International Journal of Engineering Science Invention
ISSN (Online): 2319 – 6734, ISSN (Print): 2319 – 6726
www.ijesi.org ||Volume 5 Issue 11|| November 2016 || PP. 66-67

Computational Aerodynamics Research and Vehicle Engineering Development (CAR-VED)

Ms.SujataShenkar[1], Mr. Sanjay D. Nikhade [2], Mr.Sagarkumar Banerjee [3], Mr.AmarjeetKunal[4], Mr. Santosh Mote [5], Mr. Mustufa Sadikot[6]
Department of Automobile Engineering, SavitribaiPhule Pune University,Pune, India

Abstract: *Many Persons, both from industry and also private individuals have performed research in regards to this new issue. Many have performed research on aerodynamics on certain portions of the vehicle and also on effects of shape of the body and other technologies used such as Computational Fluid Dynamics and Wind tunnel Testing.The effects of these studies is seen in the industry today. Not so long ago,the vehicles were having shapes lose to boxes and today beautiful curves dominate the vehicles bodies. These curves not only help in the beauty of the vehicle but also help the vehicle in terms of aerodynamics and fuel efficiency. In this paper we would like to highlight some important topics related with aerodynamics and how they affect the drag of the vehicles. We shall also discuss on methods used in the industry today to calculate the aerodynamic efficiency of the vehicles and their effects.*

Keywords: *Aerodynamics, Vortex and Wakes, Co-Efficient Of Drag, Computational Fluid Dynamics, Wind Tunnel*

I. Introduction

Environmental issues and increasing fuel prices are the driving forces for the auto manufacturers to develop more fuel efficient vehicles with lower emissions; large investments are aimed at minimizing the power needed for propulsion that is new downsized engines with new aerodynamics devices for drag reduction. For passenger cars the aerodynamic drag force is the dominant resistance force at higher velocity .The car body is often optimized for reducing the drag resistance. To explain the aerodynamic force in a simplified manner the resisting drag originates from the pressure difference between the stagnation pressure in the front and the base pressure at the rear. By reducing the difference between pressures the drag force will be reduced hence the fuel consumption will be reduced.

Aerodynamics is the study of forces and the resulting motion of objects through the air. For several decades, cars have been designed with aerodynamics in mind, and carmakers have come up with a variety of innovations that make cutting through that "wall" of air easier and less of an impact on daily driving. Essentially, having a car designed with airflow in mind means it has less difficulty accelerating and can achieve better fuel economy numbers because the engine doesn't have to work nearly as hard to push the car through the wall of air.Engineers have developed several ways of doing this.

For instance, more rounded designs and shapes on the exterior of the vehicle are crafted to channel air in a way so that it flows around the car with the least resistance possible. Some high-performance cars even have parts that move air smoothly across the underside of the car. Many also include a **spoiler** -- also known as a **rear wing** -- to keep the air from lifting the car's wheels and making it unstable at high speeds

One of the main causes of aerodynamic drag for sedan vehicles is the separation of flow near the vehicle's rear end. To delay flow separation, bump-shaped vortex generators are tested for application to the roof end of a sedan. Commonly used on aircraft to prevent flow separation, vortex generators themselves create drag, but they also reduce drag by preventing flow separation at downstream. The overall effect of vortex generators can be calculated by totalling the positive and negative effects. Since this effect depends on the shape and size of vortex generators, those on the vehicle roof are optimized.The aerodynamicsof a vehicle primarily depends on the vortices and wakes produced around the vehicle at different speeds.

A vortex is a region in a fluid in which the flow is rotating around an axis line, which may be straight or curved. Vortices are a major component of turbulent flow. The distribution of velocity, vortices (the curl of the flow velocity), as well as the concept of circulation are used to characterize vortices. In most vortices, the fluid flow velocity is greatest next to its axis and decreases in inverse proportion to the distance from the axis. Wake turbulence is turbulence that forms behind an vehicle as it passes through the air. Lift, drag are generated by high pressure below the wing and low pressure above the wing. As the high-pressure air moves around the wingtip to the low pressure, (high pressure always moves towards lower pressure areas) the air rotates, or creates a horizontal "tornado" behind the wings. This tornado sinks lower and lower until it dissipates.

Thu the shape of the body affects the aerodynamics. It is the primary reason for formation of wakes and drags in the movement of the vehicle. Also it is the primary contributor to the drag,. This affects the vehicles drag co-efficient. The drag coefficient of any object comprises the effects of the two basic contributors to fluid dynamic drag: skin friction and form dragThe drag coefficient of a complete structure such as an aircraft also includes the effects of interference drag.

II. How To Calculate The Aerodynamics

1) Computational Fluid Dynamics:

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, The fundamental basis of almost all CFD problems are the Navier–Stokes equations, The simulation is started and the equations are solved iteratively as a steady-state or transient. Finally a postprocessor is used for the analysis and visualization of the resulting solution.

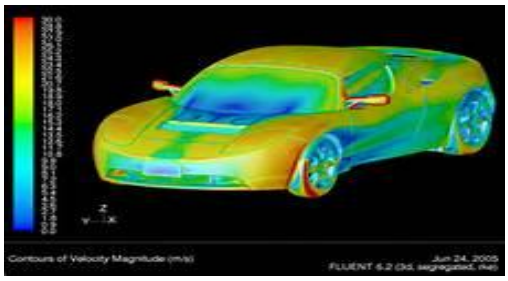

Figure 1 – CFD Analysis Software

2) Wind Tunnel Testing:

A wind tunnel is a tool used in aerodynamic research to study the effects of air moving past solid objects. A wind tunnel consists of a tubular passage with the object under test mounted in the middle. Air is made to move past the object by a powerful fan system or other means. The test object, often called a wind tunnel model, is instrumented with suitable sensors to measure aerodynamic forces, pressure distribution, or other aerodynamic-related characteristics. Pressure across the surfaces of the model can be measured if the model includes pressure taps. This can be useful for pressure-dominated phenomena, but this only accounts for normal forces on the body

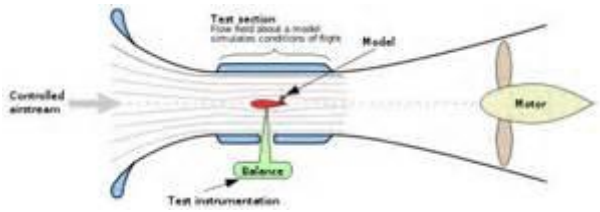

Figure 2 – Wind Tunnel Construction

III. Conclusions

Aerodynamics plays a crucial role in the role of passenger car fuel economy. Hence using all this knowledge and all these new technologies we are able to achieve attractive designs of vehicles along with the generation of streamline airflow around the vehicle which helps us achieve better economy and hence reduces the emissions of the vehicle

References

[1] Aerodynamic Analysis Of Drag Reduction Devices On The Underbody Of The Saab 9-3 By Using CFD,Chalmer's University Of Technology
[2] CFD Analysis Over A Car Body,MCADCenter, MSRSAS, Bangalore
[3] CFD Analysis of Air Flow Interactions in Vehicle Platoons,School of Aerospace, Mechanical and Manufacturing Engineering,RMIT University
[4] Aerodynamic Drag Reduction In A Passenger Vehicle Using Vortex Generator With Varying Yaw Angles .Department Of Automobile Engineering, Anna University Of Technology, Tiruchirappalli, Tamil Nadu, India
[5] Aerodynamic Drag Reduction Of A Car By Vortex Generation,International Journal Of Mechanical Engineering ISSN : 2277-7059 Volume 2
[6] www.wikipedia.org