International Conference on Mechanical, Industrial and Materials Engineering 2013 (ICMIME2013)
1-3 November, 2013, RUET, Rajshahi, Bangladesh.

Paper ID: FM-12

Effect of Roughness Height on the Turbulence Models for Ahmed Car Body Simulation

Abul Kalam Azad[1], Mohammad Arif Hossain[1], AKM Sadrul Islam[2]
[1]Department of Mathematics, Khulna University of Engineering & Technology, Khulna-9203, Bangladesh
[2]Department of Mechanical and Chemical Engineering, Islamic University of Technology, Dhaka,Bangladesh
E-mail: arifhkhan63@hotmail.com

Abstract

Effects of roughness height over various turbulence models are important for the computational simulation. To know the position of the point of separation of the recirculation, its behaviour and swirling structure around the vehicle we depend on turbulence models. But efficiency of turbulence models partly depends on the roughness height. In this study comparison between the turbulence models k-ε and RNG k-ε is done on the basis of their impact on the coefficient of drag over Ahmed car body. In the simulations all geometric quantities of Ahmed body are normalized with the body height. The channel length is about 8.8069 times of the length of the body and the blockage is about 4.27%. ANSYS-11 software is used for CFD analysis. In these simulations five roughness heights 0.0001m, 0.0002m, 0.0003m, 0.0004m and 0.0005m are used with the bulk air velocity 40m/s. The obtained drags are compared with the experimental values of LSTM.

Keywords: *Turbulence models, roughness height, CFD and aerodynamics.*

1. Introduction

Turbulence are of great importance in computational fluid dynamics .The understanding of the physics of turbulence is crucial and many different models have evolved to explain them. Sometimes the turbulence models are validated through vehicle aerodynamics. Computers have been used to solve fluid flow problems for many years. Numerous programs have been written to solve either specific problems, or specific classes of problems. From the mid-1970's, the complex mathematics required to generalize the algorithms began to be understood, and general purpose computational fluid dynamics (CFD) solvers were developed. Recent advances in computing power, together with powerful graphics and interactive 3D manipulation of models have made the process of creating a CFD model and analyzing results much less labor intensive, reducing time and, hence, cost. Advanced solvers contain algorithms which enable robust solutions of the flow field in a reasonable time.
A real-life automobile is a very complex shape to model or to study experimentally [1]. Because of the complexity of cars aerodynamics and in order to simplify studies Ahmed car body has become reference geometry (Montinat et al.,2008)[2]. The vehicle shape employed by Ahmed and Ramm (1984) is known as Ahmed body[3]. Several researchers have worked on the experiments and numerical modeling of the flow over the Ahmed body. Lienhart et al.,2000 conducted the experiments for two near slant angles (25^o, 35^o) at LSTM at Erlangen university, Germany. Craft et al. 2001 compared the performance of linear and non-linear k-ε model with two different wall functions. A dimensionless coefficient, Called drag coefficient and related to the drag force acting on the bluff body but the drag coefficient and turbulence are related to the roughness height [4]. In most CFD program a standard roughness height is need to set. But if we take much rough surface like sand grain size, we need to modify the wall function for the turbulence model with the more active roughness height. Flow over rough surface has significant importance in industrial application but much less knowledge is available for flow over rough surface. In this study comparison is done among the drag coefficients at different roughness heights over the Ahmed body for k-ε and RNG k-ε turbulence models. Velocity 40 m/s is taken in these simulations.

2. Description

The geometry of body is shown in Fig.1(a,b). All the geometric quantities are normalized with the body height H=0.288m. The values of the geometric quantities are l_r/H=2.928, G/H=0.697 and W/H=1.35, where l_r.is the length from the beginning of the slant to the front, G is the length of the slant region and W is the width of the body. The front part is rounded with a radius of R/H=0.347in the plane, y=0 and z=0. The body is placed in the channel with a cross section of $B\times F$=6.493H$\times$4.861H (width$\times$height). The front face of the body is located at a distance of 7.3H from the channel inlet and the downstream length between the rear face of the body and the channel outlet is 21H. The body is lifted from the floor, producing aground clearance of C/H =0.174 on the four pillar. The slant angle is taken as 25^o. As a result the channel length is about 8.8069 times of the body length and the blockage is about 4.27%.

