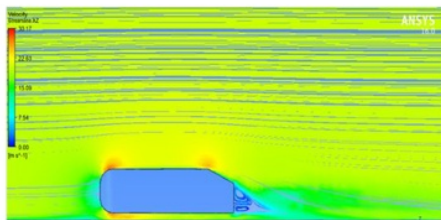


Computational fluid Dynamics

Ahmed Body Analysis

by Dilshad Rahamani

FILE	6012_DILSHAD_RAHAMANI_COMPUTATIONAL_FLUID_DYNAMICS_AHMED_BODY_ANALYSIS_176684_883427699.DOCX (1.05M)		
TIME SUBMITTED	28-APR-2019 11:44 PM (UTC+0100)	WORD COUNT	3300
SUBMISSION ID	105595357	CHARACTER COUNT	22143



University of Wales Trinity Saint David

School of Engineering

Title of Report: Computational fluid Dynamics: Ahmed Body Analysis

Student Name: Dilshad Rahamani

Student Number: P133318

MSc Mechanical Engineering

Date: 28/04/2019

Model Name and Code: Simulation and Modelling

M7X01380

Contents

List of Figures.....	1
Abstract.....	3
1.1 Introduction	4
1.1 Background Research.....	4
1.2 Objectives	5
2. Literature Review	6
3. Methodology.....	7
3.1 Geometry	7
3.2 Discretisation.....	9
3.3 Boundary Conditions.....	12
3.4 Physics Setup	13
3.5 Turbulence Study	13
3.6 Solver Setting & Convergence	15
4. Results and Discussion	17
4.1 Results for 25° slant angle	17
4.1 Results for 35° slant angle	19
5. Conclusion	22
References.....	23

List of Figures

Figure 1 Process flow of CFD analysis.....	7
Figure 2 Ahmed Body Dimensions.....	8
Figure 3 Computational Domain of Ahmed Body.....	8
Figure 4 Ahmed body symmetry section.....	9
Figure 5 Mesh Contour	10
Figure 6 Mesh Contour - Zoom view	10
Figure 7 Mesh Contour with Prism layer.....	11
Figure 8 Part Mesh Setup.....	11
Figure 9 Mesh Independence Study.....	12
Figure 10 Boundary Condition	13
Figure 11 Coefficient of Drag Vs Turbulence Model.....	14
Figure 12 Coefficient of Lift Vs Turbulence Model.....	15
Figure 13 Solver settings	16
Figure 14 Velocity Contour	17
Figure 15 Pressure Contour	17
Figure 16 Eddy Viscosity Contour	18
Figure 17 Turbulence Kinetic Energy.....	18
Figure 18 Velocity Streamline	18

Figure 19 Velocity Contour	19
Figure 20 Pressure Contour	19
Figure 21 Eddy Viscosity Contour	20
Figure 22 Turbulence Kinetic Energy	20
Figure 23 Velocity Streamline	20
Figure 24 Numerical and Experimental Results	21

Abstract

The Computational Fluid Dynamics (CFD) is one of the computational techniques used to provide real time solution for fluid dynamics problems. The CFD modelling is evaluated through various CFD software tool such as Ansys, StarCCM+, Open Foam and so on. The importance of CFD modelling in automotive industry is huge and this computational technique is widely used as problem solver. The research study of this assignment is to complete the CFD modelling over the Ahmed Body. The Ahmed Body is basic car body which is widely used in automotive industry. The primary significance of Ahmed Body is to investigate the impact of fluid flow all around the body, especially in terms of lift and drag. This assignment provide the CFD modelling over the given Ahmed Body which was completed as three software tool such as Ansys Design Modeller, ICEM CFD and Ansys Fluent. The geometry correction are made in Design Modeller and the mesh generation and mesh independence study are performed using ICEM CFD and finally the processing and post processing are performed using Ansys Fluent. The computational results are compared with experimental data to observe the accuracy of numerical results.

1.1 Introduction

In today's modern world, the necessity of vehicles is keep increasing day by day. On another hand, the impact of CO₂ emission is huge around the world. More than 50% CO₂ emission are obtained by a vehicle which leads to global warming and the impact of CO₂ emission is widely affects the climate condition. So this is the time to control the global warming through the technology. This assignment widely discuss about the vehicle aerodynamics with existing Ahmed Body car model. The primary need of vehicle aerodynamics is to reduce the drag force and increase the lift. The drag reduction leads to less fuel consumption. The less fuel consumption leads to reduce the impact of CO₂ emission. The external aerodynamics over the Ahmed Body will induce to identify the solution of drag problem when vehicle under motion.

The road vehicles such as bus, car and trucks are high impacted by the turbulent when it's under motion. The turbulent is one of the reason for drag, wake flow and vortices which high affects the speed of the vehicle. The turbulent flow can be analysed using CFD modelling. The aerodynamics features and application of vehicles shall be studied by the vehicle aerodynamics with the help of numerical analysis. The numerical simulation are performed using CFD software's. This paper widely investigates the vehicle aerodynamics of Ahmed body. Normally, the Ahmed body is basic car body model which is generally used in automotive industry. The significance of Ahmed Body is to investigate the influence of fluid flow all around the body, especially in terms of lift and drag (Parab & Mangrulkar, 2014).

1.1Background Research

The study of external aerodynamics is highly necessary to complete the efficient of vehicles in various aspects such as CO₂ emission, minimised fuel consumption, noise and comfort driving stability. Due to the advance engineering techniques, the computational simulation shall be performed and complete the numerical results with high level of flow visualisation. The computational process of vehicle aerodynamics is one of the virtual development process to

gain more advantages in flow visualisation. These virtual process will leads to identify the solution for real time problems especially in aerodynamics aspects.

