Numerical Prediction of A Passenger Car for Drag Reduction

V. Jayachandran [1], M. Samundeswari [2], R. Tamilarasan [3]
[1, 2, 3] Dept of MECH
[1, 2] Sembodai Rukmani Varatharajan Engineering College, Vedaranyam,Tamil Nadu, India.

Abstract- *Numerical prediction of incompressible turbulent flow has been performed on a passenger car body moving with a velocity of 11.11 m/s (40 km/hr). CATIA, 3D modeling software was used to model 3D surface modeling of the car. FLUENT, the computational fluid dynamics code, which incorporate k-ε turbulence model and segregated implicit solver was used to perform computation. The aerodynamic analysis was performed to study the flow behavior of the air over the car body. The analysis includes the contours of pressure and velocity that impacts the car body followed by an evaluation of the coefficient of lift and drag. In the present work, model of generic passenger car has been developed in CATIA and generated the wind tunnel and applied the boundary conditions in FLUENT platform then after testing and simulation has been performed for the evaluation of drag coefficient for passenger car. In another case, the aerodynamics of the most suitable design of tail plate is introduced and analyzed for the evaluation of drag coefficient for passenger car. The addition of tail plates results in a reduction of the drag-coefficient and lift coefficient in head-on wind. Rounding the edges partially reduces drag in head-on wind but does not bring about the significant improvements in the aerodynamic efficiency of the passenger car with tail plates, it can be obtained. Hence, the drag force can be reduced by using add on devices on vehicle and fuel economy, stability of a passenger car can be improved.*

Keywords- Drag , Tail plate, Hybrid Vehicle, Aerodynamics.

I. INTRODUCTION

The main purpose of the design of a car body is for containing and protection of the engine and accessories as well as the passenger. Layer of heavy gumming material is sprayed or brushed to the interior of the body panels. Usually the car bodies become less subjected to temperature changes due to these material acts as heat insulators. Initially an automobile body furnishing seats for the passengers was considered sufficient. But then closed car bodies became popular. With the passage of time, riding comfort with reference to seating, heating and ventilation became the target of attention.

Further due to increased cars, operation speeds of motor vehicles increased, which in turn necessitated special attention to streamlining the process of shaping the body to reduce air resistance as the engine move forward. In this case, curves instead of angles and flat surfaces are used on the body shaping. In the earlier models, the vertical front sections of the radiator and the windscreen offered considerable resistance to the car movement. Moreover, in the back of the car, air eddies formed tend to produce a drag. To overcome this air resistance, considerable power was required at intermediate and high speeds.

Wind test and actual road tests had conclusively proved that old styled closed cars had great resistance of wind of the car movement. It had been proved that if heavy end of the body (ie. rear) were placed to the front, the car would be operated more economically, by streamlining the body of the car, better speed characteristics and fuel efficiency were expected. The sloping lines provided on the streamlined car further permits it body to move more smoothly through the air due to pushing of the air up and around the car.

In comparisons to earlier models, air eddies are not formed behind the body of passenger car needs to be studied in term of aerodynamics losses.

In case of racing cars, where the speeds are of paramount importance, streamlining has great influence. For cars operating 70 to 80 kmph the advantages due to streaming are small. The fuel saving is also worthwhile only when the vehicle is operated continuously at high speed. With computational fluid dynamics codes it it is possible to study the behavior of airflow over the bodies and this will help to take critical decisions in the design processes of car body and other objects.

Aerodynamics issue is very critical to car makers. The demands for more efficient and faster car are always a priority for the marketing strategy. The importance of Aerodynamics to a Hybrid Electric Vehicle (HEV) is determination of drag estimation to know how much the car performance on the road against air resistance. It can be used to improve the stability, reducing noise and fuel consumption. In view of the fact that many of car makers, formulate research and continue to develop the HEV models focused on higher propulsion efficiency in order to integrate the energy saving by reduce the rolling resistance of wheel and reduce the drag by aerodynamically losses.

However, in this paper Computational Fluid Dynamics (CFD) analysis will be used as the technology of computer simulation to estimate the drag because it is cheaper than the conventional technique.