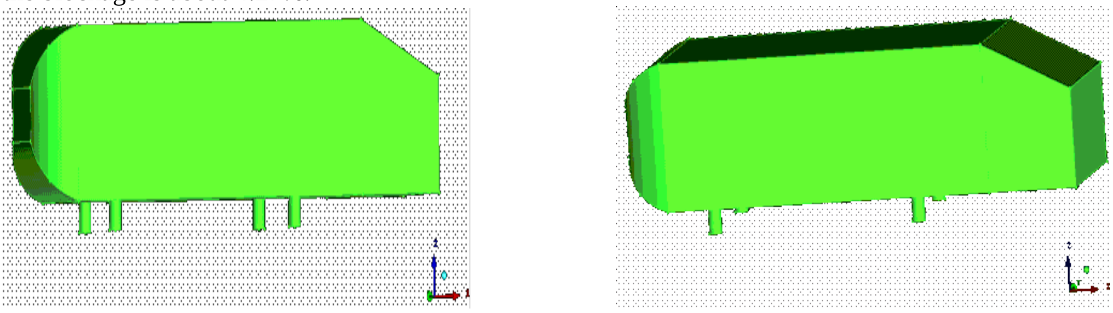

Fig. 1(a, b)- Geometric presentation of the body from different angle.

In the simulations Ansys 11© with high resolution advection scheme along with Physical timescale (physical time = length of the tunnel/velocity of the fluid) is used. The following conditions are used as simulation parameters.

Simulation type :	Stationary	Boundary condition and numerical scheme:	
Domain type	Fluid domain	Inlet:	
Fluid	Air at 25^oC	Flow region	Subsonic
Buoyancy	Buoyant	Turbulence	Low(intensity1%)
Reference pressure	1 [atm]	Temperature	298K
Gravity-	X= 0	Outlet:	
	Y= 0	Flow region	Subsonic
	Z= -9.807m/s^2	Static pressure	0 Pa
Buoyancy reference temperature	298K	Wall:	
Domain motion	Stationary	Wall influence on flow	No slip
Heat transfer option	Isothermal	Wall roughness	Rough wall
Wall function	Scalable	Roughness height	0.0001- 0.0005m
Buoyancy turbulence	Production and dissipation	Fixed temperature	298K

3. Turbulence Modeling

Generating turbulence information by solving the full Navier–Stokes equations remains incomplete at the time. Instead, analysts resort to approximate approaches, called as modeling. Turbulence modeling is based on the assumption that the real flow field may be substituted by an imaginary field of mathematically defined continuous functions. These functions usually represent physical quantities measurable in the flow field. Many turbulence modeling techniques deal with approximation to the Navier-Stokes or Reynolds-Stress equations. Any model, up to some extent can be analytically derived from Reynolds- stress equations. The main goals of turbulence modeling are: develop a set of constitutive relations valid for any general turbulent flow problem; yield sufficiently reliable answers and offer a degree of universality sufficient to justify their usage in comparison to cheaper, less general methods or to more expensive but potentially more reliable methods.
Early work on modeling turbulence was attracted by Newton's law of viscosity. An eddy viscosity a new property of turbulence was introduced and specified for different turbulent flows. Many simple models based on the eddy viscosity concept, particularly Prandtl mixing length models were developed to predict the mean velocity profiles in turbulent flows. These models continue to be in use because of their simplicity and sufficient accuracy in determining global quantities such as boundary-layer thickness, wall shear stress and point of separation. Uses of these models produce analytical solutions for many simple engineering problems. Advanced engineering applications require identification of structures of structures and calculation of statistical parameters, spectral functions, Reynolds stress distribution and turbulence heat and mass flux distributions. The model that must then be selected depends on the level of detail to be captured by the solution.

The k-epsilon model

Jones and Launder [5]developed the k-ε turbulence model in 1972. The k-ε turbulence model predicts the kinematic eddy viscosity value through the modeling of the turbulence velocity and length scale. This is achieved by solving additional transport equation of mean turbulent kinetic energy (velocity scale) and dissipation rate (length scale) and thus it falls in the category of two equation model. Here k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity and has dimension of L^2T^{-2}, and ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate) and has dimensions of k per unit time (L^2T^{-3}). The model has shown its robust and economical properties but has deficiencies to predict highly strained flow, swirling flow, rotating and separating flows.

The RNG k-epsilon model

The RNG k-ε turbulence model provides with accuracy and efficiency in the modeling of turbulent flow. This model follows the two equation turbulence modeling framework and has been derived from the original governing equations for fluid flow using mathematical techniques called Renormalization Group (RNG) method due to Yakhot and Orszag [6]. The RNG model provides a more general and fundamental model and yield improved prediction of near wall flows, separated flows, flows in curved geometries and flows that are strained by effects such as impingement or stagnation. Time dependent flows with large- scale motions, as in turbulent vortex shedding are far better predicted by RNG k-ε turbulence model.

