International Journal of Engineering, Science and Mathematics
Vol.6Issue 8, December2017,
ISSN: 2320-0294 Impact Factor: 6.765
Journal Homepage: http://www.ijmra.us, Email: editorijmie@gmail.com Double-Blind Peer Reviewed Refereed Open Access International Journal - Included in the International Serial Directories Indexed & Listed at: Ulrich's Periodicals Directory ©, U.S.A., Open J-Gage as well as in Cabell's Directories of Publishing Opportunities, U.S.A

Simulation ofAerodynamic Flow Parametersovera Simplified Sedan Car

Sujit Mishra1*
Ashok Misra2**
P.S.V.Ramana Rao3***
D.Nageswar Rao4****

Abstract

Huge demand of sedan cars due to the economic developments in the society leads to increasing competitions in automobile sectors which undergo different testings to enhance the fuel efficiency and performance of these cars where vehicle aerodynamics plays avital role. Aerodynamics affects the performance of sedan cars due to change in parameters such as lift and drag forces at high speed. With the improvement in computer technology, manufacturers are looking at computational fluid dynamics(CFD) modelling of sedan carsinstead of wind tunnel testing to reduce the testing time as well as the research & development cost.In the present study, a simulation is done with ANSYS v15 taking Second Order Upwind Schemeto obtain the results of different flow parameters viz. drag force, drag coefficient, turbulent kinetic energy and wake flow structures over a benchmark test model- 3D Ahmed body which is a simplified sedan car.It is observed thatthe results of the present simulation with regard to drag coefficient are found to be in close agreement with the existing wind tunnel experimental results.This scheme can further be used to optimize the shape of the mid-range sedan carswith safety handling capabilities even at higher speeds and to enhance their fuel efficiency.

Keywords:

CFD modelling,
Wind tunnel,
Aerodynamics,
Ahmed body,
Sedan car

Author correspondence:

Ashok Misra
Center for Fluid Dynamics Research
Department of Mathematics
Centurion University of Technology & Management, Paralakhemundi

1. Introduction

Aerodynamics is a branch of fluid dynamics concerned with the study of motion of air, when it interacts with a moving object. In the recent years, it plays a vital role in the field of automobiles. Development of automobile aerodynamics started in the early stages in 90's with different phases of shape optimization which leads to the cars from small range to luxury classes. From this wide range of cars, the sedan segment is found to be the most fiscal for the mid-range peoplenot only in aesthetics and safety comforts but also for better

*Ph.D scholar, Deptt.of Mechanical Engg.,Centurion University of Technology & Management, Paralakhemundi.
**Professor,Deptt.of Mathematics, Centurion University of Technology & Management,Paralakhemundi.
***Professor, Deptt.of Mechanical Engg.,Centurion University of Technology & Management, Andhra Pradesh.
****Advisor,Deptt.of Mechanical Engg., Centurion University of Technology & Management.

fuel efficiency. Increased fuel prices and environmental issues are the great concerns of automobile companies achieving improved engine efficiency and aerodynamic drag reduction. It could be achieved either changing theengine functioning or supplementing presently used fuel by eco-friendly fuels or changing the current automobile design. As far as engine optimization is concerned we have all most reached at a saturation level. Eco-friendly fuels are an area still under development and it will take a few more years to be adopted worldwide. Hence the easiest way to increase sedan vehicle efficiency is reducing aerodynamicsdrag.

