Computational Simulation of Cross-Wind Sensitivity of an External Body

Shishir Sirohi[1], Yagnavalkya Mukkamala[2]
Professor, Department of Thermal and Automotive Engg , School of Mechanical and Building Sciences ,Vit University,Vellore.[2]
U.G Student, School of Mechanical and Building Science, VIT University, Vellore, India1

Abstract: *The main objective of this paper is to elucidate the variation of various aerodynamic factors (drag lift) and also examine pressure variations when a body corners. This is accomplished by investigating the variation of the rear slant angle of the Ahmed body and its effect on the drag and lift coefficients through numerical simulation .To study variations in a cornering or an overtaking scenario, yaw angle is introduced in the body. The simplified vehicle geometry: Ahmed Reference Model has been used as a benchmark. Three types of slant angle categories have been chosen for the present study notch back, hatch back, square back and a comparison study has been carried out. The computational part consists of numerical simulation of the flow around the Ahmed Body employing CFD (Computational Fluid Dynamics) techniques. In numerical simulation results, pressure-based solver was utilized and the turbulence model employed was k-Epsilon realizable model with non-equilibrium wall function for near wall treatment. The designing of model used in this paper is done through the SOLIDWORKS '14; and CFD simulation carried out in FLUENT (ANSYS 15).*

Keywords*:* *Cross-wind Simulation, Ahmed body, k-Epsilon realizable model, Drag, Lift, Pressure, Cl, Cd.*

I. INTRODUCTION.

The wake region of a bluff body plays an important role in aerodynamic properties of the body.A vehicle is generally considered as a bluff body and wind tunnel testing and CFD analysis are carried out on them. From the advent of aerodynamics, excessive research has been done in the field of steady flow over a bluff body and cross wind analysis is one of the most important sub-division of vehicle simulation.

Aerodynamic analysis of vehicles in the cornering condition is an important design parameter. Despite the cornering condition being important, aerodynamicists are restricted in their ability to replicate the condition experimentally. Whirling arms, rotary rigs, curved test sections and bent wind tunnel models are experimental techniques capable of replicating some aspects of the cornering condition, but are all compromised solutions.

Numerical simulation is not limited in the same way and permits investigation into the condition. However, cornering introduces significant change to the flow field and this must be accommodated for in several ways. Boundary conditions are required to be adapted to allow for the curved flow occurring within a non-inertial reference frame. In addition, drag begins to act in a curved path and variation in Re occurs within the domain. Results highlight the importance of using correct analysis techniques when evaluating aerodynamic performance for cornering vehicles.

Ahmed body has been chosen as the benchmark for carrying out computations for studying the aerodynamic parameters. The body was first proposed by Ahmed *et al.* (1984). **[**1**]**The Ahmed body is a very simple bluff body which has its shape simple enough to allow for accurate flow simulation but retains some important practical features relevant to automobile bodies. It has a slant on its rear end, whose angle can be varied and the corresponding coefficients of lift and drag can be computed. This is done to exhibit the air flow over the different geometry sections of an automobile and in its vicinity, at different slant angles.To study cross wind yaw angle has been introduced in the body.Most of the cross wind studies to date have been performed experimentally and numerically for yawing motions or variation of yaw angles[3],[4]conducted simulations to study transient aerodynamic loading. The transient response of a vehicle to a crosswind is of utmost importance to car drivers since low level inputs can result in poor vehicle refinement, and extreme effects can result in path deviation while cornering. The purpose of this study is to simulate the time-development of the flow around a generic vehicle subjected to a sudden strong crosswind. Another objective of the work presented is to understand some physical processes involved in the aerodynamics of bluff body. Correspondingly [5] carried out simulations to examine variations in vehicle aerodynamics due to aerodynamics effect while cornering.

II.COMPUTATIONAL METHODOLOGY

The simulations carried out in the research are done using ANSYS 15.0-Workbench and modelling of prototypes are done using SolidWorks 14.

Ahmed body:

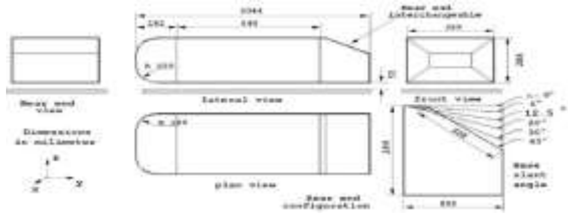


FIGURE 1:Dimensions of Ahmed Body.

