Hatchback Versus Sedan – A Review of Drag Issues

Vivek Yakkundi [1] , Shankar Mantha [2], Vivek Sunnapwar [3]

[#1] Principal , Lokmanya Tilak College of Engg , University of Mumbai ,India [#2]Shankar Mantha , Retired Prof & Head Mechanical , VJTI , India [#3]Vivek Sunnapwar , Prof Mechanical , Lokmanya Tilak College of Engg , University of Mumbai , India.

Abstract

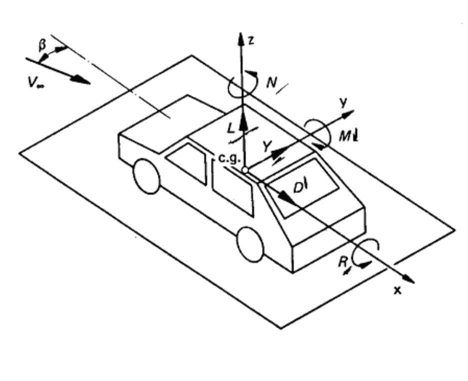
Aerodynamic drag or the wind resistance is considered to be of prime concern in vehicle design .The other important issues being the vehicle weight , fuel efficiency & emissions .The twentieth century in particular has witnessed refinement in vehicle design in terms of issues mentioned above. However the focus in this paper is on drag & related issues. It is worth noting that reduction of one count of drag i.e $\Delta C_D = 0.1$ translates into an improvement of milage of 2.60 km. As per DOT figures India has approximately 15 million cars plying on the roads as recorded during 2011. Further few hundreds are added every day. Therefore there is enormous scope for improving aerodynamics of cars looking at its growing number, which directly reflects the fuel consumption. It is observed that Hatchbacks or the two box version of cars are becoming popular than contemporary Sedan or the three box version inspite of lower drag value of Sedans, thus creating a conflict of choice amongst customers .The reasons cited are convenience of luggage boot space inside, smaller and compact size due to reduction in wheel base, better power to weight ratio, better manoeuvrability and lesser parking space. In the present paper analysis a midsegment Hatchback and its Sedan version is modelled form its image and CFD analysis is done to predict drag and other aerodynamic characteristics such as lift, flow over the car and analysis of wake / separation in the rear slant. The results of Hatchback and Sedan are compared and it is ascertained that the drag value of Sedan is lower than Hatchback.

Keywords — *Erodynamics, Turbulence, Computational Fluid Dynamics, Numerical Simulation, Drag*

I. INTRODUCTION

Cars or the road vehicles in general are bluff bodies moving in the vicinity of ground & the flow over them is complex owing to its partly attached & partly separated nature. The flow over cars is said to be turbulent eventhough the intensity of turbulence is low and nature of vortices formed around is also not very strong .Fig 1 shows forces and moments acting on a vehicle [19]. Any generic car shape without any superficial features generates three dimensional vortices in the rear slant of the car .The drag or the wind resistance of cars is made up of two parts namely Viscous drag which is found in the attached flow region due to domination of viscous effects and Pressure drag in the separated flow region where inertia effects dominate.

Fig 1. Aerodynamic forces and Moments acting on a CAR assuming six degrees of freedom.



In cars which are bluff bodies the Pressure drag accounts for 85 % and the remaining 15 % is Viscous drag [14]. As said earlier Viscous drag results from the shape / profile and Pressure drag results from sudden abrupt change in geometry / topology , where the flow separates after formation of free shear layer. The Viscous drag can be reduced by adapting to generic shapes like aeroplane wings, pontoons etc. which are known as streamlined bodies. Similarly Pressure drag can be reduced by altering the geometry and thus avoiding the extent of separation .It is also found that the design of the rear slant decides the drag or the wind resistance of the vehicle.

The vortices formed behind the car resemble that behind a cylinder , which are known as VonKarman Vortices.[1]Refer Fig 2. As these vortices are shed on a continuous basis, their kinetic energy is dissipated and as a result there is equivalent pressure loss in the rear slant region also known as Wake. Therefore the vehicle has to do excess work equivalent to the pressure loss in the wake , which is nothing but drag. Since the wind is also flowing from under the hood there is vertically upwards acting force known as Lift.It is therefore desirable that there is enough down force to counter the lift for stability and traction.

A. Problem Definition & Flow Solvers

As the flow field around the car is complex owing to its partly attached and partly separated nature, the zonal approach is required to simulate it in which the flow is assumed to be steady & incompressible. At high Reynold's numbers flowfield domains around the car can be classified as shown if Fig 2.

- Potential flow in the fore or front side, where inertial effects dominate.
- Boundary layer zone, the middle portion of roof, where viscous effects are to be accounted. In this case boundary layer method is used for computation.
- Wake zone which is also dominated by inertial effects, and is a separated flow downstream of the vehicle's base by means of free shear layer. In this case wake analysis is done.

