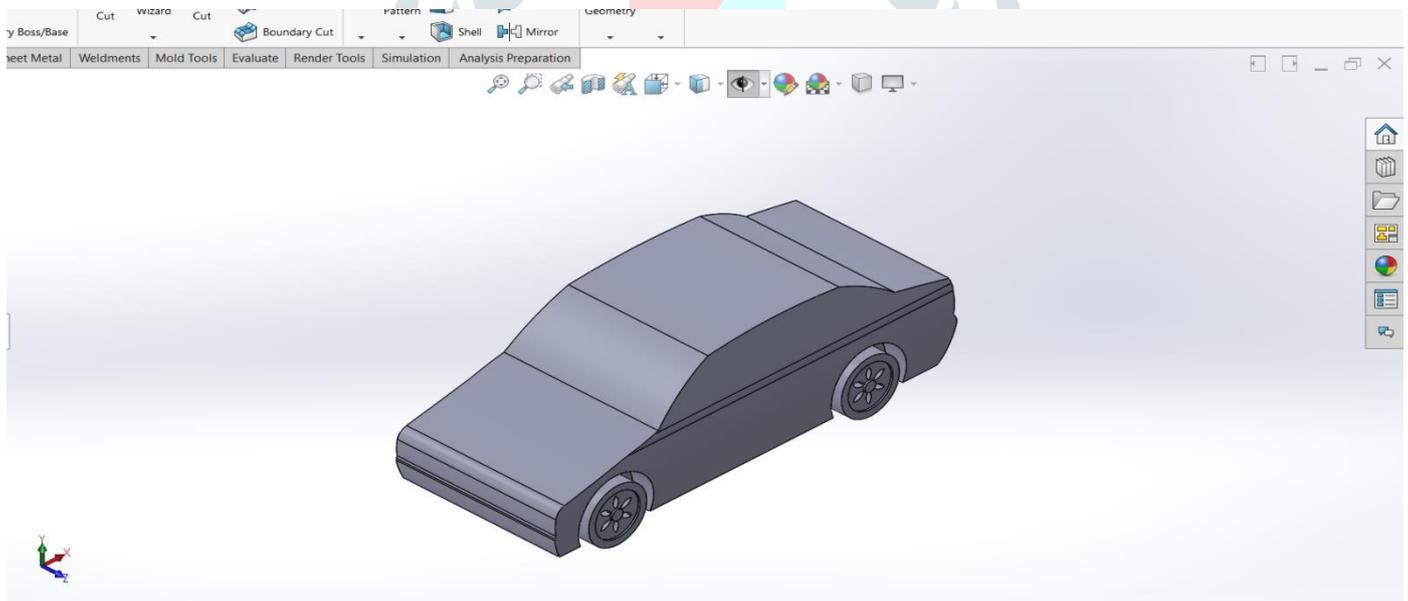


Fig 1:3d Model (front view)

### 2.2 About Ansys Software

ANSYS 14.5 conveys creative, emotional reproduction innovation propels in each significant Physics discipline, alongside upgrades in figuring velocity and improvements to empowering advancements, for example, geometry taking care of, cross section and post-preparing

### 2.3 Velocity distribution for baseline model at 100 km/hr speed

Here, first we take above baseline model and run the CFD analysis at different speed. After analysis we got velocity distribution, pressure distribution and streamline distribution for given model. The below results show velocity distribution for baseline model at 100 km/hr speed.

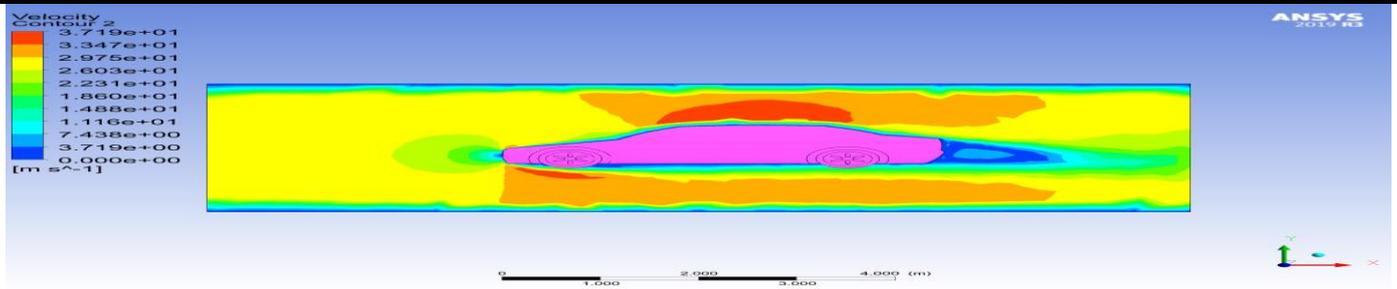


Fig 2: Velocity Distribution(100km/hr)

**2.4 Pressure distribution for baseline model at 100 km/hr speed**

The below figure shows the pressure distribution for baseline model at 100 km/hr speed. In Figure, the light green region indicates atmospheric pressure and red region is the high pressure region.

