Improvement of Bus Body Aerodynamic Shape by CFD Analysis

Ketan G. Pranami[1] Dipen V. Shah[2] Bakul T. Patel[3]
[1, 2] M.E. Student [3]Assistant Professor
[1, 2, 3]Mechanical Engg. Department
[1, 2, 3]Laljibhai Chaturbhai Institute of Technology - Bhandu.

Abstract— To save the energy and to protect the global environment, fuel consumption reduction is a primary concern of the modern bus manufacturers. Drag reduction is essential for reducing the fuel consumption. Designing a vehicle with a minimized drag resistance provides economical and performance advantages. Decreased resistance to forward motion allows higher speed for the same power output, or lower power output for the same speed. The shape is an important factor for drag reduction. To design an efficient shape of the bus that will offer a low resistance to the forward motion, the most important functional requirement today is the low fuel consumption. The main intention behind this project is to reduce the drag co-efficient & drag force of bus body by improving the aerodynamic shape using CFD software (Autodesk Flow Design). The difference of drag force as well as drag coefficient before and after the change in aerodynamic shape of Volvo bus is carried out at different speeds. We got the desirable results of the reduction of drag force about 12 to 15%.

1. INTRODUCTION

Aerodynamics is study of gases in motion. As the principal application of aerodynamics is the design of aircraft; air is the gas with which the science is most concerned. Although aerodynamics is primarily concerned with flight, its principles are also used in designing automobile and train bodies for minimum drag and in computing wind stresses on bridges, buildings, smokestacks, trees, and other structures. It is also used in charting flows of pollutants in the atmosphere and in determining frictional effects in gas ducts. The wind tunnel is one of the aerodynamicist's basic experimental tools; however in recent years, it has been supplanted by the simulation of aerodynamic forces during the computer-aided design of aircraft and automobiles.

Vehicles with an aerodynamic shape use less fuel. Air flows easily over them and less energy is needed to move them forward. At 95 Km/h 60-70% of a vehicle’s energy is used to move it through the air, compared with only 40% at 50 Km/h. Installing a sloping front roof on a lorry could save you as much as 7% of your fuel costs. Even small changes to design and shape will make a difference. Take a look at the Aerodynamic Checklist, walk around your vehicle and look at each feature to see what improvements you can make. Don’t forget, a driver’s daily vehicle check needs to include looking out for these improvements. This document covers the aerodynamic styling of commercial vehicles. Vehicles that travel at higher speeds and for longer distances will benefit most from aerodynamic styling, giving you greater savings. Drag is the energy lost pushing through air, and it accounts for most of the fuel used on long-distance journeys, regardless of vehicle type.

Overcoming drag uses approx. 60% of fuel used at cruising speeds when loaded, 70% when empty. Sharp corners, racks and parts that stick out will add “parasitic drag”, further reducing fuel efficiency.

A. History & Evolution of aerodynamics:-

Ever since the first car was manufactured in early 20th century the attempt has been to travel at faster speeds, in the earlier times aerodynamics was not a factor as the cars were traveling at very slow speeds there were not any aerodynamic problems but with increase of speeds the necessity for cars to become more streamlined resulted in structural invention such as the introduction of the windscreen, incorporation of wheels into the body and the insetting of the headlamps into the front of the car. This was probably the fastest developing time in automobiles history as the majority of the work was to try and reduce the aerodynamic drag. This happened up to the early 1950’s, where by this time the aerodynamic dray had been cut by about 45% from the early cars such as the Silver Ghost. However, after this the levels of drag found on cars began to slowly increase. This was due to the way that the designing was thought about.

Fig.1:- Evolution of Cars

B. Vehicle body aerodynamics

The constant need for better fuel economy, greater vehicle performance, reduction in wind noise level and improved road holding and stability for a vehicle on the move, has prompted vehicle manufacturers to investigate the nature of air resistance or drag for different body shapes under various operating conditions. Aerodynamics is the study of a solid body moving through the atmosphere and the interaction between the body surface and the surrounding air with varying relative speeds and wind direction. Aerodynamic drag is usually insignificant at low vehicle speed but the magnitude of air resistance becomes considerable with rising speed. A vehicle with high drag resistance tends only marginally to hinder its acceleration but it does inhibit its maximum speed and increase the fuel consumption with increasing speed.

C. Bluff body aerodynamics

In aerodynamics, a bluff body is one which has a length in the flow direction close or equal to that perpendicular to the flow direction. This spawns the characteristic that the contribution of skin friction is much lower than that of pressure to the integrated force acting on the body. Even a streamlined body such as an airfoil behaves much like a bluff body at large angles of incidence. A circular cylinder is a paradigm often employed for studying bluff body flows.