Each of these zones is governed by simplified versions of equations of motions which need to be solved separately using second order upwind schemes. Also their physical interaction has to be accounted for in an iterative process with simpler couplings for zonal solutions. [10]

Fig 3 Sub Classification of Hatchbacks

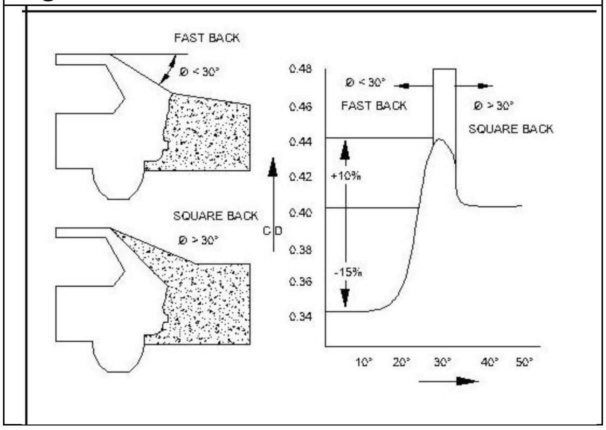
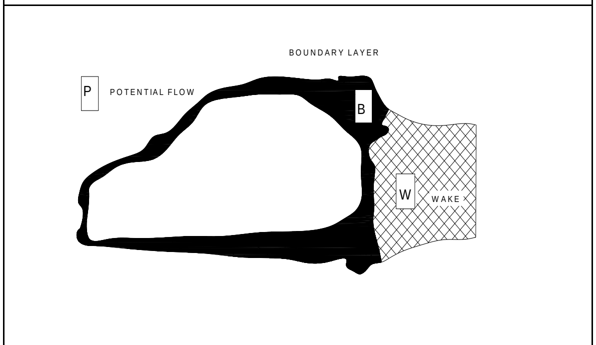


Fig 2.Flowfield domains around the car.



For the purpose, many of the CFD codes use finite volume techniques in solving flow related problems .This method was originally developed as special finite difference method for formulation .This method is well established and thoroughly validated general purpose CFD technique .The numerical algorithm of this technique consists of following steps.

- Formal integration of the governing equations of the fluid flow over all the (finite) control volumes of the solution domain.
- Discretization involves the substitution of the variety of finite difference type approximation for the terms in the integrated equations representing flow processes such as convection , diffusion and sources .This converts the integral equations into a system of algebraic equations.
- Solution of the algebraic equations by iterative methods.

The most popular solution procedures are the TDMA line by line solver of the algebraic equations and the SIMPLE algorithm to ensure correct linkage between pressure and velocity. [11]
Owing to the increased popularity of Engineering workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualization tools.

B. Classification of Cars

Broadly cars are classified as Hatchbacks – Two box & Notchbacks (Sedans) - three box based on their generic shapes. Hatchbacks are further sub classified as Fastbacks &Square backs as per the rear slant angle as shown in Fig 3. As seen in the Fig 3 the angle $30^0$is critical angle at which the drag is maximum & it is found that the drag value is minimum at an angle of 12.5^0 which increases up to $30^0$and again for greater angles the drag value decreases.

CFD Software Fluent is used as a virtual test bench for analysing two versions of cars .The Pro-e model of midsegment Hatchback is generated from its bidirectional image tracings taken on a graph paper , which are reconstituted in Pro-e and then variable section sweep tool is used to generate the car volume .Similarly by extending the rear portion to three box version Sedan is modelled as shown in Fig 4 & 5.The numerical schemes , simulation model and boundary conditions are as given in the Tables I& III for Hatchback and Sedan respectively. The graphic images of drag, static pressure contours, velocity vectors & pressure coefficients are shown in Figs 6 to 13 for specimen velocity.

Fig 4 Pro-e Model of Hatchback

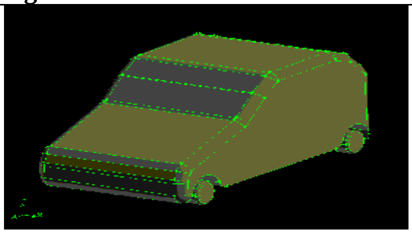
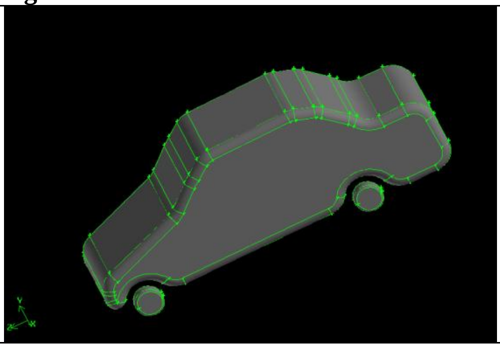


Fig 5 Pro-e Model of Sedan



Tables II and IV give breakup of pressure drag and viscous drag as found from force report function of Fluent for the two versions. The table also gives the value of pressure coefficients and analytical drag.
Graphs I,II,III, IV show the breakup of pressure and viscous drag for two versions.