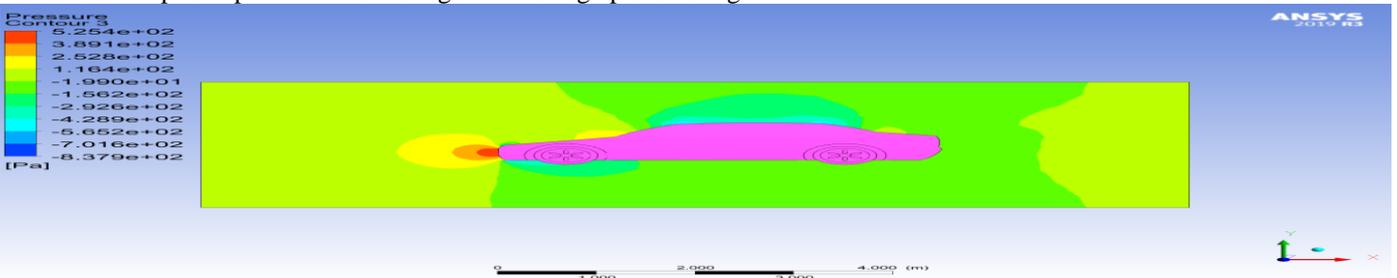


Fig 3: Pressure distribution(100km/hr)

**2.5 Streamline distribution for baseline model at 100 km/hr speed**

The below figure shows the streamline distribution for baseline model at 100 km/hr speed. Figure shows that streamline flows are disorganized for the shape of the car which should have to be smooth. So the regions where the stream lines get obstacles, is being modified. Specifically, the sharp edges of the car is modified to control the smooth stream line flow.

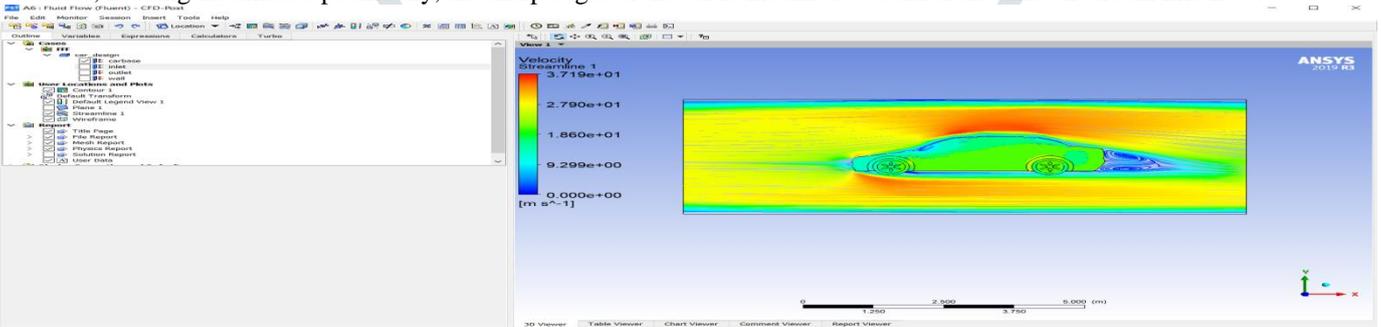


Fig 4: Streamline distribution(100km/hr)

**2.6 Value of  $C_d$  for baseline model at 100 km/hr speed**

As shown in below figure, the value of coefficient of drag is 0.51 for baseline model. The manual calculation can be done as per formula for drag force. In our case the simulation value of  $C_d$  and computation value of  $C_d$  is same.

