International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 3, March - 2014

CFD Analysis Of Flow Processes Around The Reference Ahmed Vehicle Model

(Comparison of two Turbulence Flow Models)

Shivi Kesarwani[1], Yagyesh Jayas[2], Vinay Chhalotre[3]
[1,3] Department Of Mechanical Engineering, ABES Engineering College, Ghaziabad (U.P.), INDIA
[2] Department Of Mechanical Engineering, International Armored Group, Ras Al Khaimah, UAE

Abstract— **Current Automobile industries need a firm hand on aerodynamic flow processes to improve the vehicle design. Enhanced aerodynamic design can lead to high performance engine and minimize fuel consumption. The main objective of the present work is to study the three-dimensional flow around a ground vehicle as well as understanding the different numerical flow analysis mathematics for the aerodynamic simulation purposes. This study work used the reference Ahmed vehicle model as it is a simple geometric body which generates the flow around a car. The Ahmed reference vehicle model is investigated by means of two different turbulence flow models (Reynolds stress model (RSM) and k-Ɛ model) using commercial computational fluid dynamics (CFD) code ANSYS FLUENT Version: 13. A viscous and incompressible fluid flow of Newtonian type governed by the Navier-Stokes equations is assumed. An unstructured tetrahedral mesh with finite volume discretisation is used in the computational analysis. The performances of Reynolds stress model (RSM) and k- Ɛ model have been compared. The simulated results compare well with the available experimental and simulation data for result validation.**

Keywords— Vehicle aerodynamics, Ahmed vehicle, CFD, incompressible viscous fluid, RSM, k- Ɛ model, Drag coefficient.

I. INTRODUCTION

There are a number of criteria by which to judge turbulence models. One criterion sometimes important to mathematically-minded model developers is the consistency and accuracy of the mathematics involved in the derivation of a model. A similar criterion is the belief that the best turbulence model is the one that correctly models the most and most fundamental physics of turbulence itself. Another approach to turbulence modeling looks solely at the solutions generated using a given turbulence model and compares the solutions to those generated by others and to experimental data. According to this line of reasoning, the best turbulence model is simply the one that best matches the experimental data, no matter what its origin. Still another approach concerns itself with a quality per unit cost ratio, considering that an accurate but computationally expensive turbulence model might be less useful than a slightly less accurate, inexpensive one. In the present section the turbulence models used in this study will be examined from a number of these points of view.

A reasonable beginning in the comparison of two or more turbulence models is a simple examination of the results they produce, with attention to the similarities and differences in the solutions. This paper compared the performances of Non-Equilibrium Realizable k-Ɛ model [3][4] and Reynolds stress model (RSM) for the flow processes around the reference ahmed vehicle model [7] with 12.5^{o} base slant using commercial CFD code ANSYS FLUENT Version: 13. The results (drag and vortex wake velocity) are validated with the experimental data of Ahmed et al. (1984) [1][7].

A. The Reference Ahmed Vehicle Model

The Ahmed reference model was originally developed for a time-averaged vehicle wake investigation (Ahmed et al. 1984). It is a car-like bluff body with a curved fore body, straight centre section and an angled rear end, representing a highly simplified ¼ scale lower medium size hatchback vehicle. The specific angle of the back end can be altered between 0°and 40°. The model's major dimensions are 1044mm x 389mm x 288mm. A diagram of the Ahmed reference model is shown in Figure 1. All dimensions listed in figure 1 are in mm.

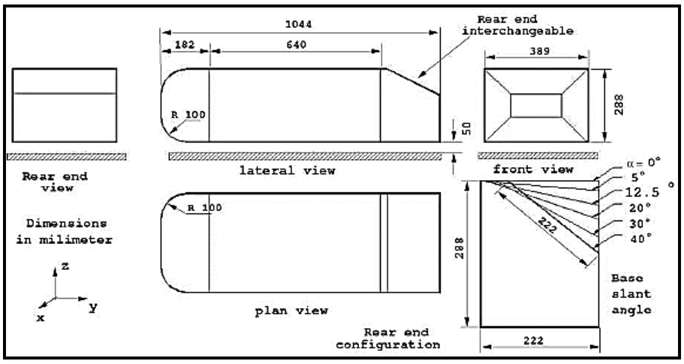

Figure 1:Schematic of the Ahmed body model[1]

II. TURBULENCE MODELS

A turbulence model is a computational procedure to close the system of *mean flow* equations. For most engineering applications it is unnecessary to resolve the details of the turbulent fluctuations. Turbulence models allow the calculation of the mean flow without first calculating the full time-dependent flow field. We only need to know how turbulence affected the mean flow. In particular we need expressions for the Reynolds stresses.