II. RESULTS AND DISCUSSION

As Fluent issimulation software it outputs results in the form of graphic images and tables .The graphic images in Figs 6 to 13 give , drag (skin friction coefficient) , static pressure contours and velocity vectors , for specimen velocity of 45 m/s . From skin friction coefficient images it is observed that the drag value for Hatchback lies between 1 to 1.20 and for Sedan it lies between 0.6 to 1 . However these figures pertain to skin friction coefficient i.e. only viscous part of drag due to profile. The total drag values found from force report of fluent are 1.40 and 0.44 for Hatchback and Sedan respectively. The discrepancies are explained as under. Generally skin friction coefficient is computed for fully attached flows in Aerospace applications. Automotive shapes being bluff bodies therefore do not give realistic values of drag as a whole. Therefore the total drag values obtained from force report can be assumed as the realistic.

Static pressure contour images indicate the regions of high & low pressures along the symmetry plane. These images enable
us to track excessive lift , which is not desirable. The images of velocity vectors enable us to analyse the flow over body & separation in the wake region .These images indicate how pronounced the trailing vortices are. Also depending upon the up wash / downwash observed the the necessary down force can be visualized. It is also observed that the static pressure and velocity values complement each other. Images of static pressure contours are pressure coefficient only presented to understand the pressures acting in various regions. However the values from these images are not mentioned as the emphasis of this paper is on drag. Refer fig 12 & fig 13 which show details of separation / wake regions. It is seen that in Fig 12 of Hatchback trailing vortices are prominent, whereas in Fig 13 of Sedan there are no vortices seen as there is no separation of flow and as a result there is downwash which also provides necessary down force.
Refer graphs I,II, III & IV which indicate that the total drag increases as the speed increases. For Sedan the pressure and viscous drag are almost 50 % each at all speeds , however for Hatchback there is variation owing to flow separation in the rear and also due to flow reversal.

III. CONCLUSION

The important conclusion which can be drawn is that the drag of Sedan (Three box version) is lower than Hatchback (Two box version) for the same car , which is ascertained from values computed.
The Hatchback has large recirculation of vortices in the separated / wake region in the rear & therefore gives rise to additional pressure drag. The Sedan or the three box version has a boat tail ramp which avoids separation and therefore the pressure drag reduces. The Sedan version is said to be an optimized version.
The drag of Hatchback version also can be minimized by idealizing the flow over it to be half Rankine oval and accordingly designing the front profile so that the pressure drag of the front streamlined part of vehicle is theoretically zero. Therefore the vehicle has only minimum viscous drag in the fore and only pressure drag in the wake region in the rear.
Also, assuming the drag to be a function of Reinhold’s number / Velocity and aspect ratio the following expressions could be written in power law. The aspect ratio in this case is ratio of width / wheel track to the diagonal chord length (Lowest front point to highest rear point).

Drag Force = $a * (V)^{b} * (A/R)^{c}$

Drag Force= $a * (Re)^{b} * (A/R)^{c}$

However the computation of these correlations need extensive experimentation. Neverthless these correlations would provide the designer with greater insight .

REFERENCES

[1] B Basara- AVL List GMBH, Graz Austria, V Przulj – AVL UK Ltd. Northwich UK , P Tibaut – AVL AST Maribor , Slovania. (On the calculation of external aerodynamics : Industrial benchmarks – SAE 2001-01-0701)

[2] C.P.Van Dam , Recent experience with different methods of drag prediction.Elsavier,Pergamon , (1999.)

[3] E.L.Houghton , P.L.Carpenter- Aerodynamics for Engineering students.Butterworth Heinman (2003)

[4] Emmauel Guilminaeu –"Computational study of flow around a simplified car body"ELSAVIER,Journal of wind engg-(2007)

[5] Hereman Lienhart & Stefan Becker – " Flow and turbulence structure in the wake of a simplified car model ." (SAE-2003-01-0656)

[6] Hitoshi Fukuda,Kazuo Yanagimoto,Hiroshi China,Kunio Nakagawa-Improvement of aerodynamics by wake control.-Elsevier-JSAE-1984.

[7] Ichinose.K. Ito.S. , Accuracy of drag prediction on bluff bodies using CFD.JSAE Rev 1998 , 19: 151-60

[8] John D Andersen – CFD – The basics with applications.

[9] Mats Ramnofors – Volvo data corp , Rikard Bensryd – Volvo data corp , Elna Homeberg – Volvo data corp , Sven Perzon – Chamlers University of Technology (Accuracy of

[10] drag predictions on cars using CFD – Effect of grid refinement and turbulence models – SAE 960681)

[11] Norburt Grun – Tesis GMBH (Simulating external vehicle aerodynamics with carflow SAE 960679)

[12] Patankar S.V. 1980 , Numerical heat transfer & fluid flow ,Hemisphere publishing corporation ,New York

[13] Pieter Wesseling – Principles of CFD.Springer(2001)

[14] Richard M Wood – Solus-Solutions and Technology (Impact of advanced aerodynamic Technology on transportation energy consumption SAE 2004-01-1306)