The computational process is involves the high level of fluid flow application such as Internal and external aerodynamics. These was achieved by the computational software and which is run by fluid flow governing equations such as law of conservation of momentum and laws of conservation mass and Navier Stokes equation. The flow equation are numerically solved by the computational software with appropriate boundary condition with respect to real time physics. The computational process is simplified, cost effective, time effective and so on. There are several advantages to use computational modelling instead experimental modelling. The experimental modelling takes more time to analyse and gather the final data but those things are easily carried out in CFD modelling. This research study deals with the external aerodynamics over the Ahmed body.

In 1984 S.R. Ahmed developed the simple car body which was used to evaluate and effect of flow patterns around the vehicles with the numerical analysis. That simple car body was further named as Ahmed body and which was accepted by automotive industry. The Ahmed body not only used to provide flow pattern and its impacts which also provide the benchmark of validation of road vehicle through various aspects.

1.2 Objectives

The primary object and the scope of work is to perform the external aerodynamics analysis over the given Ahmed body model and compare the numerical results with experimental results. The progress was done using the computational program Ansys and complete the numerical results with appropriate boundary condition. The validation and comparison of numerical results with experimental results takes places to identify the accuracy of numerical simulations.

2. Literature Review

This section describe the literature survey of existing journals and articles related to the computational analysis over the Ahmed body with appropriate investigation.

(Khan & Umale, 2014) Investigates the CFD aerodynamics modelling of the Ahmed body. The primary objective of this assignment is to investigate and analysis the Ahmed body model with K-Epsilon turbulence model. There are two Ahmed body model are analysed with separate slant angles of 25° and 35° with appropriate boundary condition. The lift and drag coefficient are complete from numerical simulation and compare those results with the experimental results for validation.

(Landge & Palande, 2016) Studied the aerodynamics characteristics of automobile car body (Ahmed body) with respect to the computational fluid analysis to calculate the lift and drag coefficient. The investigation of wake flow, vortices, and flow patter over the Ahmed body are widely described on this assignment with appropriate numerical results. The primary objective of this paper is to predict the drag production rate over the Ahmed body using computational modelling.

(Meile, et al., 2011) investigates the aerodynamics behaviour of Ahmed body via numerically and experimentally. The experiments involves with two individual slant angle Ahmed body model such as 25° and 35° with appropriate results number. For numerical simulation Ansys Fluent is used to complete the results with respect to real time physics setup.

These are the important research journals are highly investigates the external aerodynamics over the Ahmed body with appropriate boundary conditions.

3. Methodology

This section describe the complete process flow activity of the computational analysis over the Ahmed body.

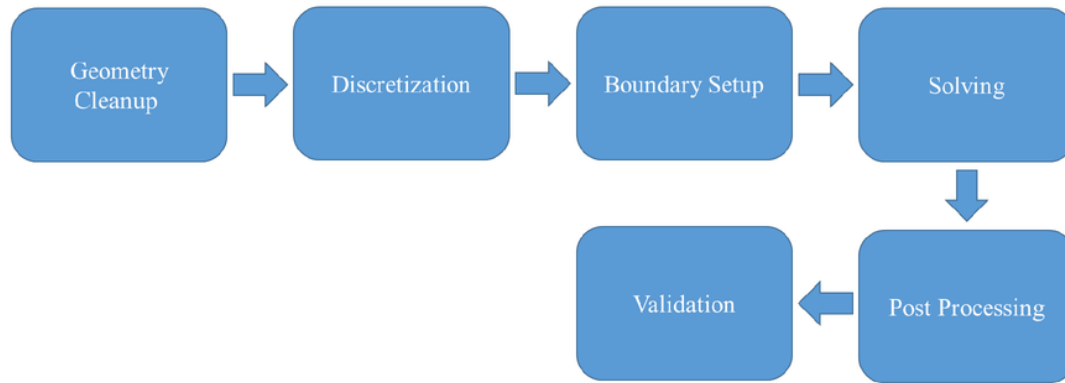


Figure 1 Process flow of CFD analysis

The above figure 1 shows the process flow of CFD analysis over the Ahmed body. The process flow holds several important stages such as geometry clean up, discretisation, boundary setup, solving, post processing and validation. Each stages holds high level of work activity to complete the numerical simulation with respect to computational software (Emmanuel Guilminea, 2008).

3.1 Geometry

This section elaborates the Ahmed body dimensions and computational domain of model with appropriate scaling.

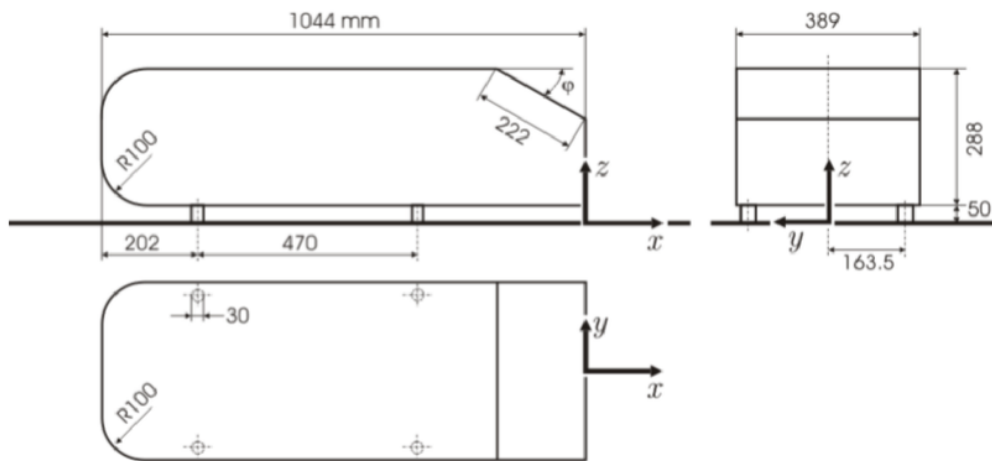


Figure 2 Ahmed Body Dimensions

The above figure 2 represents the dimensions of Ahmed body with appropriate standard. The length of the model is about 1.044meters and the height of the model is 0.288 meters and the width of the model is about 0.389meters. The distance between the ground and under body of model is appropriately represented in figure 1. There are two models are developed with the different of slant angle such as 25° and 0° and 35° . From the above mentioned dimensions the 3D CAD are being developed to perform numerical simulation.

