ISSN No:-2456-2165

Study of Flow over Car using Open Foam and Changing Different Parameters

Kummara yaleti Devendra Babu
Aerospace engineering Department,
SRM institute of science and Technology, Chennai, INDIA

Magunta Vignesh Reddy
Aerospace engineering Department, SRM
Institute of science and Technology, Chennai, INDIA

Abstract:- An important aspect affecting a ground vehicle's fuel economy is its aerodynamic drag. Thus, the goal of this research was to examine the aerodynamic performance of automobile geometry for a hatchback. By adjusting various parameters, we were able to quickly determine the Drag force and Lift force using the OpenFOAM programme. In this investigation, various geometries with marginally varied characteristics such as hood, windscreen, back, and corner radius are used. Geometries for this project are produced directly in OpenFOAM using blockMesh. Using snappyHexMesh, meshing is accomplished. This study primarily uses 10 Hatchback geometries, whose varied contours are displayed at various speeds together with the determination of their coefficients of drag and lift, leading to the identification of one Hatchback shape that is optimal.

Keywords:- *Open FOAM, Drag force, Lift force, blockMesh, snappyHexMesh.*

I. INTRODUCTION

A subfield of fluid dynamics called "aerodynamics" focuses on understanding how air moves, especially in relation to moving objects. With a lot of theory in common with fluiddynamics, aerodynamics is also an area of gas dynamics. Gas dynamics and aerodynamics are frequently used interchangeably, whereas gas dynamics to applies to all gases. Calculating the forces and moments acting on an item requires an understanding of the velocity of the air (often referred to as a flow field). Velocity, pressure, density, and temperature are common parameters that are determined for a flow field as functions of position and time. By creating a control volume around the flow field, equations for the conservation of mass, momentum, and energy may be developed and used to solve for the characteristics. The use of aerodynamics through mathematical analysis, approximation based on actual data, and wind tunnel testing forms the scientific foundation. Fuel consumption reduction is one of the top priorities for contemporary automakers as a means of conserving energy and safeguarding the environment worldwide. Lower power output at same speed or greater speed with the same power output are both possible with less resistance to forward motion. The form has a key role in reducing drag. Today, low fuel consumption is the most crucial functional criterion for designing an efficient automobile form that would give less resistance to forward motion. The drag force (or drag coefficient in non-dimensional terms), which is the name for the resistance, is a significant effect of the form of the automobile. This implies that the direction and speed of the fluid particles' motion inside the automobile matter. The exterior surface geometry of the automobile are the major focus of this article. Creating the geometries by altering several parameters, such as the windscreen angle, front hood angle, rear hood angle, front corner radius, etc., was of interest. Using OpenFOAM, one may assess the coefficients of drag, lift, drag force, and lift force. software.

II. EASE OF USE

A. Methodolgy

The airflow around the hatchback and sedan body styles of the automobile is examined using the OpenFOAM (Open Field Operation and Manipulation) CFD toolkit. OpenFOAM includes a wide range of capabilities that may be used to handle complicated fluid flow problems involving electromagnetics and solid dynamics as well as chemical interactions, turbulence, and heat transfer. With the aid of blockMesh, the automobile geometries with various external parameters are produced directly in OpenFOAM. This is followed by a stereolithography (STL) file conversion. The mesh is then created by connecting SnappyHexMesh to the STL file. PotentialFoam was used to create the first flow. Giving initial boundary conditions, simpleFoam calculations were performed, and ParaFoam was then used for post-processing.The distribution of velocity and pressure over the geometries is depicted and analysed. An influence on the automobile body's pressure and velocity contour is investigated using aerodynamic analysis. Then the force of lift and drag are estimated in accordance with each other. The tyre component is not taken into account for the flow analysis.