[15] S.R.Ahmed & G.Ramm,G.Faltin " Some salient features of the time averaged ground vehicle wake."(SAE-840300)

[16] Saeyoung Han , V Sumantran , Clark Harris , Ted Kuzmanov , Mark Heubler , Thomas Zak – General motors (Flowfield simulations of three simplified shapes and comparison with experimental measurements – SAE 960678)

[17] Stephen B Pope – Turbulent flows. Cambridge University Press (2000)

[18] T.Morel , Aerodynamic drag of bluff body shapes Charecteristics of hatchback cars.(SAE-780267)

[19] Veersteeg.H.K. & Malaseekara- Introduction to CFD-The finite volume method..Longman Scientific & Technical (1995)

[20] Wolf Heinrich Hucho – Aerodynamics of road vehicles.Butterworth,Boston(1998)

Table – I Numerical Scheme & Boundary Conditions – Hatchback Analysis	
Model	
Space	3D , Pressure based , RANS.
Time	Steady
Viscous	SST , K-ω
Wall treatment	Transitional flows.
Turbulence intensity	Less than 0.5
Flow regime /Flow conditions(Reynolds's No based on velocity & length)	$5*10^5 <$ Re $< 10^7$ (Turbulent)
Boundary conditions/ Zones	
Fluid	Airdensity-1.225 Kg/m^3, Viscosity – 1.7894e -04
Solid	AL OR MS (For wheels)
Velocity inlet (Wind tunnel inlet)	27 , 36 & 45 m/s
Wall	NA
Wall	NA
Symmetry	NA
Pressure outlet	Ambient (Gauge 0)
Default interior	Interior
Solver details	
Equations	Solved
Flow	YES
Turbulence	YES
Discretization schemes(Finite volume CFD, Segregated solver)	
Pressure	Standard-simple
Momentum	Second order upwind
Turbulent kinetic energy	Second order upwind
Turbulent dissipation rate	Second order upwind
Wind tunnel	Cuboidal
Wall motion wrt adjacent cells	Yes
Mesh surface	Triangular-pave –size 5
Mesh volume	Tetrahedral-T Grid Hybrid, Mesh size 30

Table – II Force and drag values as computed from fluent force reports for HATCHBACK

Velocity m/s	Pressure force (N)	Pressure drag(D_P)	Viscous force (N)	Viscous drag (D_F)	Total force (N)	Total drag (D= D_P + D_F)
27.00	1428220.0	0.62	930298.0	0.41	2358518.0	1.03
36.00	1084201.0	0.47	1297746.6	0.56	2381947.4	1.03
45.00	578075.0	0.25	125407,5	0.55	677332.5	0.80
60.00	1444793.1	0.63	1770340.5	0.77	3215133.6	1.40

Table – III Numerical Scheme & Boundary Conditions. Sedan Analysis

Model	
Space	3D , Pressure based , RANS.
Time	Steady
Viscous	SST-K-ω
Wall treatment	Transitional flows.
Turbulence intensity	Less than 0.5
Flow regime /Flow conditions(Reynolds's No based on velocity & length)	$5*10^5 <$ Re $< 10^7$ (Turbulent)
Boundary conditions/ Zones	
Fluid	Airdensity-1.225 Kg/m^3, Viscosity – 1.7894e -04
Solid	AL OR MS (For wheels)
Velocity inlet (Wind tunnel inlet)	27 , 36 & 45 m/s
Wall	NA
Wall	NA
Symmetry	NA
Pressure outlet	Ambient (Gauge 0)
Default interior	Interior
Solver details	
Equations	Solved
Flow	YES
Turbulence	YES
Discretization schemes(Finite volume CFD, Segregated solver)	
Pressure	Standard-simple (Patankar)
Momentum	Second order upwind
Turbulent kinetic energy	Second order upwind
Turbulent dissipation rate	Second order upwind
Wind tunnel	Cuboidal
Wall motion wrt adjacent cells	Yes
Mesh surface	Triangular-pave –size 5
Mesh volume	Tetrahedral-T Grid Hybrid, Mesh size 30

Table – IV Force and drag values as computed from fluent force reports for SEDAN.

Velocity m/s	Pressure force (N)	Pressure drag(D_P)	Viscous force (N)	Viscous drag (D_F)	Total force (N)	Total drag (D= $D_P + D_F$)
27.00	561505.60	0.18	619451.45	0.20	1180957.10	0.38
36.00	613300.00	0.20	638065.80	0.21	1251365.90	0.41
45.00	629972.00	0.21	660149.00	0.22	1290121.40	0.42
60.00	614830.40	0.20	728290.61	0.24	1343121.00	0.44