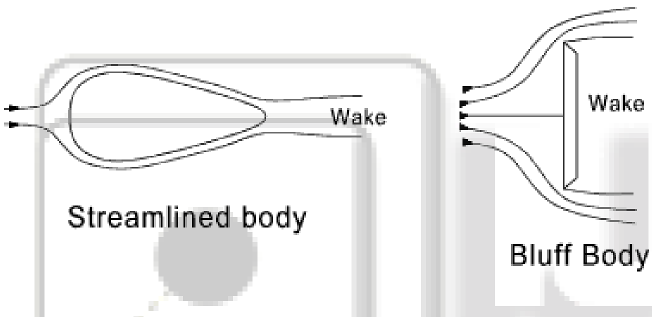


Fig.2:- The wake create by streamlined and bluff body

D. Aerodynamic forces on a body

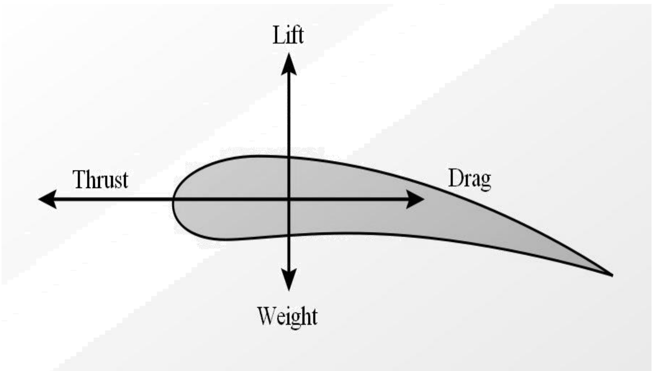


Fig.3:- Different forces acting on body

1) Lift

It is the sum of all fluid dynamic forces on a body normal to the direction of external flow around the body. Lift is caused by Bernoulli's effect which states that air must flow over a long path in order to cover the same displacement in the same amount of time. This creates a low pressure area over the long edge of object as a result a low pressure region is formed over the aerofoil and a high pressure region is formed below the aerofoil, it is this difference in pressure that creates the object to rise.

$$F = \frac{1}{2} C_L d V^2 A$$

Where:

C_L= Coefficient of Lift,
d= Density of air,
V=Velocity of object relative to air,
A=Cross-sectional area of object,parallel to wind.

2) Drag

It is the sum of all external forces in the direction of fluid flow, so it acts opposite to the direction of the object. In other words drag can be explained as the force caused by turbulent airflow around an object that opposes the forward motion of the object through a gas or fluid.

$$F = \frac{1}{2} C_D d V^2 A$$

Where:

C_D= Coefficient of Drag,
d= Density of air,
V=Velocity of object relative to air,
A= cross section of frontal area.

3) Weight

It is actually just the weight of the object that is in motion i.e. the mass of the object multiplied by the magnitude of gravitational field. This weight has a significant effect on the acceleration of the object.

4) Thrust

When a body is in motion a drag force is created which opposes the motion of the object so thrust can be the force produce in opposite direction to drag that is higher than that of drag so that the body can move through the fluid. Thrust is a reaction force explained by Newton's second and third laws, The total force experienced by a system accelerating in mass "m" is equal and opposite to mass "m" times the acceleration experienced by that mass.

E. Autodesk Flow Design

Autodesk Flow Design (formerly project Falcon) simulates air flow around vehicles, Buildings, outdoor equipment, consumer products, or other objects of your choosing in a virtual wind tunnel. Flow Design technology is extremely geometry tolerant and easy to use, enabling you to begin seeing and understanding air flow behavior within seconds of starting the application. Project Falcon will be moving forward into the world as Autodesk Flow Design. Flow Design allows you to:

- Quickly and easily visualize how design will interact with the wind. See where wakes and high and low pressure regions will form in a virtual wind tunnel.
- Start with results. Flow design to see results almost immediately and then allows you to explore the effect of different conditions.
- Use your existing design data. Flow Design reads a variety of CAD file types and can accept everything from conceptual design all the way to

fully detailed models with little or no preparation necessary.

- Get results within CAD. Inventor and Revit users have the option to use Flow Design right inside their design environment; making is easier to get insight during the design process.
- Visualize results interactively by creating dynamic 2D and 3D flow line animations, vector plots, iso surfaces, and velocity or pressure results on planes.

