iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 3 **Issue:** V **Month of publication:** May 2015

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Computational Simulation of Ahmed Body with Varying Nose radius, Ground height & Rear Slant angle

Pankaj Madharia[1], Man Mohan Tiwari[2], K .Ravi[3]
[12]*M.Tech Automotive Engineering, SMBS, VIT University, Vellore, INDIA*
[3]*Assistant Professor, Thermal and Automotive Division, SMBS, VIT University, Vellore, INDIA*

Abstract— The CFD analysis is used to analyze the complex flow behavior, flow separation and drag around the body. For this study, a simplified Ahmed body is considered as a test body which is commonly used in industry. Ahmed body consist of round nose part at front , a movable slant plane at rear end and a rectangular box , which connect rear and front part. The analysis is carried out by varying the slant angle at 25^0 and 35^0 angles. Also at each rear slant angle, the two separate cases have been studied by changing the nose radius i.e. R100 mm and R120 mm. Further modification on ground clearance from 50mm to 60mm have been carried out .The results are compared for all the cases in the form of drag coefficient and flow field. Air is assumed as an external medium and the inlet velocity of air is taken as 40m/s. K-ε turbulent model is used in order to account the turbulent flow around the Ahmed body.

I. INTRODUCTION

A. General

When developing a road vehicle it is necessary for the designer to consider the structure of flow behaviour around the vehicle. This flow pattern has great influence on parameter such as the overall shape of vehicle, aerodynamic drag and lift, fuel consumption, comfort level, noise generation and road handling. Normally the flow behavior over the vehicle is characterized by large-scale flow separation, recirculation regions, wake generation and formation of boundary layer on vehicle and ground due to the sharp edges, various corner and the other design aspects of vehicle geometry; due to this different parameters the flow around the body is turbulent. The rear part of the vehicle plays a major role in formation of wake as the flow passes over the roof of the vehicle and reach to rear, there is sudden change in geometry so flow start to separate and wake formation occurs. This wake formation leads to significant reduction in pressure at the rear part compare to front part, where pressure is higher due to stagnation point and this result into drag.

Due to above aspects study of flow is necessary for a vehicle in all perspective; means its aerodynamic design, shape, various attachment (radio antenna, mirror, door handle etc). So it is necessary for a ground vehicle to be a bluff body which should be close to ground proximity. The geometry of the actual vehicle is very complex compare to a bluff body and the flow over the body is fully three-dimensional which is characterized by the turbulent boundary layers, flow separation and large wakes .Whether typical bluff bodies or actual vehicle, the principal contribution to total drag experienced is pressure drag and major aim of vehicle aerodynamic design is to avoid or control the flow separation, so that the drag will reduce and fuel consumption can be decreased but still aerodynamically modified road vehicles have a separation of flow at its rear end.

B. Ahmed body consideration

In order to get optimum design aerodynamics analysis is done to reduce drag coefficient. The different investigation has obtained the flow behavior of newly developed turbulence models for actual complex geometry cases, than a simplified car model, which is known as the Ahmed body, has been tested by Ahmed. The Ahmed body is made up with very simple geometry, consist of a round front part, a moveable slant plane placed at rear part of the body to study the coefficient of drag and flow separation phenomena at different slant angles, and a rectangular box(at middle), which connects the front part and the rear slant plane. This is one of the most suitable and simplest bluff body for analysis which consider the minimum requirement of actual road vehicle. The wheels are aerodynamically unfavorable for vehicle analysis at very high velocity. It was observed by experimental method that the addition of wheels and wheelhouses to vehicle model leads to an increase of around 30% in aerodynamic drag, and 40% in lift. Due to short longitudinal extension of the wheel base a strong impact at the frontal part of the vehicle on the direction of the flow of the wheels occur which result in very small eddy formation. The distance between the wheel axis and front of body is in inverse proportional to the yaw angle of the flow. Thus decrease in this distance result in increase in yaw angle and due to this yaw angle increment the drag value increases. It was found experimentally that the drag coefficient for wheels in case of yaw angle of 15° is three-times the value of drag coefficient at 0° yaw angle, so the wheel and wheel base are not consider in case of Ahmed body simulation and in order to achieve the flow behavior under the body ground clearance is taken. The ground clearance plays an important role in Ahmed body design as under body wake formation also contributes to drag generation.