Fig 6) Skin Friction Coefficient –Hatchback

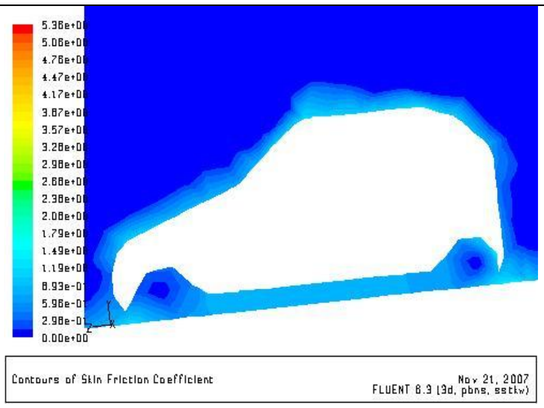


Fig 7) Skin Friction Coefficient –Sedan

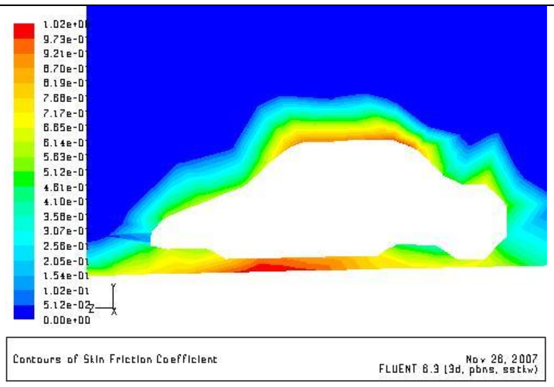


Fig 8) Static Pressure Contours –Hatchback

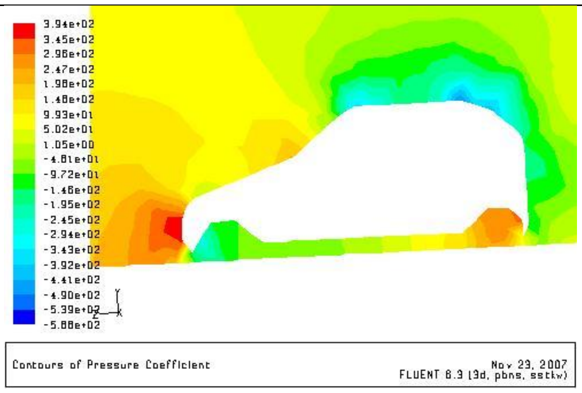


