Estimation of Drag and Lift on Ahmed Body Using CFD Analysis

M.Govardhana [#1] , Dr.B.Veerabhadra Reddy [*2]

[#]*PG student,G. Pulla Reddy Engineering College (Autonomous), Kurnool.*

govardhana.mullangi@gmail.com

[*]*Professor, Dept. of Mech.Engg. G. Pulla Reddy Engineering College (Autonomous), Kurnool.*

gprectap@gmail.com

***Abstract* -- The aim of the paper is to evaluate the drag and lift coefficients of an Ahmed body. The geometry of Ahmed body is a simplified car used to investigate the flow analysis in automotive vehicle. CFD simulations are carried out by dividing the physical domain into small finite volume elements and numerically solved the governing equations that describe the behavior of the flow, in which simplified generic land vehicle geometry was used to analyze the effect of a slanted rear end and the effect of the angle is carried. The geometry with 25° hatchback angle solid model of the car is developed using CATIA V5. The modeled car is imported to GAMBIT (version 2.4) where the fluid flow around the car is given and a tetrahedral mesh is generated. The drag and lift coefficients are obtained using the FLUENT. The commercially available FLUENT (version 12.1) has been used in the present analysis with the objective of obtaining a better flow around the car and lowering the coefficient of drag.**

***Keywords:* Airfoils, k-∈ turbulence model, Lift, Drag, hatchback angle, car profile**

I. INTRODUTION

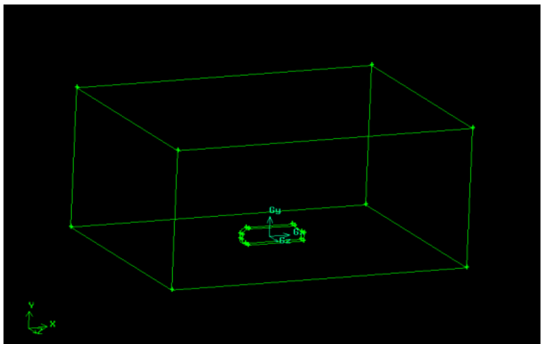
In this work, a Finite Volume Computational Fluid Dynamics (CFD) program is used to analyze the aerodynamic flow around a common vehicle. This is an important study because the aerodynamic flow has a great effect on the handling and overall performance of an automobile. Altering the geometry and studying the differences in the flow field will accomplish goals. Results from this paper should give a better understanding of how car geometry can be changed to provide better performance. Ahmed et al.[3] conducted experiments to investigate the effect of backlight angles in the range of 0^0 to 40^0. The backlight angle is the angle of depression of the rear window. In this range, two critical backlight angles (α) which were identified to have a significant influence on the flow structure were 12.5^0 and 30^0. Three ranges of backlight angles were identified which have different aerodynamic effects: $0^0<\alpha < 12.5^0$; $12.5^0<\alpha < 30^0$; and $\alpha > 30^0$. In the range of $0^0<\alpha < 12.5^0$, the flow remains attached over the rear window slant and separates at the top and bottom edges of the vertical base. In the range of $12.5^0<\alpha < 30^0$, the strength of longitudinal vortex increases and the flow becomes increasingly three dimensional. For α greater than 30^0, the flow separates at the top edge of the rear window. Bayraktar et al. [4] studied the effect of Reynolds number on lift and drag coefficients and concluded that the drag coefficient is insensitive at high Reynolds numbers (of the order of 10^6). Krajnovi et al.[13] performed LES on 25^0 Ahmed model for medium and fine grids. These studies were performed at low Reynolds number (2×10^5) to facilitate the use of LES. Large Eddy Simulation (LES) is a CFD technique where large flow structures are directly computed from Navier Stokes equations and only the structures smaller than the computational cells are modeled. The results of the study were also validated against the data from Lienhart et al. [9] and concluded that the flow structure around the model was well predicted. Kapadia et al. [12] performed Detached Eddy Simulation (DES) with a grid size of 1.74 million cells. The average drag coefficient from DES for both 25^0 and 35^0 angles was within 5% of the experimental value reported by Ahmed [3]. Kapadia et al.[12] also performed unsteady simulations using the Re-normalization group (RNG) k-ε turbulence model. Although the DES and LES have shown superior performance in predicting the overall flow structure, Reynolds averaged Navier Stokes (RANS) equation based turbulence models are chosen for automotive aerodynamics due to limitations of computer and simulation time. Lanfrit[15]. Braus et al. [6] used the Realizable k-ε model for simulation of flow on 25^0 Ahmed body with 2.3×10^6 grid size. The results suggested that although the RANS models do not predict the actual flow separation on the 25^0 base slant, the overall results including the drag coefficient are predicted with reasonable accuracy.