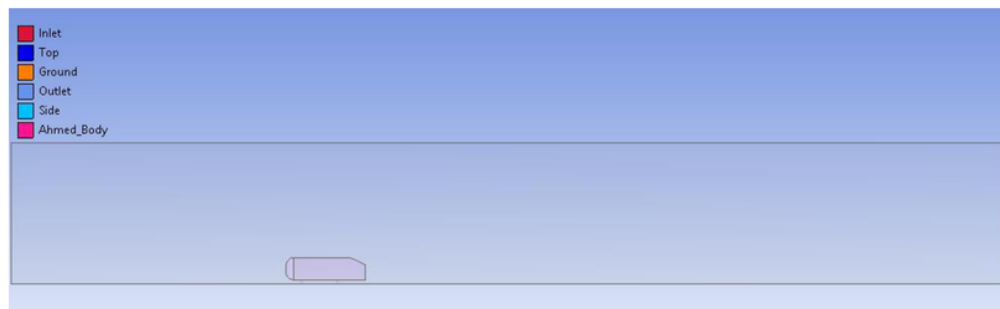


Figure 3 Computational Domain of Ahmed Body

The above 3 shows the computational domain for Ahmed body. The dimensions of computational domain are appropriately chosen for numerical simulation. The below figure 3

shows the symmetry cross section of Ahmed body. The simulation performed with symmetry section alone in order to reduce the computational time.

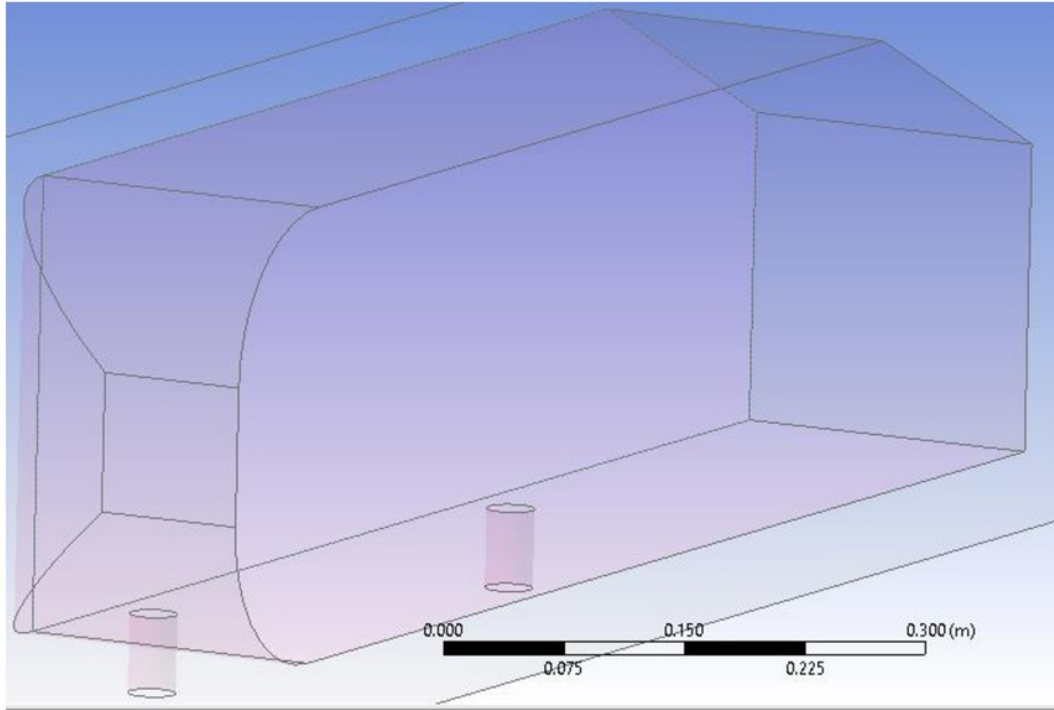


Figure 4 Ahmed body symmetry section

The dimensions of computational domain is about 8.352 meters of downstream length, 3.6meters of upstream length, height of the domain is about 1.5 meters and the distance between side section to domain is about 1 meters. While seeing the dimensions of computational domain, the upstream length is very less than the downstream length. The main purpose of increased downstream length is to predict the flow pattern promptly and also high downstream length avoids reversed flow during computation which leads to error free quality results.

3.2 Discretisation

The Discretisation is one of the most important process in CFD analysis which is also termed as meshing. During this operation, the complete domains are discretised into cells with

appropriate method. This paper performed with finite volume method with appropriate control volumes.

The discretisation or meshing are done with the help of ICEM CFD tool. The unstructured volume mesh are generated and being used for simulation.

