Effect of Aerodynamic Force in Modern Cars

M.Venkatasudhahar[1], **T.Manvelraj**[2],

[1]*Dept. of Mechanical Engineering*

CMJ University, Shillong, Meghalaya

[1]kmrflowers@gmail.com

[2]*Dept. of Mechanical Engineering*

[2]*Siva Institute of Frontier TechnologyThiruvallur, Tamil Nadu*

[2]manvelraj@rediffmail.com

Abstract- **The history of fluid dynamics is dominated by the quest to predict forces on a body moving through a fluid – ships moving through water, and in the nineteenth and twentieth centuries, aircraft moving through air, to name just a few examples. Indeed, newton's treatment of fluid flow in his principia was oriented in part toward the prediction of forces on an inclined surface. The calculation of aerodynamic and hydrodynamic forces still remains a central thrust of modern fluid dynamics. Therefore the fundamentals of compressible flow will be applied to the practical calculation of aerodynamic forces on the passenger car. In this thesis, the detailed aerodynamic analysis of car is to be carried out. Actually at the rear side pressure gradient prevails over the momentum gradient, this causes a reverse force or drag force to act of the rear side of the car. This should be analyzed by using GAMBIT as modeling software, CFD as analyzing software**

Keywords- **Aerodynamic, *Drag, GAMBIT, CFD***

I. INTRODUCTION

This study concerns about the airflow around the vehicle body. At a speed of about 70 km/hr. aerodynamic drag exceeds to 50% of total resistance to motion and above 100 km/hr. It is the most important factor. Aerodynamic forces are Lift force, Side force or cross wind force and Drag force. Aerodynamic moments are rolling moment, Pitching moment and Yawing moment.

Drag is the aerodynamic forces that oppose a car's motion through the air. Drag is generated by every part of the car (even the engines). Drag and lift are two aerodynamic forces acting on anybody traveling through the air whether it's an airplane, car, or truck. Unlike aircraft where both lift and drag are pretty much equally important, drag is far more important for vehicles, unless it's a race car or high performance sports car traveling at very high speeds. For the latter, lift is to be avoided and thus aerodynamic devices like air dams, spoilers, and wings are used to provide downward forces so the vehicle hugs the road.

Overcoming drag, the force that opposes forward motion, represents probably the largest load on the engine and thus the largest contributor to fuel usage and fuel economy. Overcoming drag, especially at higher speeds, requires much more power than other power consumers like tire rolling resistance, engine and power train friction, and powering accessories. Drag force is a function of the vehicle's drag coefficient "C_d". Frontal area (A), air density, and velocity (V) squared. While nothing can be done about reducing air density, the other factors can be reduced to improve fuel economy. As per the concept of aerodynamics and fuel economy both are all co-related by "C_d". Essentially, this is how easily a vehicle moves through the air, though drag isn't the only factor that is considered. "C_d" x Frontal Area (A) x Density of Air (ρ_a) x Speed Squared (V^2)