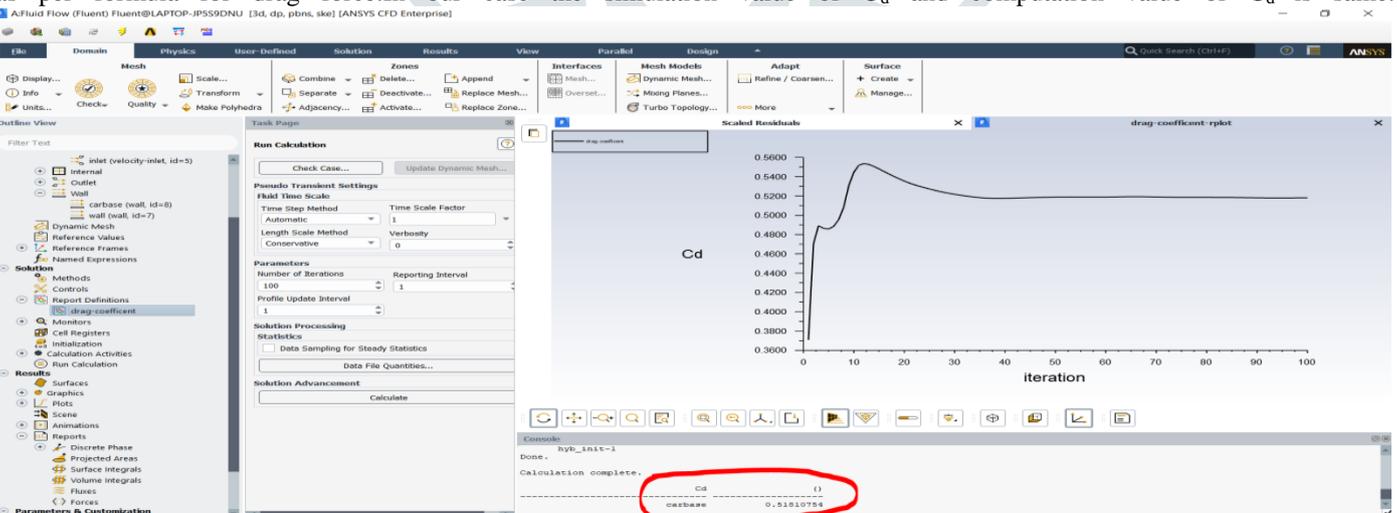


Fig 5: Value of  $C_d$  for baseline model(100km/hr)

**2.7 Value of  $C_d$  for baseline model at 60 km/hr speed**

As per above process we can do the same analysis for different speed. In below figure we get value of  $C_d$  for 60 km/hr for baseline model.



Fig 6: Value of  $C_d$  for baseline model (60km/hr)

### 2.8 Value of $C_d$ for baseline model at 120 km/hr speed

As per above process, we can do the same analysis for different speed. In below figure the speed of the vehicle is taken as 120 km/hr.

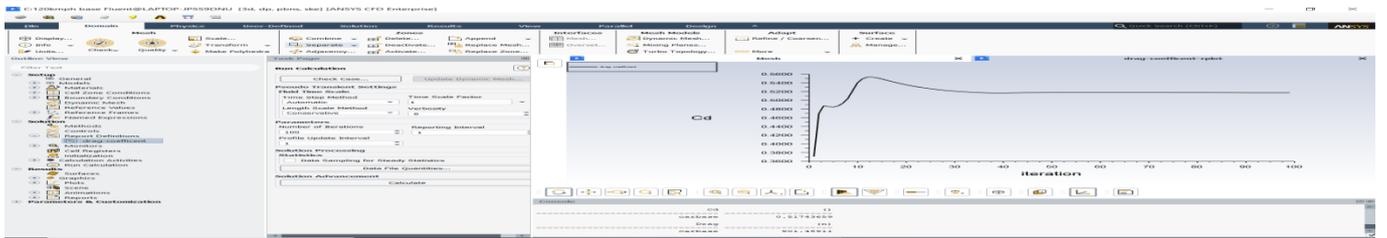


Fig 7: Value of  $C_d$  for baseline model (120km/hr)

### 2.9 Design of Modified car model

In case of modified car, we have done the changes in targeted area one by one. First we change in front line part. Then step by step we have done the changes on Rear angle, Tail angle, wide screen Angle. After changes on different spots we have done the analysis of car body.

