International Journal of Current Engineering and Technology
E-ISSN 2277 – 4106, P-ISSN 2347 – 5161
©2020 INPRESSCO®, All Rights Reserved
Available at http://inpressco.com/category/ijcet

*Research Article*

# CFD Analysis on the Effect of the Change in Profile Geometry on Drag Reduction of a Sedan Car

**Ganesh Kumar[#], Amit Sahay[^*] and Kamlesh Shivvedi[c]**

Dept. of Mechanical Engineering, Mittal Institute of Technology, Bhopal, Madhya Pradesh, India

Received 10 July 2020, Accepted 03 Sept 2020, Available online 04 Sept 2020, **Vol.10, No.5 (Sept/Oct 2020)**


### Abstract

*This study is a 3-dimensional CFD analysis of external aerodynamics of a generic car model. The aim of this study was to visualize the flow field and examine the effect of modification of some geometrical parameter on the drag coefficient of the car body. The reduction in drag allows the vehicle to operate at lower fuel consumption which eventually results in fuel economy and lower pollution. Also artificially guided air maintains stability of vehicles. Therefore, flow field analysis is important for aerodynamic analysis of a car body. The results obtained is compared and validated with existing experimental data and found to be deviated by 5.5%. The two specific name of geometrical parameter for this study which is modified are front bumper angle and hood radius. The three angles are taken into consideration for front bumper angle as 10-degree, 15-degree and 20-degree and at each front bumper angle three hood fillet radius are 200mm, 250mm and 300 mm respectively. It is found in the study that increasing the angle of inclination of front bumper angle decrease the drag coefficient and also increasing the hood fillet radius decreases the drag coefficient. The minimum drag coefficient in the study is found to be 0.3939 at front bumper angle 15 degree and hood fillet radius 300mm. The maximum reduction of the drag coefficient from the base case in which both the parameter is kept at zero-degree angle and zero fillet radius is 6.50%. The flow field suggested that as we increase the inclination angle the frontal projected area is reduced locally and pressure drag is reduced and also giving the fillet at corner the air flow is guided smoothly which also results in reduced drag.*


***Keywords**: Aerodynamic analysis, Drag coefficient, Computational fluid dynamics, Front bumper angle, Hood fillet radius.*

## 1. Introduction

*"Aerodynamics"* is a division of fluid dynamics relate to study the motion of air, mostly when it relates with a moving thing. Aerodynamics is likewise a subfield of gas dynamics, with considerable concept shared by fluid dynamics. By defining a control volume around the flow field, equations for the conservation of mass, energy, and momentum can be defined and used to resolution for the properties. Aerodynamic analysis is done with mathematical study, empirical approximations and wind tunnel testing.

Aerodynamics and its study are mainly divided into two main sub-categories, namely the internal and external aerodynamics. In External aerodynamics we studied the stream around solid things of different shapes. Estimating the drag and lift on an aircraft, the stream of air over a wind turbine blade or the shock waves that form in front of the nose of a rocket, are illustrations of external aerodynamics.

*Corresponding author **Amit Sahay** (ORCID ID: 0000-0002-0038-6957) is working as Associate Professor; **Kamlesh Shivvedi** as Assistant Professor; **Ganesh Kumar** is a Research Scholar
DOI: https://doi.org/10.14741/ijcet/v.10.5.5

On the other hand, internal aerodynamics is the study of stream through ways in solid things. For instance, internal aerodynamics includes the study of the airflow over an air conditioning pipe or through a jet engine.

A **sedan** or **saloon**, is a passenger car in a three-box structure with different segments for engine, passenger, and cargo.

It is occasionally recommended that sedans essential have four doors. However, some sources recommended that a sedan may have two or four doors. In addition, terms for example sedan have been extra lightly interpreted by car producers since 2010.