Different slant angle have been used in the research varying from 12.5° to 35°.The slant angles have been categorised in three categories

1) Notchback or Fastback (≤ 20°)
2) Hatchback (□ 20° & ≤30°)
3) Squareback (□ 30°)

A. *Enclosure Domain*:

The reference Ahmed Body is 1044mm long, 327mm wide and 288mm high. In ANSYS15 Design Modeller, a single body domain of air is created surrounding the Ahmed body walls after subtracting it from the air enclosure. It has dimensions 1m from front, 3m from the rear and 1m from the top and 1.5m on either side.

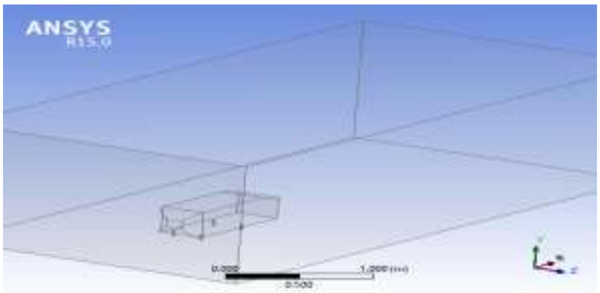
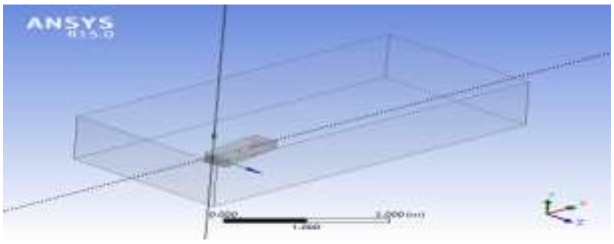


Figure 2: Air Domain.

With Yaw:

For carrying out crosswind analysis a yaw angle of 10° was introduced in the geometry and the enclosure was created. With dimensions,1m from front, 3m from the rear and 1m from the top and 1.5m on either side.

Figure 3:Air Domain demonstrating Yaw.



B. *Meshing*:

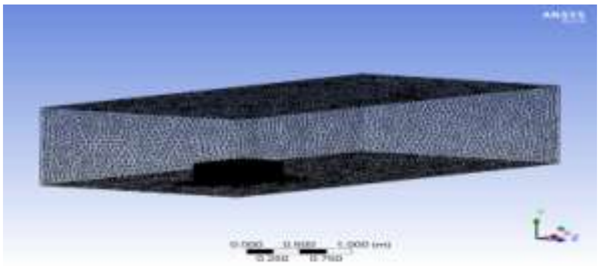
The "finite element method" technique is used in our problem. The ANSYS Meshing Tool is used for carrying out the meshing. Details of meshing :

Relevance centre: Fine, Smoothing: high, Transition: slow, Initial size seed: Active assembly, Min. size: 1mm, Max. Size: 250mm, Advanced Size function: Proximity and Curvature.\

A Sizing feature was introduced in the meshing feature to constrict the size of elements to 10mm around the body. As a result the number of nodes and elements generated are-

140568 , 756121.

Figure 4:Mesh.



C. *Simulation Conditions:*

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 has their own advantage and some disadvantage that's why other investigators still try to find which model is most suitable for the finding the actual flow behavior around the Ahmed body. In the present study Realizable k- e 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 □ 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.

Realizable k-epsilon model with non-equilibrium wall function for near wall treatment is used with Inlet velocity V = 40m/s and turbulence intensity is about 1% and turbulence viscosity ratio is 10 at inlet, turbulence intensity is about 5% and turbulence viscosity ratio is 10 at outlet. Density of air is 1.225 kg/m3,temperature is 288.16 K and viscosity is 1.7894e-05 kg/m4. Boundary conditions are: Uniform velocity at inlet, uniform pressure at outlet, wall condition lateral and at top wall of the model and stationary wall at floor. Solution method utilizes pressure-velocity coupling scheme as coupled with gradient: least square cell based method, pressure as standard and setting the momentum, turbulence kinetic energy, turbulence dissipation rate as first order upwind for the initial 100 iterations and correspondingly as second order upwind for the next 500 iterations. Moreover, turbulence viscosity factor is taken as 0.8 for the initial 100 iterations and consequently taken as 0.95 for the remaining 500 iterations.In certain the convergence occurs before 500 iterations and a convergence scale of 0.0001 is used.