In these techniques many attempts have been made since the early years in the automotive industry to reduce aerodynamic drag in order to improve performance and fuel economy. Morelli et al. [19] developed a theoretical method to determine the shape of passenger car body for minimum drag by imposing the condition that the total lift be zero. This study proved that the aerodynamic drag can be reduced substantially with an optimized body shape without any additional devices. Morelli et al. [20] proposed a new technique called "fluid tail" and applied it to the aerodynamic design of basic shape of a passenger car. To achieve a fluid tail, a ring vortex must be created at the rear of the vehicle. Maji et al. [17] developed a highly streamlined concept vehicle using only aerofoils. A single piece shell body was developed by placing selected aerofoils at their appropriate locations. Guo et al. [7] performed aerodynamic analysis of different two-dimensional car geometries using CFD. In the first part of the study, the influence of front body shape was studied. Two models were used; one with sharp edges and the other with smooth rounded edges. Hu et al.[11] conducted CFD analysis to study different diffusers with angles of 0^0, 3^0, 6^0, 9.8^0 and 12^0 on a sedan type body. The results showed that the drag coefficient first decreased from 0^0 to 6^0 and then increased from 9.8^0 to 12^0 whereas the lift coefficient consistently decreased from 0^0 to 12^0. Han et al. [10] performed aerodynamic shape optimization on Ahmad body with three shape parameters: backlight angle, boat tail angle and ramp angle. The k-ε turbulence model CFD solver was coupled to an optimization routine. This process was continued until the parameters for minimum drag were obtained. Baker et al. [5] also developed a method to generate and use polynomial approximations for design optimization of an airplane with 28 design variables. The study concluded that the response surface method provides a means to explore the design quickly and accurately. Krajnovi et al. [13] used polynomial response surface model to optimize the aerodynamic performance of a high speed train. The optimization was performed to improve the shape of the front end of the train for cross wind stability and the dimensions of vortex generators. A common method of parameterization for automotive bodies is the use of geometric parameters such as edge radius, back light angle and diffuser angle (Han, [10]). Another method is shape modification by displacing particular edges on the body in the desired direction (Peddiraju, [22]). These parameterization techniques can be implemented in all modern parametric computer aided design (CAD) systems but the drawback of using this parameterization is that only simple shapes with small changes in geometry can be studied. Samareh et al. [24] proposed a free form deformation technique for aerodynamic shape optimization using the Non-Uniform Rational B-Spline (NURBS) due to its ability to provide a better control over shape changes.

II. MODELLING AND MESHING OF CAR

Decreasing the fuel consumption of road vehicles, due to environmental and selling arguments reasons, concerns car manufacturers. Consequently the improvement of the aerodynamics of car shapes, more precisely the reduction of their drag coefficient, becomes one of the main topics of the automotive research sectors. Designing a vehicle with a minimized Drag resistance provides economical and performance advantages. Decreased resistance to forward motion allows higher speeds for the same power output, or lower power output for the same speeds. The main aim for reducing drag resistance is Fuel consumption reduction and Performance improvement. To get the above, the profile of the car is modified and a Finite Volume Computational Fluid Dynamics (CFD) program is used to analyze the aerodynamic flow around the car body flow. This is an important study because the aerodynamic flow over the car body has a great effect in the reducing the drag forces acting on the car body and reducing the overall fuel consumption rate of the vehicle. The geometry is created in GAMBIT with given dimensions. Once the model is created with vertices and edges, next step is to convert the real edges into real faces. The faces of the Ahmed body are created from the real edges. The 3-D model is created by sweeping the faces of the car generated in the above step. Thus a solid object is created. Once the 3-D model is generated the domain around it where the fluid flow is analysed is created using create volumes. Firstly, the car surfaces are meshed using face mesh option. Fig.1 and fig.2 are the geometric and the meshed models of the car faces using triangular elements with an interval size of 0.1. The real brick volume is meshed using volume mesh with tetrahedral elements and interval size of 0.2. Once the meshing is completed our next step is to assign the boundary conditions to the domain as which face is functioning as inlet or which face is functioning as outlet, etc. Car is given the boundary condition WALL as it acts as a void to the fluid flow. The inlet is given a velocity of 40 m/s (for another two cases 50 m/s and 60 m/s) as the air flows with an equal velocity of the car but in opposite direction. The outlet is given a pressure condition of 0 pressures. The top surface of the body and the left side wall are given the WALL condition. In Fluent code first the meshed geometry that was exported from the Gambit code is imported or read. This reads the entire geometry including the grids, boundary conditions, zones, etc. Once the geometry is read into the fluent code the grid is checked and is scaled to the original dimensions. After scaling the grid, the mesh is verified and visualized. The edges feature and all are displayed by using display. Now, the physical model is defined.