**Fig. 1:** Spectrum of Task for vehicle Aerodynamics (*Wolf-Heinrich Hucho*)

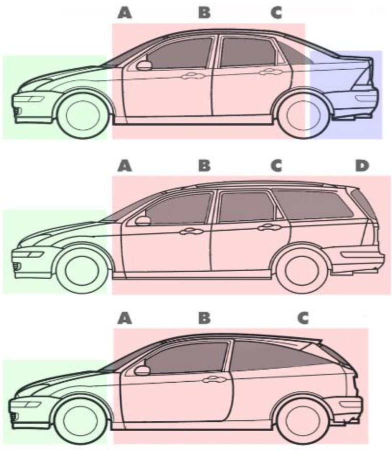



**Fig. 2:** Outlines of sedan, station wagon and hatchback types of same model (a ford focus)

## 2. Factors contributing to flow field around vehicle

The main factors which affect the flow field around the vehicle are the boundary layers, separation of flow field, friction drag and finally the pressure drag.

### *2.1 Boundary layer:*

The Aerodynamics boundary layer was first well-defined by the Aerodynamic engineer 'Ludwig Prandtl' in the conference at Germany. This permits aerodynamicists to shorten the equations of fluid flow by separating the flow field into two areas: one inside the boundary layer and the one external the boundary layer. In this boundary layer around the vehicle, the viscosity is leading and it plays a main role in drag of the vehicle

### *2.2 Separation*

During the flow above the external of the vehicle, there are certain points when the change in velocity comes to stall and the fluid starts flowing in opposite direction. This phenomenon is termed 'Separation' of the fluid flow. This generally happens at the rear part of the automobile.

### *2.3 Friction Drag*

Each wall surface or material has a distinct friction which resists the flow of fluids. Because of molecular friction, a stress acts on all surface of the automobile. The integration of the resultant force element in the free stream direction gives the friction drag. If the separation does not happen, then friction drag is one of the key reasons to cause overall drag.

### *2.4 Pressure drag*

The dull bodies like large size automobile show dissimilar drag features. On the rear part of such vehicles, there is an enormously sudden pressure gradient which leads to the parting of the stream parting in viscous flow. The front portion of the stream field indications great pressure value, while on the rear part stream separates leading to a great suction in the area. As we incorporate the force component produced by such great change in pressure, the resultant is called as 'Pressure Drag'.

## 3. Forces and moment on vehicle

While the vehicle is moving by large speed, there are numerous forces are applied to vehicle in diverse directions. Fig. 3 illustrates the details outline view of the several forces acting on the car body. As presented in the free body figure below, there are six forces acting on the car viz. Rolling Resistance, Drag, Lift, Gravity, Normal and Motor.

Rolling resistance force is because of the tires distorting when get in touch the surface of a road and differs subject on the surface existence driven on. The normal force is the force applied by the road on the vehicle's tires. Because the vehicle is not moving up and about (relative to the road), the amount of the normal forces equals the amount of the force due to gravity in the direction normal to the road. (*Tomar Akhilesh Singh et al*) (*Ansari Abdul Razzaque*)

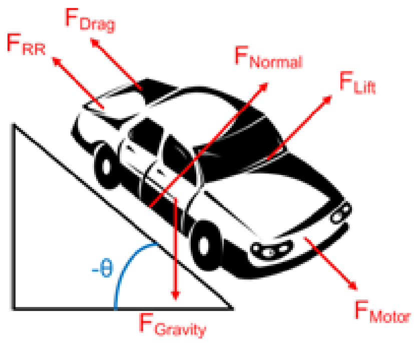



**Fig.3** Forces on the car body

Lift force acting on the car body vertically. This force causes the car to get lifted in air as applied in the positive direction, while it can result in excessive wheel down force if it is applied in negative direction. Engineers try to retain this value to a required limit to avoid additional down force or lift. (*Shelar Dipak et al*) The formula generally used to define this force is written as:

$$C_L = \frac{L}{1/2\rho A V^2}$$

Aerodynamic drag force is the force acting on the vehicle body resisting its forward motion. This force is an important force to be considered while designing the external body of the vehicle, since it covers about 65% of the total force acting on the complete body. The Aerodynamic drag force is calculated by the following formula:

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

Where;
$L$ : Lift force
$C_L$ : Lift coefficient
$D$ : Drag force
$C_D$ : Drag coefficient
A : Frontal area of the vehicle
$\rho$ : Air density
$V$ : Vehicle velocity

## 4. Methodology

The objective of this study is to analyze the effect of some geometrical parameters on the value of drag coefficient. For the study of the above mentioned analysis a 3 dimensional CFD simulation is carried out by using ANSYS FLUENT 2019 R3.