Fig 9) Static Pressure Contours - Sedan

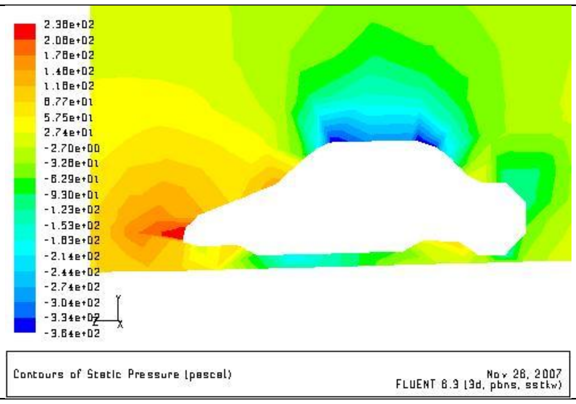


Fig 10) Velocity Vectors –Hatchback

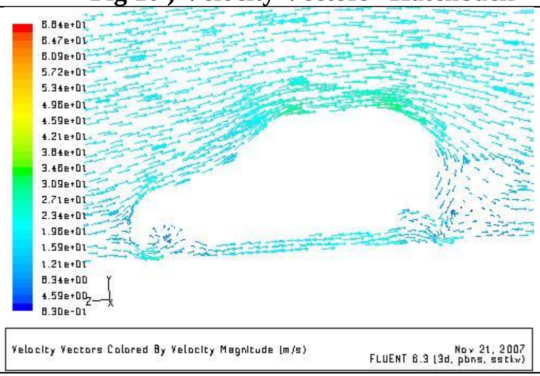
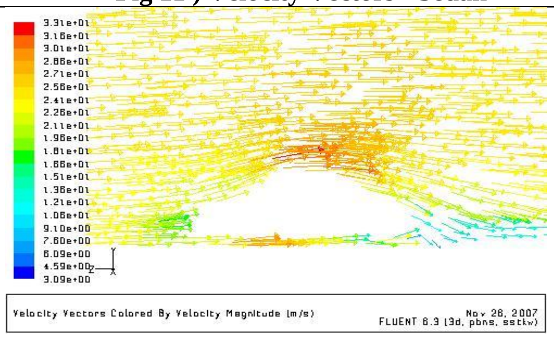
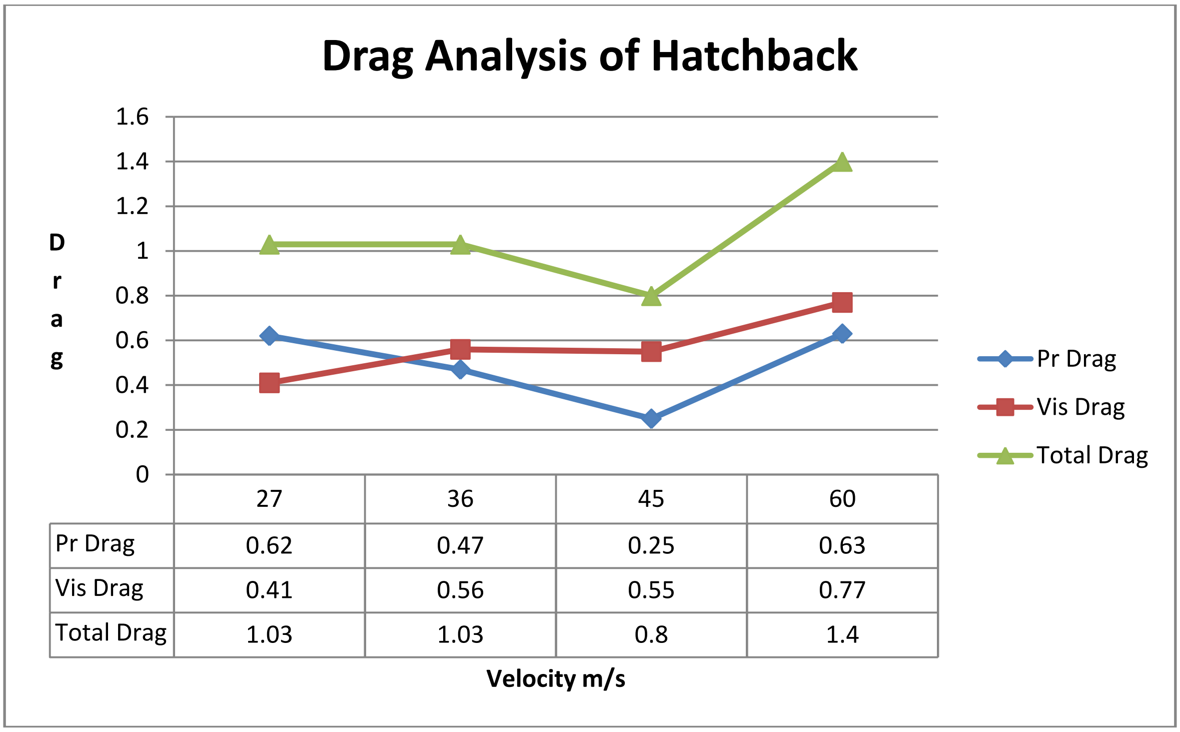


