International Journal of Advanced Research in Education & Technology (IJARET)
Vol. 3, Issue 1 (Jan. - Mar. 2016)
ISSN : 2394-2975 (Online)
ISSN : 2394-6814 (Print)

CFD Analysis of wake Region of Simplified Car Model Due To Induced Drag Force

[I]Datta Kisan Wakshe, [II]G.N. Thokal
[I]Pillai HOC College of Engineering and Technology, Rasayani
[II]Pillai Institute of Information Technology, New Panvel

Abstract

The evaluation of car since from last few decades has been extremely fast because of increased competition between the car companies. The Computational Fluid Dynamics is a power full tool used to find the flow parameters in the automobile industry today. This analysis is used to understand the stability, fuel consumption, passenger comfort and emission level before andafter of the vehicle. In automobile design the complexity involved due to the great amount of accessories and devices are used in it, makes the experimental validation unaffordable, and the flow around vehicle under normal operating condition is principally turbulent. It is highly characterized by large separation, recirculation region, a complex wake flow and interaction of boundary layer flow In this work the drag force and drag coefficient is calculated by using ANSYS CFD Workbench 14.0. For this study a simplified car model is used to investigate the flow analysis and find the wake flow around the body. Ahmed body as ground vehicle is commonly used as a test case in industry today. The present study is attempted to implement a computational study of the flow over a two-dimensional car model Reynold's averaged Navier –Stokes (RANS) with a combination of most popular k-ε turbulence model produces sufficiently accurate results. Working fluid is Air

Keywords

Aerodynamics, CFD, Drag coefficient, Turbulence, Bluff Body, N-S Equation.

I. Introduction

Nowadays, car has become one of the main vehicles to travel. Today's one of the most challenging issues within the automotive industry is reducing the fuel consumption of vehicles and to reduce the fuel consumption and emissions. The automotive industry is one of the fastest growing industries in the world. This work is motivated by the need to perform an accurate aerodynamic analysis of the drag of the vehicle Automobile industry which is one of the key sectors of the world economy.

When vehicle was invented the issues of regarding the reducing power consumption of vehicle emerged nearly at the same time. In the beginning fuelling stations were few and a low fuel consumption vehicle was needed to cover the as longest as possible distance between refuelling. Since, the number of cars has grown with the increase in fuel cost and it became again important to reduce the fuel consumption to save fuel but also the environment. Vehicle power consumption reduction can be achieved by various methods such as by improving the engine efficiency and by reducing the aerodynamic drag. Since the early 20th century, large numbers of studies were carried out in this field to improve the vehicle performance. The rear part of the vehicles make large contribution to the total induced drag over the body and in this work different form of vehicle rear section has been studied. The vehicles operate in real time situation under different air flow rate, different speeds, different direction, and different road conditions. Therefore, there is need to develop technique which will keep drag coefficient low for large range of conditions like different speed of vehicle and different air flow rate.

II. DRAG force and drag coefficient

Drag is a force that acts parallel and in the same direction as the airflow. Aerodynamic drag force increases with the square of speed therefore it becomes critically important at higher speeds reducing the drag coefficient. The drag coefficient is dimensionless quantity that describes a resistance of vehicle aerodynamics.

It is a very useful tool when comparing different vehicle shapes regardless of size and speed. The drag coefficient of an automobile impact the way the automobile passes through the surrounding air. When automobile companies design a new vehicle they take into consideration the automobile drag coefficient in addition to the other design performance characteristics of vehicle. The aerodynamic forces on a car body come primarily from differences in pressure and viscous shearing stresses. Therefore the drag force on a body could be divided into two components, when the drag is dominated by a frictional component, the body is called a streamlined body and when drag is dominated by pressure the body is called a bluff body. Mathematically drag coefficient is calculated by $\frac{1}{2}\rho A V^2$. Thus, the shape of the body and the angle of attack determine the type of drag produced in wake region. For example, an airfoil is considered as a body with a small angle of attack by the fluid flowing over it. This means that it has attached boundary layers, which produce less pressure drag on airfoil.

III. CONCEPT of Ahmed body

A computation is totally based on Reynolds-Averaged Navier Stokes Equations (RANS) which are common in automobile industry today. Although they are very successful in predicting the flow around many part of the vehicle but they are unable to predict unsteadiness produced in the wake region due to flow. In order to investigate the behavior of newly developed turbulence models for complex cases geometry a simplified car model is used. The Ahmed Body was first created by S.R. Ahmed in his research in 1984. [1] Since then, it has become a benchmark for aerodynamic car simulation tools. The Ahmed body is made up of a round front part, a moveable slant plane placed in the rear end of the Ahmed body to study the separation phenomena at different angles. The Ahmed body is a rectangular box like structure which connects the curved front part and the rear slant plane. [1] In this work the rear slant angle is kept at 30 . As the wake flow behind the vehicle is the main contributor to the drag force, which reduce the fuel efficiency of the vehicle therefore for accurate prediction of the separation of boundary layer process the wake flow are the key to the successful modeling of this case.

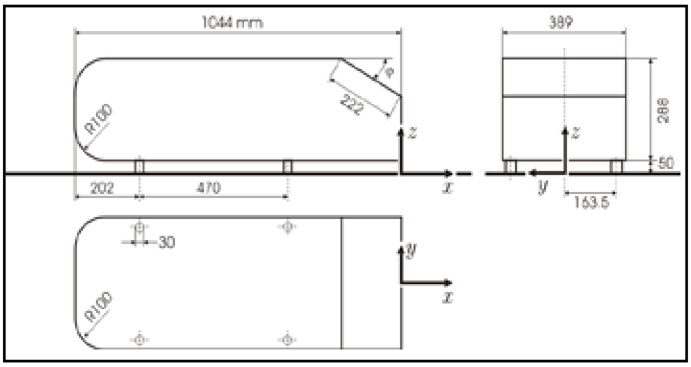

Fig .1 : Ahmed body block diagram

Following is the proposed methodology for the tasks in progress. The Computational Fluid Dynamics is a power full tool used in this work. The following steps are involved in this analysis.

A. Geometry Creation

The following geometry is created in ANSYS MODULAR The total length and height of body is L and h respectively. The distance between road and lower surface of the Ahmed body is 50 mm and θ is the slant angle measured in degree from the top side of the body. Symmetry is specified for top side of the simulation domain. The upstream length UL is 1 meter. The simulation domain height DH is 3 meter and downstream length DL is 6 meter to capture the wake region behind the body. We have to make the domain in such way that the flow should not be disturbed and got better result for simulation. The clearance is provided between the lower side of Ahmed body and lower surface of domain.

B. Grid Generation

Mesh generation is an essential step in Computational Fluid Dynamics. For grid generation ANSYS meshing is used in this work. In this work the computational simulations is requiring high quality meshes to accurately capture the complex physical phenomena. The high quality and high density meshes are required to capture accurately the complex physical phenomena but it is computationally expensive. Therefore mesh was locally refined in regions that are important and coarser mesh is used at less relevant places to reduce the computational expense with sufficient number of grid needed to be solving the problem accurately. For grid generation ANSYS meshing is used in this work. [6]

C. Governing Equation

The computational fluid dynamics is based on the three fundamental governing equations of fluid dynamics. Continuity equation, momentum equation and energy equation are used in computational methods. They are the mathematical statements of three fundamental physical principles of fluid dynamics.

- Continuity Equation (Mass equation);
- Momentum Equation (F = ma);
- Energy Equation.

$$\text{Mass Conservation} : \nabla.(\rho\vec{V}) = 0$$

$$x\text{ momentum } = \frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$$

$$\text{y momentum } = \frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g$$

$$\text{z momentum } = \frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z}$$

Momentum Equation (Newton's 2nd law)
Navier-Stokes equation 2-D ($u_z = 0$)

$$\rho\left(\frac{\partial u_x}{\partial t} + u_x\frac{\partial u_x}{\partial x} + u_y\frac{\partial u_x}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u_x}{\partial x^2} + \frac{\partial^2 u_x}{\partial y^2}\right) + \rho g_x$$

$$\rho\left(\frac{\partial u_y}{\partial t} + u_x\frac{\partial u_y}{\partial x} + u_y\frac{\partial u_y}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u_y}{\partial x^2} + \frac{\partial^2 u_y}{\partial y^2}\right) + \rho g_y$$

In this work we have considered the flow around the Ahmed body is 2-dimensional flow and it has been simulated by solving the appropriate governing equations which are mentioned below conservation of mass and momentum using ANSYS commercial CFD code. In this work heat transfer is neglected so the energy equation is not considered. For turbulence modeling standard k- ε turbulence model is selected [3]. The standard k- ε model is a two equation model [5].

In standard k-ε turbulence model the first variable is used to determine turbulent kinetic energy (k) and second variable is used for turbulent dissipation (ε) which determines the rate of dissipation of turbulent kinetic energy.

Two equation standard k-ε turbulence model given by following two equations

1. Turbulent kinetic energy (k):-

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \epsilon k)}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + 2\mu_t E_{ij} E_{ij} - \rho\epsilon$$

2. Turbulent dissipation rate (ε) :-

$$\frac{\partial(\rho\epsilon)}{\partial t} + \frac{\partial(\rho\epsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\epsilon}\frac{\partial \epsilon}{\partial x_j}\right) + C_{1\epsilon}\frac{\epsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\epsilon}\rho\frac{\epsilon^2}{k}$$

D. Boundary Conditions

In ANSYS pre-processor fluid domain is defined. There is no solid domain involved in this work. The flow in this work is turbulent, hence standard k- ε turbulence model is chosen. The boundary conditions are specified in ANSYS pre-processor and then the file is exported to the solver FLUENT. The upstream velocity is 60 m/sec. The Reynolds number is 3.2 x 10^6, based on the length of the model. Uniform velocity specified at inlet and constant pressure specified at outlet. The velocity field is obtained from the momentum conservation equations and the pressure field is extracted either from the mass conservation equation or continuity equation. In intensity and viscosity ratio the Turbulent Intensity is 2.93 % and turbulent viscosity ratio 10. The following Conditions were applied for solving the case:

- Material: Air,
- Inlet: Velocity,
- Outlet : Pressure,
- No-slip condition;
- Density of Air 1.232 kg/m^3,

E. Numerical Simulation

Solution phase is completely automatic in ANSYS CFD. The

Vol. 3, Issue 1 (Jan. - Mar. 2016)
ISSN : 2394-2975 (Online)
ISSN : 2394-6814 (Print)

Finite Element Analysis software itself computes the nodal values and its derivatives. The software itself generates the element matrices and stores the final result data in files. These files are further used in post-processing to review and analyze the results by the subsequent phase. In ANSYS the CFD flow solver used is FLUENT, which is a general purpose 2D and 3D structured and unstructured flow solver. For incompressible flow the FLUENT uses Reynolds-averaged Navier-Stokes equations (RANS). [5][3] The solver is totally based on the finite element method to build a spatial discretization of the transport equations. Equal order nodal interpolation for all variables are done while solving. The computations are performed on an Intel (R) core (TM) i5- 4440 CPU @ 3.10 GHz, 8 GB RAM 32 Bit Unicode. The total elapsed CPU time is nearly 8 hours.

IV. Results And Discussion

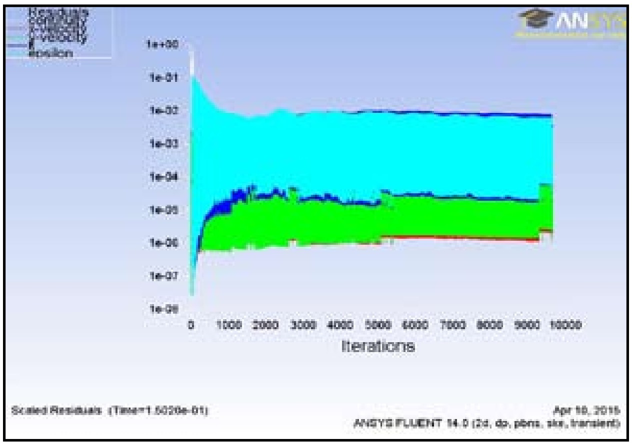

Fig. 2 : Scaled residuals (Number of Iterations)

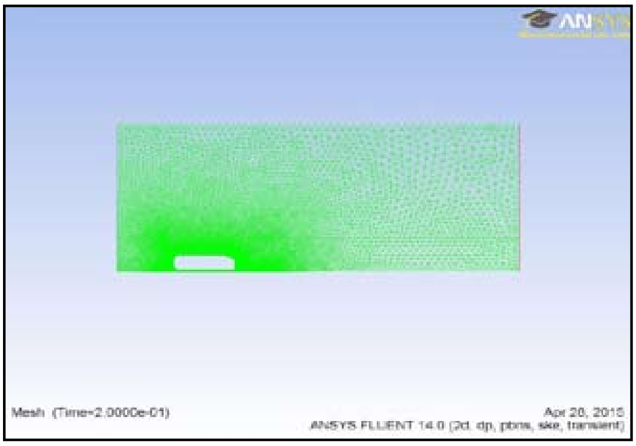

Fig. 3 : Surface mesh of Ahmed body and Domain

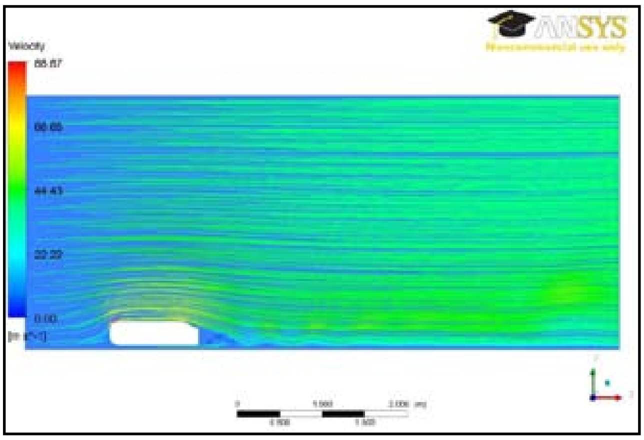

Fig. 4 : Flow of Velocity (Stream line)

