Research Journal of Recent Sciences ISSN 2277-2502
Vol. **1(9),** 1-5, September **(2012)** *Res.J.Recent Sci.*

Numerical Investigation of External Flow around the Ahmed Reference Body Using Computational Fluid Dynamics

Chauhan Rajsinh B. and Thundil Karuppa Raj R.
School of Mechanical and Building Sciences, VIT University, Vellore-632014, TN, INDIA

Available online at: www.isca.in
Received 22nd March 2012, revised 29th March 2012, accepted 30th March 2012

Abstract

This paper presents a finite-element based numerical simulation for the prediction of flow around the Ahmed body. The flow solver used is ACUSOLVE, developed by ACUSIM software. In this investigation an effort was made to investigate the fully developed turbulent flow over Ahmed Body and to evaluate the effect of slant angle. Understanding this aerodynamic phenomenon helps us in reducing fuel consumption, increase the stability and passenger comfort. In this study, the spalart-allmaras (S-A) turbulence model is used in order to reduce the computational cost at high Reynolds number. Two separate cases have been solved for two different upstream velocities and results are compared. The results are presented in the form of drag coefficient values and flow field which includes velocity contour and velocity vector fields. The validation is carried out by a simulation around the Ahmed body with the slant angle of 25° with stilts. The results are compared with the actual wind-tunnel experimental data. Moreover, the capabilities of ACUSOLVE code to predict the flow around the Ahmed Body has been analyzed by comparing the flow structure at wake region with experiments. The pre-processing & post processing for this study is carried out with the help of HYPERMESH and HYPERVIEW software respectively.

Keywords: Ahmed Body, S-A model, CFD, wake region, drag.

Introduction

The external aerodynamics of a car determines many relevant aspects of an automobile such as stability, comfort and fuel consumption at high cruising speeds[1]. The flow around vehicles is characterized by highly turbulent and three-dimensional separations, and there is a growing need for more insight into the physical features of these dynamical flows on the other hand, and powerful numerical tools to analyze them on the other hand. Computations based on Reynolds-Averaged Navier Stokes Equations (RANS) are common in industry today. Although they are very successful in predicting many parts of the flow around a vehicle, they are unable to predict unsteadiness in the wake region. The failure in predicting the base pressure is the major reason for the large discrepancy in drag prediction between experiments and numerical simulations.

In order to investigate the behavior of newly developed turbulence models for complex geometry cases, a simplified car model, known as the Ahmed body, has been tested by Ahmed[2]. 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-1.

As the wake flow behind the Ahmed body is the main contributor to the drag force, accurate prediction of the separation process and the wake flow are the key to the successful modelling of this case. To simulate the wake flow accurately, resolving the near wall region using accurate turbulence model is highly desirable. This paper will study the effectiveness of Spalart-Allmaras (S-A) turbulence models, for the modeling of the flow over the Ahmed body, and shows the behavior of S-A turbulence model, as well as the effect of the grid layout on the numerical results. This model has been selected due to the availability of the experimental results[3].

The aim of this paper is to simulate the flow around the Ahmed Body at different velocities (i.e. V=40 m/sec and V=60 m/sec) and study the variations in the flow at wake region with the change in velocities. The main importance of this work is to exhibit the capability of numerical modeling with Spalart-Allmaras turbulence model for ACUSOLVE commercial CFD code.

Methodology

Geometrical Modeling: The Ahmed Body model with domain is modeled using SOLIDWORKS CAD modeling tool based on the parameters given in figure 1. The slant angle at the rear end is kept as 25° which is identical with the experimental model.

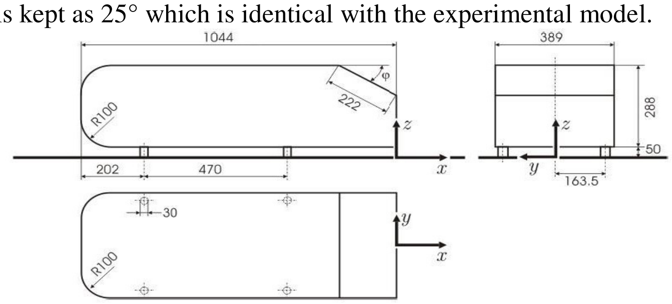


Figure-1
Schematic of the Ahmed Body model with 25° Slant angle[2]
(All dimensions are in mm)