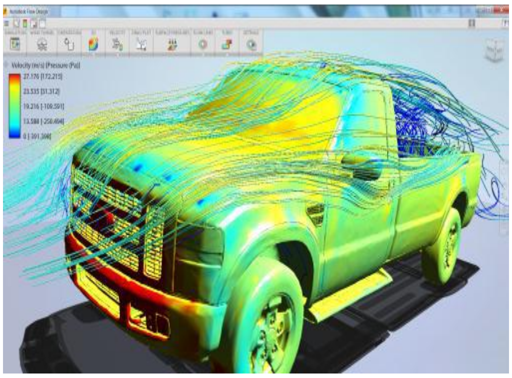


Fig.4:- Dynamic 3D flow line animations through body

In automotive applications Flow Design shows regions where air will recirculate, provide an understanding of the size and location of the wake region, and identifies high and low pressure regions on the body, and approximates the drag force and coefficient.

II. Design & Drag Force Analysis

Autodesk Flow Design (formerly project Falcon) simulates air flow around vehicles or other objects of you're choosing in a virtual wind tunnel. Flow Design technology is easy to use, enable you to begin seeing and understanding air flow behavior within seconds of starting the application.

Volvo 7900 bus

Fig. 5: - Volvo 7900 bus

Table 1: - Volvo bus specifications

Wheel base	5950 mm
Front over angle	2700 mm
Rear over angle	3450 mm
Overall length	12100 mm
Width	2550 mm
Height	3230 mm
Ground clearance	400 mm
Tyre	275/70R22.5''
GVW	19000 kg

Table 1: - Volvo bus specifications

We generate the 3D model of Volvo bus in Inventor Fusion 2013 from the above basic specifications.

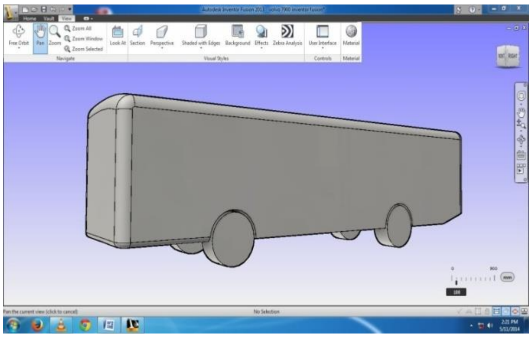


Fig.6:- Solid model of Volvo 7900 bus

After to generate 3D model of Volvo bus, we import this model in Flow Design 2014 with the following boundary conditions.

Boundary	Boundary conditions	Values
Inlet	Constant velocity Turbulent intensity Length scale	V=80 kmph V=100 kmph V=120 kmph
Outlet	Pressure outlet	Constant pressure 0 N/m^2
Bus body	No slip – stationary wall	-
Domain	Free stream	V=80 kmph V=100 kmph V=120 kmph