The CAD model, domain of fluid, meshing, applied boundary conditions and initial conditions will be discussed in detail in this chapter.

### *4.1 Model domain and geometry*

ANSYS design Modeller was used for modelling the car. Proton Iswara car model is designed and drag investigation for the identical is carried out. By varying the windscreen and hood angle inclination in the solid model, the coefficient of drag is estimated. The specifications of the car model provided below.

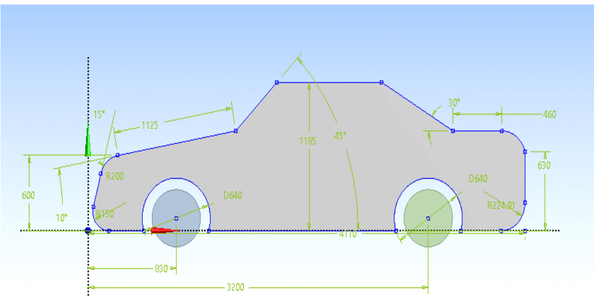


**Fig. 4:** Detail geometry modified car

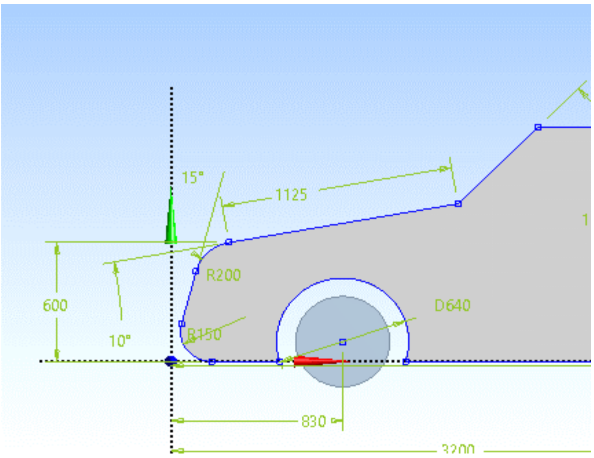



**Fig.5:** Modification of car geometry

**Table 1:** Specifications of Proton Iswara car (*Vignesh S. et al*)

| Specification | Data |
|---|---|
| Maximum Height | 1360 mm |
| Maximum width | 1655 mm |
| Maximum length | 4110 mm |
| Wheelbase | 2380 mm |
| Track (front and rear) | 1406 mm & 1356 mm |
| Gross vehicle weight | 1360 Kg |
| Maximum Output | 62 KN @ 6000 rpm |
| Maximum Torque | 109 Nm @ 4000 rpm |

We are introducing new terminology in car body of this work which front bumper angle and hood fillet radius such as shown in Fig. 5.

### *4.2 Domain Description*

The fluid domain taken for analysis is created in design modeller in ANSYS. The dimensions of the rectangular domain is taken in 5:1 ratio of the respective dimension of the car as shown in Fig. 5 except front dimension which is in the ratio of 3:1. The backside direction of the considered fluid domain is important because the vortices form behind the car when car moves which has significant role in drag and lift and the dynamic steadiness of the car in motion.

In the present analysis the full scale domain is cut by half and the symmetry model is used in CFD analysis in FLUENT. The fluid domain is rectangular and all the faces enclosing the domain except inlet and outlet is taken as wall which is not shown explicitly because FLUENT consider the unnamed outer surfaces as wall by default. All the walls of the enclosing domain is taken stationary in the study.

#### *4.2.1 CAD Models*

The modified model of car body is a cad model of two geometrical parameter modifications which is specific to the current study. The parameters taken are named as front bumper angle and hood fillet radius. The front angle is varied as 10 degree, 15 degree and 20 degree and at each angle three fillet radius of hood is considered as 200mm, 250mm and 300 mm. The name taken in the study is shown in figure for visual interpretation.