Ahmed [1] has done study of the wake formation and drag formation when vehicle subjected to different type of flow condition. He found that around rear part of the body there is large pressure drag which contributes 80-85% of total body drag. Lien hart [6] et al. found that critical angle is 30^o for rear slant angle because there is a considerable drop of drag and flow is completely separates at the rear slant part of the vehicle. After consider the Ahmed body as a bluff body for road vehicle simulation, its reduction in total weight and various modification of the external surfaces of the car are done to achieve improvement on the aerodynamics characteristics [8-12].

C. Ahmed turbulence Model

Most of the vehicle designers have gained their understanding of the air flow behavior around a vehicle through extensive wind tunnel testing. But this wind tunnel testing requires most of the resource availability, along the space and cost (for model, wind tunnel equipment) it require time for experiment. Also we cannot get the velocity and pressure value at different part of the body individually. In order to overcome all this point nowadays computational fluid dynamics (CFD) has became essential technology to calculate the quantities such as drag and lift for a ground vehicle without resort wind tunnel testing. However, if the computational models are very large it may take several day of CPU time to get the solution. In order to reduce this time scale, it is necessary to use a simplified computational technique and a model with considering the effect of turbulence. In [3] the k-ε model,k-ε-v^2 model and full Reynolds stress model with wall function have been used to investigate the flow over the Ahmed body. Therefore a powerful numerical tool is require to analyse the flow around vehicle. Computations (CFD) based on Reynolds-Averaged Navier Stokes Equations (RANS) are commonly used in automobile industry which give the accurate result for turbulent flow.

II. AERODYNAMIC ANALYSIS

A. Description

In the present work, the simplified Ahmed body was made through modelling software i.e. PRO-E and then it was imported into ANSYS15.0 WORKBENCH-FLUENT where the aerodynamic analysis was carried out on it. The size of the computation domain, around the body is 10.44m long, 2.5056m wide and 1.566m high. The Ahmed body is 1.044m long, 0.288m high and 0.389m wide, with a rear slant angle of 25°, as shown in figure 1. Generally, the CFD analysis can be classified into three categories namely pre-processing, solving and post-processing. In pre-processing, the geometric model and mesh is generated in ANSYS 15.0 and then boundary condition is applied. During solving, the various governing equations like RANS equation, momentum equation and continuity equation is solved simultaneously whereas in post-processing, the various results like velocity vector, pressure contour, drag etc. are obtained.

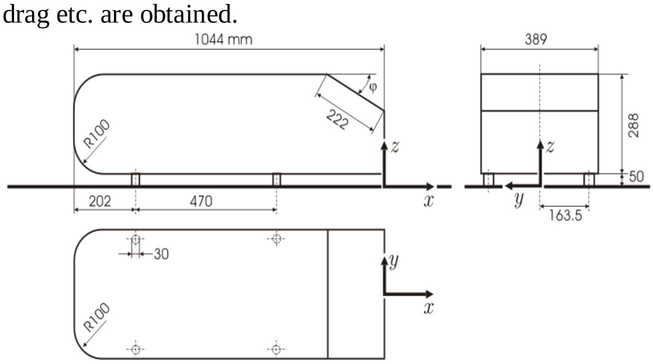


Fig.1 Ahmed Dimensions

Volume 3 Issue V, May 2015
ISSN: 2321-9653

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

B. Problem Definition

In present work, the 3D model of Ahmed body is taken into consideration and the analysis is carried out by varying the slant angle at 25^0 and 35^0. For both rear slant angle the two separate cases have been studied by changing the nose radius i.e. R100 mm and R120 mm. Also the one modification on ground clearance from 50mm to 60mm was done. The different dimension of 3D Ahmed body is used for the analysis are:

1) 25^0 Slant Angle and 100mm Nose radius
2) 25^0 Slant Angle and 120mm Nose radius
3) 35^0 Slant Angle and 100mm Nose radius
4) 35^0 Slant Angle and 120mm Nose radius
5) 25^0 Slant Angle and 50 mm ground clearance.
6) 25^0 Slant Angle and 60mm ground clearance.