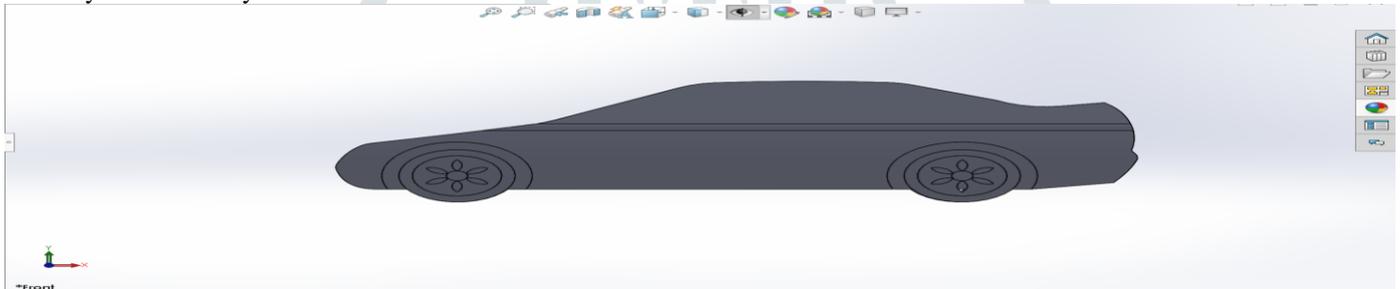


Fig 8: Modified model after change in design

### 2.10 Value of $C_d$ for modified model at 100 km/hr speed

As shown in below figure, the value of coefficient of drag is 0.41 for modified model. The manual calculation can be done as per formula for drag force. In our case the simulation value of  $C_d$  and computation value of  $C_d$  is same.

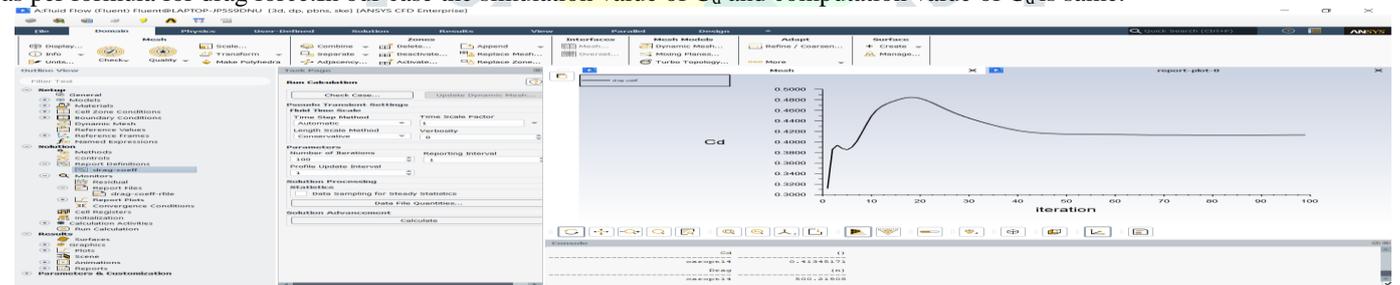


Fig 9: Value of  $C_d$  for modified model (100km/hr)

### 2.11 Value of $C_d$ for modified model at 60 km/hr speed .

As per above process we can do the same analysis for different speed. In below figure we get value of  $C_d$  for 60 km/hr for modified model.

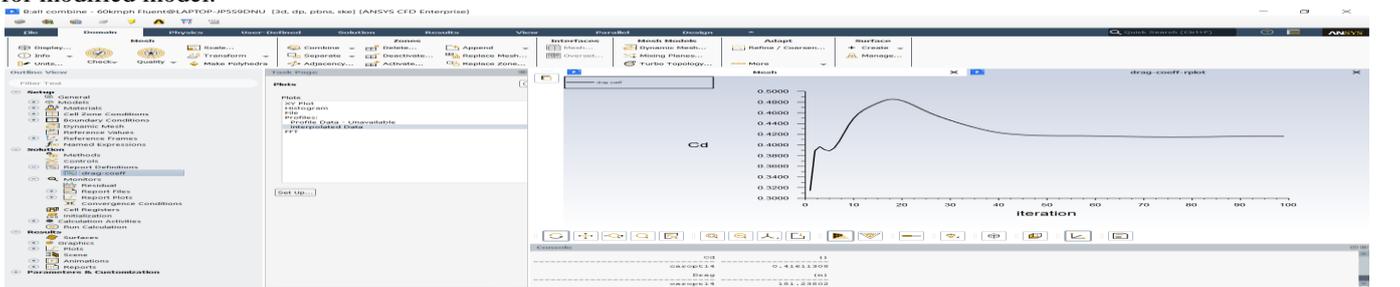


Fig 10: Value of  $C_d$  for modified model (60km/hr)

### 2.12 Value of $C_d$ for baseline model at 120 km/hr speed

As per above process, we can do the same analysis for different speed. In below figure the speed of the vehicle is taken as 120 km/hr.