II. MODELLING OF HEV BODY AND SIMULATION ANALYSIS

Nowadays, CFD can be used as the technology of computer simulation has evolved to produce good preliminary results for designers before the actual car fabrication. Firstly, the car was dismantled and most of the components were reduced. The aim here was to reduce the overall weight of the car. As an electric vehicle the engine was removed and replaced with an electric motor as the new drive train of the car. It was designed with efficiency in mind. The importance of the research was to incorporate energy saving by various method of reducing traction friction from the wheel and also to reduce the drag by aerodynamic losses. Aerodynamic losses generally increase in quadratic with speed from fluid mechanics.

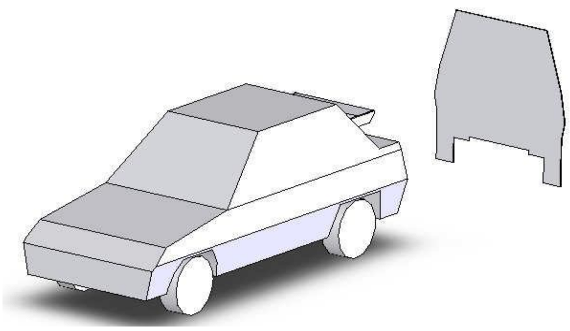

Fig 1: 3D View of a Car

As a Normal car produces lift at the rear end, an analysis of how much drag the car body can produce at different speeds is a main concern in this paper. The subdivision of drag is according to the regions on the body vehicle. But this consideration of type's classification is more difficult than an actual car, due to pressure and shear stresses more are not known with the resolution needed.

Based on the Ahmed body, for a generic car it can be divide into local drag contributions. Four geometric zones was distinguished for smooth body without attachment, there are; Front end, the rear slant, the base (i.e. vertical plane at the rear), the side panels and roof and under body. In three dimensional view the effected zones for drag are as shown in Figure.

For basic understanding in estimation of drag study on vehicle body, the external flow at the upper side of body is the best way as consideration for study in aerodynamic field.

The main concern for upper body is from the front end bonnet (hood) until the rear end (boot) or base area. Streamlines as imaginary line of flow can visualized as a pattern of air movement on the body vehicle. The Bernoulli equation theory or Venturi effect can be the basic of explanation the streamline of air velocity and the pressure over the car's body. Since the car body was modeled in CAD and refined it, the next stage is importing the CAD model into CFD software. CFD analysis was run with the COSMOSFloWorkstm software At this stage, the various car speeds was analyzed at ranging speed of 40 km/h to 110 km/h for every interval of 10 km/h. The boundary condition for this analysis is external flow with velocity moving over the car. Beside that, the types of flow considered are a mixture of laminar and turbulent flow for CFD model analysis. A strategy to meshing the model was by making automatic mesh analysis using CosmosFloWorkstm adaptive meshing capabilities. The mesh was enhanced when iterations was increased at 10 iteration, 50 iteration and 100 iteration. This increased the mesh count and narrowed the mesh location on different parts of the car more accurately to represent the flow.

III. COMPUTATIONAL WORK

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving mathematical equations that represent physical laws, using a numerical process. Similar fluctuations for pressure, temperature, and species concentration values.

The computation is performed using the finite-volume technique with upwind discretization to solve the two-dimensional compressible RANS equations. The air is considered to be a calorically perfect gas with constant ratio of specific heat. The space discretization is performed by a cell-centered formulation. To account for the directed propagation of information in the inviscid part of the equations, the advection upstream splitting method flux vector splitting is applied for the approximation of the convective flux functions. Higher-order accuracy for the upwind discretization and consistency with the central differences used for the diffusive term is achieved by the monotonic upstream scheme for conservation laws extrapolations. Time integration is performed by an explicit five stage Runge-Kutta time-stepping scheme.

To enhance convergence, a multi-grid method, implicit residuals smoothing, and local time stepping are applied.

Table 1. Solution Details

Solver	Coupled implicit formulation
Viscous model	k-epsilon
Fluid	Atm. air
Operating conditions	1 atm
Courant number	0.5

IV. CAR MODEL

The main geometric parameters of the car model are referred to Figure and the length unit is m. CATIA, 3D modeling software was used to model 3D surface modeling of the car. In the present work, model of generic passenger car has been developed in CATIA and generated the wind tunnel and applied the boundary conditions in FLUENT platform then after testing and simulation has been performed for the evaluation of drag coefficient for passenger car.