C. Governing Equations

The governing equations like momentum and continuity for 3dimensional non-steady flow is considered. The equation are given as:

Conservation of Mass: $\nabla.(\rho v) = 0$

x- Momentum Equation:

$$\nabla.(\rho u \vec{v}) = -\frac{\partial P}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$$

y- Momentum Equation:

$$\nabla.(\rho u \vec{v}) = \frac{\partial P}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho g$$

z- Momentum Equation:

$$\nabla.(\rho u \vec{v}) = \frac{\partial P}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}$$

D. Geometric Modelling

The three dimensional Ahmed body is imported to ANSYS 15.0 WORKBENCH and a control volume is created around it. The various parameters of control volume is given below:

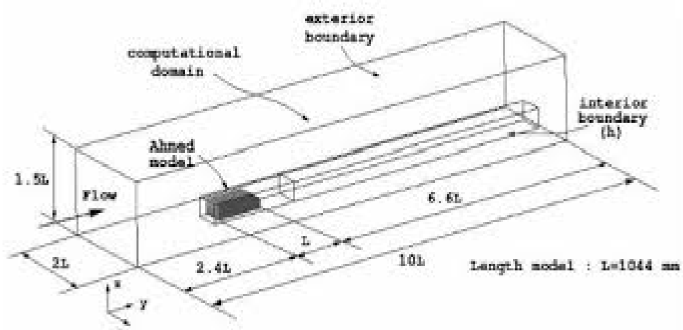


Fig.2 Ahmed body control volume

After create the control volume, the Ahmed body is subtract from control volume by using Boolean operation.

E. Mesh Generation

After physical modelling, the domain is discretized into small control volume by fine meshing of model. The face sizing of leg and Ahmed body is carried out to capture the flow behaviour. In order to study the effect of flow separation and flow reattachment, the inflation layer is generated around bluff body with a growth rate of 1.2. The meshed model of bluff body is

Volume 3 Issue V, May 2015
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

shown in fig.3. After meshing the model, the names are assigned to each part of model. The various parts of model are velocity inlet, pressure outlet, symmetry, road, Ahmed body, side and top.

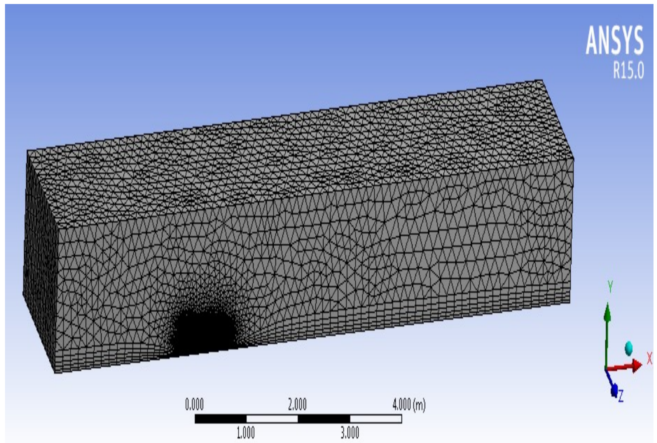


Fig.3 Meshed model of Ahmed body

F. Solver Setup

For the present work, pressure based fluent is used as a solver for solving the 3D, Steady state, incompressible flow using double precision technique. The meshed model is imported into setup and then the quality of mesh is checked. It is important to note that the volume of meshed model should always come positive. The realizable K-epsilon model is used to study the turbulent flow behaviour around the body. Also a standard wall function is adopted for near wall treatment.

The steps followed in setup are:

1) Specifying properties:

Fluid: Air

Density (kg/m^3) = 1.225

Viscosity (μ) = 1.7894e-05

Thermal conductivity (w/mk) = 0.0242

C_p (j/kg k) = 1006.43

2) *Boundary Conditions:* The boundary condition has been provided at inlet, outlet and wall condition. At inlet, the velocity magnitude of flow is given as 40m/sec in x- direction along with the turbulent intensity and viscosity ratio is provided as 0.2% and 10 respectively. The wall condition is assumed as no slip and adiabatic. The pressure outlet is given as zero gauge pressure whereas backflow turbulent intensity and viscosity ratios are taken as 5% and 10 respectively.

3) Solution Methods:

Scheme- coupled

Gradient-Least square cell based

Pressure-Second order

Momentum-Second order upwind

Turbulent kinetic energy-Power Law

Turbulent dissipation rate-Power Law

*4) Setting up Solution Controls and under relaxation factors***:**

All flow, turbulent and energy equations are used for the solution.