Table 2:- Boundary conditions for Volvo bus

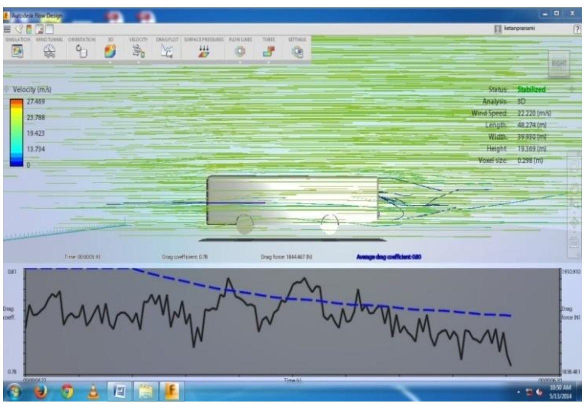


Fig.7:- Drag force 3D analysis of Volvo 7900 bus at velocity 80 kmph

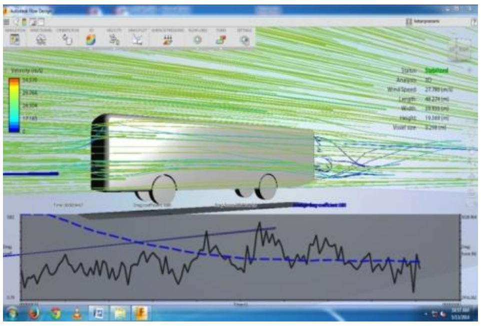


Fig.8:- Drag force 3D analysis of Volvo 7900 bus at velocity 100 kmph

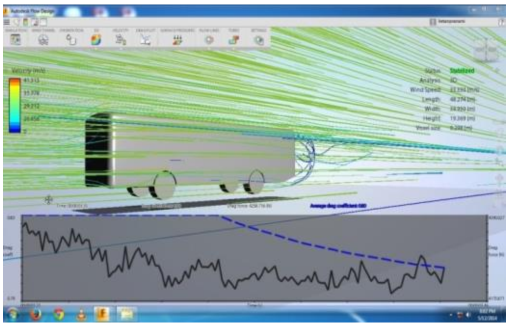


Fig.9:- Drag force 3D analysis of Volvo 7900 bus at velocity 120 kmph

We show the free vortex create behind the body is different with the different velocity of wind and also values of drag force increase with increase the speed of wind.

A. Improvement in shape of Volvo bus:-

Improvement of bus aerodynamic shape by drag reduction techniques of vehicle i.e. chamfering, rounded corners, tapered rear end etc. in Inventor Fusion 2013 and import this model in Flow Design for drag analysis.

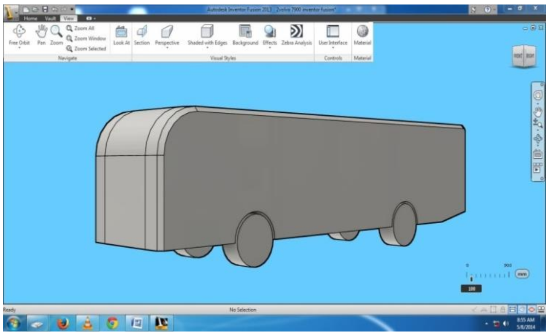


Fig.10:- Solid model of Volvo bus after change in shape

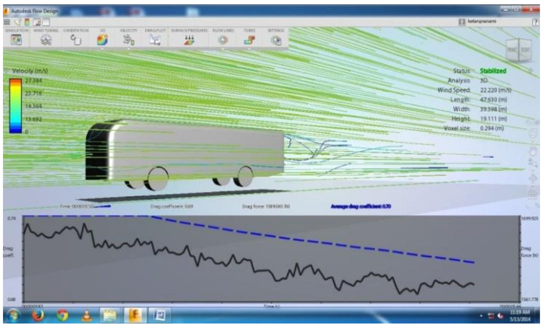


Fig.11:- Drag force 3D analysis after changed shape of Volvo

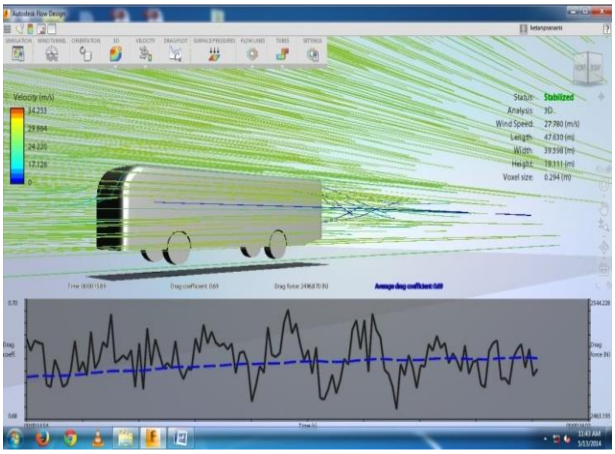


Fig.12:- Drag force 3D analysis after changed shape of Volvo bus (at 100 kmph)

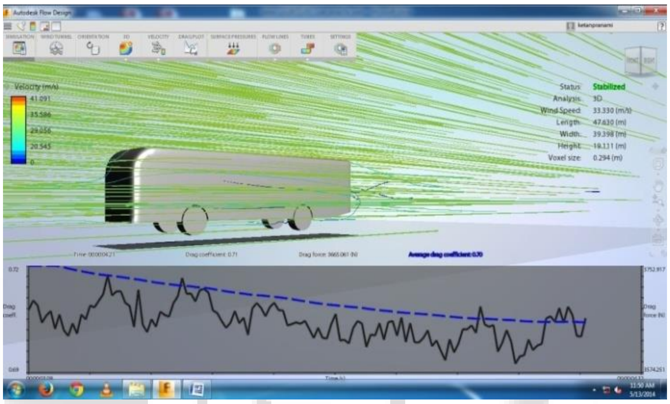


Fig.13:- Drag force 3D analysis after changed shape of Volvo bus (at 120 kmph)

The flow lines pass over the body and free vortex create near the rear side of the bus which increases with the speed of vehicles as shown in figure.