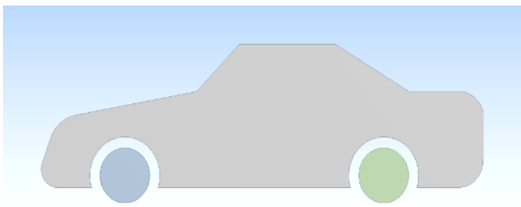


**Fig. 6:** CAD model of car with front bumper angle 20 degree and hood fillet radius of 300 mm

### *4.3 Mesh Generation*

The mesh generated in the domain hybrid mesh and consists of hex, tetra, wedge and pyramid elements. In meshing element size is 0.2 m and for capturing the wall effect an inflation of 10 layers with 1.2 growth rate is applied.

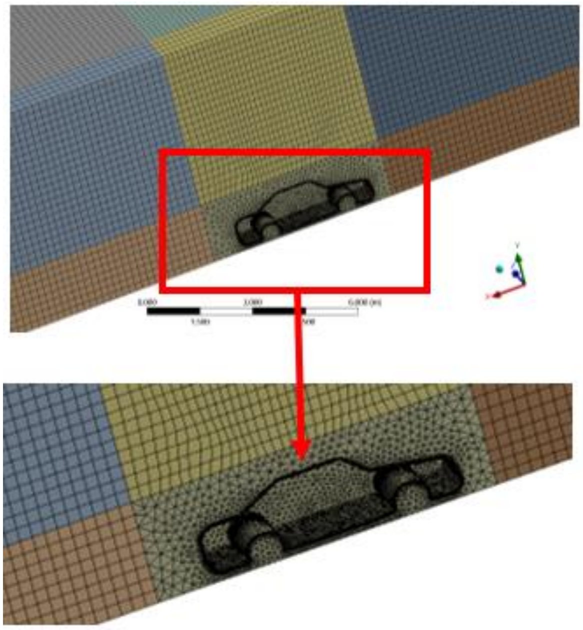


**Fig.7:** Mesh generation with modified sizing settings

*4.4 Solver Settings*

The problem of vehicle external flow numerical analysis required the solver settings to be completed before starting the simulations. The solver setting includes type of solver (3D or 2D), the viscous model, boundary conditions and solution controls. The inlet of the wind tunnel was indicated by the term “velocity-inlet”, while the outlet of the wind tunnel was termed as “pressure-outlet”.

**Governing Equations**

The equation for preservation of mass,

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\,\vec{v}) = S_m \quad (1)$$

Where $S_m$= mass included to the continuous phase or any handler sources.

**Momentum Conservation Equations**

Conservation of momentum in an inertial reference frame is described by

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \quad (2)$$

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^{\mathrm{T}}) - \frac{2}{3}\nabla.\vec{v}I\right] \quad (3)$$

*4.3.3 Transport Equations for the Realizable* **k−ε** *Model*

The modelled transport equations for kinetic energy k and rate of dissipation $\varepsilon$in the realizable model k−$\varepsilon$ are

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k v_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho \in -Y_M + S_k \quad (4)$$

And

$$\frac{\partial}{\partial t}(\rho \in) + \frac{\partial}{\partial x_i}(\rho \in v_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon \quad (5)$$

In this analysis we used k epsilon (2 equation) realizable model and non-equilibrium wall treatment function in single phase flow, where air density is 1.225 kg/m3 & viscosity is 1.7894*10$^5$. (*Ansari Abdul Razzaque*)

## 5. Simulation Results

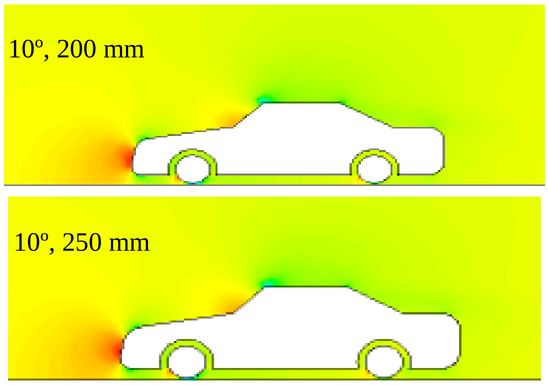
A 3D steady state, incompressible solution of the Navier-Stokes equations was implemented using ANSYS FLUENT®. Turbulence modelling was done with the realizable k-ε model using non- equilibrium wall functions. The computational results for the following cases are presented and discussed:

**Table 2:** Details of cases considered for present analysis is tabulated below (front windshield, back windshield, Hood angle & velocity is constant for all the cases which are 45º, 30º, 10º&25 m/s)

| Case | Front Bumper Angle (in º Degree) | Hood Fillet Radius (mm) |
|---|---|---|
| 1 | 10 | 200 |
| 2 | 10 | 250 |
| 3 | 10 | 300 |
| 4 | 15 | 200 |
| 5 | 15 | 250 |
| 6 | 15 | 300 |
| 7 | 20 | 200 |
| 8 | 20 | 250 |
| 9 | 20 | 300 |

*5.1 Pressure contour*

From the pressure contour distribution, it is observed that stagnation at the front end of the car results in increased pressure.