A suitable solver is specified for the model. Here the solver is taken as Pressure based as the flow is incompressible. After selecting a suitable solver, a suitable viscous model has to be defined. The material properties are specified. The material specifications of the fluid around the Ahmed body are defined. The properties of the fluid i.e., air are defined as Density = 1.225 kg/m^3, Viscosity = 1.464x10^{-5} kg/ms. Then the boundary conditions are specified. Operating conditions and boundary conditions are described. Now, the flow field is initialized by giving the velocity at inlet equal to 40 m/s.



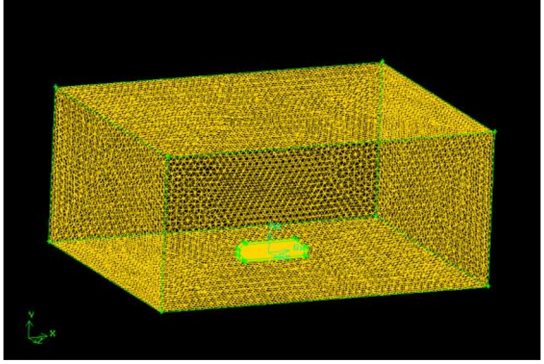


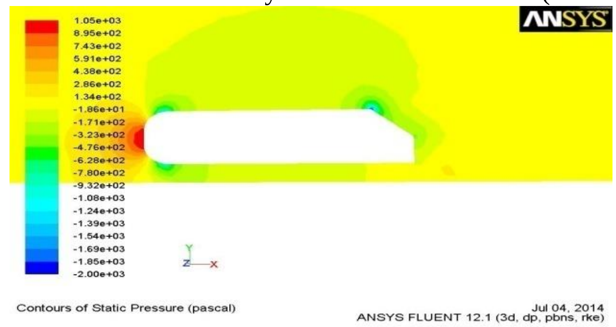
Fig1. Wire frame model Ahmed body

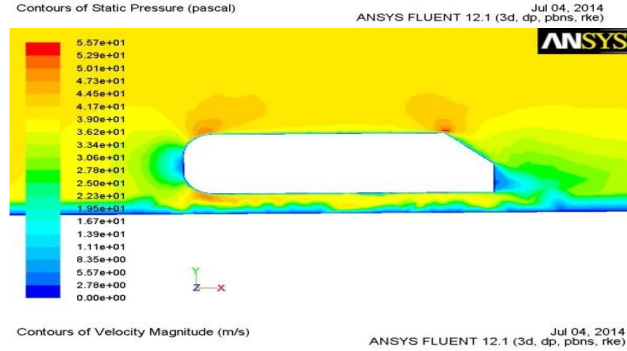
Fig2. Car domain mesh with tet elements

III.RESULTS AND DISCUSSION

The CFD analysis is carried out on the Ahmed body and the results are obtained. The analysis is carried out on three models. The velocity of the Ahmed body is varied for the aerodynamic analysis. Their corresponding variations of the pressure field, velocity field, velocity vectors of the fluid flow around the body are obtained. The corresponding drag and lift co-efficients are obtained. The results obtained for the three models are compared.