V. AERODYNAMIC FORMULAE

For lift

$$C_L = \frac{L}{1/2\rho V^2 S}$$

c_l − coefficient of lift

l − lift per unit span

ρ − density at std.atm (1.225kg/m^3)

v − velocity (40kmph)

s = span area

For drag

$$C_d = \frac{D}{1/2\rho V^2 S}$$

c_d − coefficient of drag

D − lift per unit span

ρ − density at std.atm (1.225kg/m^3)

v − velocity (40kmph)

s = span area

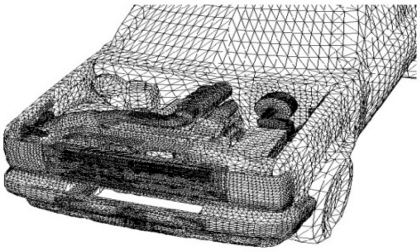

Fig 2: Unstructured Mesh

The computations have been performed using the un-structured grid. Our car model and domain is a 3-D. The dimension has been taken from the thesis, where the dimensions are in m. The meshing and cad data was done by the tool CATIA AND HYPERMESH.

VI. NUMERICAL ACCURACY ANALYSIS

The solution can be considered as converged after approximately 500 iterations, where the Courant number is 0.5. At this stage, the continuity, x-velocity, y - velocity and energy residuals, reach their minimum values after falling for over four orders of magnitude. The turbulence (k and Ɛ) residual have a five orders of magnitude decrease. An additional convergence criterion enforced in this current analysis requires the difference between computed inflow and outflow mass flux to drop below 0.5 per cent. The evaluation was performed using the UN-structured mesh. The performance of a grid sensitivity analysis confirmed that the grid resolution used here is sufficient.

Table 2. Conditions Details

velocity inlet	11.11 m/s
Pressure outlet	0 pascal
Total temperature	218 K
Operating conditions	101325 pascal

Aerodynamic Calculation (without tail plates)

For lift:

$$C_L = \frac{L}{1/2\rho V^2 S}$$

$$C_L = \frac{48.181}{\frac{1}{2} * 1.225 * 11.11^2 * 6.718}$$

$$C_L = 0.094$$

For drag

$$C_d = \frac{D}{1/2\rho V^2 S}$$

$$C_d = \frac{109.869}{\frac{1}{2} * 1.225 * 11.11^2 * 2.517}$$

$$C_d = 0.430$$

VII. RESULTS AND DISCUSSION

Though a considerable amount of data was gathered mainly in terms of line plots of static pressure, contours of velocity and turbulence contour. In the line plots, static pressure has been normalized with the free stream value. These plots are presented for the surfaces of car model, i.e. tyres, upper surface and domain. The aerodynamic phenomena such as lift and drag and flow separation are discussed.

Fig 3 Sketch of Car(Without tail plates)

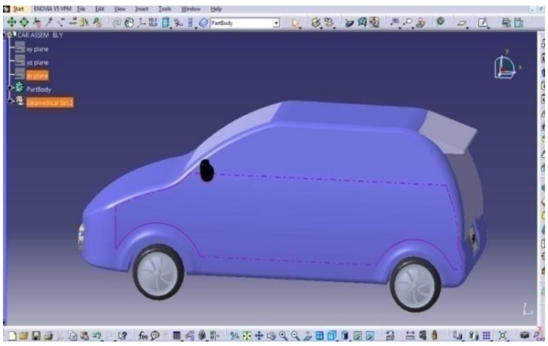

Fig 4 :- Sketch Of car(with tail plates)

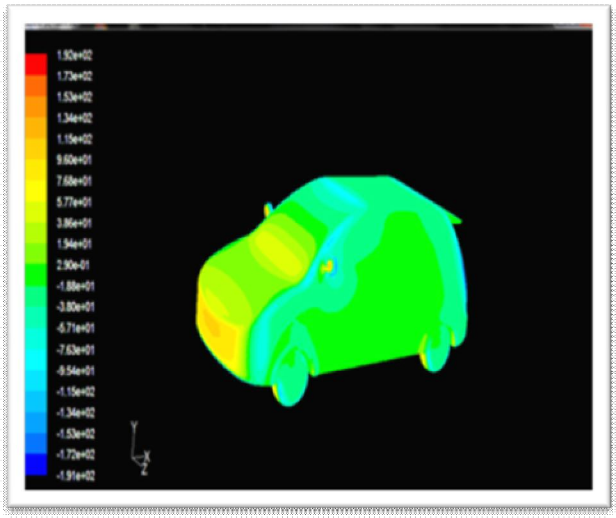

Fig 5: Pressure Contour with Tail Plate

AERODYNAMIC CALCULATION (with tail plates)