Studies have been carried out in this field formulating the techniques of flow phenomena over the different sedan shapes reducing aerodynamic drag & fuel efficient. Generally car models performed experiments through wind tunnels as well as by numerical simulations. As the air flows over the body, various discrepancies occur as we move from front to rear end. Ahmed [1] has purposed a simplified model to visualize the impact of time-average wake structures over the geometry with different configurations at the rear end side.Hucho et.al.[2]has presented the critical geometry of a streamlined car body shape governing the aerodynamic drag & lift characteristics with different alterations of rear end.Ahmed, Han, Khan, et.al [3-5]have performed a series of wind-tunnel experiments in order to examine thepressure& wake structures predicting difference in middle and rear part of the vehicle. With the growth& use of CFD packages Bijlani [6] has reviewed and investigated on different car models comparing the aerodynamics forces acting upon them with their effect on fuel consumption and stability of vehicles. Argyropouloset.al.[7] has reviewed and optimized various complex geometries of different objects with the help of Turbulence modelling using Reynolds-Averaged Navier–Stokes (RANS), Very Large Eddy Simulation (VLES), Unsteady Reynolds-Averaged Navier–Stokes (URANS), Detached Eddy Simulation (DES avoiding the test prototypes.Umesh[8] has presented Aerodynamic flow patterns of sedanand hatchback models with same frontal area using3D CATIA V5 software, ANSYS FLUENT and observed that Sedan car is more streamline than the Hatch Back car. Murtaz [9] has investigated that use of active and passive components over the real sedan car makes an advantage to be more aerodynamic with fuel efficiency.

2. Research Objective

To ascertain the simulation of 3DAhmed body with a convenient solver scheme to obtain and validate the results of different flow parameters viz. drag force, drag coefficients, turbulent kinetic energy and wake flow structures.

3. Research Methodology

3.1.Problem Definition

In the present work, the 3D model of Ahmed body consisting of inlet, outlet, nose, top bottom, slope, back, symmetry.Variation of coefficient of drag changes with rear slant angle $35^0$is numerically investigated, with different turbulent solver schemes setup to meet desirable simulating condition.The Ahmed model is a simple geometric body that retains the main flow features, especially the vortex wake flow where most part of the drag is concentrated and it is a good perfection to be used as a benchmark test.

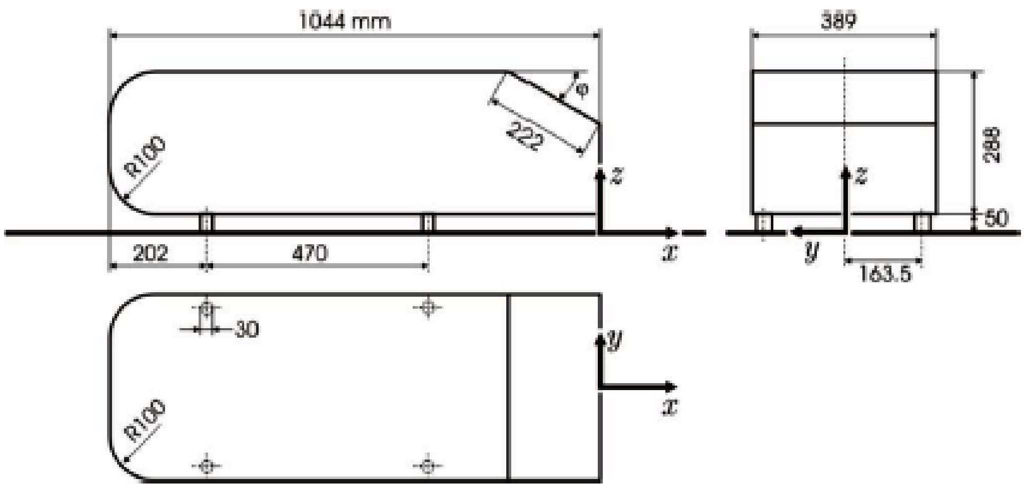

Fig 1: Geometry of Ahmed body

3.2.Governing Equations

To simulate any turbulent flow by solving the foregoing exact equations with appropriate boundary conditions using suitable numerical procedures such as two –equation known as k-e model.The velocity scale, V_s, is calculated from solution of a transport equation for KE. The KE-EP model has proved the most popular, mainly because it does not require a near-wall correction term.The dependent variable of the 2nd transport equation is not usually L_s itself, but rather the variable $KE^m L_s^n$. The standard high-Reform of the KE-EP model employs the following turbulence transport equations:

$$\rho\frac{\partial KE}{\partial t}+\rho\frac{\partial}{\partial x_i}\left[U_i KE-\frac{ENUT}{PRT(KE)}\frac{\partial KE}{\partial x_i}\right]=\rho\left(P_k+\Gamma_b-EP\right)$$

$$\rho\frac{\partial EP}{\partial t}+\rho\frac{\partial}{\partial x_i}\left[U_i EP-\frac{ENUT}{PRT(EP)}\frac{\partial EP}{\partial x_i}\right]=\rho\frac{EP}{KE}\left(C_{1e}P_k+C_{3e}\Gamma_b-C_{2e}EP\right)$$