Model 1: Velocity at 40 m/s (144 km/h)





Fig

a. Pressure contours

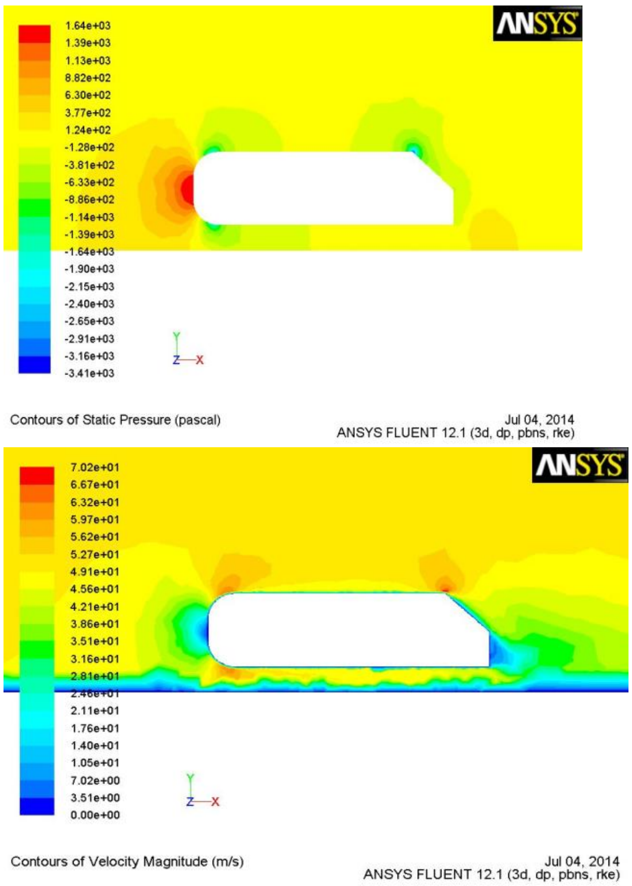
b. Velocity contours

Fig.3 Variation of pressure and velocity over Ahmed body profile of model1

Fig.3a shows the variation of the pressure field over the body. The different colored zones indicate different values of pressure. The maximum pressure is 1.05x10^{3}Pa field is highest at the front portion of the body. Fig.3b shows the variation of the velocity field over the car. The different colored zones indicate different values of velocity. The maximum velocity is 5.57x10 m/s, field is highest on top of corners the body. Fig.4a shows the variation of the pressure field over the body. The different colored zones indicate different values of pressure. The pressure field is highest at the front portion of the body. The maximum pressure is 1.64x10^3 Pa field is highest at the front portion of the body. Figure4b shows the variation of the velocity field over the car. The different colored zones indicate different values of velocity. The maximum velocity is 7.02x10 m/s field is highest on top of corners the body.

Model 2: At 50 m/s (180 km/h)

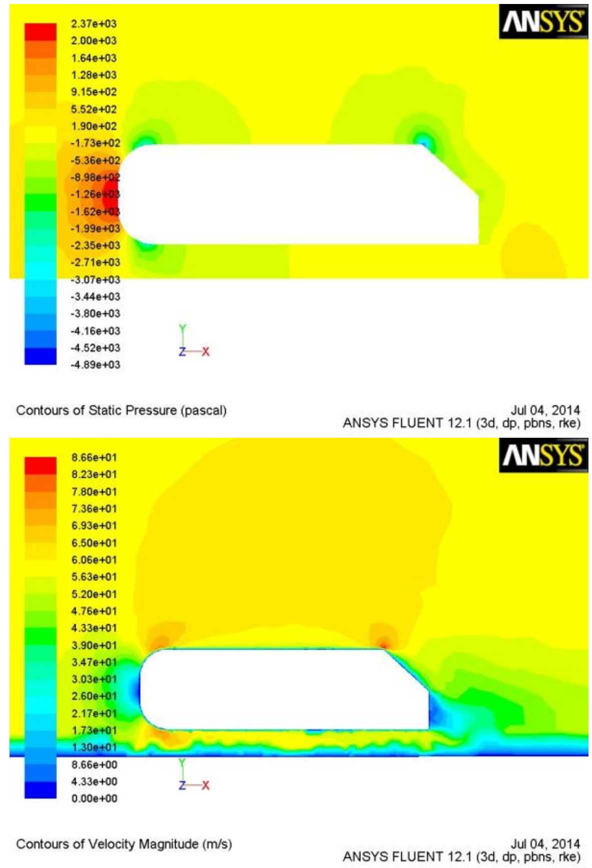
Fig.5a shows the variation of the pressure field over the body. The different colored zones indicate different values of pressure. The pressure field is highest at the front portion of the body. The maximum pressure is 2.37x10^3 Pa field is highest at the front portion of the body. Fig.5b shows the variation of the velocity field over the car. The different colored zones indicate different values of velocity. The maximum velocity is 8.66x10 m/s field is highest on top of corners the body. Table1 shows the Variation of pressure, velocity, drag coefficient and lift coefficient for different models



a. Pressure contours b. Velocity contours

Fig.4 Variation of pressure and velocity over Ahmed body profile of model 2

Model 3: At 60 m/s (216 km/h)