II. REVIEW

In recent years many researchers have come up with different ways to reduce drag Kazuo Yanagimoto, Kunio Nakagawa et al. [1] were discussed about the research on Aerodynamic drag reduction is one of the most important aspects of enhancing overall vehicle performance. Many car manufacturers have been working to establish drag reduction techniques. This paper describes the development process of a new small specialty car which achieved coefficient of drag (C\dD) of 0.25. SumontroSinha et al.[2]Gave the new method for reducing aerodynamic drag of trucks and vans has been developed. It uses Deturbulator tape to transform separated turbulent wakes into stagnant virtually solid streamlining extensions attached to the vehicle. Constrained mode flow-induced surface oscillations of the 100-¿m thick, passive, flexible-surface Deturbulator tape attenuates turbulent mixing by driving the turbulence to a pre-selected high frequency in the dissipation range. Wind tunnel tests indicated 80% drag reduction. Road tests on a Deturbulator treated minivan and pickup truck increased highway fuel economy 15-20% from reduced drag. 6% reduction in overall fuel consumption was obtained for an operational Class-8 tractor-semitrailer. Jeff, Steve Windsor et al. [3] Door mirrors have a small but measurable contribution to the overall aerodynamic drag of a road vehicle. Typically for passenger cars and SUVs this is in the range 2.5 - 5%. It can be difficult to refine the shape of door mirrors as the improvements are, sometimes, too small to measure with any accuracy. A test rig has been developed which allows a full size door mirror to be tested in a model wind tunnel facility, which has better balance resolution, where the mirror is mounted to a partial vehicle body. This also results in a faster and cheaper method to develop shapes for door mirrors. The rig is described and the initial correlation tests presented. The limitations of the rig and some further applications are discussed. Jeff, Geoff Le Good et al. [4]. Present the research on vortex structure in the wake of a car creates drag. In the case of a simple wing this drag component is well defined as a function of lift, but for road vehicles the relationship is more complex. The backlight surface has been shown to be a significant source of vortex drag and in this paper the influence of backlight aspect ratio on both vortex and base drag is investigated. The vortex drag factor is found to be independent of aspect ratio, while the base drag component is shown to be dependent on the ratio of base to frontal area..Satya Prasad, Simon Watkinset al.[5] developed Within the scope of wheel drag investigation, a mirror image method for simulating relative motions of wheels and moving ground for road vehicle aerodynamics was developed using a double-symmetry technique. An initial study using no-wheeled bluff body demonstrated feasibility. The current paper further extends this technique to a wheeled model vehicle. The concept, measuring approach and EFD and CFD evaluations are described. The double-symmetry approach enabled simulated on-road vehicle drag tests. With the ability to replicate deformations at tyre contact patch, the approach presents evaluation of rotating wheel effects on accessories and their actual contribution to the vehicle road load. Min-Ho Kim et al. [6] has done this study; a numerical simulation has been carried out for three-dimensional turbulent flows around a bluff-based bus-like body and actual bus body. The first step of this study is to verify the effectiveness of the CFD analysis. In the second step, to reduce the drag of the actual bus model, parameter studies are performed with attention to effective utilization of the rear-spoiler equipped at the roof-end of upper body. From the results of this study, it is clear that the adoption of RNG k-ε turbulence model and nonlinear quadratic turbulence model with the second order accurate discretisation scheme predicts, fairly well, the aerodynamic coefficients. The results also show that the aerodynamic drag for a commercial bus can be reduced by 14% with the use of a drag reduction device.

III. AUTOMOBILE WIND TUNNELS

Experiments to obtain aerodynamic parameters that affect automobile performance, handling, engine cooling, brake cooling, and wind noise are made with either scale models or at full scale in large tunnels. Unlike the scale of aircraft, it is quite feasible and is common practice to build tunnels that accommodate the use of full-scale automobiles. It is also advantageous to use moderate scale such as 0.25-0.4 models and conduct experiments at full-scale Reynolds numbers.

A.External Flows

There are two distinct classes of wind tunnels involved in aerodynamic experiments on automobiles. The one that is the main focus of this thesis is concerned first and foremost with the external flow characteristics. All of the major automobile manufacturersworldwide either own or have regular access to wind tunnels for such experiments of both reduced scale and full scale automobiles.

1. variable-speed fan: the 5,130-hp, 22-foot-tall fan turns at up to 360 rpm to create winds more powerful than a category 5 hurricane.

2. Heat exchanger: this keeps air temperature at a rock-solid 75 degrees Fahrenheit, so eggheads can get consistent results. 3 Rolling roads: a thin cushion of air lets the 1 mm-thick stainless steel belt run at 180 mph without burning up from the friction.