The kinematic turbulent (or eddy) viscosity and the length scale, L_s are given by:

$$ENUT=C_{\mu}C_d\frac{KE^2}{EP}l_m=C_d\frac{KE^{3/2}}{EP}$$

The model constants are: $C_{\square}$=0.5478; C_d=0.1643; PRT (KE) =1.0; PRT (EP) =1.314; C_{1e}=1.44, C_{2e}=1.92 and C_{3e}=1.0.

3.3.Numerical Implementations

Numerical implementation involves solver setting for theproblem to be analysed. The solver used in the current analysis is ANSYS FLUENT V15, where 3D modellingwas done for transient state incompressible fluid flow in CATIA V5.

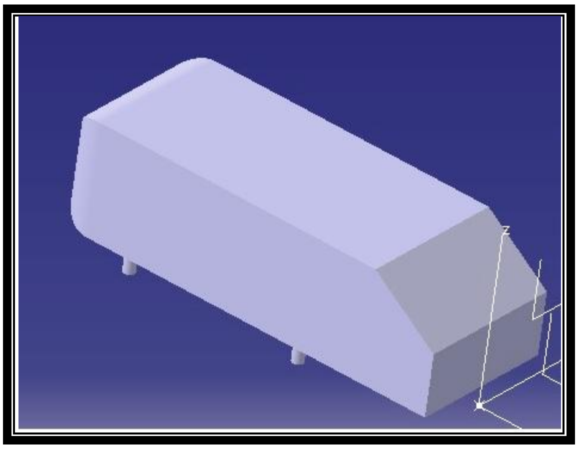

Fig2: CATIA 3D Ahmed Model

3.4.GeometricalModelling

The computational model of tested 3D car model consists of inlet, outlet, nose, top, bottom, slope back, and symmetry is developed using CATIA V5R21 part modelling as per the geometry parameters listed in table.

Table I: Design Model geometry parameters

Length	1.044m
Height	0.288m
Front radius	0.1m
Ground clearance	0.05m
Slant angle	35^o
Inlet velocity	10,20,30,40 m/s
Yaw angle	$B=0^o$
Blockage ratio	3.8%
Cross-sectional area	$A=0.112m^2$
Wind tunnel domain.	10.5m length, 3.03m wide, 5.03 height

Graphical representation of the domain model with all geometrical parameters are done in the ANSYS FLUENT workbench, shown in the Fig.3