4. Roughness height and Drag coefficient

Roughness height: The roughness height is the height of the surface irregularities for uniform sand grain roughness, or a mean height value for non-uniform sand- grain roughness. All surfaces in technical applications like the surface of a car are rough with a deviation in roughness height. We have it even for very smooth surface. In most CFD program a standard roughness is set. But if we have a much rough surface like sand corn size we have to modify the wall functions for the turbulence model with the right roughness height.
Drag coefficient: The drag experienced by a plate is purely friction drag. This can easily be determined from the equation given below.

$$D = b\int_0^l \tau_w(x)dx \quad (1)$$

From the equation (1) we get the drag of one side of the plate, where b is the width of the plate and l its length. Now the local wall shear stress is

$$\tau_w(x) = \mu\left(\frac{\partial u}{\partial y}\right)_w = \mu U_\infty \sqrt{\frac{U_\infty}{2\nu x}} f_w'' = 0.332\mu U_\infty \sqrt{\frac{U_\infty}{\nu x}} \quad (2)$$

where f_w'' is the characteristic value for the boundary layer on a flat plate at zero incidence. The skin-friction coefficient in the equation

$$C_f(x^*) = \frac{2\tau_w(x^*)}{\rho V^2} \quad (3)$$

With the reference velocity U_∞ then becomes

$$C_f(x) = \frac{2\tau_w(x)}{\rho U_\infty^2} = \frac{0.664}{\sqrt{\mathrm{Re}_x}} \quad (4)$$

Where the following form of the Reynolds number formed with the length x has been used:

$$\mathrm{Re}_x = \frac{U_\infty x}{\nu} \quad (5)$$

Combining equation (1) and (2) we find the drag on one side of the plate;

$$D = f_w'' \mu b U_\infty \sqrt{\frac{U_\infty}{2\nu}} \int_0^l \frac{dx}{\sqrt{x}} = f_w'' b U_\infty \sqrt{2\mu\rho l U_\infty} \quad (6)$$

The drag of the plate is therefore proportional to $U_\infty^{3/2}$ and to $l^{1/2}$, that is not proportional to l. This has to do with the fact that the parts of the plate at the rear contribute a relatively lower amount to the total drag than those at the front. This is because they are located in a region which has a thicker frictional layer and thus a smaller wall shear stress. If we want to estimate the value of drag coefficient in the usual manner, then we use the following equation:

$$C_D = \frac{2D}{\rho U_\infty^2 bl} \qquad (7)$$

where the wetted area *bl* serves as a reference area.

5. Results and discussion

As all the car body dimensions are normalized with the body height H so the roughness height is also normalized by the same. As a result though simulations are done with the roughness heights 0.0001m, 0.0002m etc but plots are not showing that values. Let λ denotes that normalized roughness height.

Fig. 2 and Fig 3 are representing the over all effects of roughness height on the total drag coefficient for the k-ε turbulence model and RNG k-ε turbulence model respectively. From Fig. 2 it is clear that with the increase in the roughness height the total drag primarily decreases and latter increases, which indicates that k-ε turbulence model has dependency on the roughness height. On the other hand, from Fig. 3, the total drag increases rapidly with the increase in roughness height but after certain value though it remains increasing but the rate slows down.

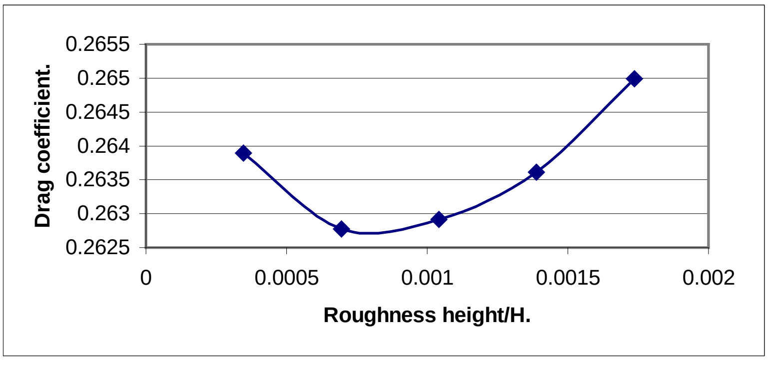

Fig.2. Drag coefficient profiles for different value of λ for k-ε turbulence model

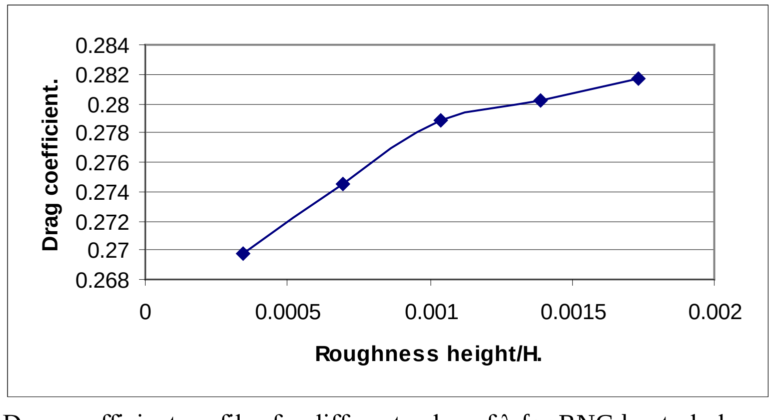

Fig.3. Drag coefficient profiles for different value of λ for RNG k-ε turbulence model