III. RESULTS AND DISCUSSION

A. Effect of drag force over Volvo bus before change in shape:-

Speed of wind (kmph)	Resolution in percentage					
	100%		150%		200%	
	F_d (N)	C_d	F_d (N)	C_d	F_d (N)	C_d
80	1844.467	0.80	1780.494	0.77	1896.123	0.82
100	2957.146		2765.283		2980.239	
120	4258.716		3995.254		4270.352	

Table 3:- Values of F_d & C_d of Volvo bus with various percentage of resolution

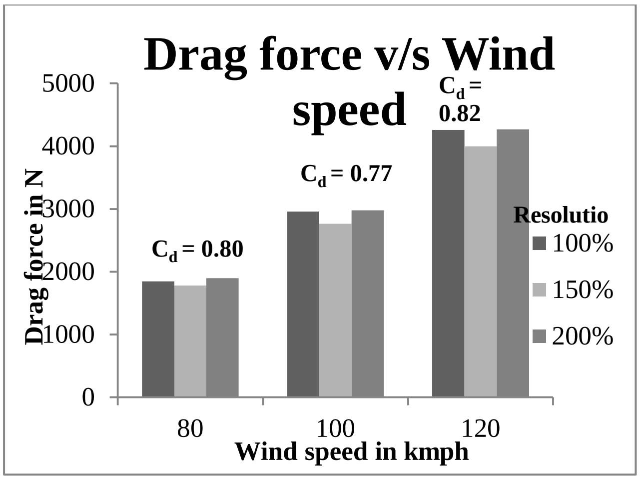


Fig. 14: Drag force v/s wind speed with different percentage of resolution

We show that drag force increases with increase the velocity of wind and vary with resolution percentage of Volvo bus before change in shape as shown in graph.

B. Effect of drag force over Volvo bus after change in shape:-

Speed of wind (kmph)	Resolution in percentage					
	100%		150%		200%	
	F_d (N)	C_d	F_d (N)	C_d	F_d (N)	C_d
80	1589.043	0.70	1542.245	0.68	1607.352	0.73
100	2496.870		2454.058		2528.541	
120	3665.061		3623.214		3698.265	

Table 4:- Values of Fd & Cd of Volvo bus after change in shape

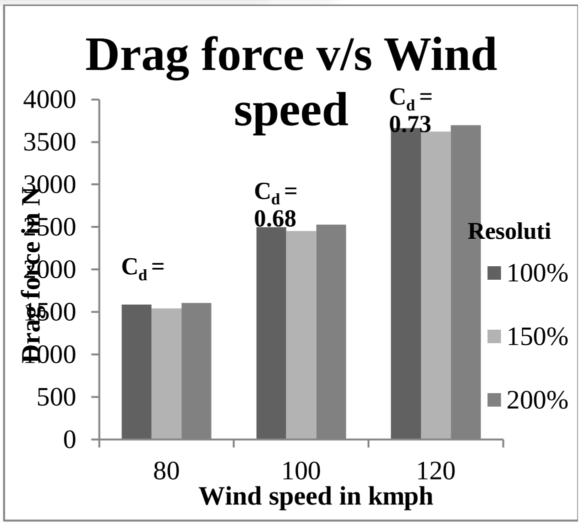


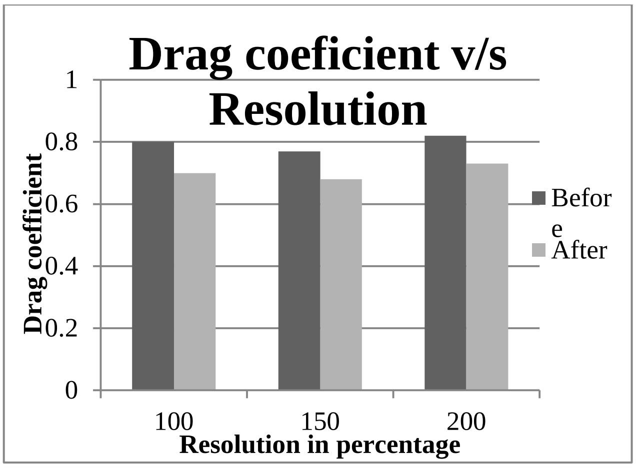
Fig. 15: Drag force v/s wind speed with different percentage of resolution after change in shape

From the above graph, we clearly show that drag force increase or decrease with different values of resolution i.e. 100, 150 & 200%.