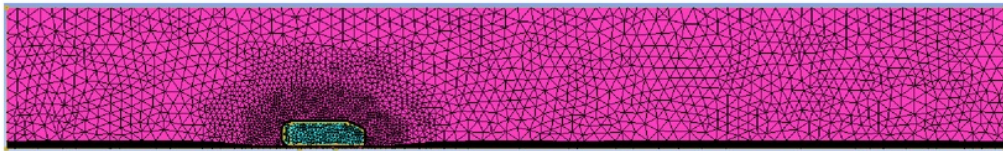


Figure 5 Mesh Contour

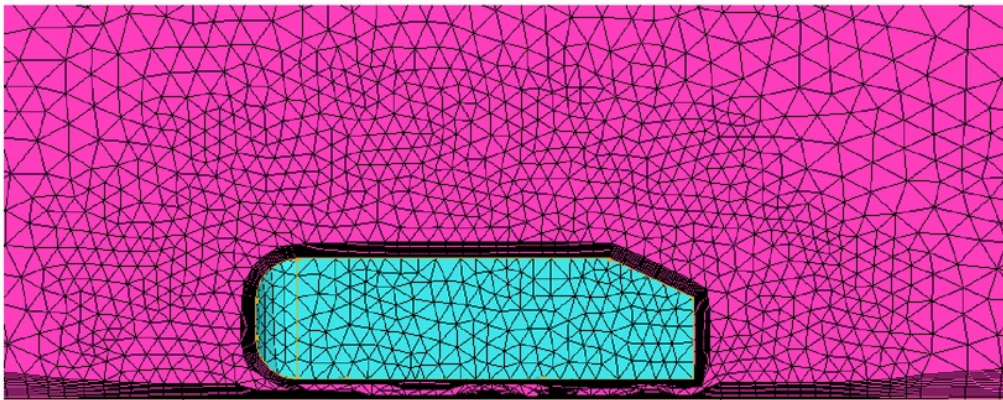


Figure 6 Mesh Contour - Zoom view

The above figures 5 and 6 shows the mesh contours of Ahmed body. The mesh was performed with ICEM CFD as discussed earlier and the mesh was generated at dense on Ahmed body region. The figure 5 clearly shows the density variation between the Ahmed body and computational domain.

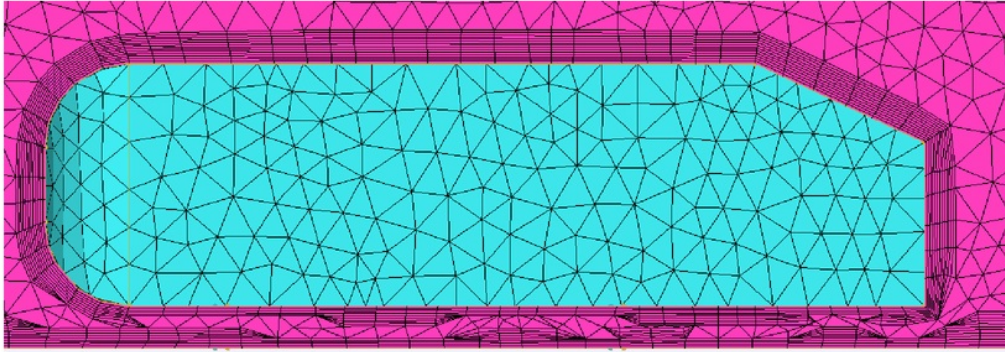


Figure 7 Mesh Contour with Prism layer

The figure 7 shows the prism layer mesh over the Ahmed body region. The main purpose of prism layer mesh is to predict the flow pattern clearly. In this model, Tetra mesh used as volume ask and the hexa mesh used for prism layer mesh. This mesh was comes under unstructured mesh condition.

There are several important factors need to considered while generating volume mesh in ICEM CFD such as scale factor, element size and part mesh setup. In this model the scale factor is considered as 8 and the maximum element seed is took as 0.02. Apart from that part mesh setup are updated appropriate with respect to domain size.

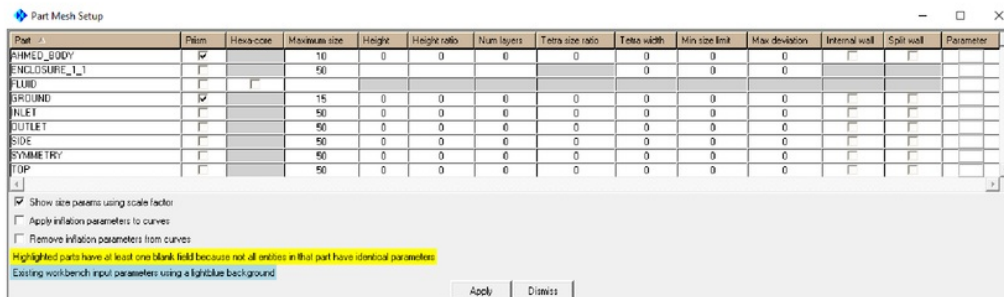


Figure 8 Part Mesh Setup

The above figure 8 shows the part mesh setup with appropriate value based on the computational domain size. The prism layer only applies on two parts such as Ahmed body and ground to predict the flow pattern appropriately. The total number of nodes is 156432 and the elements is about 749976. The mesh was chosen as medium size with respect to computational

time and processing speed. The quality parameters of mesh such as skew, aspect ratio, orthogonal angle are appropriately check.

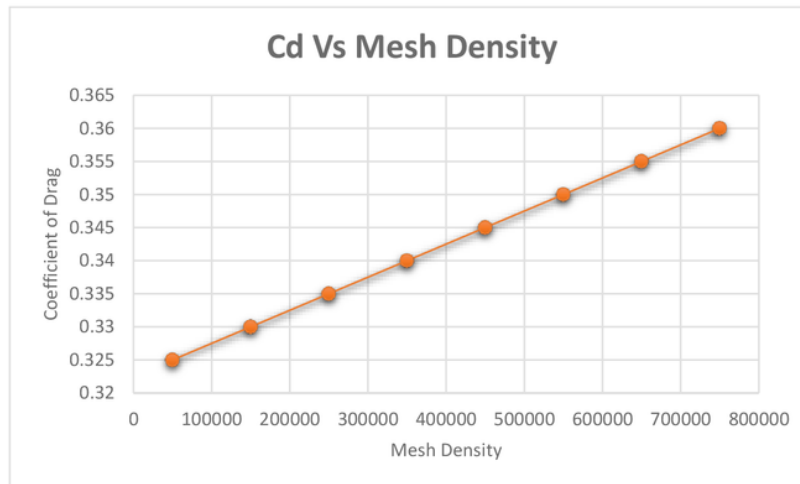


Figure 9 Mesh Independence Study

The Mesh independence study are carried out to check the impact of the mesh size over the computational results. There are three different mesh size are taken to perform mesh independence study such as Coarse mesh, medium mesh and fine mesh. As discussed earlier, this assignment performed medium based mesh. The maximum error percentage is found as 14% for coarse mesh. The medium mesh error percentage is about 6%. The fine mesh can be chose for simulation but that will leads to more computational time so for that medium mesh was finalised to complete the computation results. The figure 9 shows the mesh independence study performed for three different mesh size. The impact of each mesh size with respect to results are complete apparently, from the mesh independence study the medium mesh was finalised.