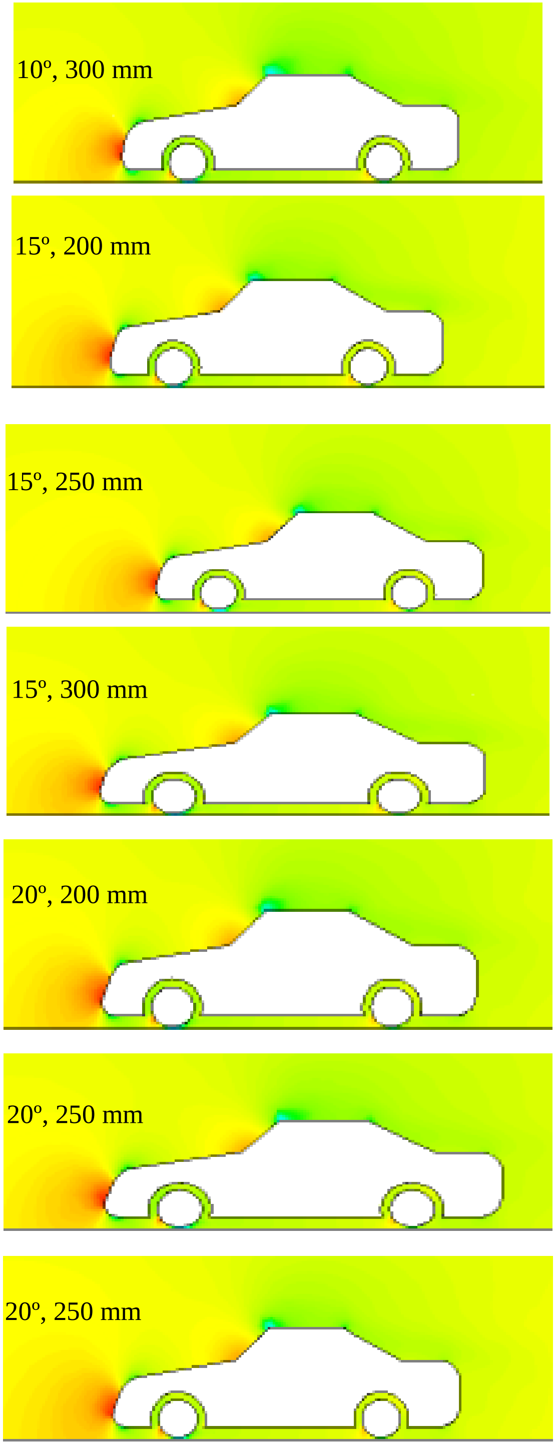



**Fig.8:** Pressure contours of car body at different conditions of front bumper angle & hood fillet radius.

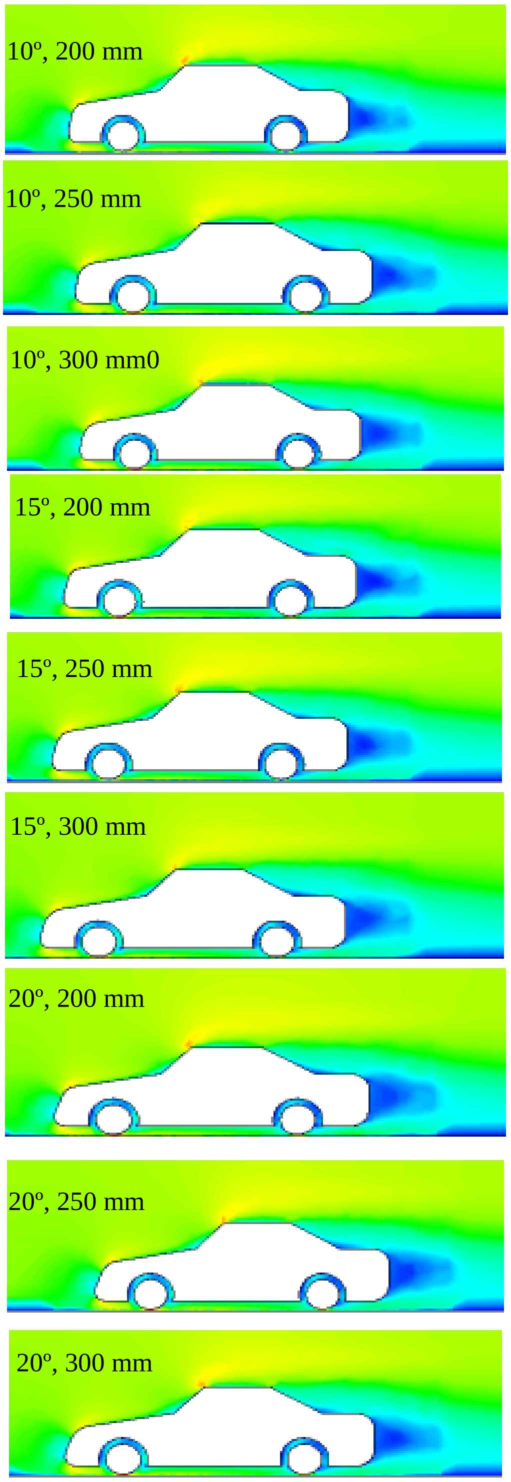



**Fig. 9:** Velocity contours of car body at different conditions of front bumper angle & hood fillet radius.

### *5.2 Velocity contour*

Viscous force is predominant considering the velocity distribution, air while passing through car get attached to the body thereby creating more viscous drag. Viscous drag is less significant regarding the total drag faced by the car. Pressure drag serves as the predominant drag in designing a vehicle aerodynamically.

### *5.3 Influence of Front Bumper Angle & Hood Fillet Radius on drag coefficient*

The flow field is observed and analysed and it is seen that as we incline the frontal projected area of bumper the drag coefficient value get reduced. It follows from the fact that pressure drag is decreased as area normal to flow direction is reduced.

The drag force is mainly due to the pressure drag the contribution of skin drag is very low in overall drag force. As we add fillet radius in hood the air flow is guided towards the front wind shield which shows that the guided air will strike the front wind shield and momentum conversion will be more over there so again the drag may be increased that is why we have a limitation in providing the fillet radius and angle of inclination of the front bumper angle.