Flow Courant number= 200

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Under Relaxation Factors:
(a)Turbulent viscosity=1
(b)Density= 1.0
(c)Body Force= 1.0
(d)Turbulent Dissipation rate= 0.8
(e)Turbulence Kinetic energy= 0.7

5) *Initialize Solution:* Solution is initialized from inlet region with the previously set initial values.

6) *Run calculation:* Number of iterations is set 500 with a reporting interval of 1.

III. RESULTS AND DISCUSSION

A. *Validation of simulation results*

In order to validate the model, the simulation results are compared with the experimental results [6] in terms of drag coefficient at corresponding rear slant angle of 25^0 and 35^0 (when nose radius is 100 mm)with speed of 40 m/s .Also the velocity contour are validated by compare the velocity profile from[7].

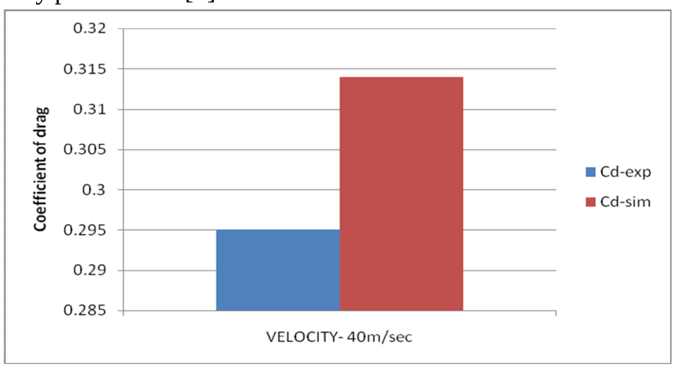


Fig.4 Velocity vs C_d graph for 25^0 rear slant angle

From the above bar chart, it can be observed that the experimental drag coefficient for 25 slant angle is 0.295[6] whereas for simulation results is 0.314.Quantitatively, the difference between the experimental and simulation result is around 6.44% for rear slant angle 25^0 which shows a good agreement with experimental data. This difference may be caused due to the major simplification made in computational models such as leakage flows are not taken into consideration. Also, the exact simulation of environmental condition is bit difficult during computational flow analysis.

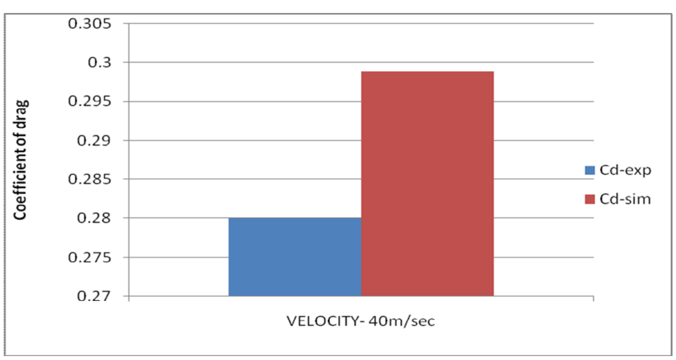


Fig.5 Velocity vs C_d graph for 35^0 rear slant angle

Volume 3 Issue V, May 2015
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

The above bar chart compares the drag coefficient at rear slant angle of 35^0 for experimental and simulation. The percentage error between simulation and experimental data is around 6.75%, and the reason for this deviation is same as mentioned above.

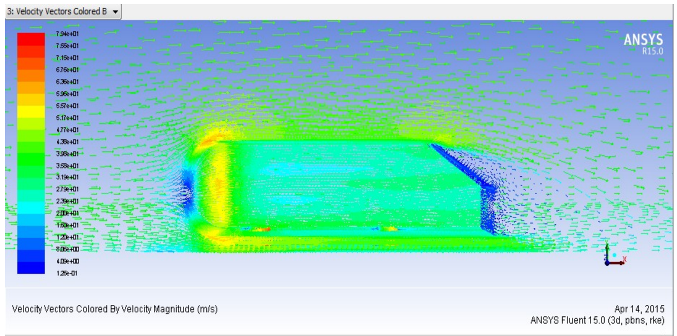


Fig.5 Velocity vector for 25^0 rear slant angle

The above fig shows the velocity vector obtained for rear slant angle at 25^0. For validation, it is compared with the velocity contour of [7]. It can be observed from fig that velocity is nearly zero at the front nose of Ahmed body thus pressure is maximum at front part which signifies the stagnation point. This velocity variation is due to sudden impact of flow at front part which results in wake formation. Similarly velocity is minimum at the rear slant part; this is because separation of flow and boundary layer formation which results in formation of vortices.