3.3 Boundary Conditions

This section describes the boundary condition for each boundary with appropriate physics. The inlet is chosen as velocity inlet with the velocity of 20m/s and the outlet is chosen as pressure

outlet, the turbulence intensity are chosen as 1% for velocity inlet and 5% for pressure outlet. The ground and the Ahmed body are taken as no slip wall condition. The top, side and symmetry section are chosen as symmetry condition. The below figure 10 shows the boundary condition for the numerical simulation of Ahmed body.

Boundary	Boundary Condition
Inlet	Velocity Inlet
Outlet	Pressure Outlet
Ahmed body	No Slip
Side, Top, Symmetry	Symmetry

Figure 10 Boundary Condition

3.4 Physics Setup

This section describe the physics setup for computational analysis. In this Ahmed body simulation the following physics setup are made

- Steady State,
- Pressure Based Solver,
- Turbulence Model (K-Epsilon, K-Omega & K-Omega SST)
- Fluid – Constant Air Density
- Default Under Relaxation Factor
- SIMPLE scheme is used for pressure-velocity coupling

These are the important boundary condition and physics setup are made in computational modelling of Ahmed body.

3.5 Turbulence Study

There are three turbulence model are performed and enumerated the results. The turbulence models are

- K-Epsilon
- K-Omega
- K-Omega SST

The standard wall function is used for both the turbulence model of K-Epsilon and K-Omega
omega

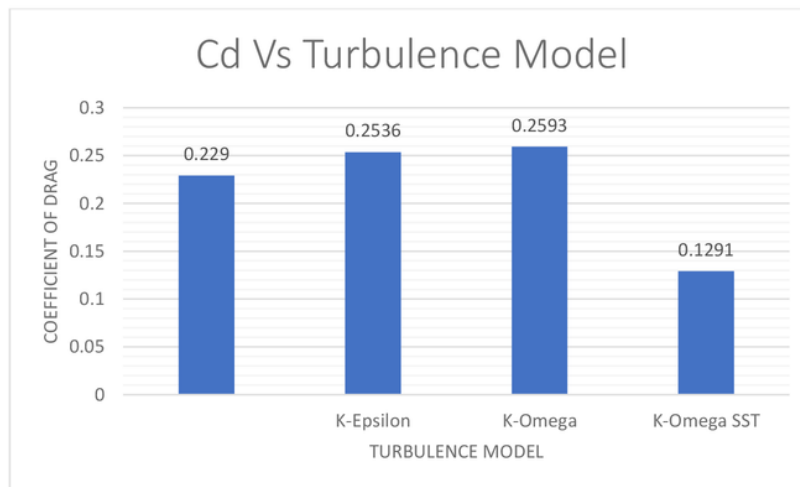


Figure 11 Coefficient of Drag Vs Turbulence Model

The above figure 11 shows the coefficient of drag value of numerical results with appropriate turbulence model. This results are taken from the model of 25° slant angle. While comparing the numerical results with experimental, the K-Epsilon comes very close value than other turbulence model. The cd value of K-Epsilon model is about 0.2536 and the experimental value is about 0.229. So the K-Epsilon turbulence model provide more accurate data than other turbulence model (Rajsinh & Raj, 2012).

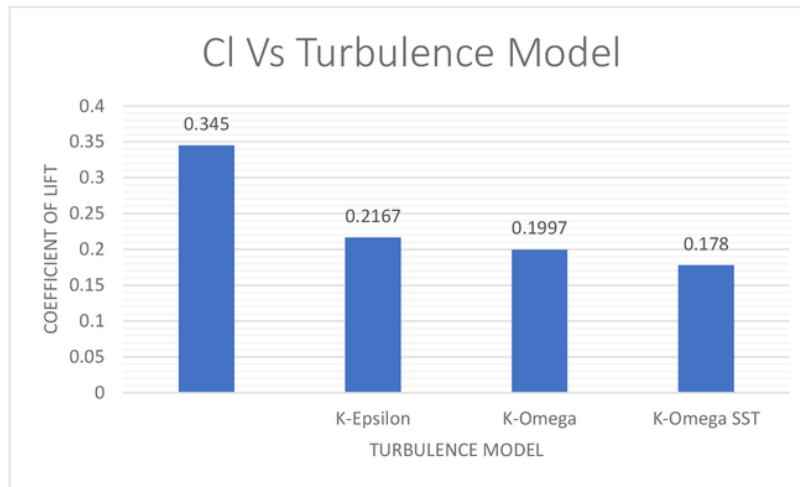


Figure 12 Coefficient of Lift Vs Turbulence Model

The above figure 12 shows the coefficient of lift value of numerical results with appropriate turbulence model. This results are taken from the model of 25° slant angle. While comparing the numerical results with experimental, the K-Epsilon comes very close value than other turbulence model. The C_d value of K-Epsilon model is about 0.2167 and the experimental value is about 0.345. So the K-Epsilon turbulence model provide more accurate data than other turbulence model.

3.6 Solver Setting & Convergence

This section describes the solver setting completed in the computational simulation. The SIMPLE scheme used for pressure-velocity coupling. Second order solution accuracy used for Pressure and momentum. The first order upwind used for turbulent kinetic energy and dissipation rate. The drag and lift is activated in monitor plot to complete the coefficient of lift and coefficient of drag value.

Solution Methods	Solution Controls
Pressure-Velocity Coupling Scheme SIMPLE	Under-Relaxation Factors Pressure 0.3
Spatial Discretization Gradient Least Squares Cell Based	Density 1
Pressure Second Order	Body Forces 1
Momentum Second Order Upwind	Momentum 0.7
Turbulent Kinetic Energy First Order Upwind	Turbulent Kinetic Energy 0.8
Specific Dissipation Rate First Order Upwind	

Figure 13 Solver settings

The above figure 13 shows the solver setup such as solution methods and its control for the computational simulation. The RMS convergence criterion is used for this simulation. The convergence limit is set as 1×10^{-5} for continuity, x-velocity, y-velocity, z-velocity, k, epsilon, omega, SST residuals. The solution was converged between 500-800 iteration with different simulation cases.

4. Results and Discussion

This section briefly describes the numerical results of Ahmed body with appropriate contour representation

4.1 Results for 25° slant angle

The following section demonstrates the computational results of 25° slant angle Ahmed body model,

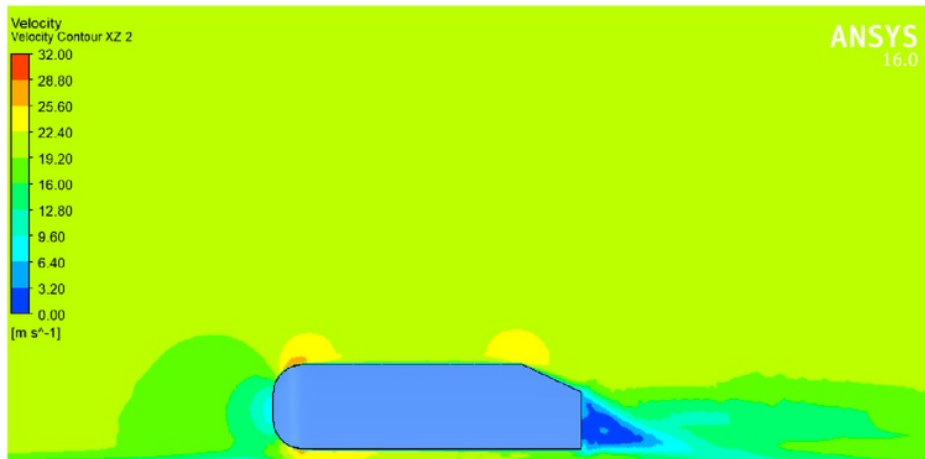


Figure 14 Velocity Contour

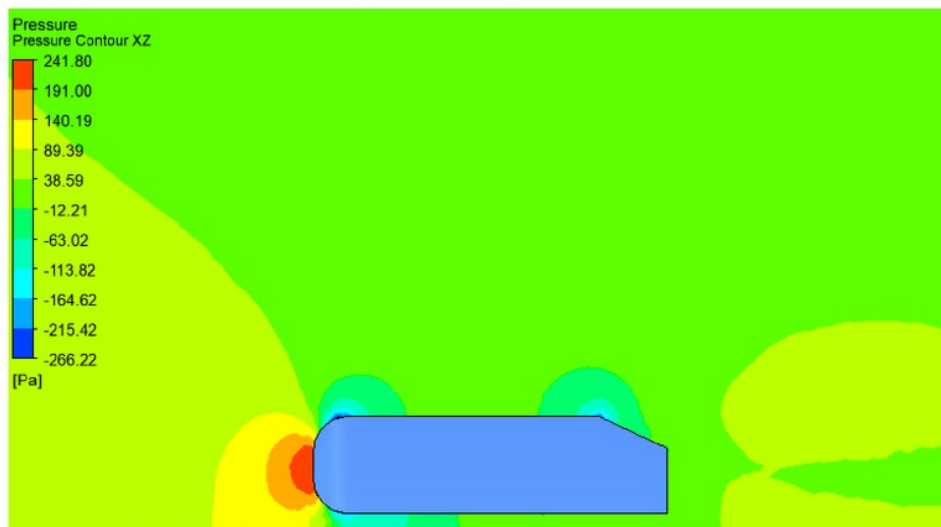


Figure 15 Pressure Contour

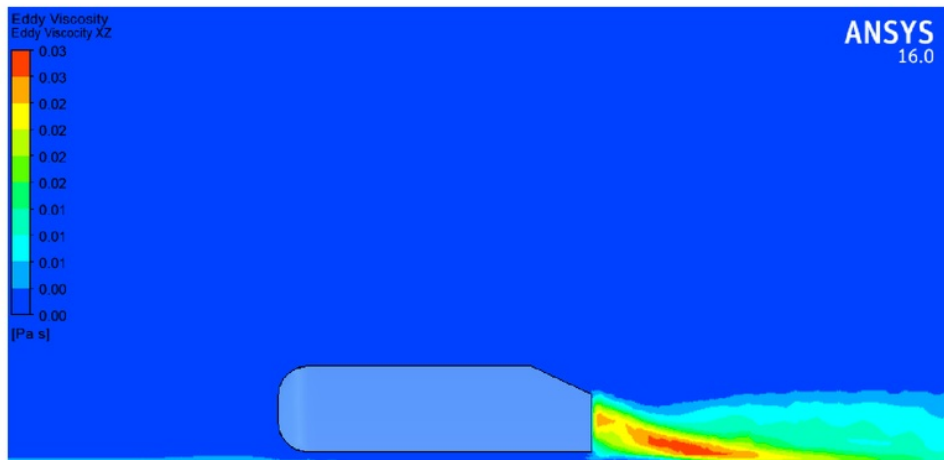


Figure 16 Eddy Viscosity Contour

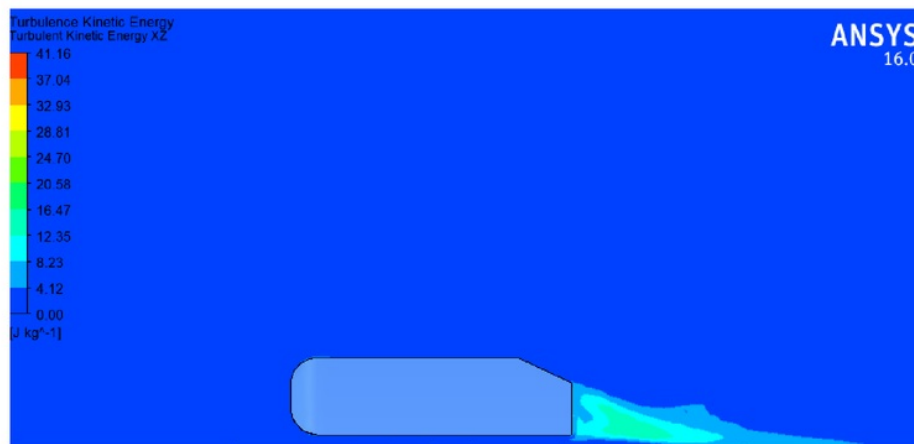


Figure 17 Turbulence Kinetic Energy

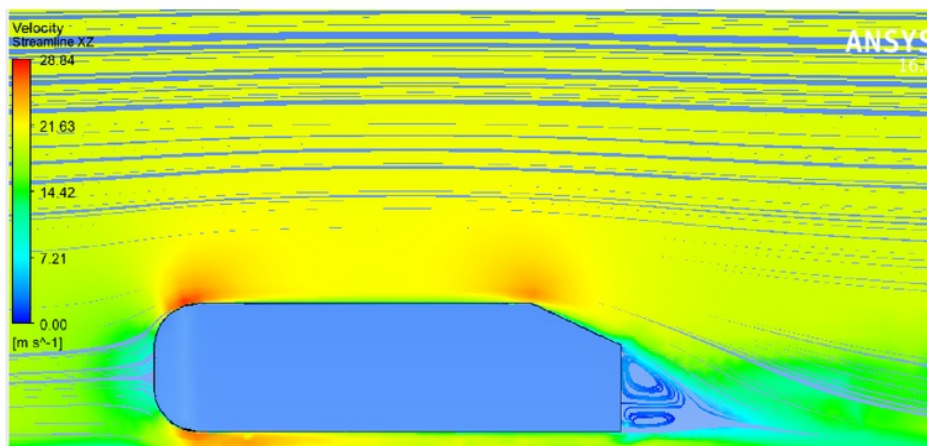


Figure 18 Velocity Streamline

The above figures 14-18 shows the computation results of Ahmed body with 25° slant angle. The velocity contour shows the velocity regime with appropriate deviations. The rear region loads very less velocity than the front end. The pressure contour eddy viscosity, turbulence kinetic energy are appropriately plotted. The velocity streamline flow clearly shows the vortex region on the rear end. The velocity are plotted as user specified instead global or local for clear outline.

4.1 Results for 35° slant angle

The following section demonstrates the computational results of 35° slant angle Ahmed body model,