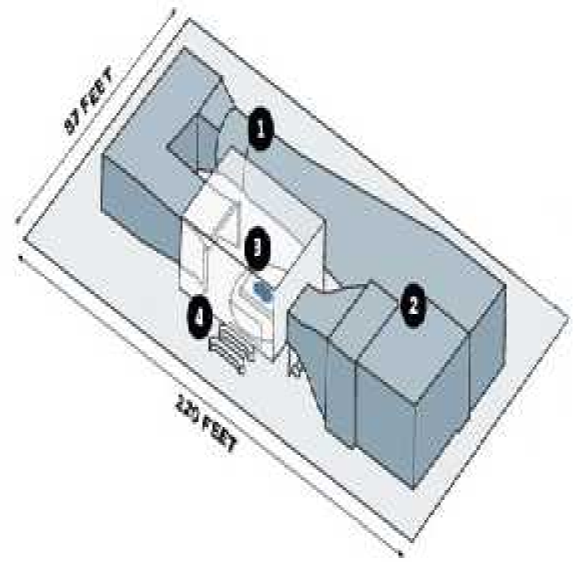

Fig. 1. Automobile wind tunnel

4 Control room: Team engineers inside will scrutinize their car's behavior as operators tweak factors like ground clearance and yaw.

A.Wind Tunnel Testing

Fig.2 Hi-speed car with wind flow

A wind tunnel is a research tool or a testing laboratory with a controlled environment used to understand aerodynamically what is happening to a particular shape or object. Or in this case... TATA INDIGO we taken in the consideration of the drag force. It is used to quantify and validate what is going on aerodynamically to the vehicle by measuring forces exerted on the body through a 6 component balance underneath the floor that the vehicle is fixed to by use of adjustable wheel pads.

Adding a rear wing to your car might gain you down force, but a wind tunnel will tell you in fact if it is, how much, and how it affected the overall front to rear balance of your car. Each test will output the drag, down force (front & rear), and side force (front & rear), yaw/pitch/roll moments, in order to see the big picture of what is really happening for any given configuration. Even though we test at 85mph we can scale the force data to virtually any speed you would see with your vehicle to see what the aero forces would be at that speed.

Have you ever wondered why wind tunnel testing seems to be so vital amongst the top race teams, and why every time you turn on a NASCAR or F1 race someone is talking about how they just had their car in the wind tunnel? It is hard to imagine that something you cannot see could have such a significant influence on the performance of a vehicle. More and more racers are catching on to the importance of aerodynamic testing and the value it can bring to just about any application. Whether you have a specific handling issue, or you just looking for more top speed, a wind tunnel is a good place to make some measurable differences.

A.Aerodynamic Drag: Aerodynamic drag is a force in the opposite direction of the vehicles forward movement. You need enough power and torque to overcome this force and move the air around the vehicle. At low speeds drag is not a major concern, but as the speed increases, the aerodynamic drag will increase with the square of the speed (speed^2). This means, in order to double the speed you will increase the drag force by four times.

- If your drag force is 70 lbs at 50 mph, then the force at 100mph is 280 lbs. At 200 mph the drag force would be 1120 lbs, and so on.

Just when you get over that fact, there is one more that will really get your attention. Horsepower is affected by the speed as well, but HP = Drag force x velocity. This means as the speed increases the Horsepower required is now the cube of the speed. Every time the speed doubles the HP required to overcome the drag forces is increased by 8 times!!

- If it requires 15 HP to overcome the drag forces at 50 mph, then it would take 120 HP to overcome the drag force at 100 mph. To reach 200 mph you would need 960 HP. 300mph would require 2700HP!! This is why you can add 50, even 100HP to your engine and not see a big gain in top speed.

Example of how a 15% reduction in C_D would translate into Horse Power and Drag Force.

