ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 3, May 2013

Computational Study of Flow around a Simplified 2D Ahmed Body

Deepak Kumar Kalyan, A.R. Paul
PG Student, Applied Mechanics Department, MNNIT, Allahabad
Assistant Professor, Applied Mechanics Department, MNNIT, Allahabad

***Abstract*— The Ahmed body is a simplified car used in automotive industry to investigate the flow analysis and find the wake flow around the body. The present study is tried to implement a computational study of the flow over a two-dimensional car model and try to understand the flow behavior with the real life vehicles problem. Study the flow structure on drag with 25° hatchback angle. Numerical results agree well with the reported experimental results. Simulation carried with several turbulent models and found that realizable k-ε turbulent model gives good agreement with experimental result. Present study also measured time averaged velocity profile at various transvers planes in the wake.**

Index Terms— **Ahmed body, Computational Fluid Dynamics, Turbulence model.**

Nomenclature:

U_x = axial velocity, U_0 = free-stream velocity, θ = slant edge angle of Ahmed body.
UL = upstream length, DL = downstream length, DH = height of simulation domain,
GL = distance between road and lower surface of the body, h = height of the body, L = length of the body.

I. INTRODUCTION

External flow analysis over car geometry is always an interested part of research in aerodynamic field. The study performed by experiments in wind tunnels as well as by computational software. Due to high expensive of experimental study the recent year the computational fluid dynamics (CFD) gradually becoming the most efficient tool in vehicle design and in aerodynamic fields. Ahmed [3] Main purpose is to study of the wake and drag around the body in different flow condition. Ahmed found about 80-85% of body drag is pressure drag and it will show the rear part of the body. Lien hart [6] et al. found during experiments the critical angle (rear slant edge) is 30^0 because there is a sudden drop of drag coefficient and flow fully separates over the rear slant edge. CFD software provides to visualize the complex flow phenomena using different color contour plots. Also compute the many derived parameters along with the graphical operation of the interested regions. The main objective of the present work is to undertake a comparative experimental and numerical investigation over a two-dimensional Ahmed body. Visualization techniques were employed to examine the structure of the wake and time-averaged velocity measurements were made on the centerline plane and at transverse planes in the wake. During the recent past CFD is gradually becoming established as an efficient tool in vehicle design. CFD technology allows for the visualization of complex flow phenomena in a virtual environment that can significantly enhance the learning experience. The simplest “complete models’’ of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. In this paper the aim is simplify the complex simulation effort and show the better result by the same tool of software.

II. COMPUTATIONAL METHODOLOGY

Most of the computations performed in this research are done using commercial CFD code Ansys-Workbench, and modeling software Catia V5. Ansys-Workbench is a combined module of CFD code including pre-processing solver and post-processing. The problem to be solved is unsteady turbulent flows over the Ahmed body (2D).The Ahmed body is made up of a round front part, a moveable slant plane placed in the rear of the body to study the separation phenomena at different angles, and a rectangular box, which connects the front part and the rear slant plane, as shown in Figure above (Fig. 1, Fig. 2).

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 3, May 2013

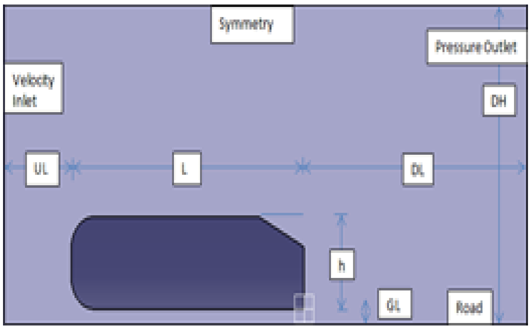

Fig.1 Schematic diagram of solution domain


Fig.2 Geometry of domain with Ahmed Body

III. GEOMETRY DESCRIPTION

The following geometry (Fig.1, Fig.2) is created in Ansys design Modeler. The total length of body is L and h is the height of the body. Distance between road and lower surface of the body is GL and θ is the slant angle in degree measured from the top surface of the body. The uniform velocity mentioned at the inlet and constant pressure specified at outlet. "Symmetry" is specified for top of the simulation domain. The upstream length UL is 1m. The height of simulation domain DH is 3m and downstream length DL is 6m for capture the wake region behind the body. In this paper the Ahmed body slant edge angle is 25^0.This figure made in the design software Catia v5. We have to make the domain in such way that the flow is not disturbed and got better result. There is a clearance between the lower surface of body and lower surface of domain. The clearance GL is 50mm.The bottom corner of rear part of body is origin.