The geometry of the Ahmed body is shown in figure-1. The slant angle is adjustable and is the main variable model-parameter in the experimental investigations. In this study, only 25° slant angle was investigated. The Ahmed body, of length (L) 1044 mm, height (H) 288 mm, and width (W) 389 mm, was placed at 50 mm from the ground (G). The model is mounted on four cylindrical struts with a diameter (f) of 30 mm. The reference axis (X, Y, and Z) is linked to the model. The origins of these axis lies at the point O located on the floor of the ground of the wind tunnel, on the base of the model and in the symmetry plane of the model. The computational domain starts 2 × L in front of the model and extends to 5 × L behind the model. The width of the domain is 1.87 m and its height is 1.4 m. These dimensions are recommended as per the ERCOFTAC workshop on Refined Turbulence Modeling[4].

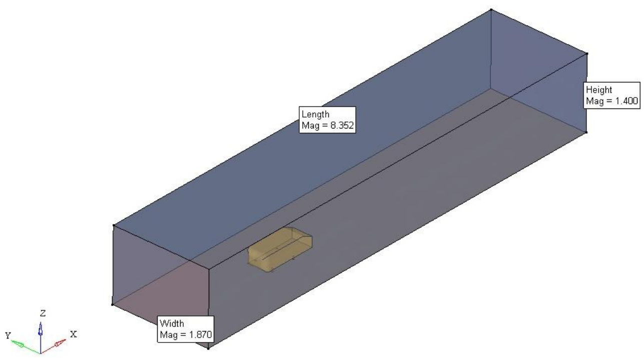


Figure-2
Schematic of the Ahmed Body model with Computational Domain

Grid Generation: The 3-D model is discretized in HYPERMESH, which a highly efficient generic pre-processing tool. In order to capture both the pressure and velocity boundary layers the entire model is discretized using tetrahedral mesh elements which are accurate and involve less computation effort when solved with ACUSOLVE CFD code. Fine control on the tetrahedral mesh near the Ahmed Body walls allows capturing the boundary layer gradient accurately. The entire wind tunnel with Ahmed Body is considered as a single fluid domain. There is only surface mesh on the Ahmed Body; no solid mesh has been created for body as it is not required: i. element type: tetrahedral, ii. total number of nodes: 2.85 Millions, iii. total number of elements: 15.55 millions, iv. No. of boundary layers around Ahmed Body : 16, v. first layer thickness near body surface: 0.2mm, vi. boundary layer growth rate: 1.2

The fluid mesh is made finer along the body surfaces so that there is good control over the number of nodes. Smooth Transition between the element of last boundary layer and core tetra element has been assured as any aggressive transition may not give the correct flow behavior in the results. Gradual increment for core tetra mesh achieved with the use of appropriate growth rate.

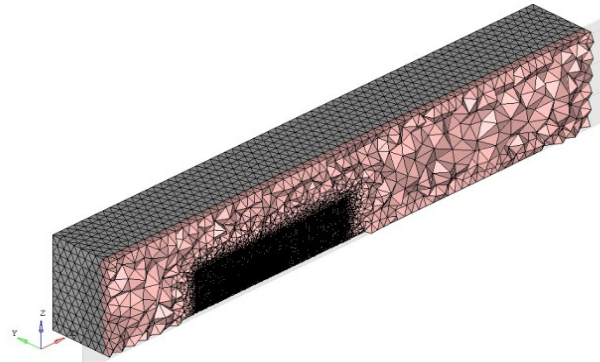


Figure 3
Fluid Mesh: Cut plane along symmetry axis

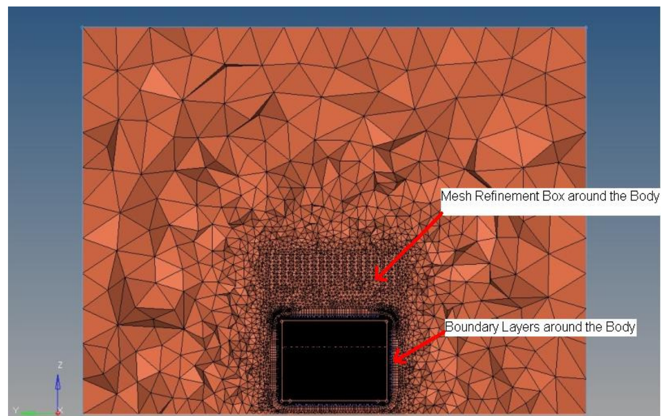


Figure-4
Fluid Mesh: Cut plane along X- axis

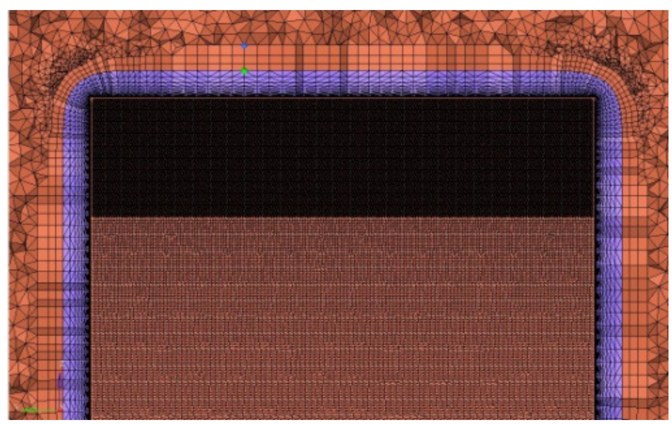


Figure-5
Grid Distribution near the Ahmed Body

Governing Equations: The 3-dimensional flow around the Ahmed Body has been simulated by solving the appropriate governing equations viz. conservation of mass and momentum using ACUSOLVE commercial CFD code. The energy equation is not considered because the heat transfer is considered to be zero. Turbulence is taken care by standard Spalart-Allmaras (S-A) model. The Spalart–Allmaras model is a one equation model for the turbulent viscosity. It solves a transport equation for a viscosity-like variable $\tilde{\nu}$[5].

Conservation of mass : $\nabla.(\rho\vec{V}) = 0$

$$x\text{-momentum} : \nabla.(\rho u\vec{V}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$$

$$y\text{-momentum} : \nabla.(\rho u\vec{V}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g$$

$$z\text{-momentum} : \nabla.(\rho u\vec{V}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z}$$

The one-equation S-A model is given by the following equation:

$$\frac{\partial \hat{\nu}}{\partial t} + u_j \frac{\partial \hat{\nu}}{\partial x_j} = c_{b1}(1-f_{t2})\hat{S}\hat{\nu} - \left[c_{w1} f_w - \frac{c_{b1}}{\kappa^2} f_{t2}\right]\left(\frac{\hat{\nu}}{d}\right)^2 + \frac{1}{\sigma}\left[\frac{\partial}{\partial x_j}\left((\nu+\hat{\nu})\frac{\partial \hat{\nu}}{\partial x_j}\right) + c_{b2}\frac{\partial \hat{\nu}}{\partial x_i}\frac{\partial \hat{\nu}}{\partial x_i}\right]$$

Boundary Condition Setup: In AcuConsole (pre-processor for ACUSOLVE) the fluid domain is defined. There is no solid domain involved in study. The flow in this study is turbulent, hence S-A turbulence model is chosen. The boundary conditions are specified in AcuConsole pre-processor and then the file is exported to the solver. The upstream velocities for two cases are: 40 m/sec and 60 m/sec. The Reynolds number, based on the length of the model, is 2.78×10^6. Besides, following Conditions were applied for solving the case: Material: Air, Inlet: Velocity, Outlet: Pressure (Atmospheric)

Numerical Solution Approach: Solution phase is completely automatic. The FEA software generates the element matrices, computes nodal values and derivatives, and stores the result data in files. These files are further used by the subsequent phase (post-processing) to review and analyze the results.

The CFD flow solver used is ACUSOLVE, which is a General purpose 3-dimensional, unstructured flow solver which uses the incompressible Reynolds-averaged Navier-Stokes equations. The solver is based on the finite element method to build a spatial discretization of the transport equations. The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure equation. All equations are solved using Garlinkan Least Square method. Equal order nodal interpolation for all variables are done while solving. The computations are performed on a HPC Cluster (12 CPU @ 2.66 GHz, 32 GB RAM). The total elapsed CPU time is nearly 13 hours for a total of 80 time steps.

Two simulations have been carried out with the same model dimensions, grid, boundary conditions and modeling technique except the change in upstream velocity. The variation for velocity is kept as 40 m/sec and 60 m/sec

Results and Discussion

Experimentally, for the 40 m/sec case, the two strong counter-rotating vortices emanating from the slant are present and the flow separates in the middle region of the top edge and reattaches on the slant[6]. An illustration of the experimental flow is illustrated by the sketch given below. Consequently, the turbulence model must predict separation and reattachment on the slant.

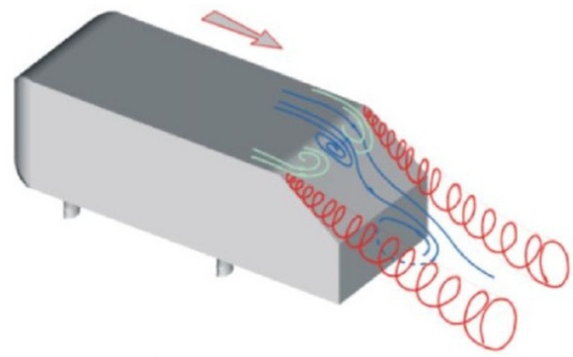


Figure-6
Development of the flow for the Ahmed Body with 25° slant angle

Case: 1 Velocity: 40m/sec: The spalart–allmaras model for which the flow remains attached, predicts separation just past the top of the slant, as in experiment, but does not predicts the same reattachment seen in the experiment. The figures below show the wake behind the body predicted using S-A turbulence model and using experiments. We see that the simulation with the S-A turbulence model predicts a massive separation on the slant. The reattachment on the slant is not present clearly which can be a possible limitation for S-A turbulence model.

Case: 2 Velocity: 60m/sec: For higher velocity also, the flow behavior remains same as in case-1. It is almost matching with the experimental results of 40 m/sec. The major change that is identified is the recirculation size and location in the wake, which is not similar to case-1. Apart from that, a recirculation bubble is seen on the slant end which is not clearly visible in case-1.

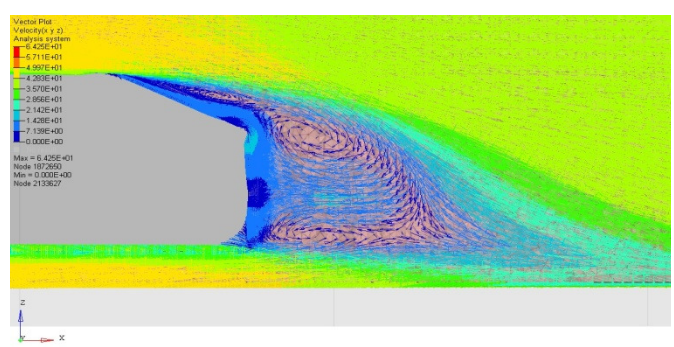


Figure-7
Prediction for Wake region behind the Ahmed Body for case-1

Research Journal of Recent Sciences ____ ISSN 2277-2502
Vol. **1(9),** 1-5, September **(2012)** *Res. J. Recent Sci.*

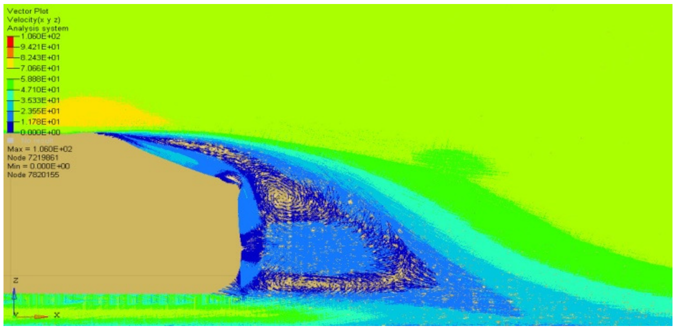


Figure-8
Prediction for Wake region behind the Ahmed Body for case-2

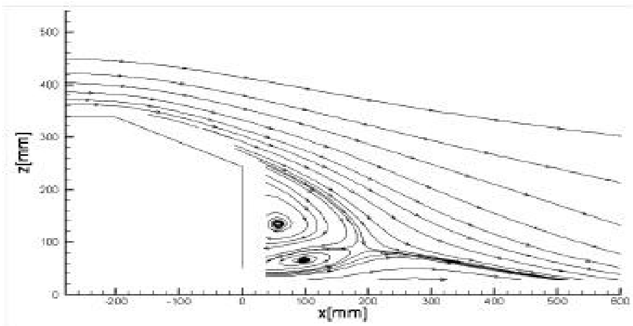


Figure-9
Experimental Prediction for Wake region behind the Ahmed Body for case-1

The tendency of S-A to predict a massive separation is clearly visible in figure-7 and 8, which presents the rear slant, centerline streamlines. We can see that with S-A model, the wake is in good agreement for both cases with experiments. But, the flow over the rear slant is not well predicted. The experiments show separation at the top edge and reattachment of the flow about half the length along the slant back. The computation with the S-A turbulence model shows a massive separation on slant but it does not predict the reattachment as in the experiments for both the cases.

Drag Coefficient Comparison: The drag coefficient value obtained with the S-A model is 0.266 for case-1 and 0.740 for case-2. The same for the experiments by Ahmed is 0.290. This drag coefficient computed takes into account the drag of the model and that of the feet[7]. The S-A model gives the same tendency but the drag on the slant is over-estimates. These differences are not surprising because the simulation do not predict correctly the flow on the slant.

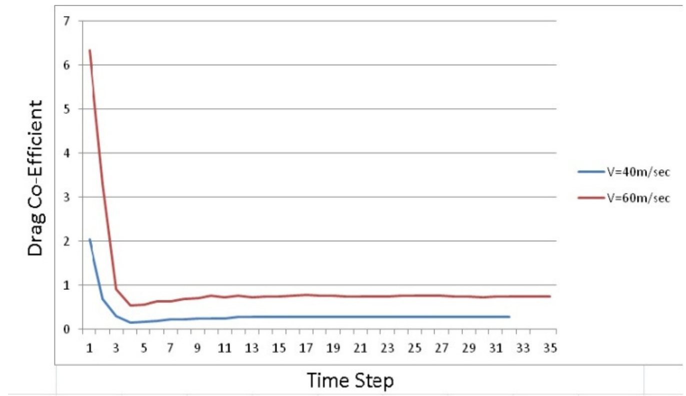
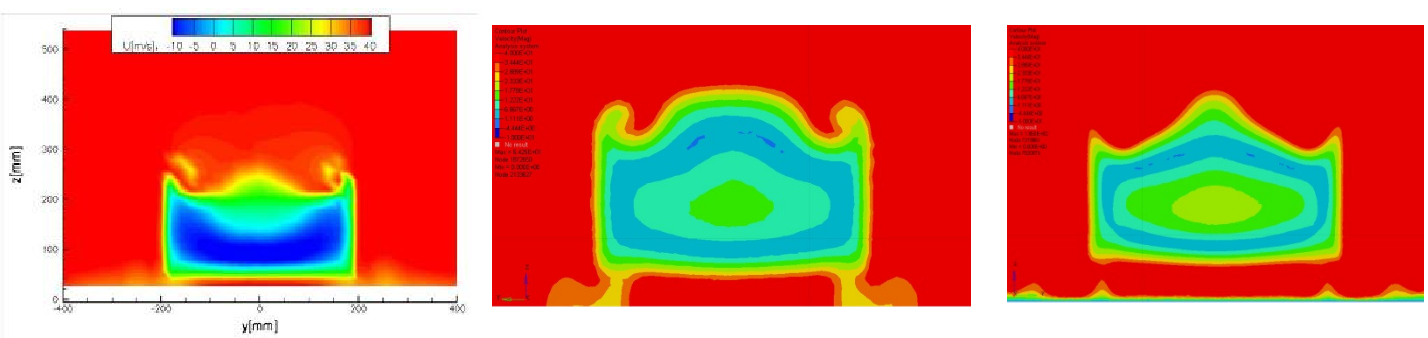


Figure-10
Drag Co-Efficient Comparison

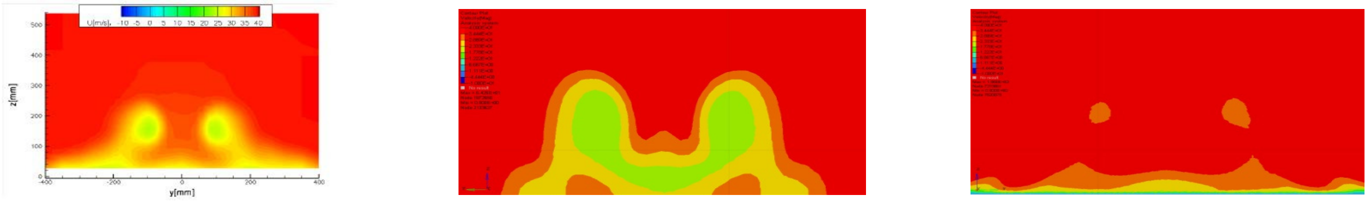
As shown in figure-10, the drag coefficient for 60 m/sec velocity is almost three times when compared to that of 40 m/sec velocity. Hence, it can be stated that increasing velocity by 20 m/sec gives significant rise in drag coefficient.

Wake Region Flow Structure Analysis: The analysis of the flow structures in the wake region of the Ahmed body is depicted in figures 11 to 13. The figures show the streamwise velocity component at different planes behind the body[8]. The wakes of the C-pillars are slightly weak in the simulation than in experiments[9]. The structural analysis of the Ahmed body can be carried out using Finite Element Techniques similar to[10,11].



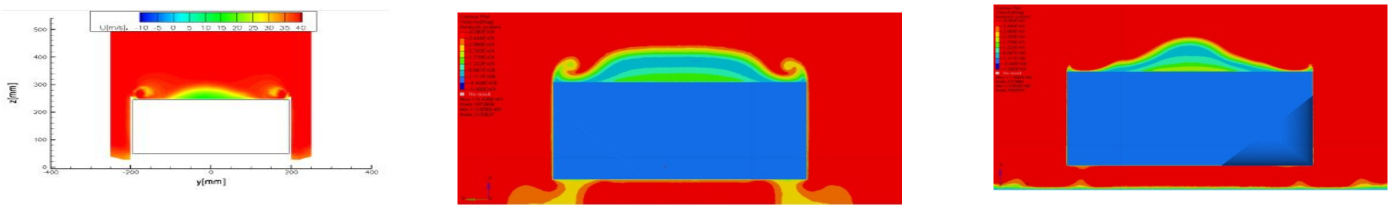
Drag Co-Efficient Comparison
Experiments V=40 m/sec V=60m/sec

Figure-11
Comparison of Wake flow at X=0



Experiments V=40 m/sec V=60m/sec

Figure-12
Comparison of Wake flow at X=80mm



Experiments **V=40 m/sec** **V=60m/sec**

Figure-13
Comparison of Wake flow at X=500mm

Conclusion

The flow field for the external flow over the Ahmed body can be simulated by computational approach. Simulation with S-A turbulence model, has been carried out for two different velocities for the generic Ahmed body with 25° slant angle. The turbulence model predicts the topology of the flow correctly. At the 25° slant angle, the simulation predicts massive separation but the flow over the rear slant is not well predicted. Whereas the experiment shows reattachment about half-way down the center of the face. Following are some other outcomes that can be listed out from this study: The comparison for the drag coefficient for both velocity values has been carried out and it is found that increasing velocity by 20 m/sec gives thrice the more drag coefficient value. The numerical modeling of the flow around Ahmed Body gives clear understanding flow structure near the slant and at the wake region of the body. The flow prediction capabilities for ACUSOLVE code is observed and found that with the use of S-A turbulence model, it is able to predict the flow well in agreement with the experimental results.

Acknowledgments

The support of the presented work by Altair Engineering-India is gratefully acknowledged. Special thanks to Dr. Marc Ratzel and Mr. Rajesh Krishnan for many helpful discussions and for the attribution of HPC.

References

1. Hucho W.H., Aerodynamics of Road Vehicles, SAE International, Warrendale, PA **(1998)**

2. Ahmed S.R., Ramm G. and Faltin G., Some Salient Features of the Time-Averaged Ground Vehicle, SAE Paper 840300 **(1984)**

3. Lienhart H. and Becker S., Flow and Turbulence Structures in the Wake of a Simplified Car Model, SAE Paper 2003-01-0656 **(2003)**

4. Manceau Rand and Bonnet J.P., Proceedings of 10th Joint ERCOFTAC (SIG-15)/IAHR/QNET-CFD Workshop on Refined Turbulence Modeling, Poitiers, France **(2002)**

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

6. Guilmineau E., Computational Study of Flow around a Simplified Car Body, *Journal of Wind Engineering and Industrial Applications*, **96**, 1207-1217 **(2008)**

7. Guilmineau E., Numerical simulation with a DES approach, SAE Paper 2010-01-0758 **(2010)**

8. Minguez M., Pasquetti R. and Serre E., High-Order Large Eddy Simulation of Flow Over the “Ahmed Body” Car Model, Physics of Fluids, **20**, 095101 **(2008)**

9. Kapadia S., Roy S. and Wurtzler K., Detached eddy simulation over a reference Ahmed car model, AIAA paper no. 2003-0857 **(2003)**

10. Kumar Krishan and Aggarwal M.L., A Finite Element Approach for Analysis of a Multi Leaf Spring using CAE Tools, *Research Journal of Recent Sciences*, **1(2),** 92-96 **(2012)**

11. Purkar T. Sanjay and Pathak Sunil, Aspect of Finite Element Analysis Methods for Prediction of Fatigue Crack Growth Rate, *Research Journal of Recent Sciences,* **1(2)** 85 -91 **(2012)**