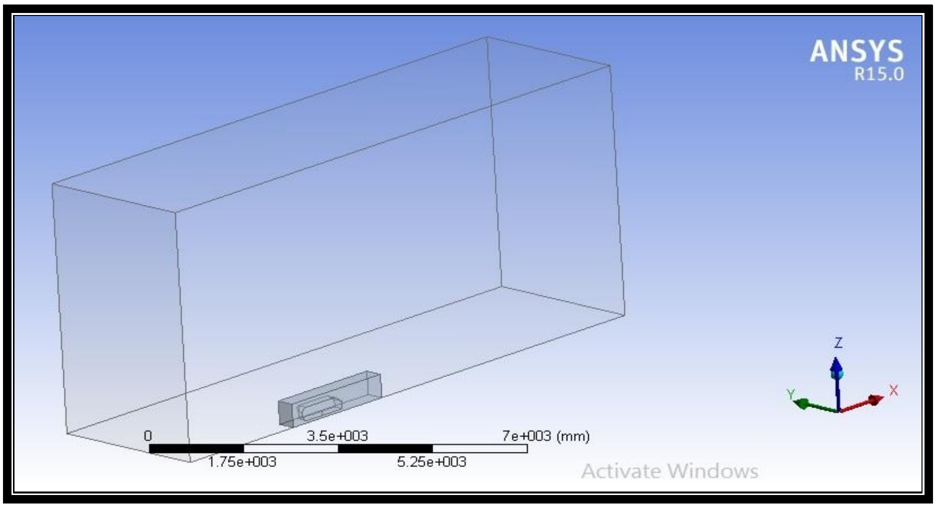

Fig.3: Domain representation of the Ahmed model test section

3.5. Mesh Generation

After physical modelling, the 3D computational domain is discretized which has triangular elements. Grid independence tests are carried out to ensure that a nearly grid independent solution can be obtained. Initially coarse mesh was generated for the car model with approximately 1275972 grid elements; later on the mesh was refined by increasing the grid elementswith 1354249, 1511745, 1603744 & 1806071. The same problem was made to run i.e. at the same velocity but with different mesh size. Fig4 shows the mesh domain of Ahmed body.

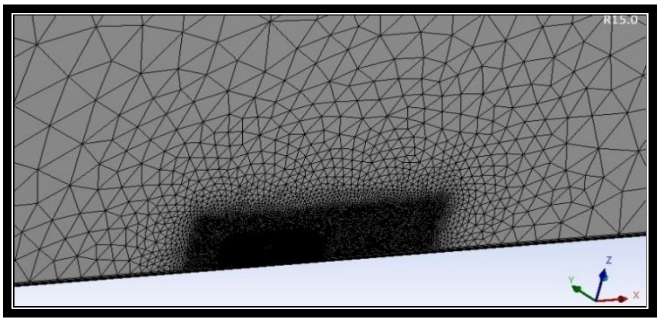

Fig 4: Mesh generation of Ahmed model setup

In the final mesh, grid element count was limited to 1806071 as there were no appreciable changes in drag co-efficient results.Drag coefficient Cd is a strong function of the Reynolds number at low values of Re andCd often levels off for Re above some threshold value as suggested by Yunus [16] can be shown in Fig.5.

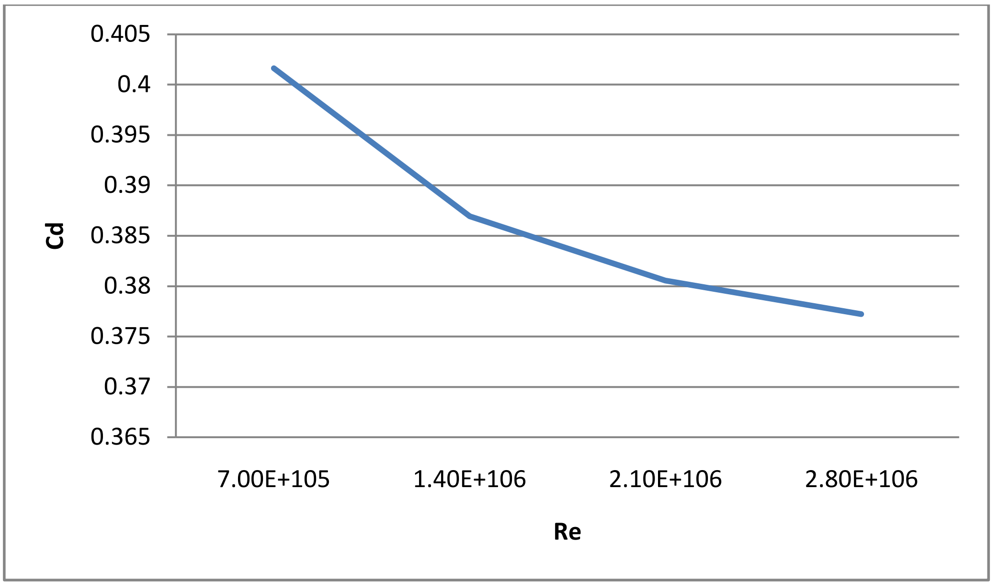

Fig 5: Variation of Drag Co-efficient (C_d)with Reynolds Number

Table II. Variation of Drag Co-efficient with Grid Elements

Sl. No.	Grid Elements	Co-efficient of Drag
1	1275972	0.401
2	1354249	0.393
3	1511745	0.389
4	1603744	0.387
5	1806071	0.385

3.6. Solver Setup

For the present work the solver used is ANSYS Fluent V15. The following steps are taken for the setting up the solving techniques in the solver for 3D, steadystate& incompressible.