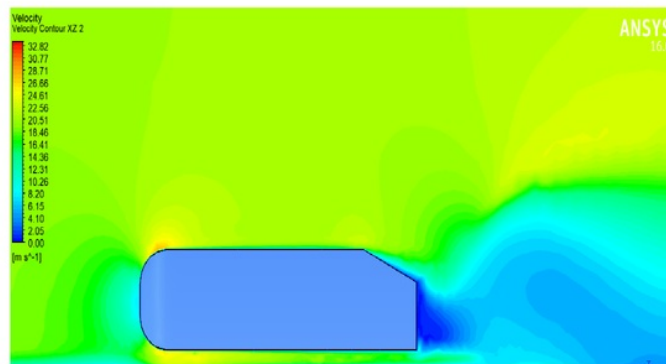


Figure 19 Velocity Contour

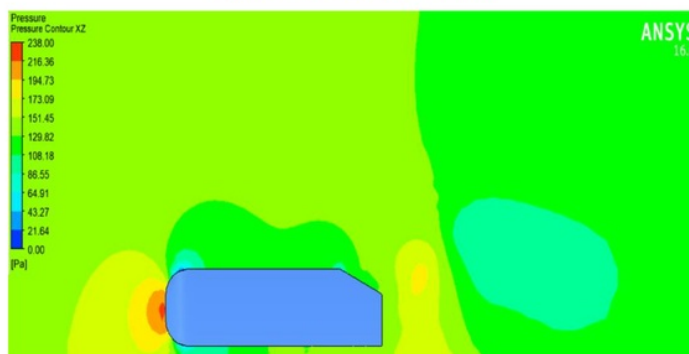


Figure 20 Pressure Contour

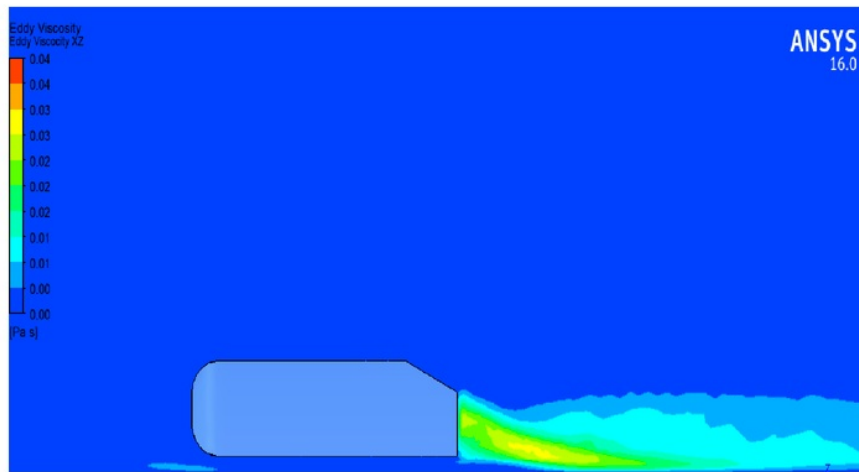


Figure 21 Eddy Viscosity Contour

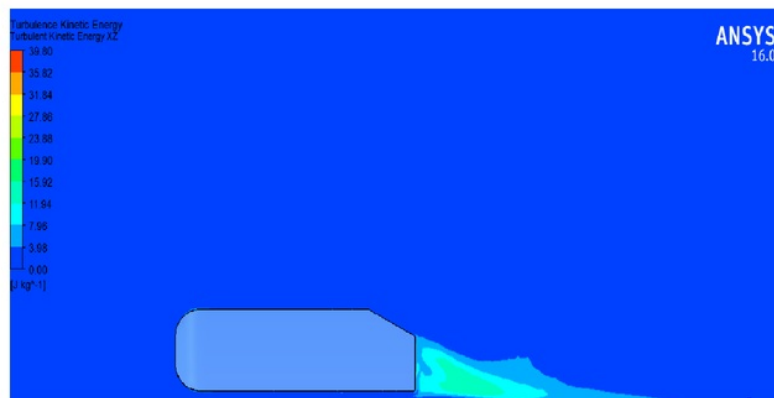


Figure 22 Turbulence Kinetic Energy

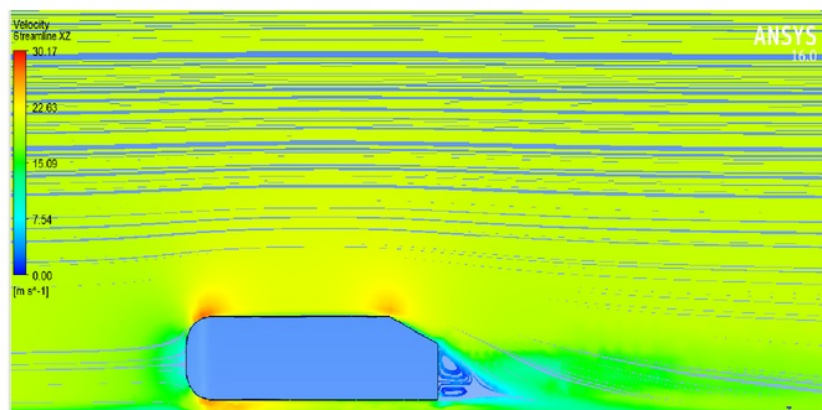


Figure 23 Velocity Streamline

The above figures 19-23 shows the computation results of Ahmed body with 35° slant angle. The velocity contour shows the velocity regime with appropriate deviations. The rear region loads very less velocity than the front end. The pressure contour eddy viscosity, turbulence kinetic energy are appropriately plotted. The velocity streamline flow clearly shows the vortex region on the rear end. The velocity are plotted as user specified instead global or local for clear outline.

Parameter	Experimental at 25°	CFD Results at 25°
Cd	0.229	0.2536
Cl	0.345	0.2167
Parameter	Experimental at 35°	CFD Results at 35°
Cd	0.279	0.3128
Cl	0.004	0.0066

Figure 24 Numerical and Experimental Results

The above table 24 shows the numerical results of Ahmed body model for 25° and 35° slant angle respectively. The numerical results of Ahmed body is not that much accurate than experimental results. The variation of results due to the mesh size, turbulence model, solver monitors and controls. The computational modelling over the Ahmed body model are analysed and compared with experimental results (Kalyan & Paul, 2013).

5. Conclusion

The flow characteristic over the Ahmed body model are complete with computational software with different turbulent condition. This vehicle aerodynamic project provide huge learning in flow regime over the Ahmed body with appropriate boundary conditions. Initially the computational domain are generated with design modeller and the meshing was done using ICEM CFD and the fluent solver was used for numerical simulation with post processing. Each and every contour are represented appropriately for both the model of 25° and 35° slant angle of Ahmed body. The K-Epsilon turbulence model was found as more accurate than K-omega and K-Omega SST for this simulation. The mesh independence study, turbulence study are made appropriately. The numerical results are compared with experimental results for validation and found some variation which is due to mesh and other boundary condition. The assessment of aerodynamic analysis over the Ahmed body is enumerated with appropriate learning outcomes.

References

- Emmanuel Guilminea, u., 2008. Computational study of flow around the simplified car body. *Journal of Wind Engineering and Industrial Aerodynamics*.
- Kalyan, D. K. & Paul, A., 2013. Computational study of flow around a simplified 2D ahmed body. *IJESIT*.
- Khan, R. S. & Umale, S., 2014. CFD Aerodynamic Analysis of Ahmed Body. *International Journal of Engineering Trends and Technology*, Volume 18.
- Landge, A. A. & Palande, D. D., 2016. Aerodynamics Study of Automobile Car Ahmed Body using CFD Simulation to Predict the Drag Coefficient and Down Forces. *International Engineering Research Journal*.
- Meile, W. et al., 2011. Experiments and numerical simulations on the aerodynamics of the Ahmed body. *ISSR Journals*.
- Parab, A. & Mangrulkar, A., 2014. Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5. *International Journal on Recent Technologies in Mechanical and Electrical Engineering*, 1(4).
- Rajsinh, C. B. & Raj, T. K. R., 2012. Numerical investigation of external flow around the ahmed reference body using computational fluid dynamics. *Research Journal of Recent Sciences*.