B. Mathematical Formulation

Drag force(FD) The force restraining a vehicle's forward motion is called an aerodynamic drag force. Because it makes up around 65% of the total force acting on the entire body, this force must be taken into account while designing the outside of the vehicle. A helpful tool for comparing various vehicle forms is the drag coefficient, a dimensionless number that quantifies aerodynamic resistance of a vehicle. The aerodynamic drag force is calculated using the formula shown below:

FD=1 qV2CDA (1)

where: Drag Force (FD)
Drag Coefficient, or CD
A: The front of the vehicle
V = Wind Speed
Air Density = q
Lift Force (FL)

When applied in a positive direction, lift force allows the vehicle to be raised into the air; however, when applied in a negative direction, lift force can create excessive wheel down force. In order to prevent too much down force or lift, engineers work to keep this number within a set limit. The formula usually used to define this force is written as:

FL = 1 qV2CLA (2)

where:
FL=Lift Force
CL=Lift Coefficient
A=Frontal Area Of The Vehicle
V=Wind Velocity
q=Air Density

III. IN TERMS OF HATCHBACK GEOMETRY

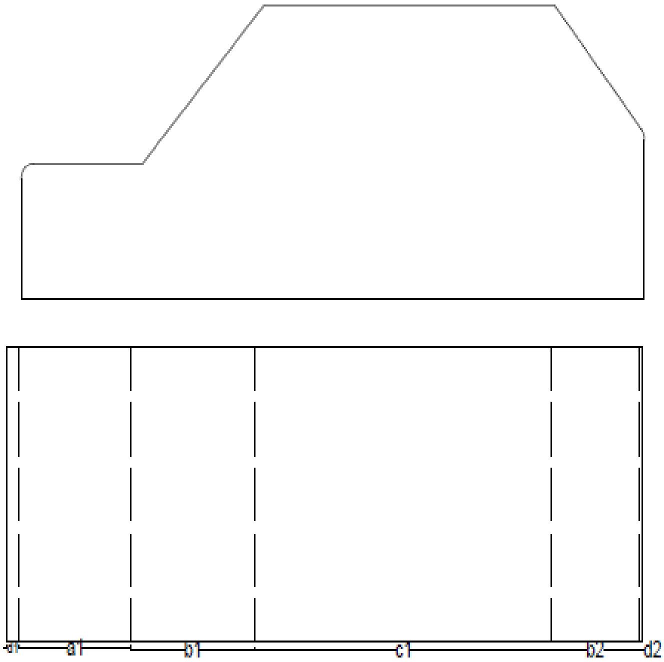


Fig. 1: Dimensions of a hatchback

Case No.	d1	d2	Hood angle	Rear angle
1	0.05	0.001	0	0
2	0.1	0.001	5	0
3	0.15	0.001	10	0
4	0.2	0.001	15	0
5	0.25	0.001	20	0
6	0.05	0.001	0	5
7	0.1	0.001	5	10
8	0.15	0.001	10	15
9	0.2	0.001	15	20
10	0.25	0.001	20	25

Table 1: Geometric parameters for the hatchback

A. Computional setup

To meet the objectives, many instances were taken into account. The technique is the same for all scenarios since the simulation setup is the same in all circumstances. The actual mesh must first be entered into OpenFOAM's file structure in order to start a new simulation. However, in our project, we are utilising blockMesh within OpenFOAM to create the geometry. After that, it is transformed into a stereolithography (STL) file.

B. Domain-specific information

The area of computation is intended to result in a free flow with negligible obstruction, or, more specifically, a box having an inlet, an exit, two sides, a roof, and a ground level. All of the domain's faces were given names when the geometry was established during the construction of computational mesh. The front face and back face of the domain, which correspond to the pressure outlet and velocity inlet, are the names of the inlet and outlet planes at x= 0 and x= L. Outer wall as wall is the name of the planes at y=L, z=0, and z=L. The model's names are "vehicle as a wall."

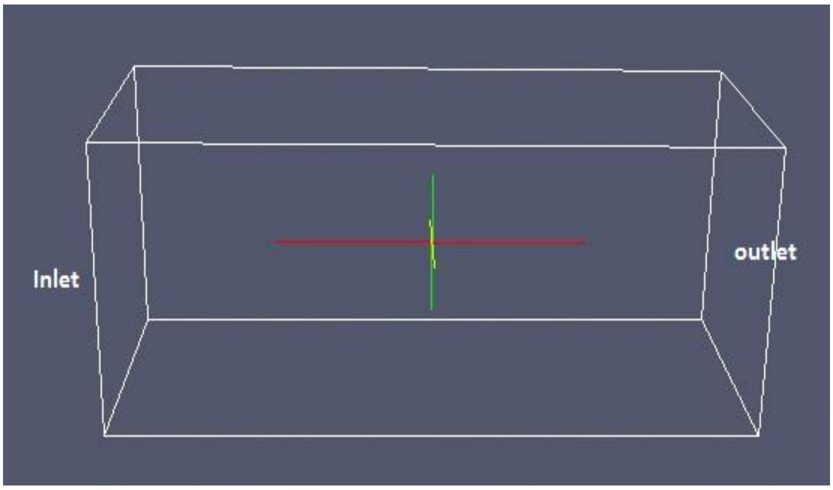


Fig. 2: Defining the Domain

C. Mesh

Snappyhexmesh, a meshing tool built with OpenFOAM, is used to construct the mesh. To generate high quality hexdominant meshes from random geometry, use the utility snappyHexMesh. Le snappyHexMeshDict settings govern its operation. It may be carried out concurrently. The feature edges are preserved, and wall layers are added. In the table below, mesh size information is provided.

