International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Impact Factor (2012): 3.358

CFD Analysis of Aerodynamic Drag Reduction of Automobile Car - A Review

Jaspinder Singh[1], Jagjit Singh Randhawa[2]

[1]Research scholar, Department of production engineering
PEC University of technology, Chandigarh-160012

[2]Assistant Professor, Department of production engineering
PEC University of technology, Chandigarh-160012

Abstract: ***Computational Fluid Dynamics (CFD) is the numerical techniques to solve the equations of fluid flow. CFD tool is found very useful in automobile industry ranging from system level (exterior aerodynamics, ventilation, internal combustion engines) to component level (disk brake cooling).CFD simulations are carried out by dividing the physical domain into small finite volume elements and numerically solved the governing equations that describe the behaviour of the flow. The governing equation (Navier Stokes equations) is very complex and almost impossible to solve analytically thus requires numerical techniques. In the present paper, research work in the field of "CFD analysis of aerodynamic drag reduction of automobile car" has been reviewed.***

Keywords: CFD Simulation, Drag force, Drag coefficient, Ahmed body, Aerodynamics

1. Introduction

Computational Fluid Dynamic (CFD) is the branch of fluid dynamics providing a cost-effective mean of simulating real flows by the numerical solution of the governing equation. The governing equations for Newtonian fluid dynamics, namely the Navier-Stoke equation. The differential equation governing the system is converted to a set of algebraic equation at discrete points, and then solved using digital computer. For such a complex interaction, CFD analysis is probably the only efficient tool in order to assess specific design parameterization of a generic car shape. With the recent increase in threat to the environment considerably due to vehicular pollution it has become a need of the hour to increase automobile efficiency. In order to achieve this either change have to be brought about in engine functioning, or supplementing presently used fuel by eco-friendly fuels or by enhancing current automobile design. As far as engine optimization is concerned we have reached a saturation point. Using eco-friendly fuels is an area still under development and it will take a few more years for it to be adopted worldwide. So it can be easily concluded that enhancing automobile design is the easiest way to increase vehicle efficiency.

In optimization of car aerodynamics, more precisely the reduction of associated drag coefficient (C_D), which is mainly influenced by the exterior profile of car, has been one of the major issues of the automotive research centres all around the world. Average C_D values have improved impressively over the time, from 0.7 for oldboxy designs of car to merely 0.3 for the recent more streamlined ones [Desai (2008)].The description of the fuel energy used in a modern vehicle at urban driving and highway driving. The shape of the vehicle uses about 3 % of fuel to overcome the resistance in urban driving, while it takes 11% of fuel for the highway driving. This considerable high value of fuel usage in highway driving attracts several design engineers to enhance the aerodynamics of the vehicle using minimal design changes [Krishnani (2009)].The effect of drag on the moving vehicle is proportional to the square of velocity, so with increase in velocity (at approximately 50 km/h), aerodynamic drag becomes one of the most prominent factors contributing to the total drag experienced by the vehicle [Singh (2004)].

Aerodynamic evaluation of air flow over an object can be performed using analytical method or CFD approach. On one hand the analytical method of solving air flow over an object can be done only for simple flows over simple geometries like laminar flow over a flat plate. If air flow gets complex as in flows over a bluff body, the flow becomes turbulent and it is impossible to solve Navier-Stokes and continuity equations analytically. On the other hand obtaining direct numerical solution of Navier-Stokes equation is not yet possible even with modern day computers. In order to come up with a reasonable solution, a time averaged Navier-Stokes equation is being used (Reynolds Averaged Navier Stokes Equations- RANS equations) together with turbulent models to resolve the issue involving Reynolds Stress resulting from time averaging process.

2. Literature Review

A brief review of literature has been provided in the following areas: (a) Ahmed body(b) Drag reduction(c) Car body aerodynamic shape optimization (d) NURBS.

Ahmed body

Ahmed et al. (1984) 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 C 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. Reynolds number: Ahmed et al. (1984) were performed at a wind speed of 60 m/s. This corresponds to a Reynolds number of 4.29×10^6 based on model length. Bayraktar et al. (2001) studied the effect of Reynolds number on lift and drag coefficients. Thus he was concluded that the drag coefficient is insensitive at high Reynolds numbers (of the order of 10^6).Computational investigations on the Ahmed body: 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 modelled. Since the size of turbulent vortices decreases with increasing Reynolds number, LES is performed at moderate Reynolds numbers so that most of the turbulent vortices can be directly solved rather than modelled. Krajnovi et al. (2004) 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. The results of the study were also validated against the data from Lienhart et al. (2003) concluded that the flow structure around the model was well predicted. Kapadia et al. (2003) performed Detached Eddy Simulation (DES) with a grid size of 1.74 million cells. This study was performed on 25^0 and 35^0 Ahmed bodies. The average drag coefficient from DES for both 25^0 and 35^0 angles was within 5% of the experimental value reported by Ahmed (1984). Kapadia et al. (2003) also performed unsteady simulations using the Re-normalization group (RNG) k-ε turbulence model. The results suggested that the RNG k-ε model over predicts the drag coefficient. 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 2005).Braus et al. (2001) used the Realizable k-ε model for simulation of flow on 25° 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° base slant, the overall results including the drag coefficient are predicted with reasonable accuracy.

Drag reduction: 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. Morelliet al. (1976) 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. (2000) 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. (2007) developed a highly streamlined concept vehicle using only aerofoils. A single piece shell body was developed by placing selected aerofoils at their appropriate locations. Guoet al. (2011) 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. Larger stagnation areas were observed on the sharp edged geometry as compared to smooth and rounded edged geometry. Smooth edged geometry also showed reduced pressure areas at bottom of the front end. In the second part of the study, different rear geometries with different backlight angle were studied. Hu et al. (2011) 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 12^0whereas the lift coefficient consistently decreased from 0^0 to 12^0.

Aerodynamic shape optimization**:** Han et al. (1992) 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. (1998) 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 space. Krajnovi et al. (2009) 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. The results of the study suggested that the response surface method is a practical solution to the complex problem of aerodynamic shape optimization.

NURBS: Geometry parameterization is an essential part of the design exploration and shape optimization process and there are several methods of generating parametric geometries. 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, 1992, and Muyl, 2004). Another method is shape modification by displacing particular edges on the body in the desired direction (Peddiraju, 2009).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. Samarehet al. (2004) proposed a free form deformation technique for aerodynamic shape optimization using the NURBS due to its ability to provide a better control over shape changes. The NURBS parameters changed in this study were the NURBS control points.

3.Conclusions from Previous Research Work

After going through the research work carried out in the field of CFD analysis of automobile car for aerodynamic drag reduction, it has been concluded that there are many thrust areas (shape optimization, meshing type and boundary condition) which affect the CFD analysis of drag coefficient. CFD approach is arguably the way of the future in promising faster turnaround simulation time with cheaper running cost.

At the same time, it offers superior capability than the experimental approach in terms of post processing of data and graphical representation of flow analysis. Provided CFD approach is to be reliable and can produce quantifiable results with a high degree of accuracy, the role of wind and aero acoustics tunnel testing in the future might be reduced for only validation purposes. However, more work should be done on improving CFD technological advancement for a more efficient and easier way to tackle problems associated with aerodynamics and aero acoustics.

4. Acknowledgement

The author wishes to express their sincerest thanks to Mr. Jagjit Singhrandhawa for their guidance in my research work.

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 GmbH.
[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.

Author Profile

Jaspinder Singh received the B. Tech degree in Mechanical Engineering from S.B.B.S Institute of Technology in 2011 and is currently perusing M.E in Industrial Design from PEC University of technology.