IV. GRID DESCRIPTION

Mesh generation is an essential step in computational simulations requiring high quality meshes to accurately capture the complex physical phenomena. High quality and high density meshes are required to accurately capture the complex physical phenomena but it is computationally expensive. Therefore mesh was locally refined in regions that are important and coarser mesh was used at less relevant places to reduce the computational expense with sufficient number of grid needed to be solving the physics accurately. For grid generation Ansys meshing is used. Multi block structured mesh (Fig.3) with 80698 elements is created. The selection of element is based on the grid independence test (Fig. a).The convergence criteria of the solution is 10^{-4} (fig. b).

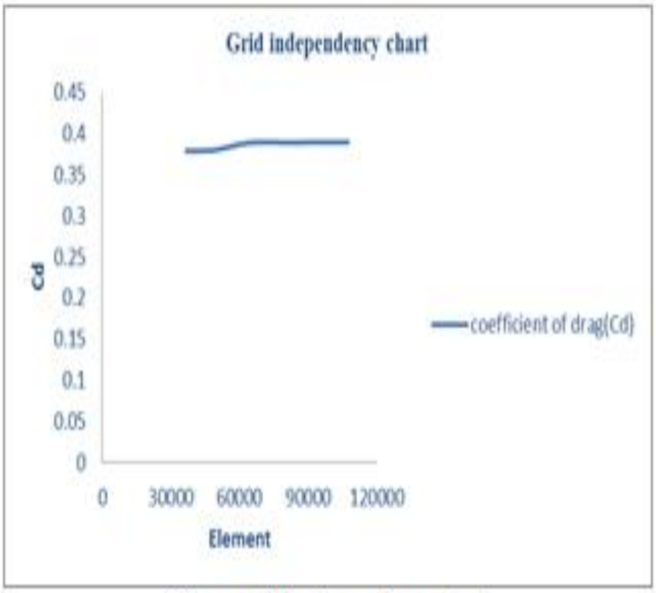

Fig: a grid independency test

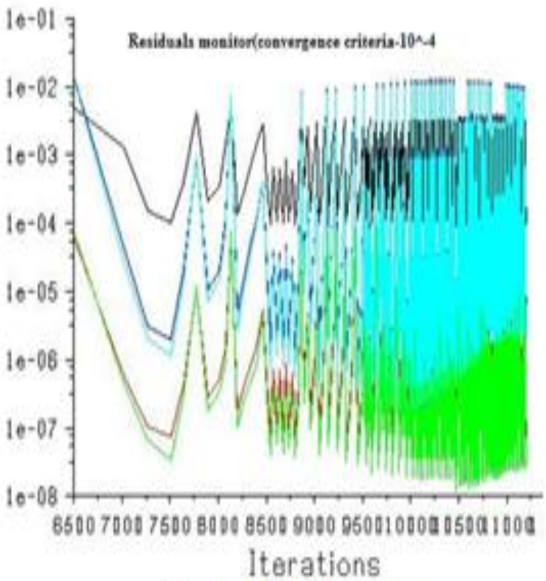

Fig: b residuals monitor

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 3, May 2013

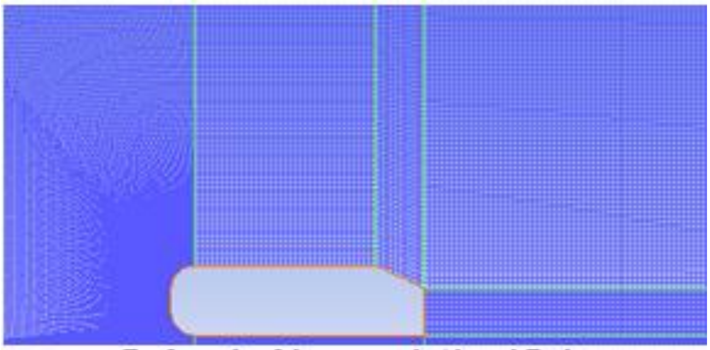

Fig.3 mesh of domain with Ahmed Body

V. SOLUTION METHODOLOGY

In present investigation uniform velocity of 40m/s with 2.93% turbulence intensity and viscosity ratio 10 is used at inlet. Fluid is considered to be incompressible air and flow is unsteady. Air flow over the body is governed by the Reynolds Average Navier-Stokes equations (RANS). Fluent (post processor) in order to investigate the flow around the bluff body. There are several turbulence models to identify the flow behavior around the body. Every Model have their own advantage and some disadvantage that's why lots of author still try to find which model is most suitable for the finding the actual flow behavior around the Ahmed body. In this paper Realizable k- ε turbulence model is used to solve the flow analysis. The car surface is treated as wall and no-slip condition. Reynolds number based along the length of the car and free stream velocity. This model comes under two equation groups of model in which two extra quantity turbulence kinetic energy k and its dissipation rate ϵ need to solve and try to achieve better result. The incoming air is located one meter upstream from the nose body. Transport equation for momentum and turbulence parameter is solved with quick discretization.

VI. MATHEMATICAL MODELS

A. Governing equation:

The Reynolds Averaged Navier-Stokes (RANS) equations solved by fluent are presented in Equations. The RANS approach of permitting a solution for the mean flow variables greatly reduces the computational effort. If the mean flow is steady, the governing equations will not contain time derivatives and a steady-state solution can be obtained economical.

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = \frac{\partial p}{\partial x_i} + A + B$$