Number of Points	Number of cells	Number of Faces	Number of Internal Faces
1969583	1824778	5617183	5515085

Table 2: information about mesh size

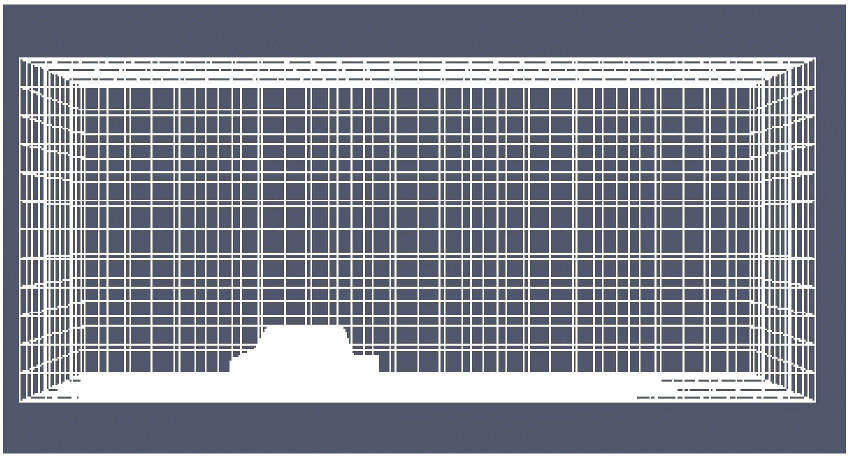


Fig. 3: After meshing, a side/view

D. Boundary conditions

The tests were conducted at a variety of operating speeds, including 60, 70, 80, 90, 100,110, 120, and 130 km/hr. This implies that any hatchback's geometry may go across this range of speeds. Consider case 1 as an example. The inlet boundary condition's velocity is set to 60 km/h (16.16 m/s) and the temperature is 300 K. (26.850C). Pressure outlet with a gauge pressure of 0 pa is the outlet boundary condition. The air's viscosity is adjusted at 1.7894 x 10-5 kg/m3, and its density is 1.125 kg/m3 (ms).

E. Turbulence Simulation

The kmSST turbulence Simulation was used in all of the simulations to account for turbulence. This model was chosen because it has a track record of being reliable in separation zones and can combine a solid free stream model with a solid boundary layer model.

F. Turbulence Intensity

The turbulence intensity, *I*, is defined as the ratio of the root-mean-square of the velocity fluctuations, u`, to the mean free stream velocity, u.

$$I = uu. \quad (3)$$

uTurbulence intensity for internal flows may be rather high, with suitable inlet values falling between 1% and 10%. A fully developed duct flow's core turbulence intensity may be calculated as:

$$I = 0.16Re{-1/8} \quad (4)$$

Depending on the flow conditions, the value of turbulence intensity at the freestream for external flows can be as low as 0.05%. The amount of turbulence we are taking into account is 0.02%.

G. Concerning the Turbulence Length Scale

The magnitude of the huge eddies in turbulent flows is quantified by the turbulence length scale, or l. An approximative length scale can be obtained using an empirical connection between the physical size of the obstacle (or characteristic length), L, and the size of the eddy, l.

l = 0.07L (5)

H. Kinetic Energy(K) and Specific Dissipation Rate of Turbulence (m)

The following equations may be used to get the required turbulent kinetic energy (K) and specific dissipation rate (m), and table no. 5.2 provides these values for the operating speed range we have chosen.