Not only the total drag is the matter of interest but also the drags at the nose, rear slope and at the back are of interest. Thus the drag coefficients at the nose, at the back and at the rear slope along with their sum treated as the total drag coefficients are tabulated for different values of the normalized roughness height in Table 1 and 2 for k-ε and RNG k-ε turbulence model respectively.

Table 1. Drag coefficient values for k-ε model.

Roughness height/Body height	Drag coefficient at the nose	Drag coefficient at the rear slope	Drag coefficient at the back	Total drag coefficient
0.0003472	0.01400	0.13589	0.11400	0.26389
0.0006944	0.01795	0.12892	0.11590	0.26277
0.0010416	0.02058	0.12505	0.11728	0.26291
0.0013888	0.02270	0.12203	0.11888	0.26361
0.0017360	0.02466	0.12012	0.12021	0.26499

It is seen from Table 1 that with the increase in the normalized roughness height the drag coefficient at the nose is continuously increasing. The drag coefficient at the back is also monotonously increases but the drag at the rear slope decreases with the increase in the normalized roughness height. These up and down effects finally have produced primary decrease and then increase effect. From Table it is observed that with the increase in the normalized roughness height the drag coefficient at the nose grows primarily faster but afterward slows down.

Table 2. Drag coefficient values for RNG k-ε model.

Roughness height/Body height	Drag coefficient at the nose	Drag coefficient at the rear slope	Drag coefficient at the back	Total drag coefficient
0.0003472	0.014552	0.115008	0.140176	0.269736
0.0006944	0.017966	0.115004	0.141573	0.274543
0.0010416	0.020612	0.114932	0.143282	0.278826
0.0013888	0.024207	0.113039	0.143021	0.280267
0.0017360	0.024788	0.114038	0.142814	0.281640

The drag coefficient at the back primarily increases but afterward decreases, whereas that at the rear slope primarily decreases and then increases, but the sum, the total drag coefficient increases continuously with the increase in the normalized roughness height.

Finally the minimum obtained coefficient of drags at the rear slant and at the back are tabulated in Table 3 to compare among k-ε and RNG k-ε turbulence models with LSTM benchmark values. It is observed that the simulations produced drags are comparable with the LSTM benchmarks.

Table 3. Comparison between drag coefficients of different models

Model	Drag coefficient at the rear slope	Drag coefficient at the back
LSTM	0.121	0.129
k-ε	0.1289	0.1159
RNG k-ε	0.115008	0.140176

6. Conclusion

In this paper Ahmed car body is used to view the effect of roughness height on k-ε and RNG k-ε turbulence models. On the basis of the observations it may be mentioned that

i) k-ε turbulence model has dependency on the roughness height, so in simulations care must be taken to choose this parameter.

ii) Drag coefficients at different positions do not show the same behaviour.

iii) Both k-ε and RNG k-ε turbulence models have the capability to estimate drags comparable to the benchmark.

Wherever Times is specified, Times Roman or Times New Roman may be used.

7. References

[1] H. Lienhart, C. Stoots, S. Becker, “Flow and turbulence structure on the wake of a simplified car model (Ahmed model)”*, DGLR Fach. Symp. der AG ATAB,Stuttgart University*, 2000.

[2] G. Martinat, R. Bourguet, Y. Hoarau, F. Dehaeze, B. Jorez and M. Braza “ Numerical Simulation of the Flow in the Wake of Ahmed Body Using Detached Eddy Simulation and URANS Modeling”. *Peng S-H and Haase W (Eds.)*: *Adv. in Hybrid RANS-LES Modeling; NNFM 97,* pp. 125-31, 2008.

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

[4] T.J. Craft,S. E.Gant, H. Iacovides and B.E. Launder, “ Computational study of flow around the Ahmed car body”, *9 th ERCOFTAC workshop on refined turbulence modeling, Darmstadt University of Technology, Germany*, 2001.

[5] W.P. Jones and B.E. Launder, “The prediction of laminarization with a two-equation model, *Int. J. Heat and Mass Trans.*,, Vol. 15, pp. 301-314, 1972.

[6] V.Yakhot and S. A. Orazag , “Renormalization Group analysis of Turbulence, I. Basic Theory”, *Journal of Scientific Computing*, Vol. I, pp. 3-51, 1986.