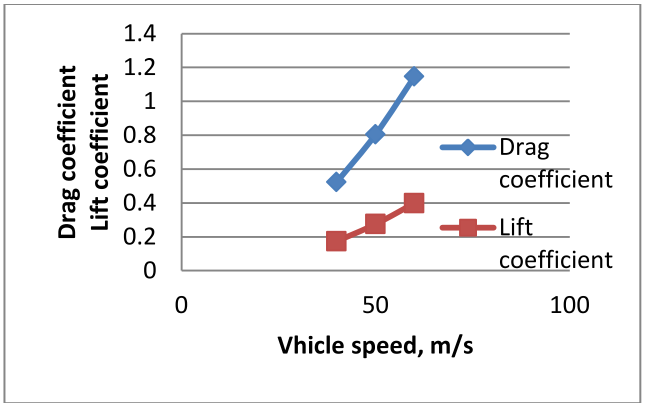
a. Pressure contours b. Velocity contours

Fig.5 Variation of pressure and velocity over Ahmed body profile of model 3

Table1: Variation of pressure, velocity, drag coefficient and lift coefficient for different models

S.No.	Vehicle Speed, m/s	Pressure, Pa	Velocity, m/s	Drag Coefficient	Lift coefficient
1	40	1.05e3	5.57e1	0.524	0.174
2	50	1.64e3	7.02e1	0.805	0.275
3	60	2.37e3	8.66e1	1.146	0.399

The fig.6 shows the variation of drag and lift coefficients with respect to the vehicle speed over Ahmed body profile. In this case as vehicle speed increases drag increases more than the lift coefficient. The fig.7 shows the variation of pressure and velocity with respect to the vehicle speed over Ahmed body profile. As the vehicle speed increases pressure increases, and velocity also increases simultaneously.



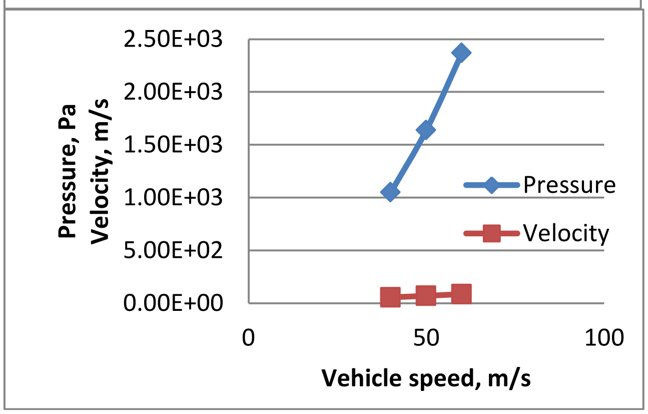


Fig6. Variation of velocity and pressure with respec
Fig7. Variation of drag and lift coefficients with
to vehicle speed over a Ahmed body profile
respect to vehicle speed over Ahmed body profile

IV. CONCLUSIONS

The drag coefficient and lift coefficients of an automobile are found by changing the vehicle speed of the Ahmed body. The modeling is done in GAMBIT and the analysis in CFD. The aerodynamic analysis on Ahmed body showed a maximum drag coefficient of 1.146 and lift coefficient of 0.399. As the speed of Ahmed body increases the lift coefficient increased gradually and drag coefficient increased sharply and is highest at the top speed. As the vehicle speed increases, the drag coefficient increases which in turn increases the fuel consumption. The fuel consumption is the highest at the top speed. As the vehicle speed increases, the pressure in front of the vehicle has a maximum value of $2.37x10^3$ Pa. At the highest speed of the vehicle the velocity of air is maximum is at top side corners with a value of 8.66x10 m/s.