III. RESULTS AND DISCUSSION.

The drag ,lift and pressure coffecients are calculated for the above 6 models.The variaton being in slant angle and yaw angle.Each body is tested with yaw and also for without yaw angle to get a comparison between the changes of the coefficients when a body tend to corner in real-life scenario.

Flow field around a bluff body is characterized by several flow separation regions.The flow separates at the front hood, fenders, and the front head lamps, windshield wipers etc. However, the separated flow re-attaches itself at the rear of the car. These dead water regions are essentially quasi 2-D, and don't possess much re-circulation or energy. Flow separation also occurs at the rear of the vehicle due to a slanting back.These vortices are also of the first type, "quasi 2-D".They exhibit themselves as "contra-rotating" vortices in the tail wake, and don't possess much energy.The shape and size of these vortices depends on the specific rear end design.The second type of flow separation results in a 3-D vortex.These vortices have considerable re-circulation, and energy.They originate from the sides of the vehicle, at the "A"
and "C" pillar regions, and propagate downstream.

When subjected to yaw motion there is disturbance in the flow field around the body especially from "A" pillar region which disturbs the entire flow around the body which gives variations in aerodynamic charateristics.

Categories	Slant Angle	Cd	Cl
Notchback	12.5°	2.9528e-01	7.4175e-02
Hatchback	25°	3.1439e-01	3.0881e-01
Squareback	35°	3.4987e-01	2.1609e-01

Table 1: Simulation Results of Body without Yaw Angle.

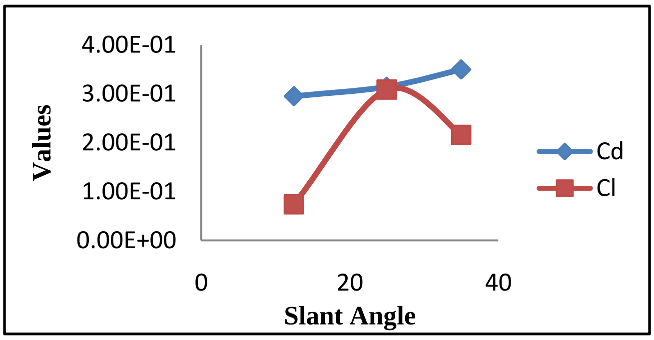


Figure 5 : Graph between drag & lift coefficients and different rear slant angles.

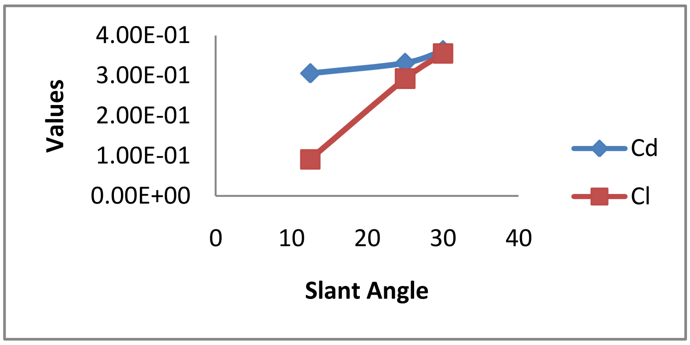
It is observed from **Table 1** that as the slant angle increases the coefficient of drag increases due to increase in vortex shedding in the wake region as the strength of the trailing vortices increses with increase in slant angle and caused due to boundry layer sepration around the C pillar.

A significant increase in lift is also obseverd and a drop is noticed after the critical angle of 30° where maximun vortex shedding occurs. Moreover, the lift coefficient linearly increases as the rear slant angle is increased from 12.5° up till 30°[9].

Table 2:Simulation Results of Body with 10° Yaw Angle.

Categories	Slant Angle	Cd	Cl
Notchback	12.5°	3.0531e-01	9.1622e-02
Hatchback	25°	3.3124e-01	2.9241e-01
Squareback	35°	3.6308e-01	3.5451e-01

Figure 6 : **Graph between drag & lift coefficients and different rear slant angles and yaw angle.**

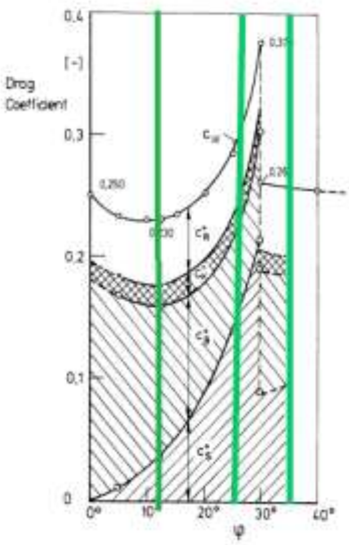


From **Table 2** it is revealed that when a body was subjected to yaw angle a higher value of drag coefficient are encountered as predicted.This is due sepration of flow at "A" plillars which disturbs the flow around the body and hence increasing drag and corresponding increase in lift coefficient which in turns also increases the induced drag.