For lift

$$C_L = \frac{L}{1/2\rho V^2 S}$$

$$C_L = \frac{35.22}{\frac{1}{2} * 1.225 * 11.11^2 * 5.7}$$

$$C_L = 0.074$$

For drag

$$C_d = \frac{D}{1/2\rho V^2 S}$$

$$C_d = \frac{120.45}{\frac{1}{2} * 1.225 * 11.11^2 * 1.11}$$

$$C_d = 0.352$$

VIII. CONCLUSION

The main objective is to estimate the drag coefficient and flow visualization is achieved. Aerodynamics drag for my car is 0.430 and with tail plate is 0.352 at ranging velocity between 40km/h. The analysis shows aerodynamics drag in term of drag forces or drag coefficient proportionally increased to the square of velocity. The contour plot of velocity and pressure were shown the in aerodynamics drag analysis as a visualization analysis. The pattern of visualization for every velocity depict quite same either for velocity contour plot or pressure contour plot.

The project of CFD simulation over a passenger car for aerodynamic drag reduction is done and the drag is reduced by 0.078%. Hence it is achieved by a popular software packages like CATIA V5r18, HYPERMESH, AND FLUENT.

Table 4. Result Comparison

Contents	Co efficient Of Lift	Coefficient Of Drag	% Drag Reduction
With Tail Palte	0.094	0.430	0.708
Without Tail Plate	0.074	0.352	

REFERENCES

[1] Wolf-Heinrich Hucho. Aerodynamic of Road Vehicle. Fourth Edition. Society of Automotive Engineers, Inc. 1998.

[2] Heinz Heisler. Advanced Vehicle Technology. Second Edition. Elsevier Butterworth Heinemann. 2002.

[3] Rosli Abu Bakar, Devarajan Ramasamy, Fazli Ismail, Design and Development of Hybrid Electric Vehicle Rear Diffuser, Science, Technology & Social Sciences 2008 (STSS), Malaysia.

[4] Guido Buresti. The Influence of Aerodynamics on the Design of High- Performance Road Vehicles. Department of Aerospace Engineering University of Pisa, Italy. 19 March 2004.

[5] Luca Iaccarino. Cranfield University Formula 1 Team: An Aerodynamics Study of the Cockpit. School of Engineering. Cranfield University. August 2003.

[6] Wong H.M, D. Ramasamy, Series Hybrid Electric Vehicle Cost-Effective Powertrain Components Development, RDU 070305, UMP Research Grant2007.

[7] Mark Coombs and Spencer Drayton, Proton Service and Repair Manual, Haynes Ptd. Ltd, P Ref 1, 2003, USA

[8] Amir Shidique, Simulation and Analysis of Hybrid Electric Vehicle (HEV) by Addition of a Front Spoiler, p39, Thesis, Universiti Malaysia Pahang, 2007.

[9] Bruce R. Munsan. Donald F. Young and Theodore H. Okiishi. Fundamental of Fluid Mechanics. Fifth Edition. John Wiley & Sons (Asia), Inc. 2006.

[10] Dr. V. Sumantran and Dr. Gino Sovran. Vehicle Aerodynamics. Society of Automotive Engineers, Inc. 1996.

M.Samundeswari was born in Tamilnadu, India in 1990. She received her B.E. degree in Manufacturing Engineering from College Of Engineering Guindy-Anna University, Chennai in 2013. She is finished her M.E.(CAD/CAM) in Anna University. She is currently working as an Assistant Professor in the Department of MECHANICAL Engineering at Sembodai Rukmani Varatharajan Engineering college, Vedaranyam.

.Jayachandran was born in Nagapattinam Tamilnadu, India in 1990. He received his B.E. degree in Mechanical Engineering from Anjalai Ammal Mahalingam Engineering College.He finished his M.E.(Manufacturing Engineering) from the Annamalai University,Chidambaram. He is currently working as an Assistant Professor in the Department of MECHANICAL Engineering at Sembodai Rukmani Varatharajan Engineering college, Vedaranyam.

R.Tamilarasan was born in Nagapattinam Tamilnadu, India in 1992. He received his B.E. degree in Mechanical Engineering from AVC Engineering College in 2013.He finished his M.E.(CAD/CAM) in Anna University. He is currently working as an Assistant Professor in the Department of MECHANICAL Engineering at Sembodai Rukmani Varatharajan Engineering college, Vedaranyam.