Fig 11) Velocity Vectors - Sedan

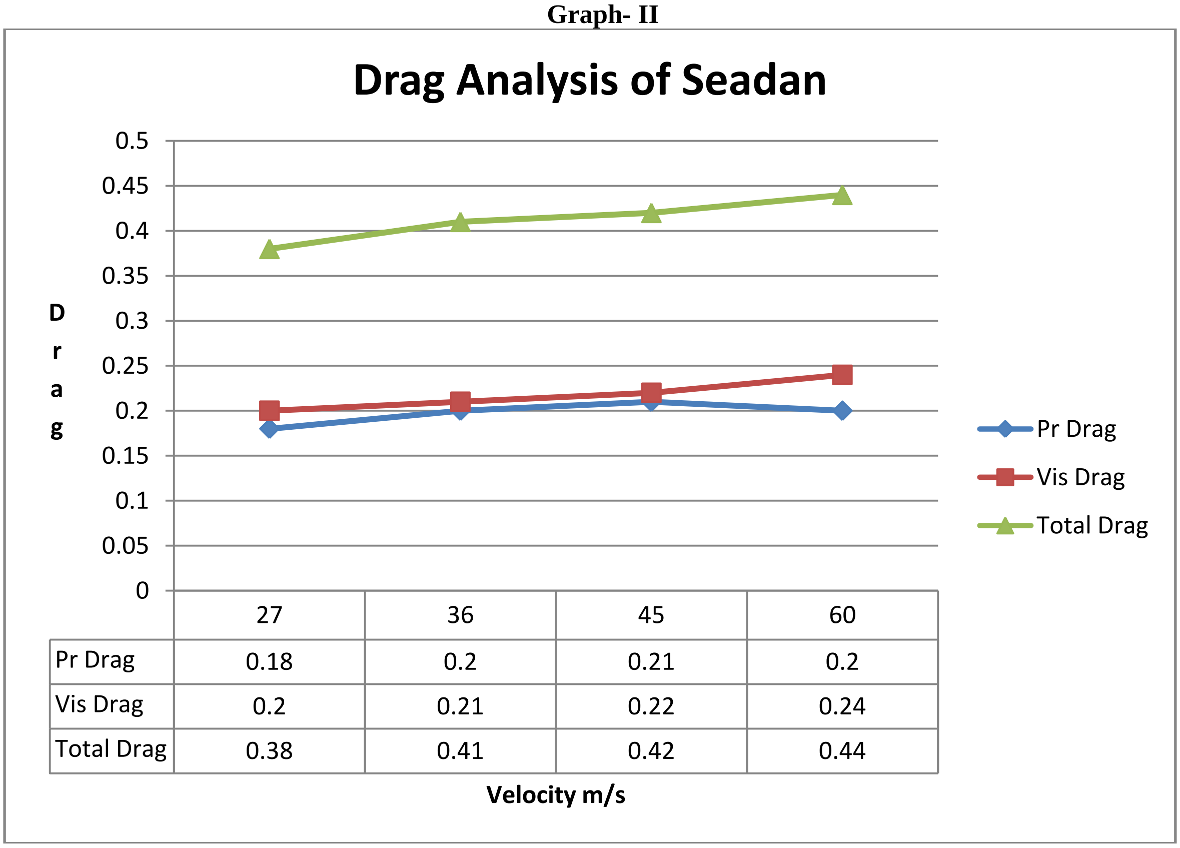


Graph – I



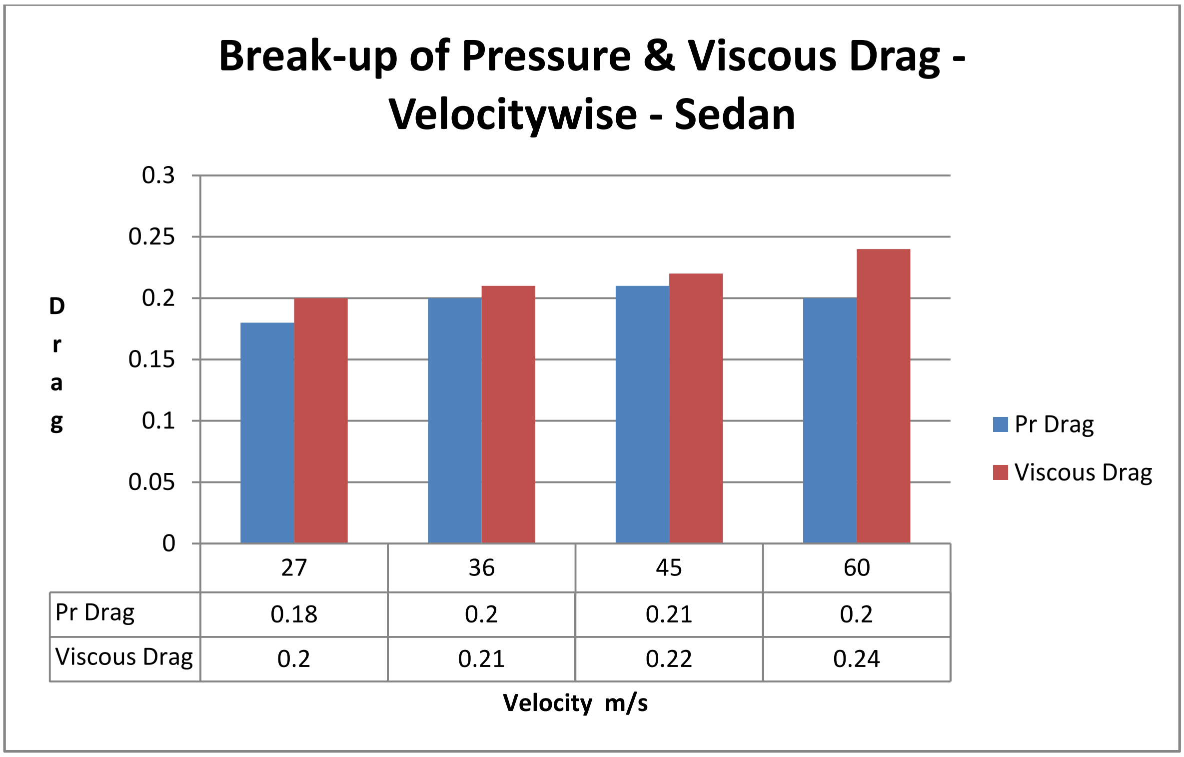
	27	36	45	60
Pr Drag	0.62	0.47	0.25	0.63
Vis Drag	0.41	0.56	0.55	0.77
Total Drag	1.03	1.03	0.8	1.4

Graph- II



	27	36	45	60
Pr Drag	0.18	0.2	0.21	0.2
Vis Drag	0.2	0.21	0.22	0.24
Total Drag	0.38	0.41	0.42	0.44