- Imported meshed model over the solver workbench and check the mesh quality.
- Scale the mesh as per the dimensions specified with appropriate scaling factor.
- Setup the solver specifications such as pressure based, transient, and absolute velocity formulation.
- Defining models such as viscous model (standard k-ε model) is done. A standard wall function is opted fornear-wall treatment.

The model constants in k-ε equation are:
C_{μ} = 0.09 (model constant for Turbulent viscosity)
$C_{1\varepsilon}$ = 1.44 (model constant for transport equation)
$C_{2\varepsilon}$ = 1.92 (model constant for transport equation)
σ_k = 1.0 (Turbulent kinetic energy Prandtl number)
σ_{ε} = 1.3 (Turbulent dissipation rate Prandtl number)

Materials Properties:

Fluid: Air
Density (ρ) = 1.225 kg/m3

Viscosity (μ) = 1.7894e-05 kg/ms

Boundary Conditions:

The Input boundary condition needed for the simulation has been taken from the experimental data presented by H.Lienhart et al [15]. The boundary conditions applied to simulate the performance of Ahmed body at different dimensions are as follows:
-Inlet-velocity inlet
-Outlet-pressure outlet (atmospheric)
-Wall condition-no slip and adiabatic wall condition.

Solution Methods:

Different solverschemeschosen at each stage are shown in Table III.

Table III: solver scheme used for model analysis

Stage	I	II	III	IV
Pressure-velocity coupling scheme	Simple	Simple	Simple	Simple
Spatial Discretization:				
Gradient	Green Gauss cell based	Green Gauss cell based	Green Gauss cell based	Green Gauss cell based
Pressure	Standard	Standard	Second order	Second order
Momentum	1st order upwind	2nd order upwind	1st order upwind	2nd order upwind
Turbulent KE	1st order upwind	2nd order upwind	1st order upwind	2nd order upwind
Turbulent Dissipation Rate	1st order upwind	2nd order upwind	1st order upwind	2nd order upwind
Iteration number	200	200	200	200
Relaxation factors	Pressure :0.3 Density :1 Body force: 1 Momentum :0.7 Turbulent kinetic energy:0.8 Turbulent Dissipation rate:0.8 Turbulent viscosity :1	Pressure :0.3 Density :1 Body force: 1 Momentum :0.7 Turbulent kinetic energy:0.8 Turbulent Dissipation rate:0.8 Turbulent viscosity :1	Pressure :0.3 Density :1 Body force: 1 Momentum :0.7 Turbulent kinetic energy:0.8 Turbulent Dissipation rate:0.8 Turbulent viscosity :1	Pressure :0.3 Density :1 Body force: 1 Momentum :0.7 Turbulent kinetic energy:0.8 Turbulent Dissipation rate:0.8 Turbulent viscosity :1
Velocity for each case	10-40 m/s	10-40 m/s	10-40 m/s	10-40 m/s

4. Results &Discussion:

Present result of oveall Drag coefficient (C_d) is better approachable to that of the experimental results obtained by Ahmed [1] as compared to the simulation results of Parab et.al [17] &Khan et .al [5], shown in Fig 6.