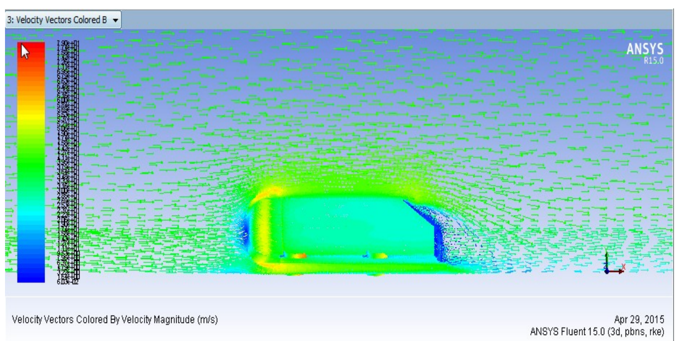


Fig.6 Velocity Contour for 35^0 rear slant angle

After simulation for 35^0 rear slants angle using k-ε model, the velocity vector which we get over the body is shown in fig.6. From fig it can be observed that flow is fully separated over the rear slant and strong vortices do not form and this result in small intensity wake formation.

B. Simulation of Ahmed body for Nose Radius of 100mm and 120mm at 25^0 rear slant angle

For having a better understanding of drag coefficient on Ahmed body, the body with different front nose radius at 100mm, 120mm was designed and analyzed by keeping all other parameter such as ground clearance, length, velocity same; at 25^0 slant angle. Development of model, grid generation, boundary conditions and solver setup are carried out in the same way as explained before. Simulated results obtained after successful simulation of Ahmed body for different nose radius at 25^0 slant angle, suggest the optimum shape and better working conditions for low coefficient of drag.

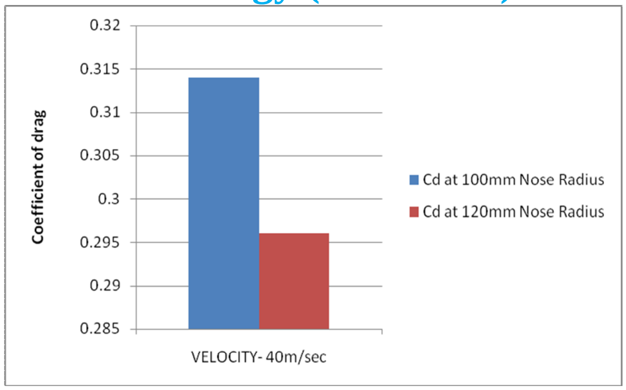


Fig.8 Velocity vs C_d graph at 25^0 slant angle with various nose radius

Figure 8 shows, the velocity vs drag coefficient graph for different nose radius at 25^0 slant angle. It can be observed from the graph that drag in case of 100mm nose radius is more compare to 120mm.It was observed that with the increase in nose radius from 100mm to 120mm, there was a reduction in coefficient of drag by 6.08% which ultimately reduces fuel consumption as well as noise. When nose radius is low at the front part then it acts like a sharp edge so the flow at the front start separate from this edge, which results in increase in drag; however with increase in nose radius, it forms a edge with fillet like section so the flows were not separated suddenly and thus low vortices formed.

C. Simulation of Ahmed body for Nose Radius of 100mm and 120mm at 35^0 rear slant angle

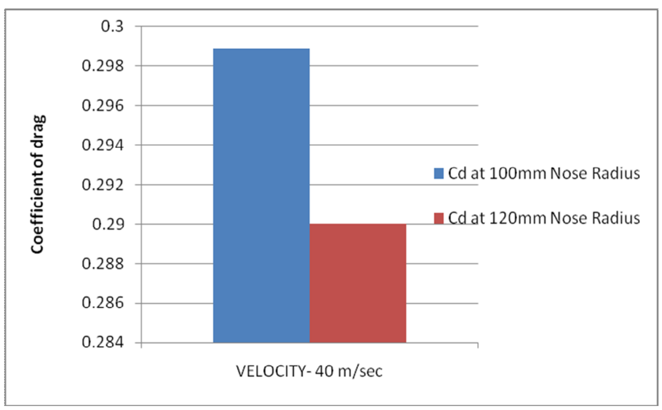


Fig.9 Velocity vs C_d graph at 35^0 slant angle with various nose radius

Figure 9 shows, the coefficient of drag value with nose radius of 100mm and 120mm at 35^0 rear slant angle with velocity of 40 m/sec. It can be seen from the figure that variation in drag for different nose radius is same as that of variation for 25^0 slant angle. It has been found that that coefficient of drag decreases by 2.97% when nose radius increases from 100mm to 120mm for 35^0 slant angle. The reason is same as mentioned above.