A. Validation:

As per results of the 1st Simulation on an ahmed model of slant angle 25° the Cd values without yaw angle corresponds to be 0.29528 which are in corroboration with the findings of (Ahmed, 1984) of value 0.295 [1]. The following graph is a schematic from the (Ahmed, 1984).The graph depicting the similar values, therefore the present research work has been carried out.

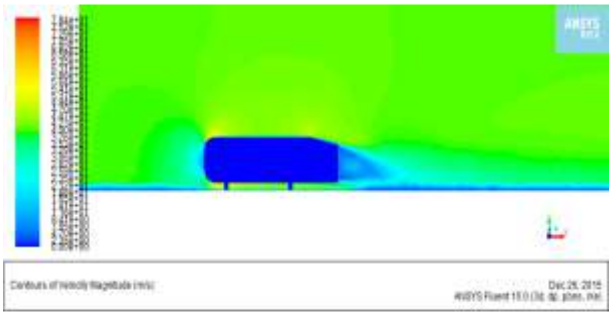
Figure 6: Characteristic drag coefficients for the Ahmed body for various slant angles.



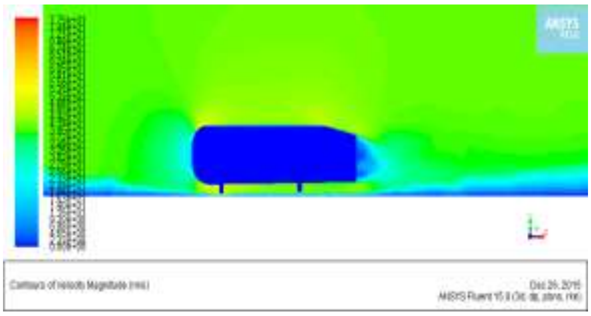
B. Velocity Countours:

1) Slant Angle 12.5°

WithoutYaw

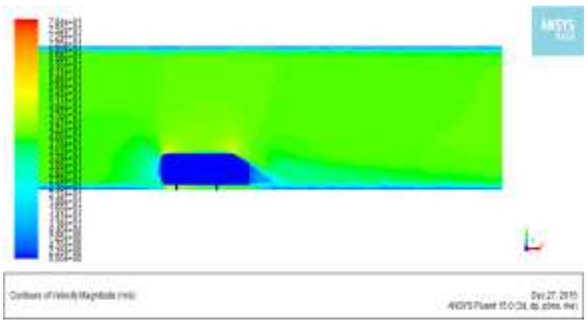


With Yaw

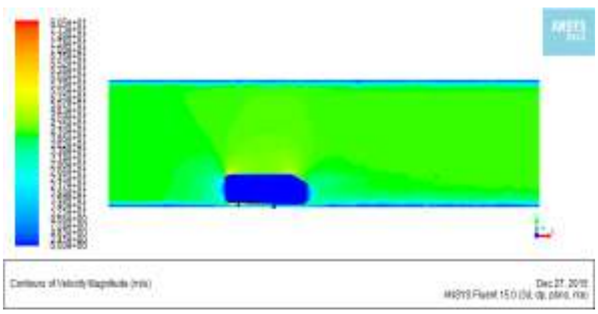


2) Slant Angle 25°

Without Yaw

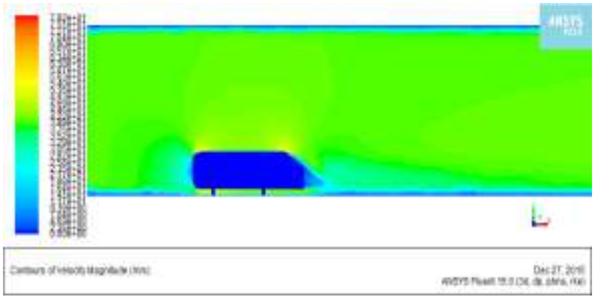


With Yaw

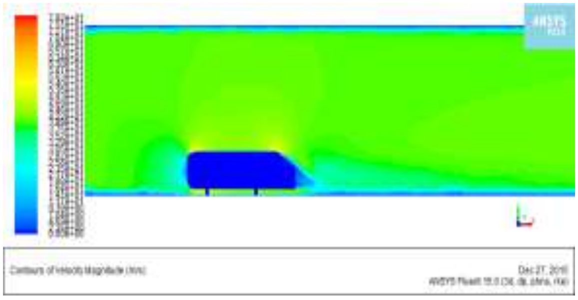


3) Slant Angle 35°

Without Yaw



With Yaw

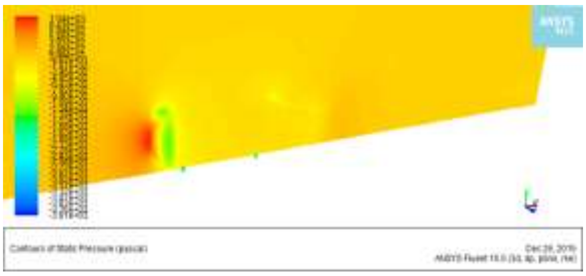


Gives us the velocity of air along different geometry sections of the Ahmed body. The green area corresponds to the velocity 40m/sec, while blue area show low velocity. The velocity contours clearly indicate that the wake region, and consequently the form drag, continually decreases as the rear slant angle is increased from 0-30°and thereafter, boundary layer separation occurs on the rear slant. However, there is an increase in lift with increase in rear slant angle due to larger pressure difference generated between top and bottom surfaces of the vehicle, and this results in larger lift induced drag.. As the wake area is reduced, the drag-causing eddies (vortices), generated due to the pressure difference between the top and bottom of the Ahmed body at the rear trailing end, come closer together. This results in less kinetic energy being dissipated due to these smaller eddies and less drag. The eddies in the yaw simulations are distorted in nature due higher vortices shedding and resulting in higher drag values.