$$here,\ A = \frac{\partial\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2\delta_{ij}}{3}\frac{\partial u_i}{\partial x_i}\right)\right]}{\partial x_j}\ and B = \frac{\partial\left(-\rho\overline{u_i' u_j'}\right)}{\partial x_j}$$

This approach is generally adopted for all practical engineering calculations, and is used with turbulence models.

Realizable *k*- $\mathcal{E}$ Turbulence Model:

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 3, May 2013

$$\frac{\partial(\rho\kappa)}{\partial t}+\frac{\partial(\rho\kappa u_j)}{\partial x_j}=\frac{\partial\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial\kappa}{\partial x_j}\right]}{\partial x_j}+G_\kappa+G_b-\rho\varepsilon-Y_M+S_\kappa$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_j)}{\partial x_j}=\frac{\partial\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]}{\partial x_j}+\rho C_1 S\varepsilon-\rho C_2\frac{\varepsilon^2}{\kappa+\sqrt{\nu\varepsilon}}+C_{1\varepsilon}\frac{\varepsilon}{\kappa}C_{3\varepsilon}G_b+S_\varepsilon$$

$$here, C_1=\max\left[0.43,\frac{\eta}{\eta+5}\right],\eta=S\frac{\kappa}{\in},S=\sqrt{2S_{ij}S_{ij}}$$

Here, G_k and G_b are generation of turbulent K.E. due to mean velocity gradients and buoyancy respectively. Y_M is represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. S and S_ε are user-defined source terms for ε and k respectively.

VII. RESULTS AND DISCUSSION

To find the drag and flow analysis it is essential to know about the static pressure variation around the body and how turbulence effect behind the body. All these effects can be easily find out by the help various contours which are obtained by the help of post-processing (fluent). The results of CFD analysis Correspond to different position are discussed in the form of static pressure, turbulent viscosity ratio and stream function contour. In fig.4 Study of static pressure contours help in understanding of sudden deceleration of flow and red color indicates that static pressure raises lower half of the nose section. At top of the nose part flow is accelerate and create suction zone. And velocity gradually decreases at top surface of body. In fig.5 the contour of turbulent viscosity ratio shows the some vortex pattern created behind the Ahmed body means the flow separation occurs in this regime. And back flow happens and as a result vortex created. In fig.6 Shows the contour of stream function and it captured the flow separation behind the slant edge of the body. In 2-D model we see a little flow separation at the rear part. Recirculation zone of flow gives the idea about the strength of form drag or pressure drag. In fig.7 shows that velocity vector contour where clearly shows the circulation appears behind the body. And more velocity at top frontal part of the Ahmed body.