Turbulent Kinetic Energy (K) = 3/2 (UI)2 (6)

Specific Dissipation Rate(m) =k1/2/Cμ¼*s (7)

S. No.	Speed in km/hr	Speed in m/sec	K	m
1	60	16.16	0.1665	1.061
2	70	19.44	0.2268	1.2421
3	80	22.22	0.2962	1.4197
4	90	25	0.375	1.5972
5	100	27.27	0.4629	1.7747
6	110	30.55	0.56	1.9522
7	120	33.33	0.666	2.1296
8	130	36.11	0.7824	2.3071

Table 2: K and m values

IV. RESULT AND DISCUSSION

At 100 mph, a comparison of all hatchback shapes All ten hatchback designs were compared at a speed of 100 km/h, and pressure fluctuation was seen in the vector plot, pressure contour, and velocity contour as well as the front, side, and top views.

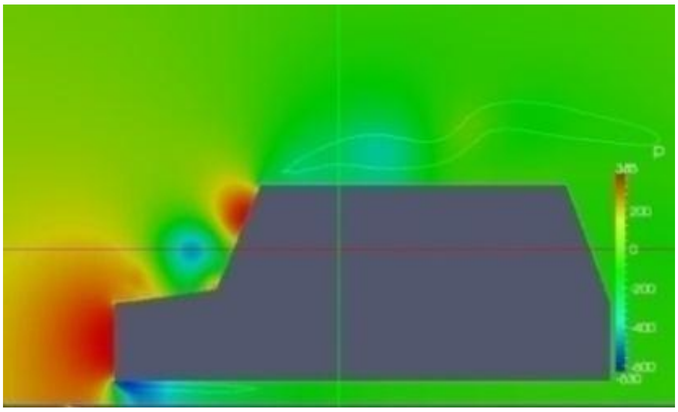


Fig. 4: Pressure contour in Case 1

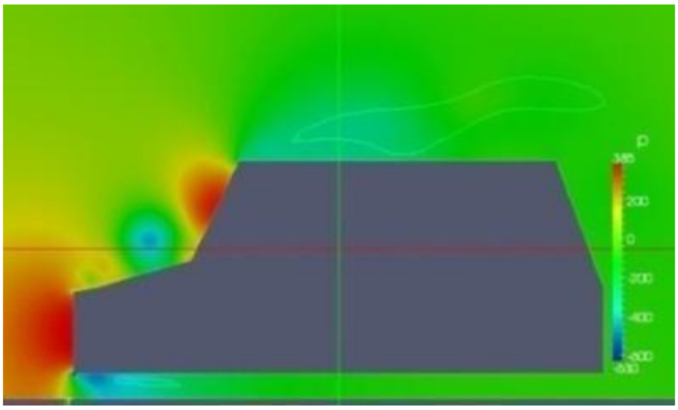


Fig. 5: Pressure contour in Case 2

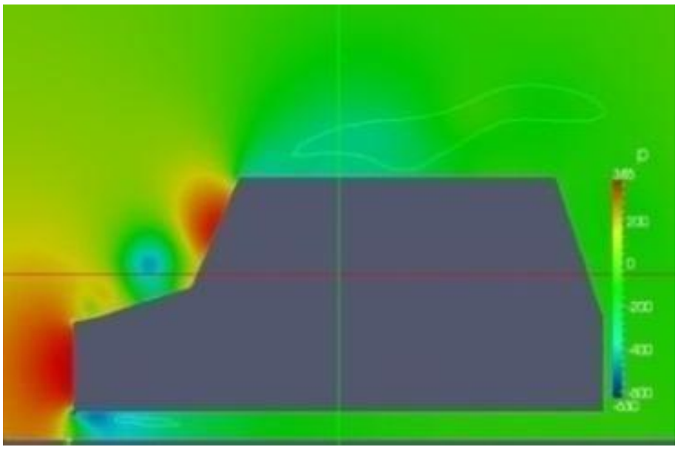