The influence of Front Bumper Angle & Hood Fillet Radius on drag coefficient is found to be very strong effect on aerodynamic analysis of car body. In CFD results minimum drag coefficient in case 9 at 300 mm hood fillet radius & 20° front bumper angle. In most of the cases minimum drag coefficient found at maximum front bumper angle with larger fillet radius. This clearly indicates that increase front bumper angle will be reduce the drag coefficient of car body with above mentioned constraints.

This study has mainly focused on to reduce the drag in car body with the help of CFD analysis. The drag coefficient and drag force values has found in analysis which has tabulated. It has shown by tabulated values that if we increase the front bumper angle and hood fillet radius, the value of drag force has decreased with the limitation discussed above.

**Table 3:** Values of drag coefficient and drag force in different cases

| Case | Front Bumper Angle (Degree) | Hood Fillet Radius (mm) | Drag coefficient $C_D$ |
|---|---|---|---|
| 1 | 10 | 200 | |
| 2 | 10 | 250 | 0.4213 |
| 3 | 10 | 300 | 0.4169 |
| 4 | 15 | 200 | 0.4101 |
| 5 | 15 | 250 | 0.4198 |
| 6 | 15 | 300 | 0.4167 |
| 7 | 20 | 200 | 0.4083 |
| 8 | 20 | 250 | 0.4030 |
| 9 | 20 | 300 | 0.3943 |

The comparison of drag coefficient value of case 1 case2 & case 3 in graph at 10 degree front bumper angle. It clearly shows in fig.10 that if hood fillet radius increased, drag coefficient will decreased. Minimum drag coefficient value has found at case 3 which is 0.4101.