IV. COMPUTATIONAL ANALYSIS

There are many CFD packages available in the market today. Fluent and Gambit packages improve the capability of analyzing a flow and meshing the element respectively. Fluent is one of the most popular flow analysis packages in use today. The analysis of car aerodynamics can present a significant challenge, requiring the simulation of many different configurations and positions of both car and new attachment. Wind tunnel analysis with a rolling road is often impractical. The deployment of CFD Fluent 6.1 and Gambit within the design process, however, enables such studies to be carried out with relative case. When air flows over the surface of a car, a boundary layer forms where there is a large velocity gradient. In order to capture this phenomena correctly, the mesh around the surface of the body must be very fine. To perform this boundary layer study, I will be creating and solving two different meshes. All of the mesh parameters will be staying the same between the two meshes except for the boundary layers. The possible values for creating geometry vertices are using this vertices do the face of model in gambit then creating 3 D model. The car has a length (x L), a depth (y L), and a height (z L).The car length is aligned with the x axis, the depth with the *y* axis, and the height with the *z* axis. The flow is assumed to be symmetric about a plane that bisects the car in the y-direction and therefore only half the car is modeled. One corner of the car is assumed to lie at the origin. Table shows the car geometry. Generate the outer limits of the computational domain.

In Operation, In Create volume at enter -10, 10, and 0 for x, y, and z respectively. Then an appropriate vertex is created. In a similar manner, create the volume for the bottom left, bottom right, top right, middle left, and middle right, at locations

Height	=	20 m
Length	=	35 m
Breath	=	20 m

Now create volume of domain then it's subtract with the model and creating the flow volume of car model in domain volume.

A.Meshing

An inflated boundary of prismatic elements was used near the car surface to improve spatial resolution and gain a better understanding of boundary layer phenomena. An unstructured mesh with polyhedral elements was used for volume meshing. Simulations were carried out with the turbulence model, coupled with a blend factor of 0.5 for the advection scheme.

The computational mesh was constructed automatically using polyhedral cells mesh, surrounded at solid boundaries by three prismatic extrusion layers. Because polyhedral cells fill space more efficiently than tetrahedral elements, fewer cells were required than might otherwise have been needed, significantly aiding the goal of using a small desktop machine to perform such aerodynamic analyses.

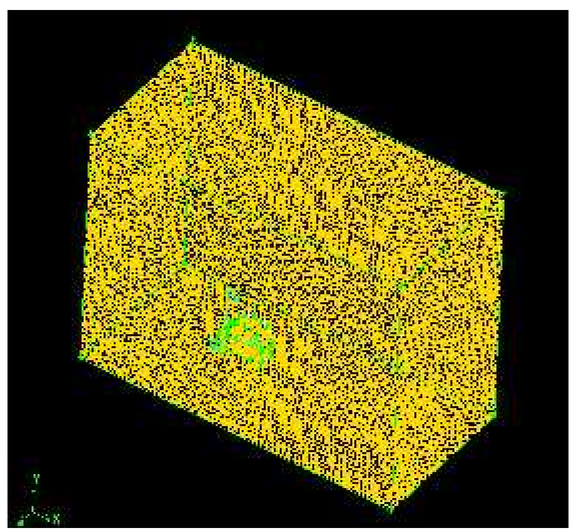

Fig.3. 3D view of final polyhedral mesh with volume source visible around the car

A model that has already been meshed and it has only 130’000 polyhedral cells. Note that at least 5 million cells, with hexagonal in the near-wall regions, would be necessary to obtain reliable and detailed results in such a case.

The computational domain extends far upstream of the car where the boundary condition will be a velocity inlet. The top and bottom of the computational domain are "periodic" boundary conditions, which mean that whatever flows out of the top goes directly into the bottom. It is assumed that since the outer limits of the computational domain are so far apart, this car behaves as if in an infinite free stream. In order to adequately resolve the boundary layer along the car wall, grid points will be clustered near the wall. Far away from walls, where the flow does not have large velocity gradients, the grid points can be very far apart. A hybrid grid will be used in this problem. Grid adaptation within the flow solver, Fluent, will increase the grid density even more near the wall and wherever else needed.

B. Define the Geometry

When the geometry was defined in the creation of the computational mesh, all faces of the domain were assigned names. The names of the inlet and outlet planes (at *x* = 0 and *x* = *L)* are front face and back face of domain as velocity inlet and pressure outlet respectively. The names of the planes at *y* = *L*, *z=0, and z=L* are outer wall as wall. The names of the model are car as a wall. And bottom face is defined as road.

Now create volume of domain then it’s subtract with the model and creating the flow volume of car model in domain volume.

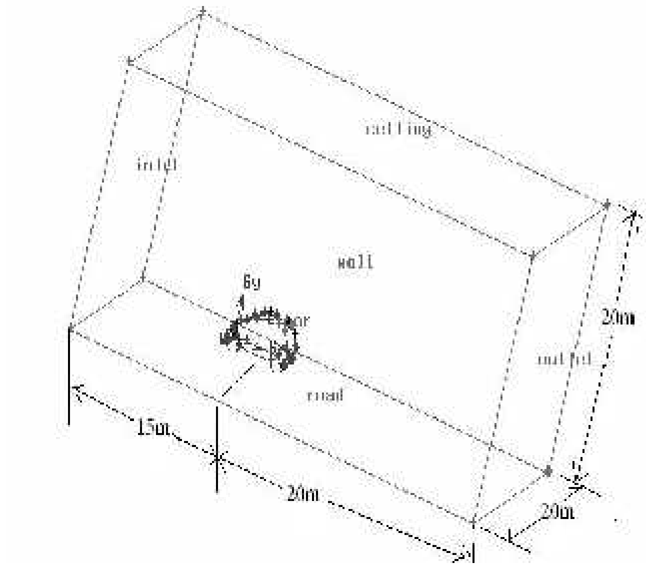

Fig 4.Defining the geometry

Dimension of the domain,

Height = 20 m

Length = 35 m

Breath = 20m

C. Pre-Processing

Fluent reads the grid with about 130’000 cells from gambit file. Grid Check is sure there is no negative volume or face area and there is no warning of any kind. Define the model of Solver is Segregated for Continuity equation is first solved for all cells, then Momentum and then turbulences. This works well for incompressible and moderate compressible flow. Applying the Implicit for each equation is solved for all cells together with actual dates. The implicit solver brings faster convergence. Define the model as 3D and Steady (car velocity will be constant and we don’t Expect instabilities). It is Absolute there is no moving mesh zone in the mesh. Define the Model is Viscous as k-epsilon for a robust and efficient turbulent model which gives good results in most cases where turbulences have an isotropic repartition. Define the model of energy equation.

The material is air and its properties are

Density (ρ_a) = 1.225 kg/m^3

Viscosity (μ_a) = 1.464e-5 kg/m/s

Those values correspond to the ICAO norm. Fluent means dynamic viscosity as we consider air as incompressible and are not looking for heat transfer problematic, we don't need to specify properties.

D. Operating Conditions

Let the 101325 Pa which corresponds to the ICAO-Norm. Fluent works with relative pressure.

Boundary Conditions

Car model is "wall" with "car" (in the field "Zone Name").We considers our model as a wind-tunnel model. So the car is a stationary wall, the viscosity makes the air stick at the car coachwork, so no slip the coachwork is very smooth, so a roughness of zero. Ceiling of the wind-tunnel and Side wall of the wind-tunnel are specified shear for this will allow the air to slip on the ceiling wall. This is not realistic, but so, we can use a very coarse mesh without boundary layer problems.Road is specified as Moving Wall. As the car doesn't move, the road will have a velocity in the positive x-direction, so that the flow under the car will be correctly modeled. Velocity is 25 m/s in the Speed field. Correspond to 90km/h. and 0.05m in the "Roughness Height" field. Inlet is 25 m/s in the "Velocity Magnitude" field as the car doesn't move, the air has to in the positive x-direction. Outlet is Zero Pa in the "Gauge pressure" field means we have atmospheric pressure at the outlet. Initialize the "Compute From"-inlet. This will attribute to all cells of the model, the velocity, pressure and turbulences values that we defined for the inlet.

E.Processing

Calculate a solution for using corresponding selected condition.

F.Post-Processing

Normally we would have to enable better numerical schemes (2nd or 3rd order and run until a much better convergence of the flow solution is reached, but this would take about 3 hours with this case and about 2 weeks with an adequate mesh refinement). So we simply visualize the actual results.

Six important results were obtained from the analysis.

1. Drag force variation along the car model.
2. Calculated co-efficient of drag.
3. Static pressure variation along the car model.
4. Total pressure variation along the car model.
5. Velocity vectors.

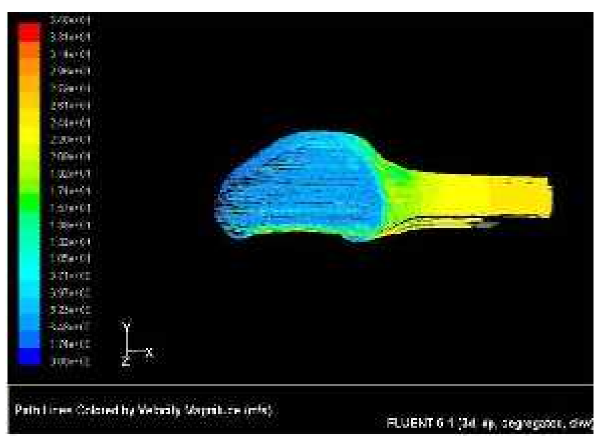

Fig.5. Path line velocity and magnitude variation

Table 1. Comparison of Drag between existing Car and Modified car

Conditions of car / Parameters	Car before Modification	Car after Modification
Drag of the Car	592.2357 N	568.8502 N
C_d of the car	0.4616	0.4566

The effect of attachment is estimated that the separation point is shifted to downstream, which in turn narrows the flow separation region. The flow field was thus investigated in order to verify the correctness of this estimation Modified the car model is reducing drag compare without modification.

V. RESULT AND CONCLUSION

The cad model was analyzed by using the software and, wind tunnel test also conducted by the help of scaled model both the results are tabulated for the car model. So, this attachment tends to reduce the drag coefficient and also upstream of the flow separation point in order to control separation of airflow above the rear corner of car and improve the aerodynamic characteristics. Application of the attachment shows a 0.005 reduction in drag coefficient.

As a result of the verifications, it is confirmed that the attachment creates stream wise vortices, and may reduce the fuel consumption of car. We could predict that the attachment cause the pressure of the vehicle's entire rear surface to increase therefore decreasing drag.

VI. PROPOSED APPROACH

This thesis work could further be improved to a certain extent by conducting experimental analysis of this modification. Also a several prototype models could be made and analyzed in the automobile wind tunnel and CFD. Then the experimental results can be compared, verified and examined to make it a complete solution.

From those analyses, aerodynamic properties and aesthetics of the Car could be improved. Also optimum fuel efficiency may be achieved.

VII. REFERENCE

1. Shibata, H.(1983) 'MMC's Vehicle Wind Tunnel', Automobile Research Review (JARI) Vol. 5, No. 9.
2. Hucho, W. H.(1998) 'Aerodynamics of Road Vehicles', Fourth Edition, SAE International.
3. 'Wind Engineering and Industrial Aerodynamics', Vol.22, Elsevier Science Publishers BV, Amsterdam, 1986.
4. Florence (1991) 'Emission reduction in road vehicles by kinetic energy recuperation', Procs. ISATA conference on electric and hybrid vehicles.
5. Autumn (1992) 'Assessment and reduction of wind buffeting in an open cabriolet' Int. Jnl. of Vehicle Design, vol. 13, nos 5/6, pp. 486-493, ISSN 0143-3369.
6. I Mech E, NEC Birmingham(1994) 'Improving the wind environment in open cars Int. Conf. on Vehicle NVH and refinement, paper C487/021/94, pp. 73-78.