ANSYS FLUENT provides a large suite of turbulence models within the context of Reynolds-averaged Navier-Stokes (RANS) approach [3]. So, in this study two practically simple models have been adopted: Realizable k- ε model [3] [4] and Reynolds stress model (RSM) [10] with Non-Equilibrium wall functions.

A. Realizable k-ε model

This model uses the Boussinesq hypothesis [9] to relate the Reynolds stresses to mean velocity gradients:

Equation (1);

$$-\rho \langle u_i u_j \rangle = 2\mu_t S_{ij} - \frac{2}{3}\rho k \delta_{ij}$$

Where, μ_t is the coefficient termed turbulence "viscosity" (also called the eddy viscosity),

$$k = \frac{1}{2}\left(\langle u_1 u_1 \rangle + \langle u_2 u_2 \rangle + \langle u_3 u_3 \rangle\right)$$

is the mean turbulent kinetic energy,

$$S_{ij} = \frac{1}{2}\left[\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right] - \frac{1}{3}\frac{\partial U_k}{\partial x_k}\delta_{ij}$$

is the *mean* strain rate.

Now, the *Realizable k-ε model* comes under two-equation group of models in which two additional transport equation for turbulence kinetic energy, *k*, and its rate of dissipation, *ε,* need to be solved in order to achieve closer:

Equation (2);

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\epsilon - Y_M + S_k$$

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

Where,

$$C_1 = \max\left[0.43, \frac{\eta}{\eta + 5}\right], \quad \eta = S\frac{k}{\epsilon}, \quad S = \sqrt{2S_{ij}S_{ij}}$$

In these equations, P_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. P_b is the generation of turbulence kinetic energy due to buoyancy.

And the values for all constants in above equations have been set by the solver as recommended:

$$C_{1\epsilon} = 1.44, \quad C_2 = 1.9, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.2$$

B. Reynolds stress model (RSM)

The Reynolds Stress Models (RSM), also known as the Reynolds Stress Transport (RST) models, are higher level, elaborate turbulence models. The method of closure employed is usually called a *Second Order Closure*. In RSM, the eddy viscosity approach has been discarded and the Reynolds stresses are directly computed. The exact Reynolds stress transport equation accounts for the directional effects of the Reynolds stress fields.

The Reynolds stress model involves calculation of the individual Reynolds stresses, $\rho\overline{u_i' u_j'}$, using differential transport equations. The individual Reynolds stresses are then used to obtain closure of the Reynolds-averaged momentum equation.

The exact transport equations for the transport of the Reynolds stresses, $\overline{u_i' u_j'}$, may be written as follows:

Equation (3);

$$\frac{\partial}{\partial t}\left(\rho\overline{u_i' u_j'}\right) + \frac{\partial}{\partial x_k}\left(\rho u_k \overline{u_i' u_j'}\right) = -\frac{\partial}{\partial x_k}\left[\rho\overline{u_i' u_j' u_k'} + \overline{p'\left(\delta_{kj}u_i' + \delta_{ik}u_j'\right)}\right]$$

$$+\frac{\partial}{\partial x_k}\left[\mu\frac{\partial}{\partial x_k}\left(\overline{u_i' u_j'}\right)\right] - \rho\left(\overline{u_i' u_k'}\frac{\partial u_j}{\partial x_k} + \overline{u_j' u_k'}\frac{\partial u_i}{\partial x_k}\right) - \rho\beta\left(g_i\overline{u_j'\theta} + g_j\overline{u_i'\theta}\right)$$

$$+\overline{p'\left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i}\right)} - 2\mu\overline{\frac{\partial u_i'}{\partial x_k}\frac{\partial u_j'}{\partial x_k}}$$

$$-2\rho\Omega_k\left(\overline{u_j' u_m'}\epsilon_{ikm} + \overline{u_i' u_m'}\epsilon_{jkm}\right) + S_{user}$$