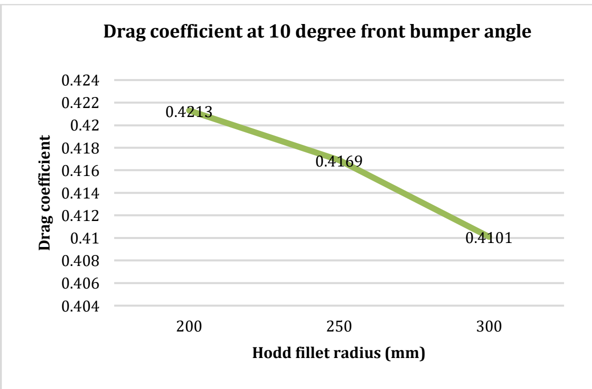



**Fig. 10:** Comparison of Drag Coefficient at 10degree front bumper angle with different hood fillet radius

The comparison of drag coefficient value of case 4 case 5 & case 6 in graph at 15 degree front bumper angle. It clearly shows in fig.11 that if hood fillet radius increased, drag coefficient will decreased. Minimum drag coefficient value has found at case 6, which is 0.4083.

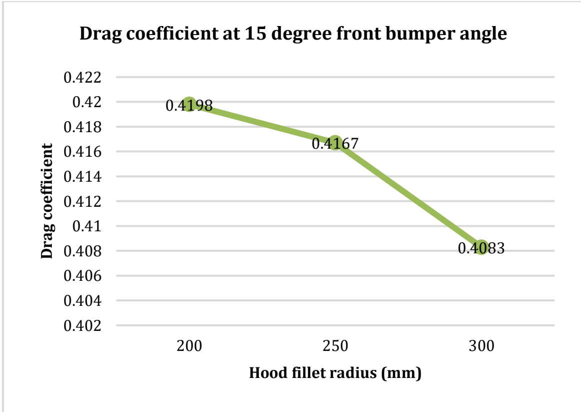



**Fig. 11:** Comparison of Drag Coefficient at 15 degree front bumper angle with different hood fillet radius.

The comparison of drag coefficient value of case 7 case 8 & case 9 in graph at 20 degree front bumper angle. It is clearly shows in fig. 12 that if hood fillet radius increased, drag coefficient will decreased. Minimum drag coefficient value has found at case 9, which is 0.3939. It has minimum that's why this is our optimum result among all cases.

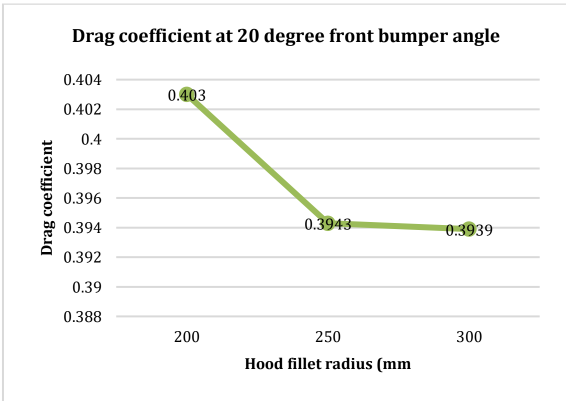



**Fig. 12:** Comparison of Drag Coefficient at 20 degree front bumper angle with different hood fillet radius.

## Conclusions

The aerodynamic drag and flow characteristics of a high-speed generic sedan passenger vehicle with modified geometrical dimension situations were numerically investigated. The objective of the study was to check the effect of a geometrical design parameter on the drag coefficient of the car. For this purpose different cases with modified CAD model were simulated and the conclusion was drawn on the basis of the result observed and visualization of flow field.

Followings are the conclusions drawn in the study:

- After analysing the results of the simulation of front angle inclination increasing from 10 - 20 degree at constant hood fillet radius of 200 mm the percentage reduction in drag coefficient has found 4.34%.
- After analysing the results of the simulation of front angle inclination increasing from 10 - 20 degree at constant hood fillet radius of 250 mm the percentage reduction in drag coefficient has found 5.42%.
- After analysing the results of the simulation of front angle inclination increasing from 10 - 20 degree at constant hood fillet radius of 300 mm the percentage reduction in drag coefficient has found 3.95%.
- After analysing the whole data from all cases it was found that the maximum reduction of drag coefficient is from 0.4213 to 0.3939 which is 6.50%.

We concluded in the study that increasing the front bumper angle reduces the drag coefficient, also it is concluded that increasing the fillet radius of hood reduces the drag coefficient up to some extent.

Further the study can be proceeded to find the effect of providing the fillet at the corners of front wind shield and back wind shield on drag coefficient and lift coefficient of the vehicle. Further the effect of moving wall boundary condition on the bottom wall of the fluid domain can be analysed for the accuracy of the CFD solution. The design of wheel also plays an important effect on the drag coefficient therefore the drag due to wheel can be studied in future. A design for the guide-way of air striking the front bumper can be optimized for the stability and drag reduction so that the flow pattern can be fruitful in terms of fuel economy.

## References

Ansari Abdul Razzaque (March 2017), "CFD analysis of aerodynamic design of tataindica car", International Journal of Mechanical Engineering and Technology (IJMET),Volume 8, Issue 3, pp. 344–355 Article ID: IJMET_08_03_038.

Bauskar Manoj P. , Dhande Dinesh Y. , Vadgeri Shivraj, Patil Sunil R. (2019), "Study of Aerodynamic Drag of Sports Utility Vehicle by Experimental and Numerical Method", Materials Today: Proceedings 16, 750–757.

Cheng See-Yuan, Chin Kwang-Yhee, Mansor Shuhaimi, Rahman AbdBasidAbd (2019), "Experimental study of yaw angle effect on the aerodynamic characteristics of a road vehicle fitted with a rear spoiler" Journal of Wind Engineering & Industrial Aerodynamics 184, 305–312.

Das Rubel Chandra, Riyad Mahmud (2017) "CFD Analysis of passenger vehicle at various angle of rear end spoiler"; 10th International Conference on Marine Technology, MARTEC 2016. Procedia Engineering 194, 160 – 165.

J Abinesh and J Arun kumar (October, 2014), "CFD Analysis of Aerodynamic Drag Reduction and Improve Fuel Economy", IJMERR, ISSN 2278 – 0149 Vol. 3, No. 4.

Kulkarni Shreenidhi R , Kothari Abhishek , Pavate Chetan , Dodmani Chinmay "Aerodynamic simulation of a truck to reduce the drag force ",International Journal of Engineering Research, Volume No. 4, Issue No. 11, pp : 613-617.

Kumar V. Naveen , Narayan K. Lalit , Rao L. N. V. Narasimha and Y. Sri Ram (2015), "Investigation of Drag and Lift Forces over the Profile of Car with Rea spoiler using CFD", International Journal of Advances in Scientific Research,ISSN: 2395-3616 ,pp.331-339.

R.Varun, .S.Sankar, Rajiv Varma, Sreejith K.V. (May 2018), "CFD Analysis of Aerodynamics of Car", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 7, Issue 5.

Shelar Dipak, Dabb Abhishek,Phatangare Aniket , Gupta Rohit "Aerodynamics Analysis of Passenger Vehicles using CFD Modeling" International Conference on Ideas, Impact and Innovation in Mechanical Engineering (ICIIIME 2017), ISSN: 2321-8169.

Tomar Akhilesh Singh , Prajapati Anuj, Sharma Anuj ,Shrivastava Shubham (7 May, 2019) "CFD Analysis on the Aerodynamic Effects of Spoiler at Different Angle on Car Body ", International Journal of Innovative Technology and Exploring Engineering (IJITEE), ISSN: 2278-3075, Volume-8 Issue.

Toukir Islam (2016), "Numerical Study on Aerodynamic Drag Reduction of Racing Cars", Science Direct, pp.308 – 308.

Vignesh S. ,Ganga D Vikash Shridhar , Jishnu V. ((2019)) "Windscreen angle and hood inclination optimization for drag reduction in cars", 14 Global Congress on manufacturing and management (GCMM – 2018), Procedia Manufacturing 30 685–692

Wolf-Heinrich Hucho (1998) "Aerodynamics of Road Vehicles", SAE International, Warrendale, PA