REFERENCES

[1] Anderson CFD book: the basic with application McGraw-Hill series in aeronautical and aerospace engineering.
[2] ANSYS® Academic Research, Release 13.0, Help System, FLUENT Theory guide, ANSYS, Inc.
[3] Ahmed, G. Ramm, and G. Faltin. (1984). some salient features of the time averaged ground vehicle wake. SAE Paper 840300.
[4] Bayraktar, I., Landman, D., and Baysal, O. (2001). Experimental and Computational Investigation of Ahmed Body for Ground Vehicle Aerodynamics. SAE Technical Paper 2001-01-2742.
[5] Baker, C. A., Grossman, B., Haftka, R. T., Mason, W. H., & Watson, L. T. (1999). HSCT configuration design space exploration using aerodynamic response surface approximations. AIAA-98-4803.
[6] Braus, M., Lanfrit, M. (2001). Simulation of the Ahmed Body. 9th ERCOFACT/IAHR Workshop on Refined Turbulence Modelling.
[7] Guo, L., Zhang, Y., & Shen, W. (2011). Simulation Analysis of Aerodynamics Characteristics of Different Two-Dimensional Automobile Shapes. Journal of Computers, 6(5), 999-1005. doi:10.4304/jcp.6.5.999-1005.
[8] Gustavsson, T. (2006). Alternative approaches to rear end drag reduction. KTH, Department of Aeronautical and Vehicle Engineering, Royal Institute of Technology, TRITA-AVE 2006:12.
[9] H. Lienhart and S. Becker. (2003). Flow and turbulent structure in the wake of a simplified car model. SAE Paper 2003-01-0656, 2003.
[10] Han, T., Hammond, D. C., and SAGI, C. J. (1992). Optimization of bluff body for minimum drag in ground proximity. AIAA Journal, Vol. 30, No. 4, pp. 882-889.
[11] Hu, X., Zhang, R., Ye, J., Yan, X., and Zhao, Z. (2011). Influence of Different Diffuser Angle on Sedan's Aerodynamic Characteristics. Physics Procedia, Volume 22, Pages 239-245.
[12] Kapadia, S., Roy, S. (2003). Detached Eddy Simulation over a reference Ahmed car model. AIAA Paper 2003-0857.
[13] Krajnovi´c, S., and Davidson, L. (2004). Large-Eddy Simulation of the Flow around Simplified Car Model. 2004 SAE World Congress, SAE Paper No. 2004-01- 0227, Detroit, USA, 2004.
[14] Krajnovi, S. (2009). Optimization of aerodynamic properties of high-speed trains with CFD and response surface models. The Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains, 197-211.
[15] Lanfrit, M. (2005). Best practice guidelines for handling Automotive External Aerodynamics with FLUENT. (Version 1.2), Fluent Deutschland.
[16] M. M. Islam et al. (2010), "COMPUTATIONAL DRAG ANALYSIS OVER A CAR BODY" The International Conference on Marine Technology Dhaka, Bangladesh.
[17] Maji, S. and Almadi, H. (2007). Development of Aerodynamics of a Super Mileage Vehicle. SAE Technical Paper 2007-26-060, 2007, doi: 10.4271/2007-26-060.
[18] M. Desai, S A Channiwala and H J Nagarsheth, (2008)"A comparative assessment of two experimental methods for aerodynamic performance evaluation of a car," Journal of scientific &Industrial research, Vol. 67, pp 518 – 522.
[19] Morelli, A., Fioravanti, L., and Cogotti, A. (1976). The Body Shape of Minimum Drag. SAE Technical Paper 760186, 1976.
[20] Morelli, A. (2000). A New Aerodynamic Approach to Advanced Automobile Basic Shapes. SAE Technical Paper 2000-01-0491.
[21] P. N. Krishnani, (2009) "CFD study of drag reduction of a generic sport utility vehicle," Master's thesis, Mechanical engineering department, California State University, Sacramento.
[22] Peddiraju, P., Papadopoulous, A., Singh, R. (2009). CAE framework for aerodynamic design development of automotive vehicles. 3rd ANSA & μETA International Conference.
[23] S. N. Singh, L. Rai, A. Bhatnagar (2004) "Effect of moving surface on the aerodynamic drag of road vehicles" Proceeding of IMechE. Vol. 219 Part D: J. Automobile Engineering, pp 127- 134.
[24] Samareh, J. A. (2004). Aerodynamic shape optimization based on free-form deformation. AIAA,4630.
[25] Wolf-Heinrich Hucho (1998) "Aerodynamics of Road Vehicles", SAE International, Warrendale.