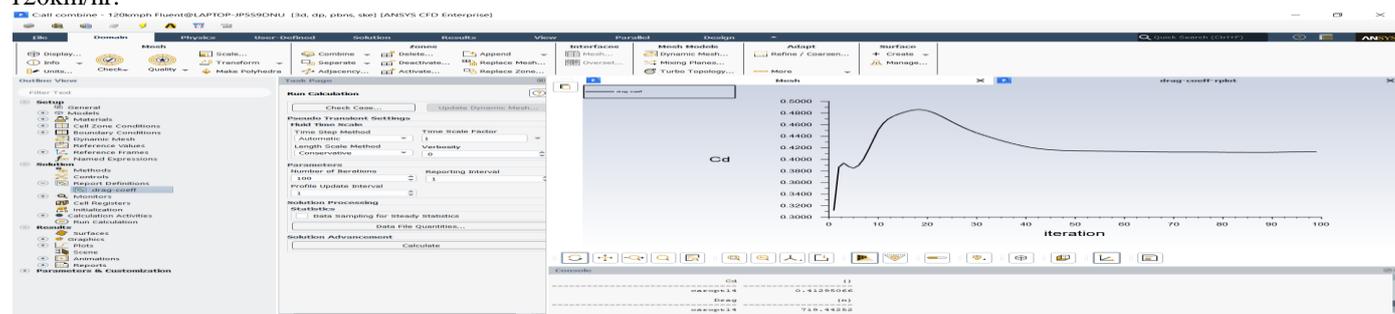


Fig 11: Value of  $C_d$  for modified model (120km/hr)

## IV. RESULTS AND DISCUSSION

### 4.1 Results of two different models

Table 4.1: Value of  $C_d$  for baseline and modified model

BASE LINE MODEL		
SPEED	DRAG FORCE	COEFFICIENT OF DRAG
60	226.62	0.52
100	626.84	0.51
120	901.48	0.51
MODIFIED MODEL		
SPEED	DRAG FORCE	COEFFICIENT OF DRAG
60	181.86	0.416
100	500.21	0.41
120	719.21	0.41

Result Shown on above table 4.1 are for two prototypes. One is for Baseline and another is for modified model. The value of drag force and  $C_d$  is shown for different Speeds.

In the process of redesigning, exterior styling with improved aerodynamics of existing model is done and then, a detailed computational analysis has been done. The Two car body has been modeled for performing numerical analysis using CFD software. The boundary conditions are assumed as for inlet zone velocity of 100 km/hr and outlet zone pressure is set to zero Pascal. Velocity given to the fluent analysis is 100kmph. The resultant drag Coefficient of the baseline model is 0.51 and the modified model we get is 0.41. So that The Coefficient of drag is reduced to approximately 10%. Reduction in Drag Force Improves the performance and Reduce the fuel requirement. After going through the research work carried out in the field of CFD analysis of automobile car for aerodynamic drag reduction, it has been concluded that there are many thrust areas (shape optimization, meshing type and boundary condition) which affect the CFD analysis of drag coefficient. CFD approach is arguably the way of the future in promising faster turnaround simulation time with cheaper running cost. At the same time, it offers superior capability than the experimental approach in terms of post processing of data and graphical representation of flow analysis. Provided CFD approach is to be reliable and can produce quantifiable results with a high degree of accuracy, the role of wind and aero acoustics tunnel testing in the future might be reduced for only validation purposes. However, more work should be done on improving CFD technological advancement for a more efficient and easier way to tackle problems associated with aerodynamics and aero acoustics

### Acknowledgment

The submission of the report gives me opportunity to all those who have helped me in completion of project work. I hereby would like to acknowledge the in valuable support of my project guide Prof. Swati Prajapati, Department of Mechanical Engineering, PIET, Vadodara and Prof. Noman Mohammed, Department of Mechanical Engineering, PIET, Vadodara. They helped me through their invaluable suggestion and support throughout the project and have been an unending source of inspiration to me. I also thank my other faculty members who have been directly or indirectly helped me whenever and wherever it was required by me. I owe my special thanks to my friends for supporting and encouraging me in all possible ways and also all faculty members of Mechanical Engineering Department for the knowledge they imparted me throughout my project.

### REFERENCES

- [1] Aerodynamic drag reduction for low carbon vehicles by J P Howell
- [2] Road vehicle aerodynamics had been summarized by A R Barnard et al. (2014). And Hucho and Sovran
- [3] A parametric analysis of aerodynamic force in a simplified body by Mahmud Khaled, Hicham Hage, Fabien Harambat.