Graph -III



Graph - IV

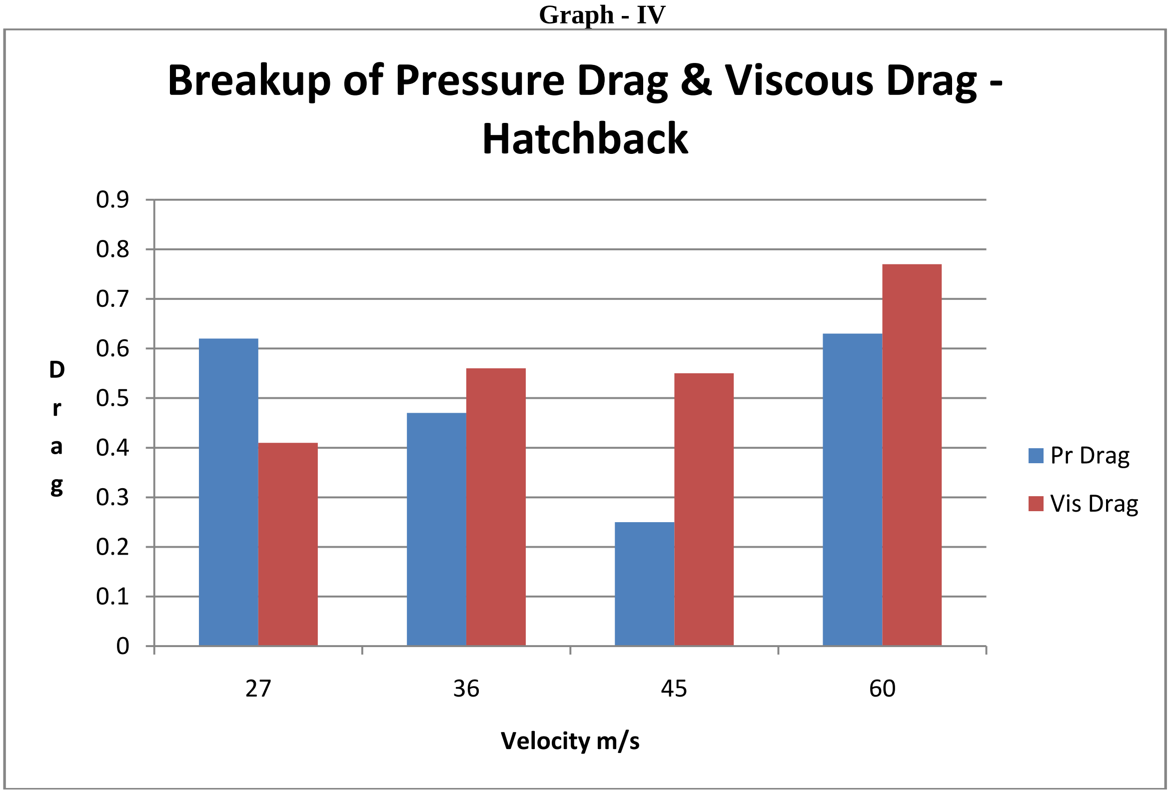


Fig 12 Hatchback - Vortices in the rear slant / Wake region in the lower region creating loss of pressure.

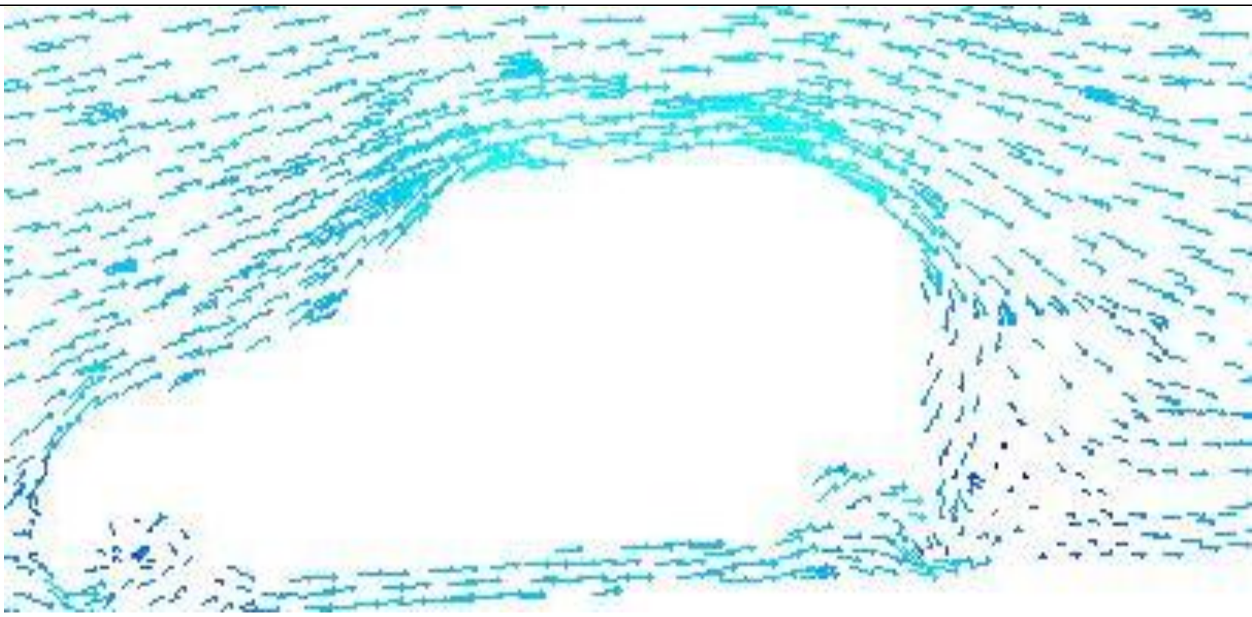


Fig 13 Sedan – No Vortices seen in the rear slant / Wake region & there is a downwash providing down force.