D. Simulation of Ahmed body for Ground Clearance of 50mm and 60mm for 25^0

In order to achieve a better body design in the prospect of ground clearance the simulation was done by changing the ground clearance from 50mm to 60mm.Figure 10 is showing simulation result for Ahmed body i.e. variation of coefficient of drag with two ground clearance .

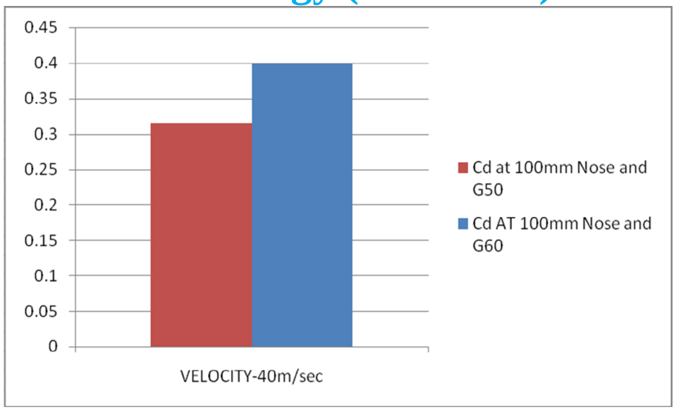


Fig.10 Velocity vs Cd for G50 and G60 at 25^0 slant angle.

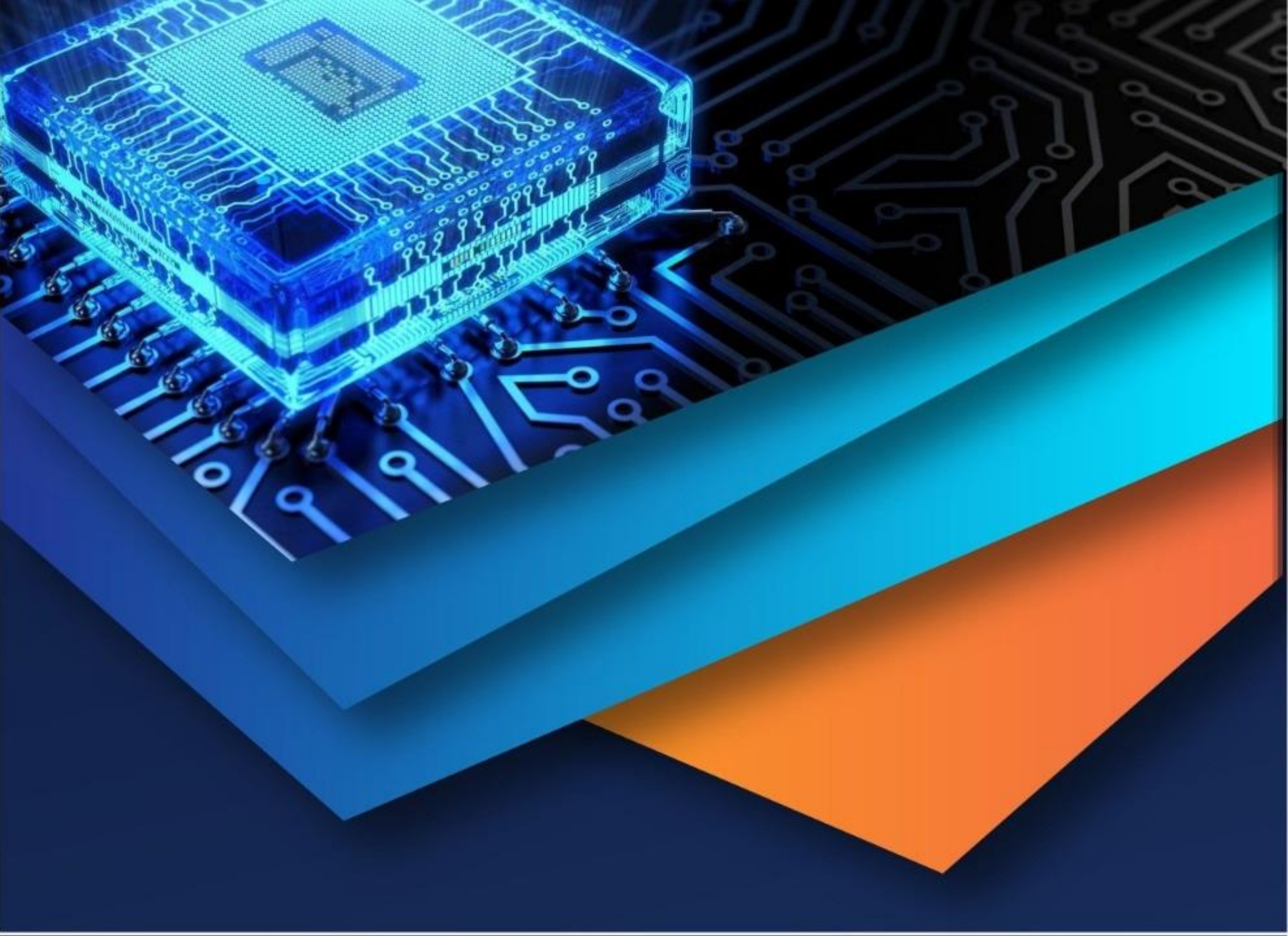
It can be observed from the above bar chart that with the increase in ground clearance from 50mm to 60mm, drag of Ahmed body increases by 26.75% which results in increased fuel consumption, noises and loss of vehicle handling.