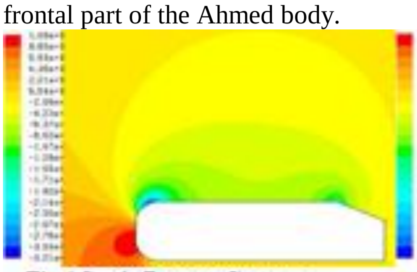
Fig.4 Static Pressure Contour

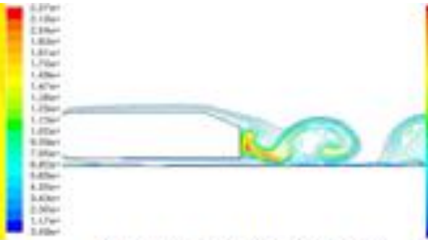
Fig.5 Turbulent Viscosity ratio.

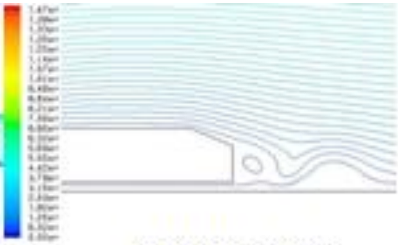
Fig.6 Steam function

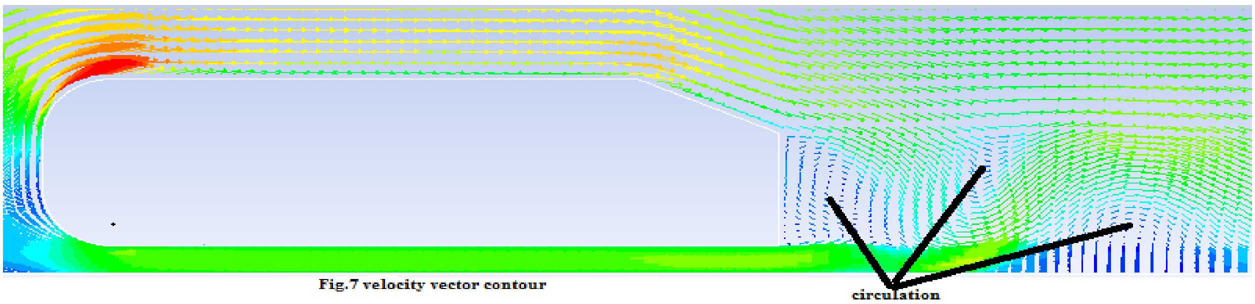

Fig.7 velocity vector contour

VIII. COMPARISON OF VELOCITY PROFILE

The following graphs on the non-dimensional axis show the time-averaged velocity profiles where CFD results compared with experimental data for slant angle equals 25 degree. For a particular position along the block we studied the velocity behavior at different location above the body. The computational result calculated by k- ε turbulence model. Here we calculate velocity profile at different axial location(x/h).when the flow is smooth there is a good agreement with both graph but at rear position there is a more variation among both graphs value. It studied that there is some uncertainty or standard deviation among the result. The deviation varies at different position(x/h).Time-averaged velocity graph (Fig.8) gives the idea how and where the velocity variation occurs around the Ahmed body.

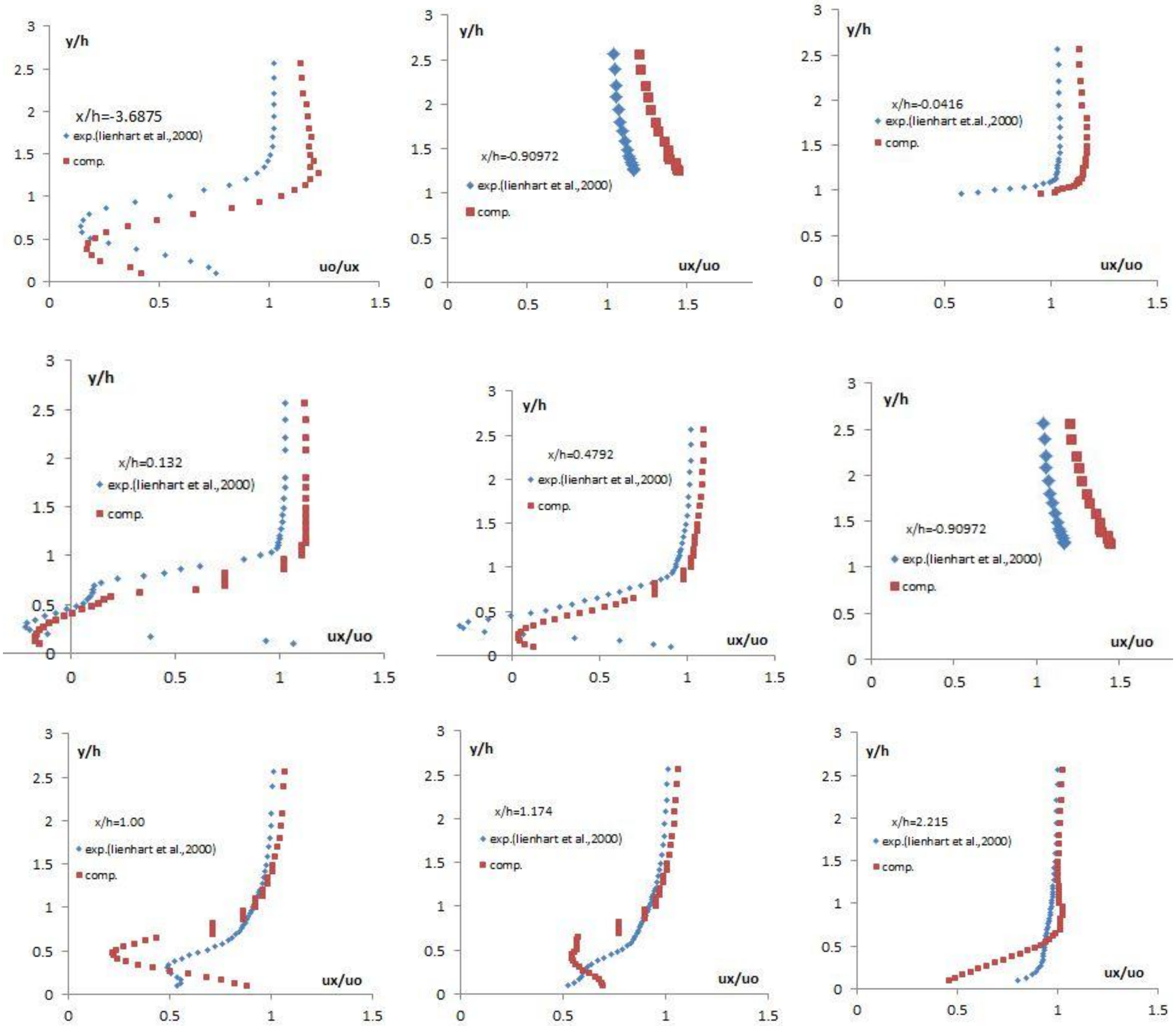

Fig 8: Time –averaged axial velocity at various X/h position around the car

IX. CONCLUSION

For the two-dimensional, 25° Ahmed body it can be able to capture separation along the slant edge for different turbulence models. The Wake region created behind the body which in nicely captured in 2d case also. so large energy losses are often associated with boundary layer separation. Two-dimensional study helps educational experience and concept of flow analysis around bluff body in less time compare to three dimensional Ahmed body. Present studied shows that the Nature of Time average velocity profile for both computational and Experimental are similar and it is a satisfactory result because the experimental result comes from the three dimensional geometry. The Reliable k-ε turbulence model gives the better result compare the other models. This level of study is very helpful for the beginners of CFD. Because in less period of time they completed external turbulent flow analysis works on different type of bluff bodies.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 3, May 2013

REFERENCES

[1] Parihar and A. Kulkarni, F. Stern and T. Xing , S. Moeykens, Using Flow Lab, An Educational Computational Fluid Dynamics Tool, To Perform A Comparative Study Of Turbulence Models, Computational Fluid Dynamic Journal, 2006,VOL 15;NUMB1,Pages 175-182.

[2] Emmanuel Guilmineau, Computational Study of flow around a simplified car body, journal of wind engineering and industrial aerodynamics 96, pp. 1207-1217, 2008.

[3] S.R. Ahmed and G. Ramm, Some Salient Features of the Time Averaged Ground Vehicle Wake, SAE Technical Paper, 840300, 1984.

[4] W.Angelis, D.Drikakis, F.Durst, W.Khier, Numerical and Experimental Study of the flow over a two-dimensional car model, journal of wind engineering and industrial aerodynamics 62(1996) 57-79.

[5] S.R.Ahmed, Wake structure of typical automobile shapes. Transactions of ASME, journal of fluids engineering V105, P429-434, 1983.

[6] H. Lienhart, C. Stoots, and S. Becker, “Flow and Turbulence Structure on the Wake of a Simplified Car Model (Ahmed Model)”, DGLR Fach. Symp. Der AGATAB, Stuttgart University, 2000.

[7] T.J.Craft, S.E.Gant, H.lacovides, B.E.Launder, computational study of flow around the Ahmed car body, 9th ERCOFTAC workshop on refined turbulence modeling, Darmstadt University of Technologgy, Germany, 2001.

[8] S. Kapadia, S. Roy, M. Vallero, K.Wurtzler, and J. Forsythe, Detached-Eddy Simulation over a Reference Ahmed Car Model, AIAA-2003- 0857.

[9] CFD Lab4 (Ahmed Car), Technical Note, http://css.engineering.uiowa.edu/~me_160/docUments/CFD_lab4.pdf.

[10] 10th joint ERCOFTAC (SIG-15)-IAHR-QNET/CFD Workshop on Refined Turbulence Modeling, http://cfd.me.umist.ac.uk/ .

[11] F.R.Menter, 1993, Zonal two-equation k-w turbulence models for aerodynamic flow. In AIAA 24th Fluid Dynamics Conference, Orlando, FL.AIAA paper 93-2006.

[12] T. Weinkauf, H.-C. Hege, B.R. Noack, M. Schlegel and A. Dillmann. Coherent structures in a transitional flow around a backward-facing step. Phys. Fluids, Special Section: Gallery of Fluid Motion, 15(9), S3, 2003.

[13] Spalart P.R., Jou W.H., Strelets M., Allmaras S.R.: Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach, 1st AFOSR Int. Conf. on DNS/LES, Aug. 4-8, 1997, Ruston, LA. In: Advances in DNS/LES, C. Liu and Z. Liu Eds., Greyden Press, Columbus, OH, USA (1997).

[14] Spalart, P.R. and Allmaras, S.R., 1994, “A One-Equation Turbulence Model for Aerodynamic Flows,”La Recherche Aerospatiale1, pp. 5-21.

[15] Krajnovic, S. Davidson, L., 2004, Large-eddy simulation of the flow around simplified car model.

[16] Kapadia, S., Roy, S., 2003, Detached eddy simulation over a reference Ahmed car model, pp. 1-10.

[17] Gohlke, M., Beaudoin, J. F., Amielh, M., Anselmet, F., 2007, Experimental analysis of flow structures and forces on 3D-bluff-body in constant cross-wind , Exp Fluids , vol. no.43 , pp. 579-594.

[18] Beaudoin , J.F. , Aider , J.L.,2008, Drag and lift reduction of a 3D bluff body using flaps , Exp Fluids , vol. no. 44, pp. 491–501.

AUTHOR BIOGRAPHY

Deepak Kumar Kalyan, a PG student at MNNIT, Allahabad. He had completed Bachelor degree in year 2009 from aeronautical engg department from Aesi and presently pursuing master degree with specialization FLUID ENGG. He is currently working on modeling and CFD analysis on centrifugal pump impeller. **Email-deepakkalyan007@gmail.com**

A.R. Paul, an Assistant professor of Applied Mechanic Department at MNNIT, Allahabad. His area of research Fluid Dynamics, Aerodynamics, Flow Control, CFD, Turbulence, Fluid Machineries, Bio-Fluids .The author having more than 10 years of teaching, and research experience. He has about 39 national and international publications. He is also author of various engg. Books.