C. Comparison of drag coefficient before and after change in shape:-

Resolution in percentage	Drag coefficient	
	Before	After
100	0.80	0.70
150	0.77	0.68
200	0.82	0.73

Table 5:- Comparison of drag coefficient before and after change in shape



Graph 3:- Comparison of drag coefficient w.r.t resolution

Above graph shows comparison of drag coefficient before and after change in shape with different percentage of resolution. At resolution of 150%, the value of drag coefficient is less than 100% resolution & after 150% it becomes increases.

The reduction of drag force as well as drag coefficient is about 12 to 15% from the above analysis.

IV. CONCLUSION

The aerodynamic design of vehicles is an area where a lot of improvements will appear in the near future, in concern of drag reduction. The guidelines pointed out in the text are of a general nature that can be implemented in most modern road going vehicles; Smooth vehicle shape, rounded corners, High rake angle for the windscreen, Tapered rear end, Minimized body seams, Optimized rear view mirrors and Smooth underbody. Wheel sides, wheel covers kept smooth and minimizing gap between wheelhouse and wheels.

The aerodynamics of road vehicles have been described in order to get notions about how to reduce drag resistance, and sometimes even theoretically ideal techniques have been recommended. However, we design an automobile, have to deal with a whole lot of other performance, functionality and styling issues which sometimes, and not so few, still tend to rule when meeting incompatibilities with aerodynamic ones.

We have completed successfully the analysis of Volvo bus in 3-D by using CFD software Flow Design 2014. By taking in account the boundary conditions; velocity, pressure, density of air and the frontal area of the bus, we can design the new shape by the help of drag reduction techniques of the bus in 2D as well as in 3D in the future. Thus we will be able to produce the vehicle's body shape with optimum frontal area that will offer less resistance to the air moving in the opposite direction

Some of these incompatibilities are very hard to overcome since some of those non aerodynamic characteristics of a vehicle often have an exceptionally narrow range of possible alternatives.

Keeping to these guidelines we should make it possible to have commercial vehicles with a CD value between 0.50 and 0.55 in the coming future.

REFERENCES

[1] F. R. Grosche, G.E.A. Meier "Research at DLR Gottingen on bluff body aerodynamics, drag reduction by wake ventilation and active flow control", Journal of Wind Engineering and Industrial Aerodynamics 89 (2001) 1201–1218
[2] Simon Watkins, Gioacchino Vino "The effect of vehicle spacing on the aerodynamics of a representative car shape" Journal of Wind Engineering and Industrial Aerodynamics 96 (2008) 1232–1239
[3] M.Gohlke, J.F.Beaudoin, M.Amielh, F.Anselmet "Shape influence on mean forces applied on a ground vehicle under steady cross-wind" Journal of Wind Engineering and Industrial Aerodynamics 98 (2010) 386–391
[4] Mahmoud Khaled, Hicham El Hage, Fabien Harambat, Hassan Peerhossaini "Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body" Journal of Wind Engineering and Industrial Aerodynamics 107–108 (2012) 36–47
[5] Andras Gulyas, Agnes Bodor, Tamas Regert, Imre M. Jánosi "PIV measurement of the flow past a generic car body with wheels at LES applicable Reynolds number" International Journal of Heat and Fluid Flow (2013)
[6] Chien-Hsiung Tsai, Lung-Ming Fu, Chang-Hsien Tai, Yen-Loung Huang, Jik-Chang Leong "Computational aero-acoustic analysis of a passenger car with a rear spoiler" Applied Mathematical Modeling 33 (2009) 3661–3673
[7] Subrata Roy, Pradeep Srinivasan "External flow analysis of a truck for drag reduction" Society of Automotive Engineers, Inc.
[8] A.Muthuvel, M.K.Murthi, Sachin.N.P, Vinay.M.Koshy, S.Sakthi, E.Selvakumar "Aerodynamic exterior body design of bus" International Journal of Scientific & Engineering Research, Volume 4, Issue 7, July-2013
[9] Sachin Thorat, G.Amba Prasad Rao "computational analysis of intercity bus with improved aesthetics and aerodynamic performance on Indian roads" International Journal of Advanced Engineering Technology.
[10] www.thetruthaboutcars.com
[11] Google Images
[12] Wikipedia – The free encyclopedia
[13] A Text Book on Automobile Chassis and Body Engineering by Sri. N.R.HEMA KUMAR.
[14] The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains by Rose McCallen, Fred Browand, James Ross
[15] www.autodesk.com
[16] https://grabcad.com
[17] www.3Dcadbrowser.com
[18] www.volvobuses.com