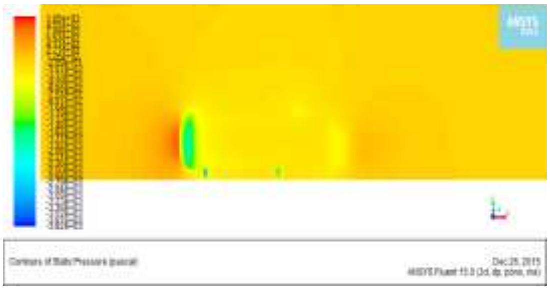
C. Pressure Distibution

1) Slant Angle 12.5°

Without Yaw

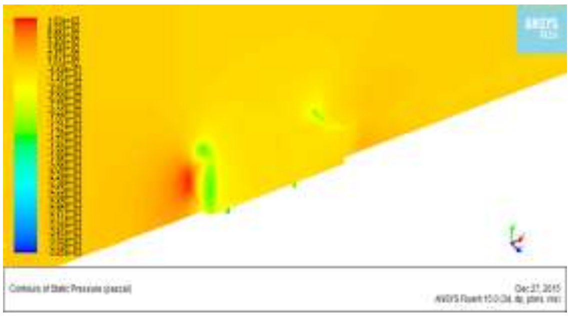


With yaw

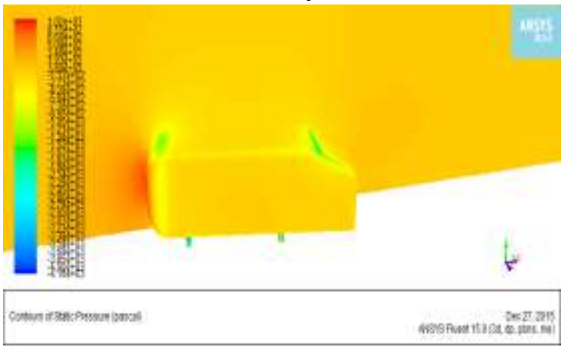


2) Slant angle 25°

Without Yaw

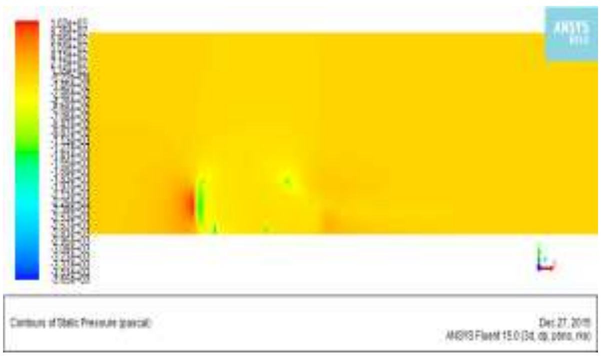


With yaw

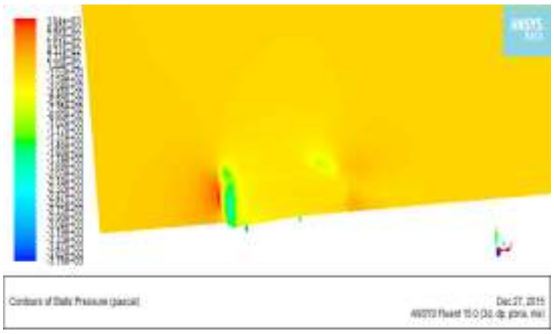


3) Slant Angle 35°

Without Yaw



With Yaw

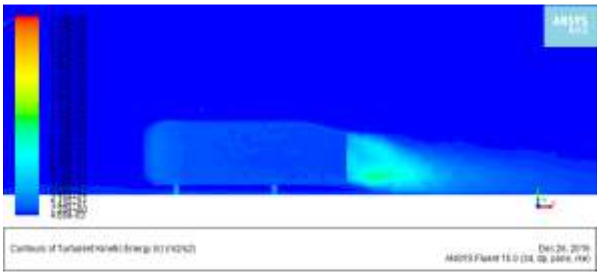


The air becomes almost stagnant as it strikes the vehicle which results in air exerting very high pressure on front engine grill of the vehicle represented by the red area. The airflow then gets divided between the upper and lower surface of the vehicle. The higher pressure air on front surface accelerates as it travels over the curved nose surface of Ahmed body, causing the pressure to drop. This lower pressure creates lift over the roof surface as the air passes over it. As the air continues to flow and make its way to the rear, a notch is created by the rear slant owing to flow separation, leaving a vacuum or low pressure space which is not filled up properly by theair. The resulting pressure difference generates drag .

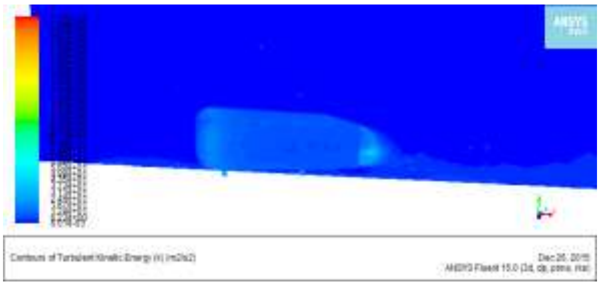
D. Turbulence Contours:

1) **Slant *Angle 12.5°***

Without Yaw

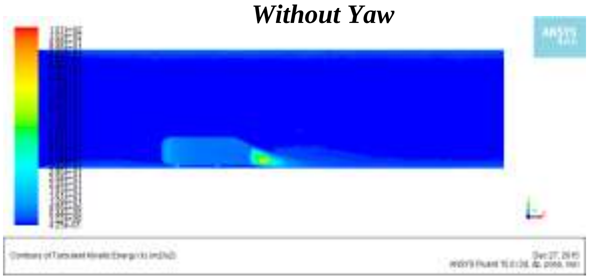


With Yaw

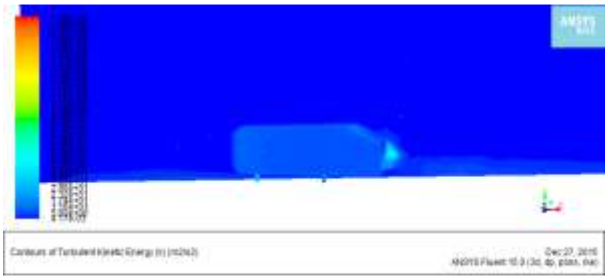


2) Slant angle 25°

Without Yaw

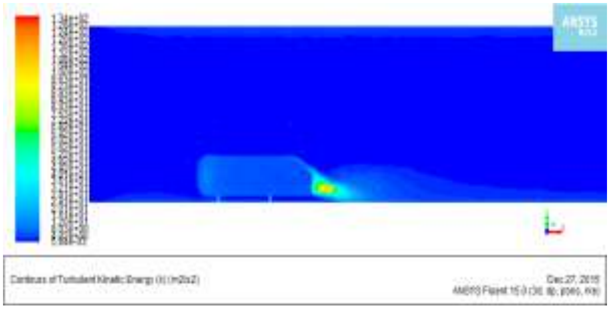


With Yaw

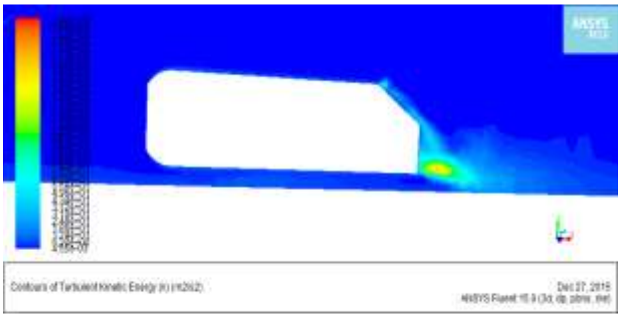


3) Slant Angle 35°

Without Yaw



With Yaw



The turbulence contours showcase the rear wake region and turbulence created in this region as vortex shedding occurs and displays the changes as the boundary layer tends to re-attach after dissipating all the energy.

IV. CONCLUSION:

CFD results for multifarious manipulations of the rear slant angle of the Ahmed Body are presented after the geometries are designed and then developed. Comparison and examination by plotting a graph of the lift and drag coefficients obtained for 3 types of body.As predicted that yaw will result in a higher drag the simulation clearly validate it Further study can be carried out by varying the turbulent models, Detached eddy simulation (DES) models for larger yaw angles between 15°-20° should be utilised to obtain proper results for larger wakes.

REFERENCES:

1.Ahmed, S.R., Ramm, G., Faitin, G., "Some Salient Features of the Time – Averaged Ground Vehicle Wake", Society of Automotive Engineers, Inc., Warrendale, PA. (SAE-TP-840300) 1984.

2.Garry, K.P., and Cooper, K.R., "Comparison of Quasi-Static and Dynamic Wind Tunnel Measurements on Simplified Tractor-Trailer Models," *Journal of Wind Engineering and Industrial Aerodynamics*22:185-194, 1986.

3.Passmore, M. and Mansor, S., "The Measurement of Transient Aerodynamics Using an Oscillating Model Facility," SAE Technical Paper 2006-01-0338, 2006, doi:10.4271/2006-01-0338.

4.Mansor, S., and Passmore, M.A., "Estimation of Bluff Body Transient Aerodynamics Using an Oscillating Model Rig," *Journal of WindEngineering and Industrial Aerodynamics* 96:1218-1231, 2008

5.Guilmineau, E. and Chometon, F., "Numerical and Experimental Analysis of Unsteady Separated Flow behind an Oscillating Car Model," *SAE Int. J. Passeng. Cars - Mech. Syst.*1(1):646-657, 2009, doi: 10.4271/2008-01-0738.

6.Bearman, P.W., and Mullarkey, S.P., "Aerodynamic Forces on Road Vehicles Due to Steady Side Winds and Gusts," Royal Aeronautical Society Vehicle Aerodynamics Conference, Loughborough, UK, 1994.

7. Passmore, M.A., Richardson, S., and Imam, A., "An Experimental Study of Unsteady Vehicle Aerodynamics," *Proceedings of the*338 *Kawakami et al / SAE Int. J. Passeng. Cars - Mech. Syst. / Volume 5, Issue 1(May 2012)*

8.Mankowski, O., Sims-Williams, D., Dominy, R., Duncan, B. et al., "The Bandwidth of Transient Yaw Effects on Vehicle Aerodynamics," *SAEInt. J. of Passeng. Cars - Mech. Syst.*4(1):131-142, 2011, doi: 10.4271/2011-01-0160.

9.Saurabh Banga, Md. Zunaid, Naushad Ahmad Ansari , Sagar Sharma , Rohit Singh Dungriyal "CFD Simulation of Flow around External Vehicle: Ahmed Body" IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE),Volume 12, Issue 4 Ver. III (Jul. - Aug. 2015).

10."*Fundamentals of Aerodynamics*",5th edition, John D. Anderson, McGraw-Hill Book Co.