Or,

$$C_{ij} = D_{T,ij} + D_{L,ij} + P_{ij} + G_{ij} + \phi_{ij} - \epsilon_{ij} + F_{ij} +$$

Local Time Derivate +

User-Defined Source Term

Where,
C_{ij} is the Convection-Term,
$D_{T,ij}$ equals the Turbulent Diffusion,
$D_{L,ij}$ stands for the Molecular Diffusion,
P_{ij} is the term for Stress Production,
G_{ij} equals Buoyancy Production,
ϕ_{ij} is for the Pressure Strain,
ϵ_{ij} stands for the Dissipation and
F_{ij} is the Production by System Rotation.

Of these terms, C_{ij}, $D_{L,ij}$, P_{ij}, and F_{ij} do not require modeling. After all, $D_{T,ij}$, G_{ij}, ϕ_{ij}, and ϵ_{ij} have to be modeled for closing the equations.

And the values for all constants in above equations have been set by the solver as recommended:

$$C_s \approx 0.25, C_l \approx 0.25, C_\gamma \approx 0.25$$

III. METHODOLOGY

A. Geometric Parameters

The Ahmed model has a length *L* = 1044 mm, the height *H* and the width *B* are defined according to the ratio (*L*: *B*: *H*) = (3.36:1.37:1). It has three main geometrical sectors: the front one, with boundaries rounded by elliptical arcs to induce an attached flow, a middle sector which is a box shaped sharp body with a rectangular cross section and, finally, a rear end sector. The 12.5º slant angle is analysed here, where the slant length is kept fixed to 222 mm.

The dimensions of the computational flow domain are taken relative to *L* as shown in the Figure 2 (G. Franck *et al.*, 2009) [5] [8]. The parallelepiped domain has 10L× 2L× 1.5L in the streamwise x, spanwise z and stream-normal y (vertical) Cartesian directions respectively. The body of length L is placed at a vertical distance of 50 mm from the ground (See Figure 1). The inlet flow section is placed 2.4 L upstream of the model front while the outlet flow section is placed 6.6 L downstream from the model rear end. The incoming flow is at 40000 mm/s with 1% turbulence intensity (%). Airflow is assumed to be incompressible. Outflow is assumed fully developed and the zero-gradient velocity boundary condition is imposed. The wind tunnel roof and walls are treated as no-slip (See Figure 2).

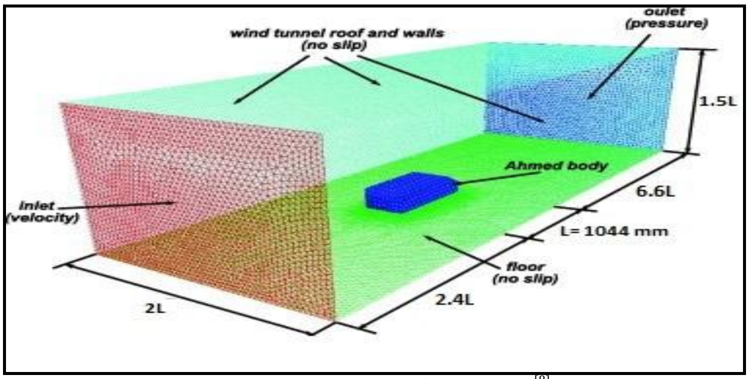

Figure 2: The computational flow domain [8]

B. Grid Description

The meshing process is performed with the ANSYS ICEM CFD mesh generator. The unstructured tetrahedral grid approach is applied. It involves a basic tetrahedral grid generation and the addition of layers of wedge elements for a better resolution close to the body surface. The total number of tetrahedral elements is 275, 8201 with 494,827 nodes. Due to the low-Reynolds turbulence model, the distance between the first fluid points and the walls is fixed to 0.01 mm. The distance y+ [6] [8] is near to 1 for the Reynolds number (Re) = 2.784×10^6. Figure 3 shows a mesh view of the symmetric plane.