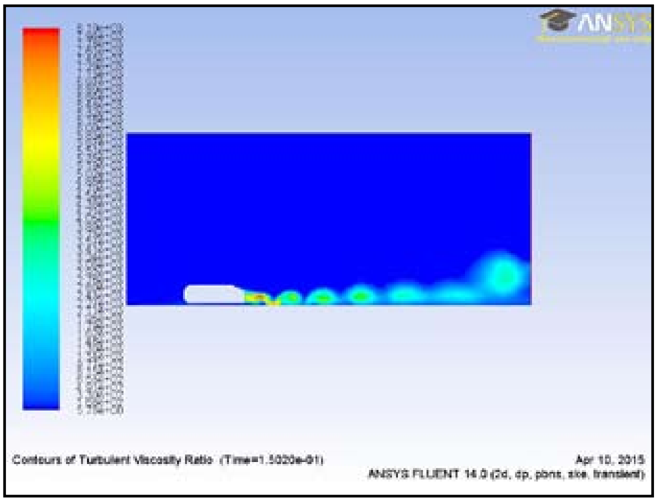

Fig .5 contours of turbulent Viscosity ratio

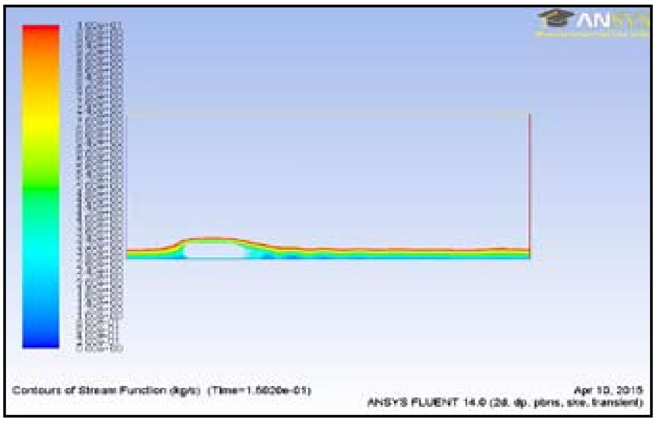

Fig. 6 : Contours of stream function

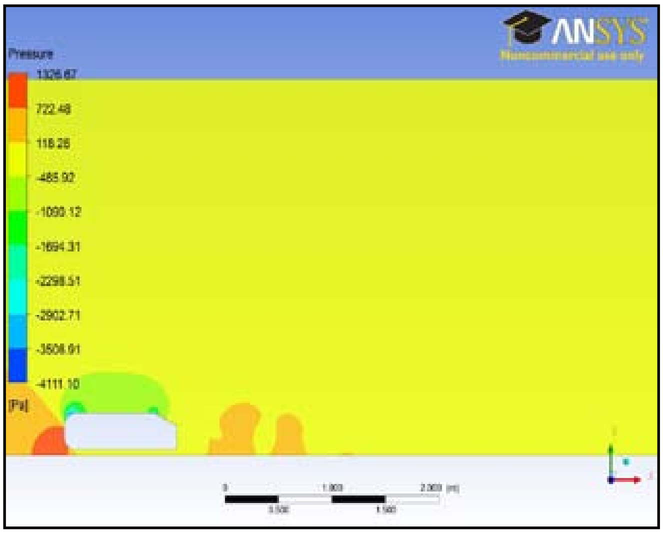

Fig.7 : Contours of pressure

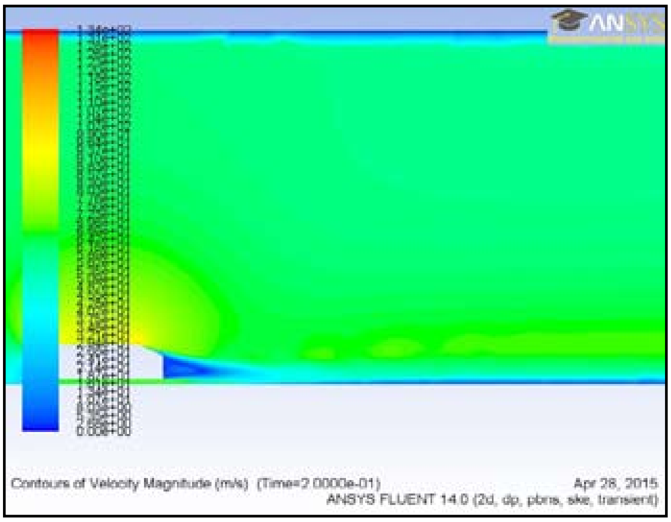

Fig. 8 : Contours of velocity magnitude (Time 0.2 s)

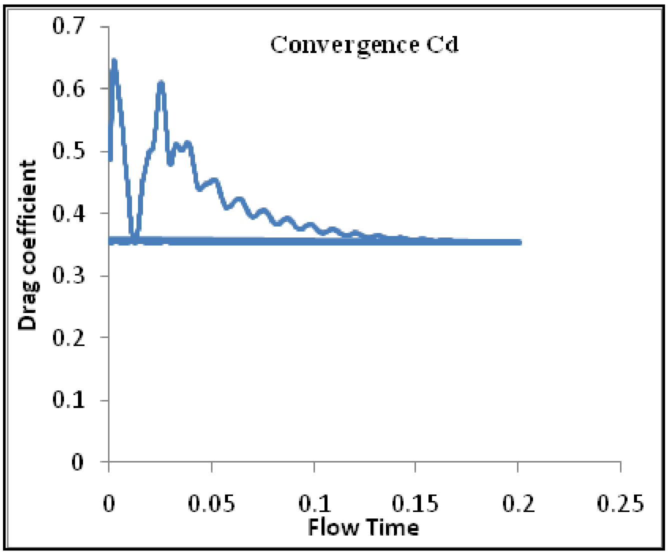

Fig. 9 : Graph Drag Coefficient Vs Flow Time

Table 1 : Pressure Forces and Coefficients on Car Body

Zone	Pressure(N)	Coefficients
Bottom	8.5823113	0.0135146
Top	9.7647552	0.015376599
Back	59.903235	0.094329863
Nose	-29.715639	-0.046793335
Slope	144.06501	0.22685975
Wall Surface Body	32.523358	0.051214662
Net	225.12303	0.35450214

V. Conclusion

The 2-D wake analysis of the simplified car model known as Ahmed body rear angle at 30 has been successively done. The Reynolds no based on the length and velocity was of the 3.2 x 10^6 .The current investigation leads to a number of conclusions in terms of static and dynamic pressure as well as coefficient due to strong wind impact on the car model (Ahmed body). The drag coefficient is constant at 0.354 in the range of flow time 0.11s to 0.2s. On nose the pressure is negative which indicate that lift is produced.

VI. Future Work

- Pressure effects and drag is also challenging on car structures. Proper study is to be investigated for such type of body through prototype.
- Prototype to be designed and miniature model to be prepared. The basic idea underlying the approach is to test the model physically and running through the simulations.
- To understand the real time physical flow behavior around the car model it required experimentation in wind tunnel.

References

[1]. S. R. Ahmed, R. Ramm, and G. Falting. Some Salient Features Of The Time-Averaged Ground Vehicle Wake. In SAE technical paper series 840300, Detroit, 1984.

[2]. Ram Bansal and R. B. Sharma Drag Reduction of Passenger Car Using Add-On Devices , Journal of Aerodynamics Volume 2014 Journal of Aerodynamics Volume 2014 (2014), Article ID 678518.

[3]. Jurij SODJA, Turbulence models in Computational Fluid Dynamics, University of Ljubljana Department of physics, CDF (Theory).

[4]. Mr. P. R. Sonawane, Prof. S. P. Shekhawat Aerodynamic Analysis of the Car Body for Minimum Fuel Consumption Journal of information, knowledge and research in mechanical ISSN 0975 – 668X| NOV 10 TO OCT 11 | VOLUME – 01

[5]. John D. Anderson, Jr. Computational Fluid Dynamics-basics with application, McGraw-Hill series in mechanical engineering (1995).

[6]. http://www.ansys.com, ANSYS, Inc. Release 15.0 South pointe November 2013 Canonsburg, PA 15317 certified to ISO.