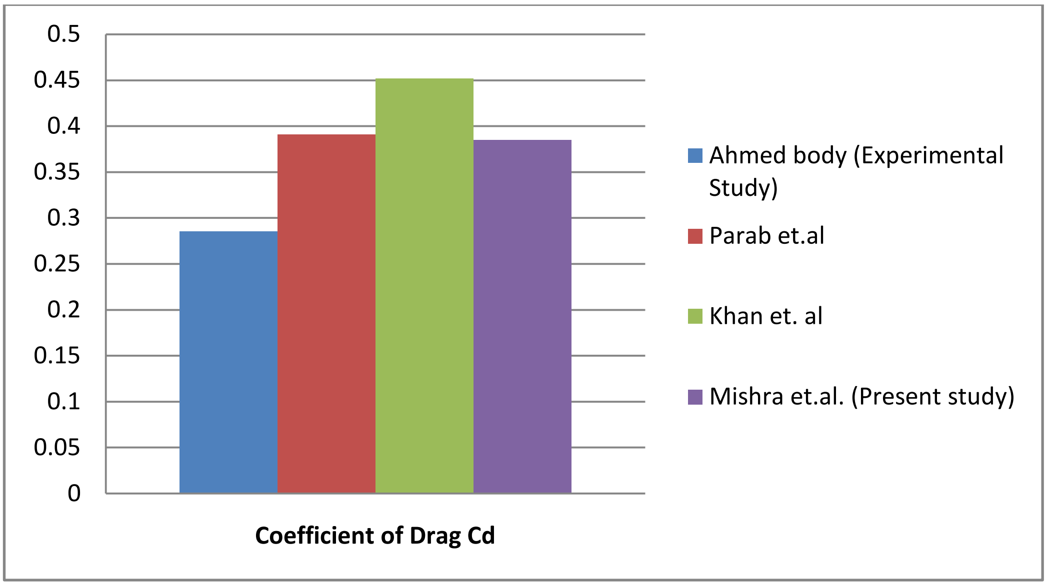

Fig 6: Comparison of C_d Values at velocity 40m/s.

The velocity, pressure, turbulent kinetic energy, turbulent dissipation rate contours are shown at follows:

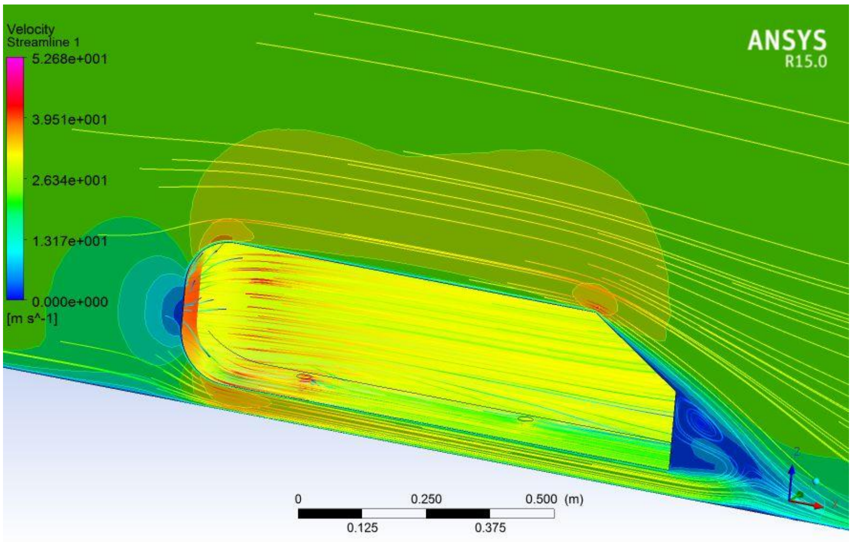

Fig 7: 3D Velocity contours at velocity 40m/s

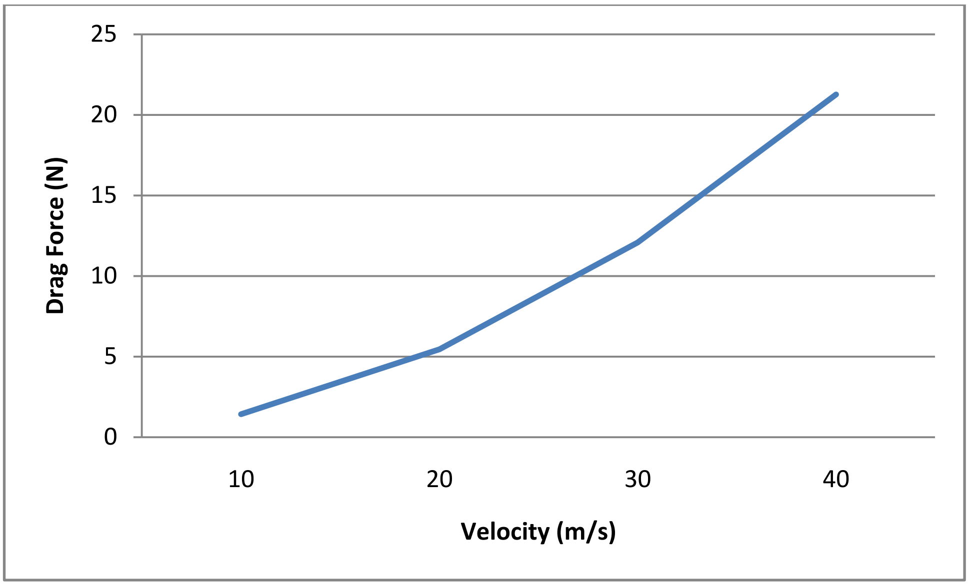

Fig8: Variation of F_d with Velocity

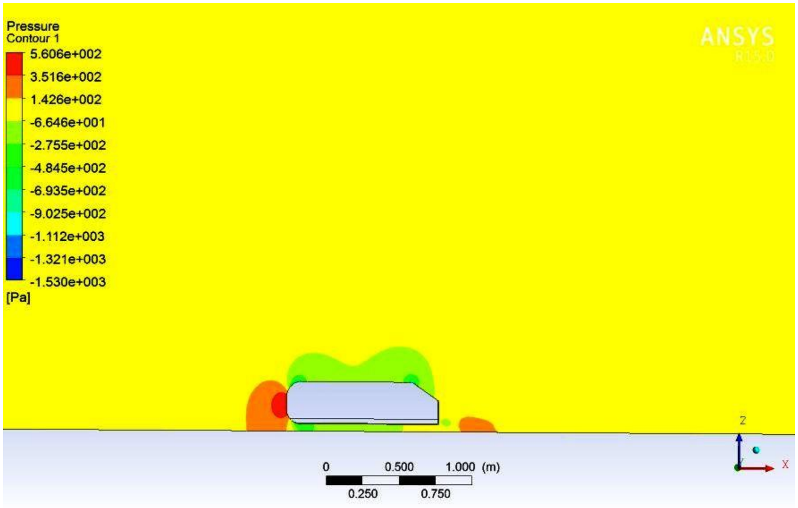

Fig 9: Pressure contours at 40m/s

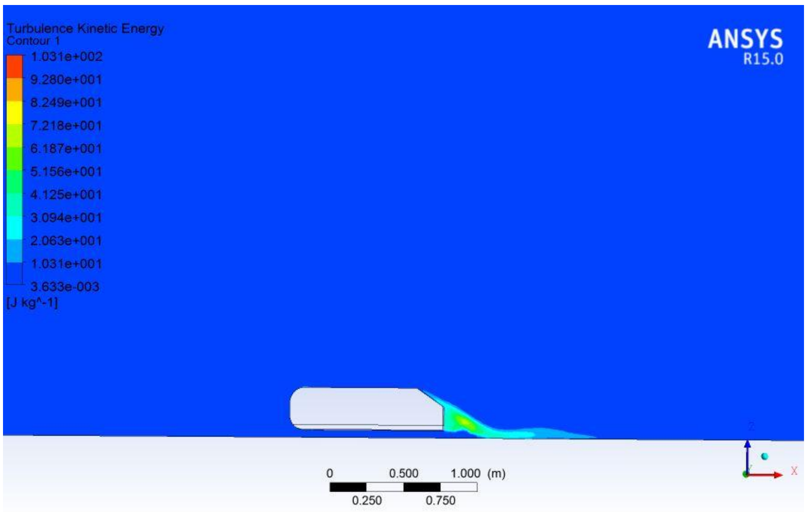

Fig10: Turbulent Kinetic Energy at 40m/s

From the above results, it is revealed that

- 3D Streamline patterns are observed from the Fig 7 over the Ahmed model which continues up to the separation point where the flow gets detached arising the eddies and vortices towards the rear part of the body.
- Fig.8shows that the drag force gets increased with the increase in velocity.
- It is evident from the Fig. 7 & Fig.9 that the high pressure (stagnation Point) acts at the front nose part and this pressure gets decreased with the increase in velocity.
- Separation zone is clearly visualized at the rear slant part region in Fig.7.
- It is envisaged from Fig.10 that the Turbulent Kinetic Energy increases at the mid section showing the development in vehicle wakes.

5. Conclusion:

CFD analysis with Second Order Upwind schemewas successfully carried out over the Ahmed 3D Benchmark model. The results of the present simulation with regard to drag coefficient were found to be in close agreement with the wind tunnel experimental results.

6. Future work

The main purpose of CFD simulation is to saves huge amounts of money, by avoiding the need to build and test prototypes for various complex geometries and various design parameters with optimization results. Proper solver schemes leading towards accuracy and convergence must be chosen to validate the results. Once the validity of the simulation was achieved the next step was to make modifications in the geometry and selection of a better sedan car model which could positively affect the performance characteristics with improve handling capabilities at higher speeds & overall safety of the vehicle.

References

[1] Ahmed, S.R. and Ramm G., "Some salient features of the time averaged ground vehicle wake," SAE Technical Paper 840300, 1984.