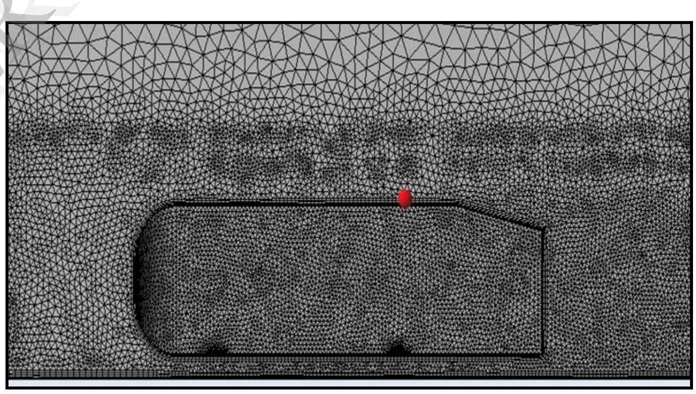

Figure 3: Mesh view of the symmetric plane

IV. Results And Discussions

The simulation results of two flow models: Reynolds stress model (RSM) and k-Ɛ model, are the averaged value over several iterations of the flow field equations and compared with the experimental data of Ahmed model. Here, a selection of results have been provided to show the main difference in the behavior of the two turbulence models with same wall function (Non-Equilibrium Wall function).

A. *Drag Production*

In Figure 4, the unsteady nature of the drag coefficient C_D (Iteration-evolutions) is shown. In each case, oscillations in its values are observed during startup but, after some time-steps, these oscillations become small and C_D approaches a constant value.

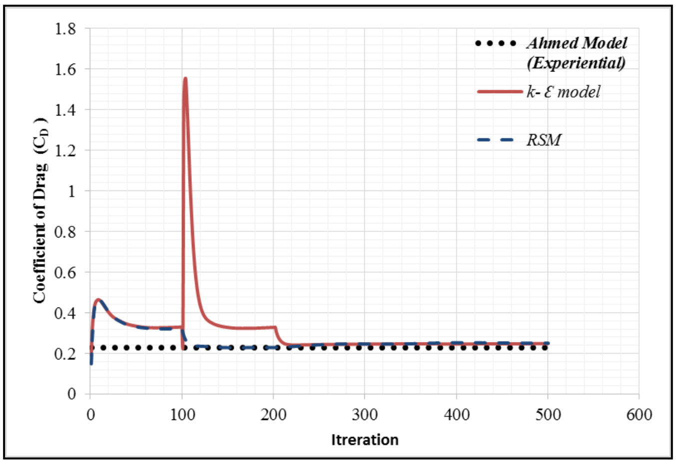

Figure 4: The unsteady nature of the drag coefficient C_D

The mean drag coefficient measured in the wind tunnel tests (Ahmed et al., 1984) for a slant angle of 12.5^o was C_{Dexp} =0.230 (See Figure 5) [1] [7], while the numerical simulation values are C_D=0.251 for the Reynolds stress model (RSM), and C_D= 0.249 for the k-Ɛ models, with a percentage relative error of $Ɛ_r$ %= + 9.130 and $Ɛ_r$ %= +8.607 respectively (See Table 1).

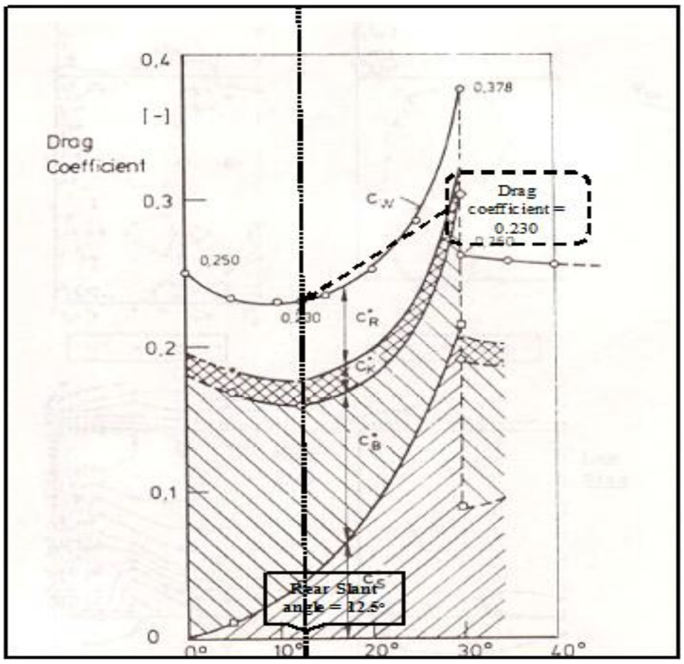

Figure 5: Characteristic drag coefficients for the Ahmed body for various rear slant angles [Experimental Data]

Table 1: Comparison of C_D and validation

Ahmed Model	*Drag Coefficient (C_D)*	*Percentage Error ($Ɛ_r$ %)*
Experimental[1] [7]	0.230	-----
Reynolds stress model (RSM)	0.251	+ 9.130
Realizable k-Ɛ model	0.249	+ 8.607

B. *Pressure Variations*

The pressure coefficient (C_P) for a slant angle 12.5^o is plotted in Figure 6 as a function of the streamwise coordinate at the top body surface. It can be observed that at the front end and rear slope, the value of C_P varies steeply for both the models.

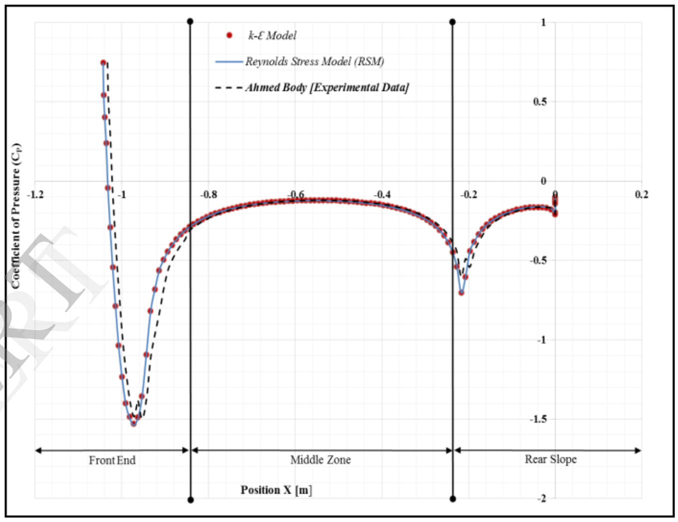

Figure 6: Comparison of C_P as a function of the streamwise coordinate

And it can also be observed that the turbulence models are in good agreement with the experimental data [7].

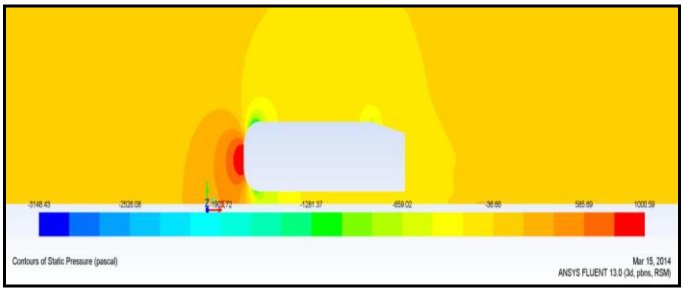

Figure 7: Pressure Contour Plot (RSM)

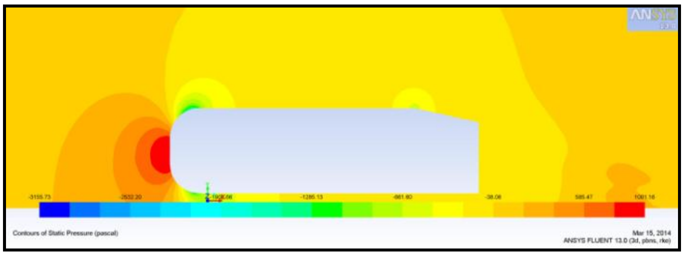

Figure 8: Pressure Contour Plot (k-Ɛ model)

The Figure 7 & 8 are presenting the contour plots of the pressure field for Reynolds stress model (RSM) and k-Ɛ model respectively. Here, k-Ɛ model is little bit over predicting the static pressure compared to RSM at the rear end of the Ahmed body (due to excess estimation of rear slant vortex flow by k-Ɛ model).

C. Velocity Field

The Figure 9 & 10 are showing the contour plots of mean velocity flow field in the symmetry plane of the Ahmed body. Here, it can be visualised that there is a flow separation between the roof top and the slant of the body due to sharp gradient in the geometry. Flow detachments occur on the sharp edges of the body. Vortical structure in k-Ɛ model is more extended than Reynolds stress model.

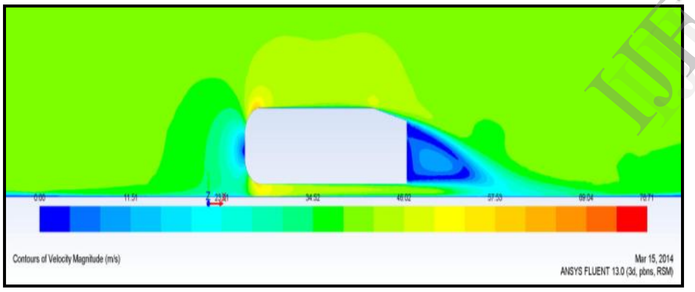

Figure 9: Mean velocity for Reynolds stress model

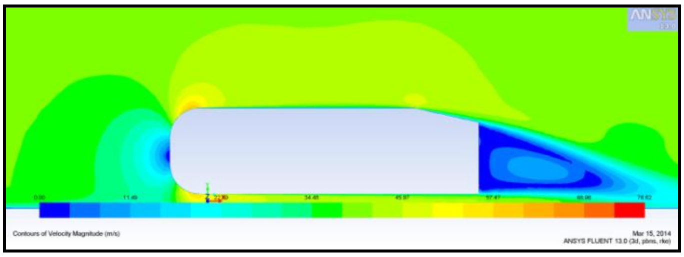

Figure 10: Mean velocity for k-Ɛ model

Figure 11 & 12 give a mean velocity profile comparison of Reynolds stress model (RSM) results with the realizable k-Ɛ model for the separation zone. Geometrical parameters are normalized by the height of the Ahmed body, ‘h’ (288 mm). Compared with the realizable k-Ɛ model, the Reynolds stress model gets better results for velocities above the rear slant and behind the Ahmed body, because the velocities predicted by the Reynolds stress model fit well with the experimental data[4]. The numerical results of the RSM and the k-ε model predicted a wake being recovered too soon at the downstream and predicted velocities have larger discrepancies when compared to the experiment data.

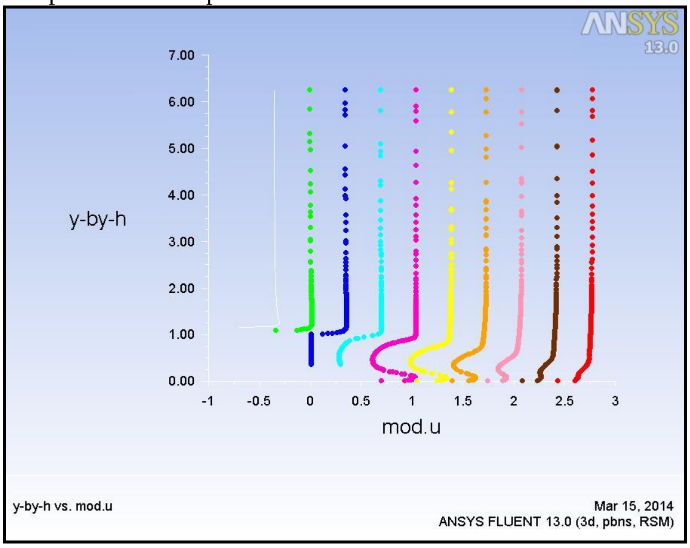

Figure 11: Mean velocity profile for Reynolds stress model

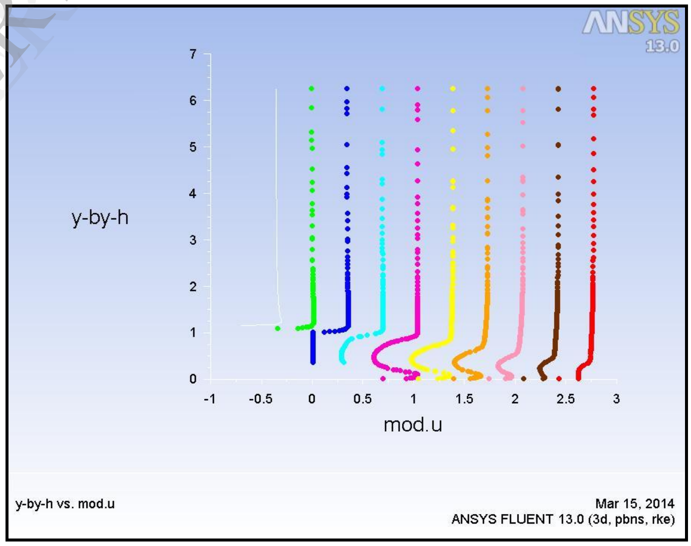

Figure 12: Mean velocity profile for k-Ɛ model

V. CONCLUSIONS

- It can firstly be concluded that the Reynolds Stress model (RSM) provides a more accurate simulation of the Ahmed body than the Realizable k-ε model. Despite this, both models significantly over predict the pressure drag over the front end, and thus cannot be used for accurate pressure drag force predictions.

In general, Reynolds stress models can model many flows where Realizable k-ε model fails; examples are: Flows where streamline curvature or curvature of solid boundaries is important, flows near stagnation points, rotating flows.

- The simulations predicted that the changes in pressure over the front end would have the most significant effect on drag force. This suggests that despite the absence of accurate drag predictions from the CFD, the flow structure and how it is altered by the inclusion of a wall is well modelled in this region.
- The simulation of the changes to the vortices shed from rear slope was also, in general, well predicted. Further experimental testing is required to ascertain whether the trends in flow velocity predicted by the CFD for the near wall vortex are mirrored by experimental data.

REFERENCES

[1] **Lienhart, H. and Becker, S.**, "Flow and Turbulence in the wake of a simplified car model," SAE Technical Paper Series, 2003-01-0656. Reprinted from: Vehicle Aerodynamics 2003 (SP-1786), 2003 SAE World Congress, Detroit, Michigan, March 3-6, 2003.

[2] **Smagorinsky, Joseph** (March 1963). "General Circulation Experiments with the Primitive Equations". General Circulation Research Laboratory, U.S. Weather Bureau, Washington, D.C.,Volume 91, Issue 3 (March 1963).

[3] **Liu Yunlong, Moser Alfred**, "Numerical modeling of airflow over the Ahmed body," CFD 2003, 11th Annual Conf. of the CFD Society of Canada, Vancouver, British Columbia, Canada, May 2003.

[4] **Durbin P.A.**: Separated Flow Computations with the k-ε-v^2 Model, AIAA Journal, V33, N4, PP659-664, 1995.

[5] **G. Franck, N. Nigro, M. Storti and J. D'elía**, "Numerical Simulation of The Flow around the Ahmed Vehicle Model," Latin American Applied Research 2009.

[6] "Best practice guidelines for handling Automotive External Aerodynamics with FLUENT" http://www.ansys.com.

[7] **Ahmed, S.R., Ramm, G., and Faltin**, G., "Some Salient Features of the Time-Averaged Ground Vehicle Wake," SAE 840300, 1984.

[8] **Shivi Kesarwani, Sujata Saha, N.K. Singh**, "CFD analysis of drag production flow processes around the reference Ahmed vehicle model," Manfex 2013, 2nd Annual International Conference on Manufacturing Excellence, Amity University Uttar Pradesh, May 2013, PP: 99-102 ISBN: 978-93-83083-17-6.

[9] **F. Menter.**, Two-equation eddy-viscosity turbulence model for engineering applications. AIAA Journal, 32:1598-1605, 1994.

[10] **Launder, B. E., Reece, G. J. and Rodi, W. (1975)**, "Progress in the Development of a Reynolds-Stress Turbulent Closure." Journal of Fluid Mechanics, Vol. 68(3), pp. 537-566.

Definitions, Acronyms, Abbreviations

CFD: Computational Fluid Dynamics
RSM: Reynolds Stress model
Unit Used: MMGS

C_D: Drag Coefficient = $\dfrac{Drag\ Force}{\frac{1}{2}\,U_\infty^2\,S}$

C_P: Pressure Coefficient
U_∞: Free-Stream velocity
u,v,w: velocity components in X, Y and Z directions respectively
Mod U: Modified velocity components in X
X: Streamwise co-ordinate
Y: Vertical co-ordinate
Z: Transverse co-ordinate
L: Ahmed Model length = 1044 mm
h: Ahmed Model height = 288 mm
ρ: air density
S: Frontal area of Ahmed model = 112,032 mm^2