Fig. 6: Pressure contour in Case 3

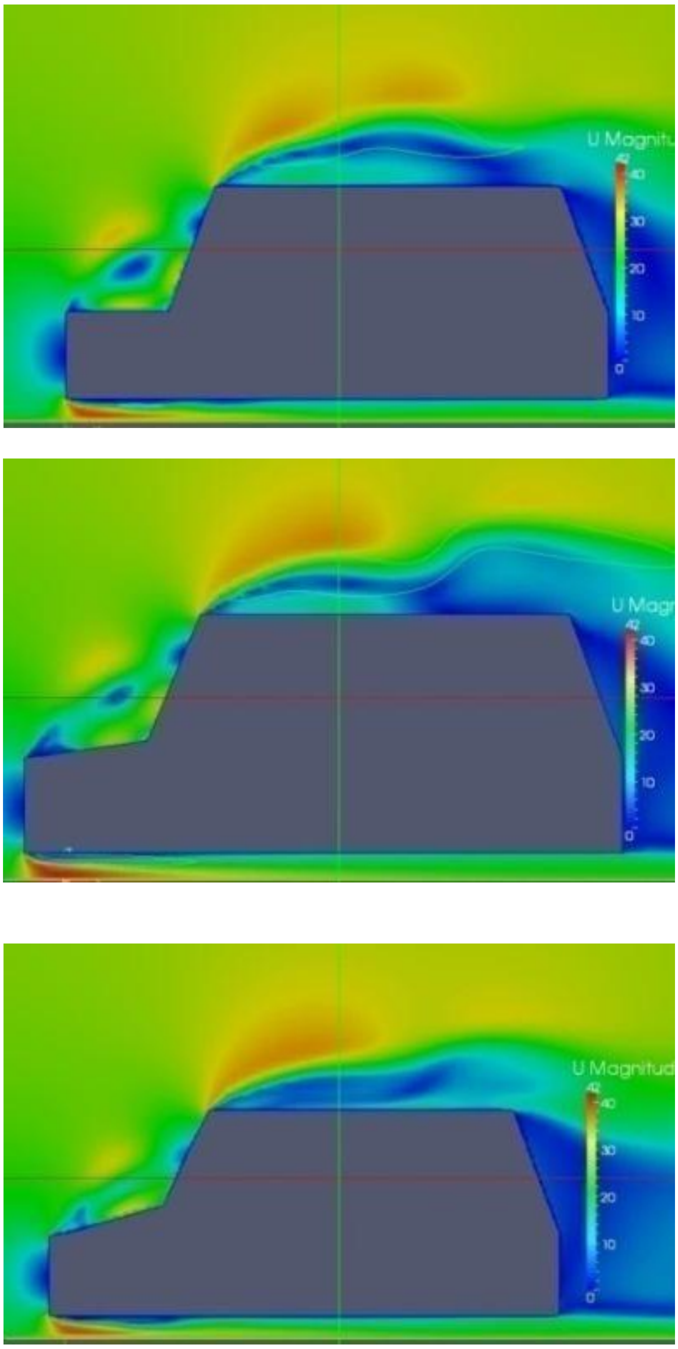


Fig. 7: Case 1,2,3 Velocity profile

ISSN No:-2456-2165

- **pressure gradient** – All of the pressure contours show that there was a greater pressure concentration in front of the car. Particularly when it comes to the air slows down in front of the automobile, which causes more air molecules to condense into a smaller area. When the air in front of the automobile becomes stagnant, The sides and roof of the automobile are among the places it looks for reduced pressure. When the air goes over the car's hood, the pressure
- **Velocity curve -** All of the velocity contours show that the air speed is decreasing as it approaches the car's front. The air speed then increases away from the front of the car.

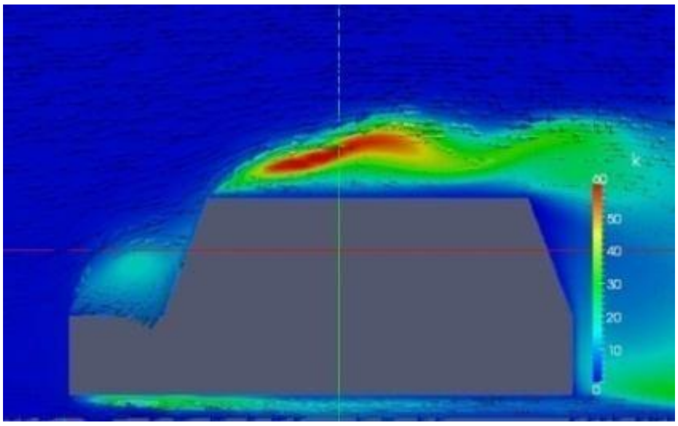


Fig. 8: Vector plot in Case 1

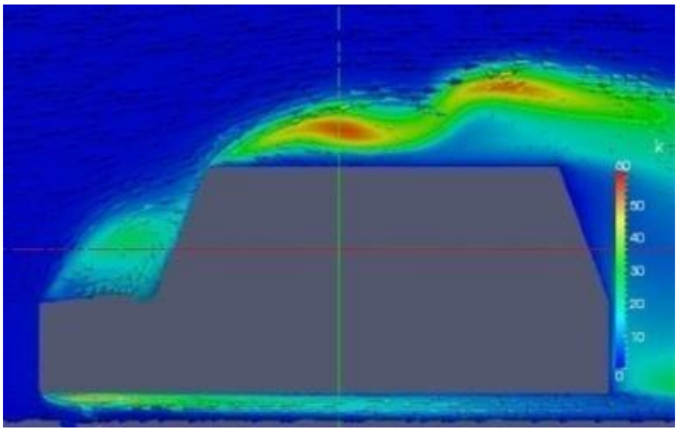


Fig. 9: Vector plot in Case 2

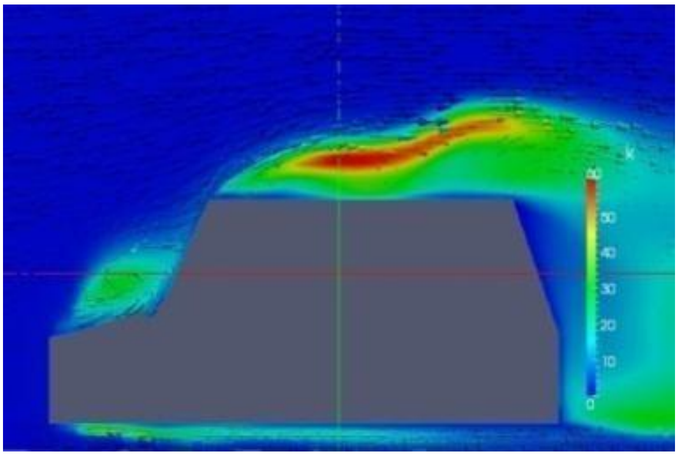


Fig. 10: Vector plot in Case 3

A. Velocity for KE

These figures indicate that a lot of turbulence is generated at the front of the windscreen in designs with flat hoods. On the other hand, the strength of turbulence is minimal in geometries with a specific degree of hood angle. While blue vectors have the least kinetic energy, red vectors have the most.

ISSN No:-2456-2165

B. Drag force and coefficient for hatchback shapes.

S. No.	Speed in km/hr	C_d	
		Case 1	Case 2
1	60	0.654951	0.594011
2	70	0.654393	0.593766
3	80	0.65299	0.590715
4	90	0.652445	0.589211
5	100	0.651024	0.619261
6	110	0.652086	0.5846395
7	120	0.651617	0.585798
8	130	0.654798	0.585073

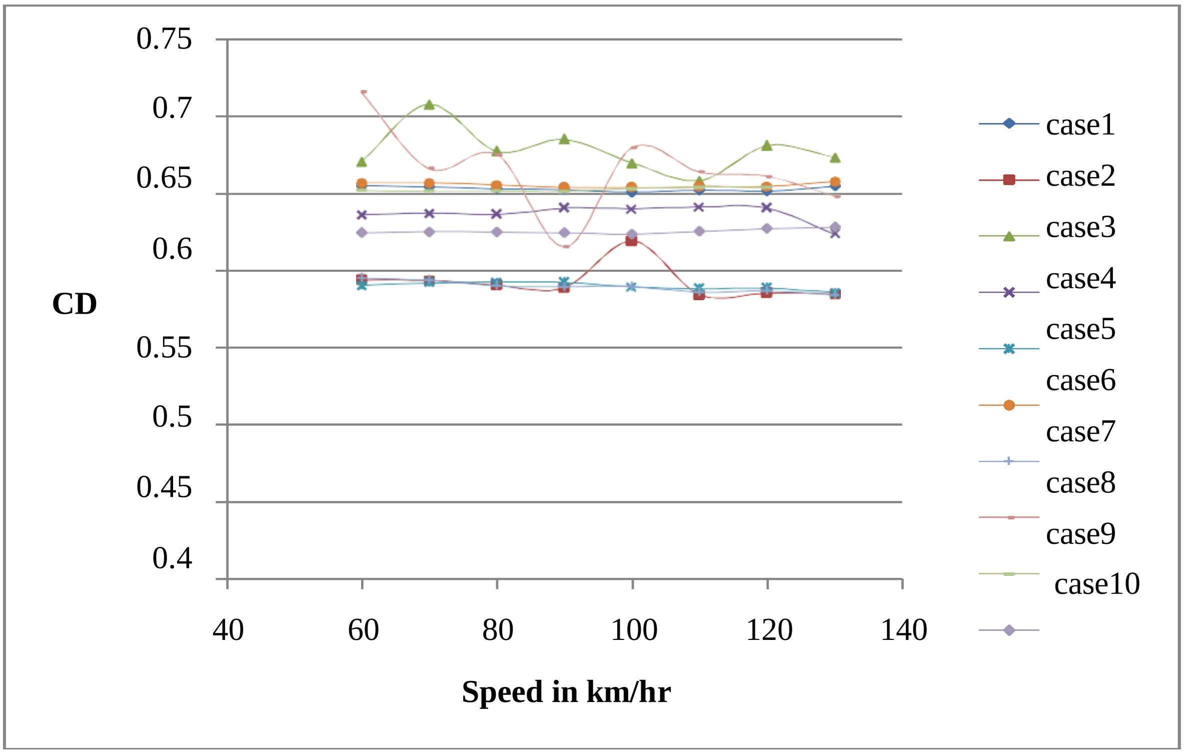
Table 3: Coefficient of Drag for Hatchback shapes

S. No.	Speed in km/hr	C_d	
		Case 3	Case 4
1	60	0.670589	0.636342
2	70	0.707524	0.637281
3	80	0.677222	0.636532
4	90	0.685012	0.640736
5	100	0.66963	0.640041
6	110	0.658186	0.641367
7	120	0.680911	0.640709
8	130	0.673265	0.623777

S. No.	Speed in km/hr	c	
		Case 7	Case 8
1	60	0.595456	0.715567
2	70	0.594008	0.666141
3	80	0.590271	0.675141
4	90	0.589647	0.615444
5	100	0.589988	0.679625
6	110	0.586203	0.663983
7	120	0.587196	0.66095
8	130	0.584413	0.648114

S. No.	Speed in km/hr	Cd	
		Case 5	Case 6
1	60	0.590656	0.656767
2	70	0.592152	0.656787
3	80	0.592919	0.655578
4	90	0.592731	0.653927
5	100	0.589595	0.653934
6	110	0.588576	0.654424
7	120	0.5889	0.654499
8	130	0.586087	0.65789

S. No.	Speed in km/hr	cd	
		Case 7	Case 8
1	60	0.651984	0.624737
2	70	0.651568	0.625417
3	80	0.651533	0.625077
4	90	0.651276	0.624649
5	100	0.653126	0.623819
6	110	0.654069	0.625451
7	120	0.6538655	0.627333
8	130	0.652157	0.628304



Graph: A comparison of every CD for different cases Speed V/S

ISSN No:-2456-2165

V. CONCLUSION

The essay makes several broad recommendations that may be applied to the majority of current road-going automobiles, including: smooth vehicle design, rounded corners, high windshield angle, tapered tail end, and minimal body seams. The car model's aerodynamic drag can be decreased by the use of the rear screen angle. Both the layout of the model's back component and the size, shape, and placement of the displays have an impact on minimising drag.

REFERENCES

[1.] Darko Damjanović et.al. " CFD Analysis Of Concept Car In Order To Improve Aerodynamics" 2011
[2.] Chainani. A, Perera. N, "CFD Investigation of Airflow on a Model Radio Control Race Car"July 2–4, 2008, London, U.K.
[3.] Mahmoud Khaled et.al. "Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body" J.Wind Eng.Ind.Aerodyn.107-108(2012)
[4.] Kunal Kumar Tirkey, Suresh Kumar " CFD Analysis Of Car Body"
[5.] JUSTIN MORDEN, "Aerodynamic Analysis And Testing Of A Formula Renault Race Vehicle Using Analytical And Computational Methods Validated By Real World Test Data
[6.] Prof. G Sivaraj, Mr. Gokul Raj "Optimum way to increase the fuel efficiency of the car using base bleed." June 2012 Vol. 2 IJMER
[7.] Shailesh Jindal , "Numerical Investigation of Road Vehicle Aerodynamics Using Immersed Boundary RANS Approach",2005 SAE International
[8.] Dr. Marc J. Richard , "Parallel CFD of a prototype car with OpenFOAM" June 21-24,2010
[9.] W. Meile et.al, "Experimental And Numerical Simulation On The Aerodynamics Of Ahmed Body" vol.3(1)-March 2011.