[2] Hucho W H, Aerodynamic of road Vehicles (Butterworth, London) 1997.

[3]Ahmed S.R., (1981), "Wake structure of typical automobile shapes", Trans. ASME, J. Fluids Eng. 103, 162-169.

[4] Han T., (1989), "Computational analysis of three dimensional turbulent flow around a bluff body in ground proximity", AIAA J. 27(9) 1213- 1219.

[5]Khan,R.S and Umale S,"CFD Aerodynamics Analysis of Ahmed Body,"International Journal of Engineering Trends and Technology (IJETT) – Volume 18 Number 7 – Dec 2014

[6]Bijlani et al., "Experimental And Computational Drag AnalysisOf Sedan And Square-Back Car," IJAET, IV, II, 2013,63-65

[7] Argyropoulos C.D, Markatos N.C," Recent advances on the numerical modelling of turbulent flows," Applied Mathematical Modelling 39 (2015) 693–732

[8] Sharath Kumar S N " Analysis of external aerodynamics of Sedan and Hatch back car models by Experimental and Computational Methods" IJRMET ,Vol. 6, Issue 1, 2016

[9]Ahmed A, Murtaza M A,"Cfd Analysis Of Car Body Aerodynamics Including Effect Of Passive Flow Devices – A Review",IJRET,05, 03, 2016.

[10]D. Y Dhande, "Experimental analysis of aerodynamic aspects of sport utility vehicle," *IJEST*, vol. 5(7), pp. 1476-1480, 2013.

[11] 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.

[12]A. I. Heft, T. Indinger and N. A. Adams, "Introduction of a new realistic generic car model for aerodynamic investigations," *SAE International*, 2012.

[13] S.R.Ahmed," Wake Structure of Typical Automobile Shapes", Journal of Fluidsngineering, 162/Vol. 103, 1981

[14]Fukdaet.al,"Improvement of vehicle aerodynamics by wake control",JSAE Review 16(1995),151-155

[15] H. Lienhart, C. Stoots and S. Becker, Flow and turbulence structures in the wake of a simplified car model (Ahmed Model).

[16] Yunus A Cengel, John M Cimbala "Fluid Mechanics Fundamentals and Applications", McGraw-Hill Publications, 2006.

[17]parabet.al,"Aerodynamic Analysis of a car Model using Fluent-Ansys 14.5,"IJRMEE,vol 1,Issue 4,2014