IV. CONCLUSION

In the present work, the simulation of Ahmed body is carried out for different slant angle and nose radius along with variation in ground clearance. It has been found that there is reduction in drag as the nose radius increases from base value for both rear slant angle. Also, it has been found that there is large wake formation of low intensity with the increase in slant angle, which results in the dissipation of energy. Increasing the ground clearance has a counter effect on coefficient of drag. Compared to k-ε model, a more accurate model can be used to get a better insight into flow separation and flow reattachment.

REFERENCES

[1] S.R. Ahmed and G. Ramm, "Some Salient Features of the Time Averaged Ground Vehicle Wake", SAE Technical Paper, 840300, 1984.
[2] T.J. Craft, S.E. Gant, H. Iacovides, B.E. Launder and C.M.E. Robinson, "Computational Study of Flow Around the 'Ahmed' Car Body", 9th ERCOFTAC/ IAHR Workshop on Refined Turbulence Modelling, 2001.
[3] Yunlong Liu, Alfred Moser, "Numerical modeling of airflow over the Ahmed body", Swiss Federal Institute of Technology, ETH-Zentrum WET A1, CH-8092 Zurich, Switzerland.
[4] Hucho W.H., Aerodynamics of Road Vehicles, SAE International, Warrendale, PA (1998).
[5] Ahmed, S.R., " Influence of base slant on the wake structure and drag of road vehicles", Transactions of the 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] W. Meile, G. Brenn, A. Reppenhagen, B. Lechner and A. Fuchs ,"Experiments and numerical simulations on the aerodynamics of the Ahmed body," ISSR, vol. 3(1), pp. 32-39, 2011.
[8] S. Yongling, W. Guangqiang and Xieshuo, "Numerical simulation of the external flow field around a bluff car," Shanghai Tongji University Shanghai, China.
[9] C. Rajsinh B. and T. K. Raj R. "Numerical investigation of external flow around the Ahmed reference body using computational fluid dynamics," Research Journal of Recent Sciences, vol. 1(9), pp. 1-5, 2012.
[10] D. Y Dhande, "Experimental analysis of aerodynamic aspects of sport utility vehicle," IJEST, vol. 5(7), pp. 1476-1480, 2013.
[11] M. Desai, S. A. Channiwala and H. J. Nagarsheth, "A Comparative assessment of two experimental methods for aerodynamics performance evaluation of car," Journal of scientific and industrial Research, vol. 67, pp. 518-522, 2008.
[12] A. I. Heft, T. Indinger and N. A. Adams, "Introduction of a new realistic generic car model for aerodynamic investigations," SAE International, 2012.



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY