Ansys-CFX Analysis on a Ahmed Body

Roopsandeep Bammidi - M.Tech (CAAD); Dr.B.V.Ramana Murty- (Professor)
Gayatri Vidya Parishad College of Engineering (A), Madhurawada, Visakhapatnam-48.

*Abstract***:** **The Aerodynamic behaviour of the Ahmed body, which seems to be small trucks is investigated using Ansys-CFX analysis at two different slant angles 50° and 65°at front end of the body with each at different geometry. The analysis was done for the Ahmed body at a speed of 60km/h. The result shows that such as contours, vectors turbulence kinetic energy and also used to analyze the characteristics of streamlines flow or boundary layer that occurs on the body of this model and the variations in drag coefficient at two different slant angles considered for the same body. All the analysis and modifications have been carried out computationally in the CFD software "ANSYS CFX 15" and the part modeling in "CATIA V5R19".**

Keywords: Ahmed body; vehicle aerodynamics; drag coefficient; contours; vectors; turbulent kinetic energy; streamlines.

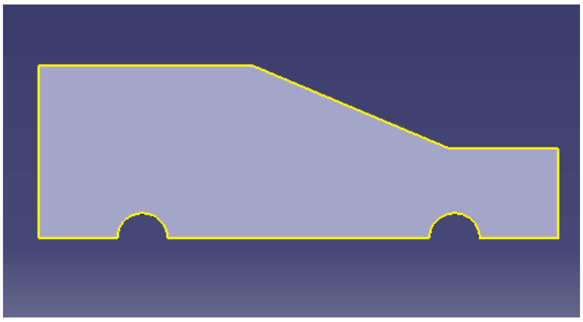
INTRODUCTION

The Aerodynamics has become crucial since have realized its importance. The study gives brief idea about characteristics of aerodynamics of simple automobile designs. The coefficient of drag varies from body to body; hence manufacturers allot big proportion of their attention to aerodynamics. To achieve high vehicle performance, much of the attention focuses on lowering the vehicle drag coefficient. Aerodynamics has a strong impact on body design. Two elements that have major

Influence on the drag coefficient of a bluff object is the roundness of its front corners and the degree of taper at its rear end.

Catia Model:

Model-1



Model-2

Dimensions of the Models:

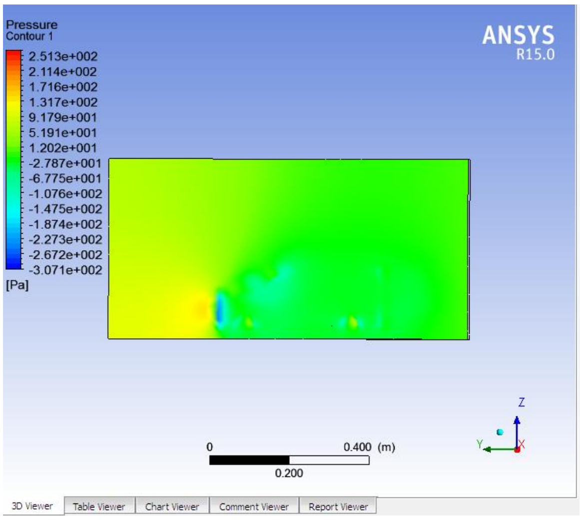
Dimensions	Model-1	Model-2
Length	**4138.48 mm**	**4216.40 mm**
Width	**1500.00 mm**	**1500.00 mm**
Height	**1364.95 mm**	**1397.00 mm**
Slant Angle	**50 degrees**	**65 degrees**

CFD SETUP: The boundary condition for the flow is also an important factor. Boundary conditions used for this analysis are given as follows:

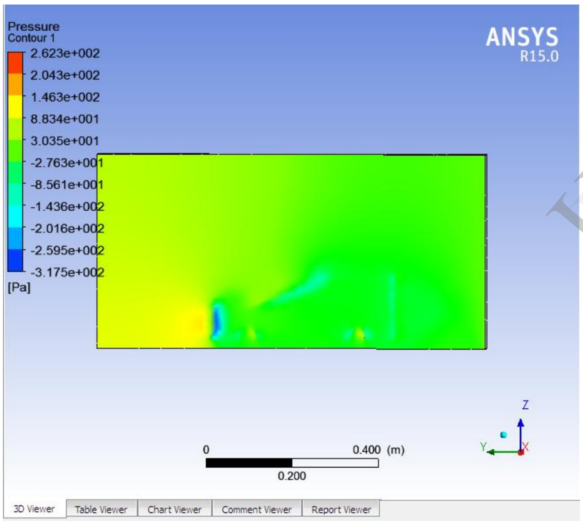
Velocity Inlet	**16.67 m/s**
Sides and Top	Symmetric Walls
Road	Wall

Analysis of the Designed Models: The analysis involves the detailed study of the basic models. And to read the flow and encounter the regions responsible for high drag is the main purpose of this step. This includes the Contours Vector Plots, Turbulent Kinetic Energy, Streamline Flows and Drag Coefficients of both models.

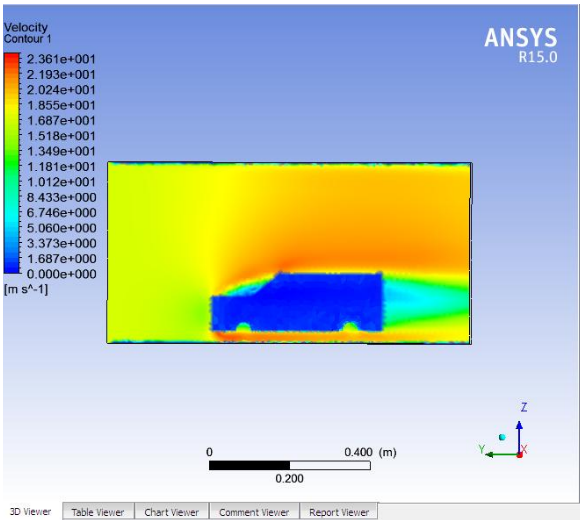
The Pressure and Velocity Contours of both the models are as follows:



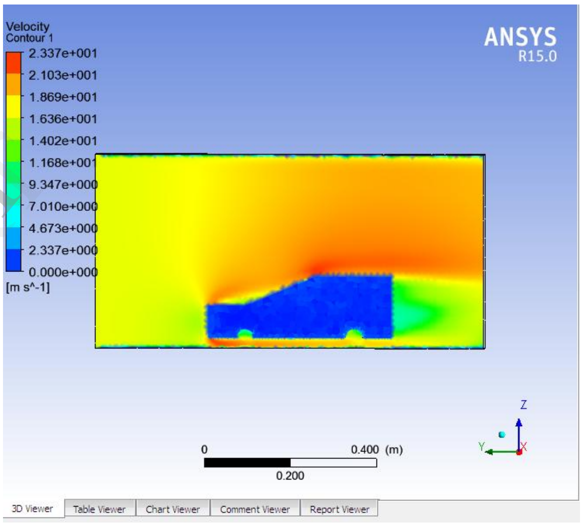
Pressure Contours of Model-1



Pressure Contours of Model-2

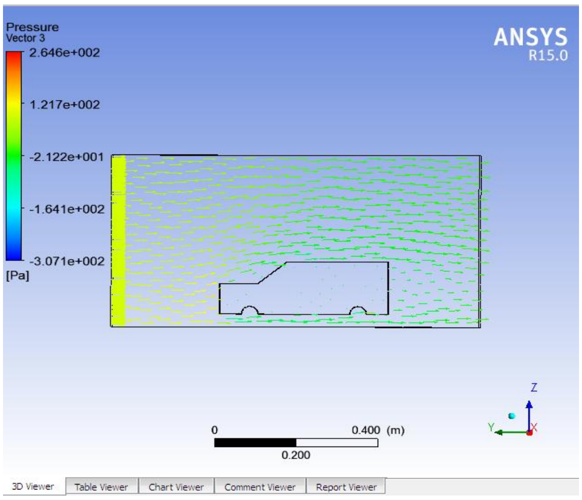


Velocity Contours of Model-1

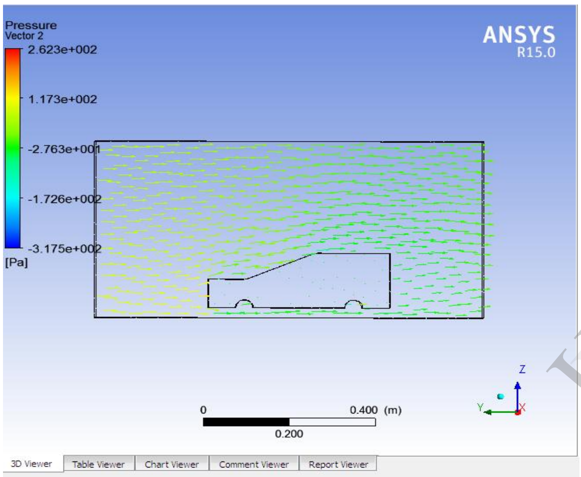


Velocity Contours of Model-2

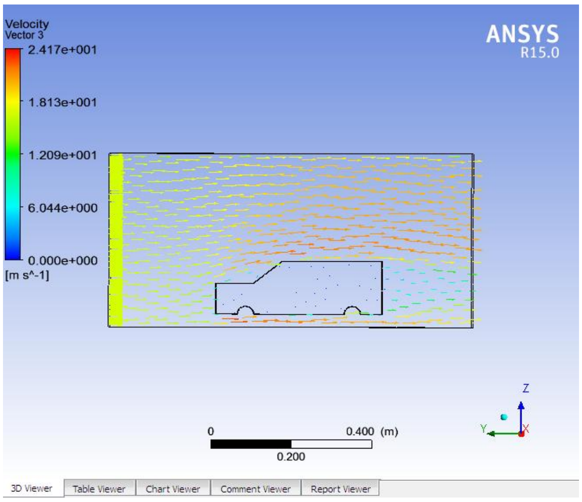
The Pressure and Velocity Vectors plots of both the models are as follows:



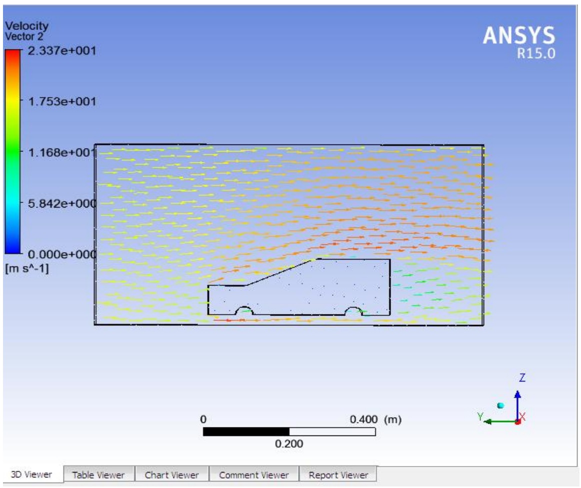
Pressure Vector Plot of Model-1



Pressure Vector Plot of Model-2



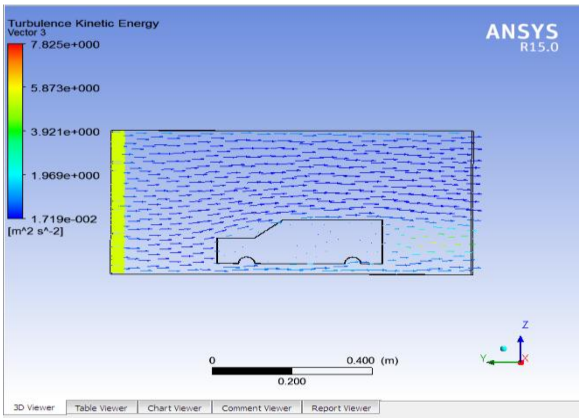
Velocity Vector Plot of Model-1



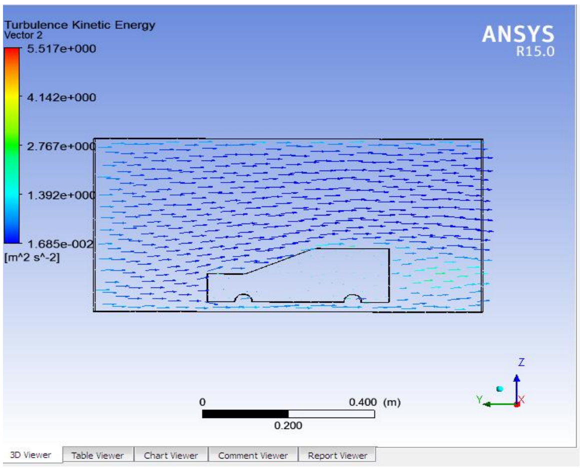
Velocity Vector Plot of Model-2

Due to the high mass flow rate and complex flow patterns associated with platoon study, the flow is said to be turbulent. And the flows are characterized by fluctuating velocity fields which the Fluctuations mix transported quantities such as momentum, energy, and species concentration, in a way that causes the transported quantities to fluctuate as well. Since these fluctuations are of small scale with high frequency and they are too computationally expensive which simulates directly in practical engineering calculations.

The Turbulent Kinetic Energy of both models are as follows:

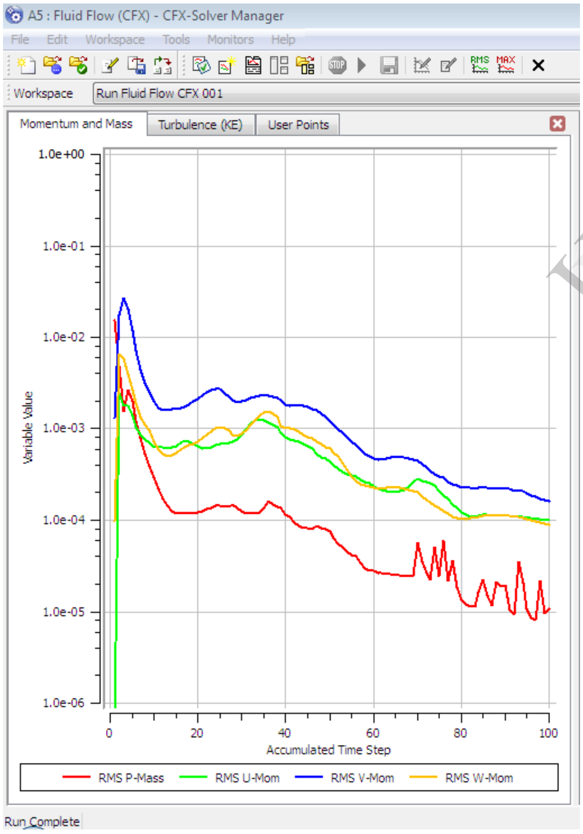


Turbulent Kinetic Energy of Model-1

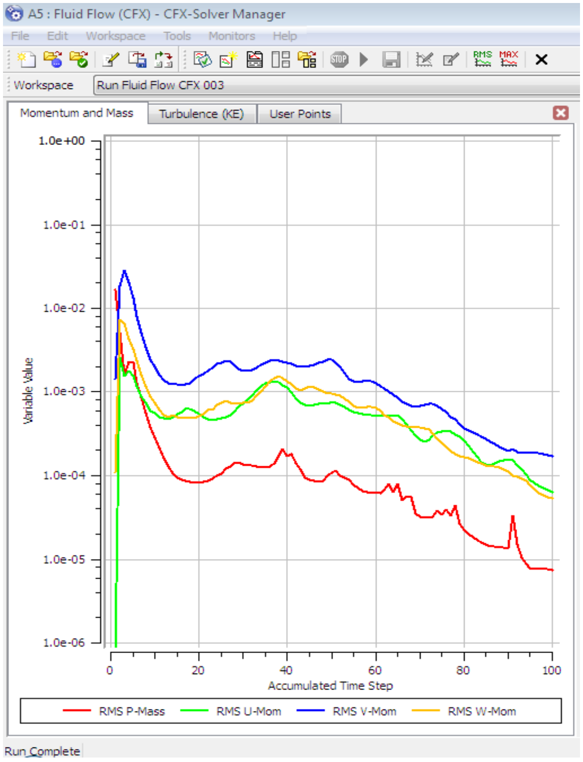


Turbulent Kinetic Energy of Model-2

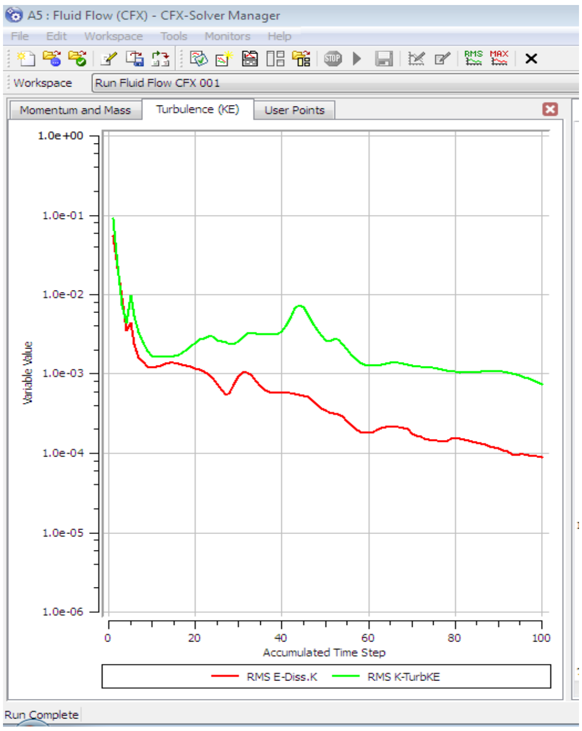
The graphs associated with these models such as Momentum and Mass, Turbulence, and Drag Coefficient are given below:



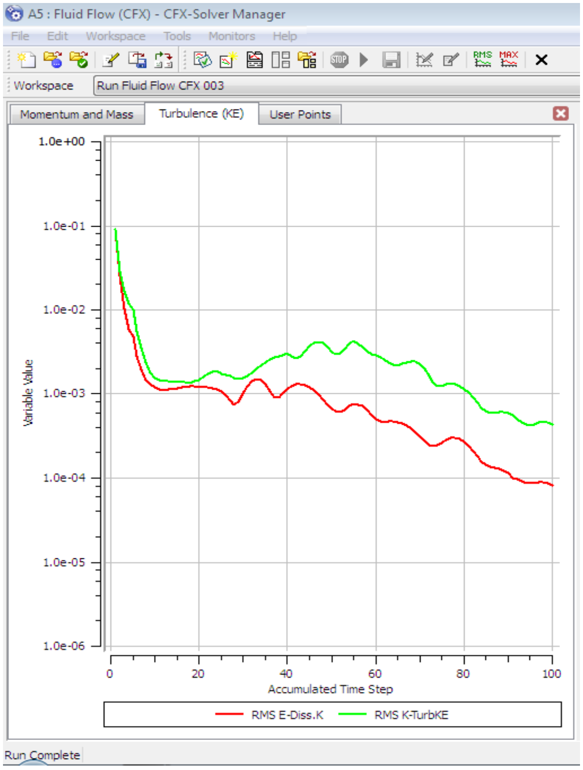
Momentum and Mass graph of Model-1



Momentum and Mass Graph of Model-2



Turbulence graph of Model-1

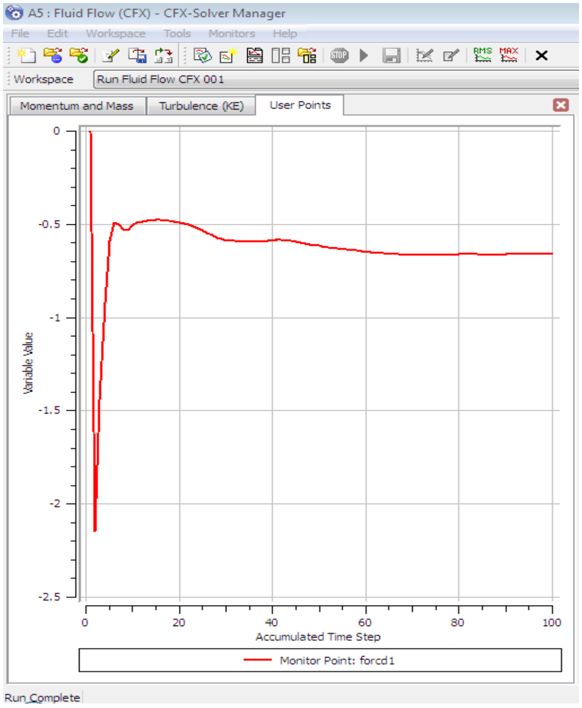


Turbulence graph of Model-2

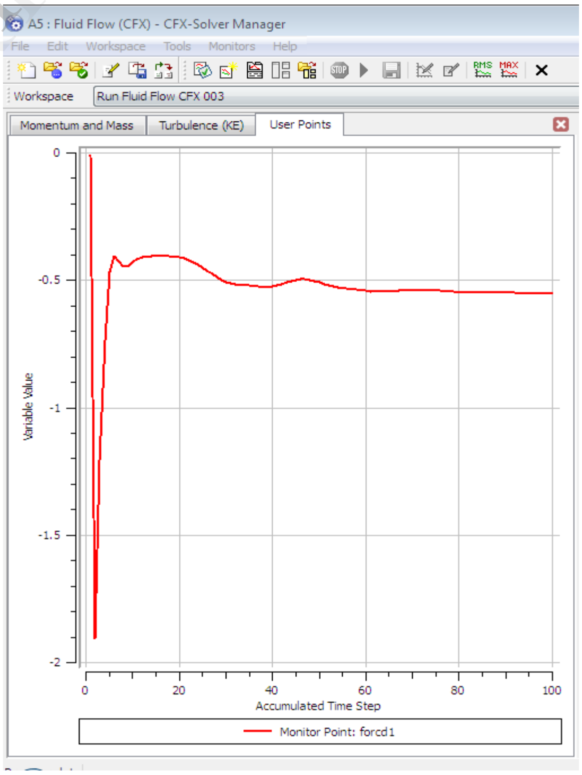
The study of aerodynamics involves the drag force that opposes the motion of a car and the fuel consumption of car has become crucial to think about as emission of CO2 in the environment which shows negative consequences. And also there are many regions in the car that contributes in increasing the drag coefficient. So, in this paper concentrated on slant angles at front end which would variant the drag coefficient. This results the best model from Ansys-CFX Analysis.

The major factors that impact on drag is the frontal area of the body. The drag reduction is essential in improving fuel efficiency and protecting the global environment and driving performance. And the body also attains top speed for a criterion given power.

The drag coefficient graphs of both the models are shown below:



Drag Coefficient Graph of Model-1

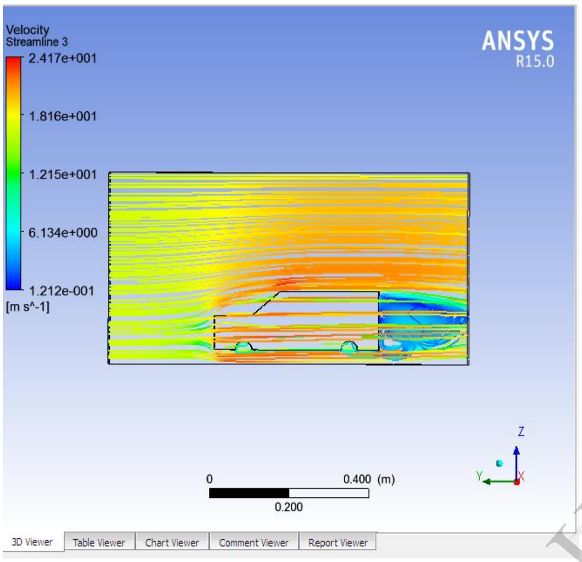


Drag Coefficient Graph of Model-2

NOTE: The variable values in drag coefficient graphs are in negative as the front end of model and the domain are in opposite facing. So, values are negative, neglecting the facing and the values obtained are in positive which the drag coefficient for these simple models are 0.659010 and 0.551017.

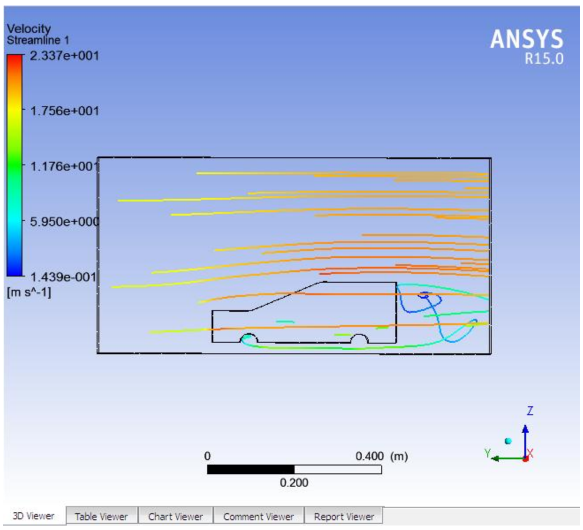
Ahmed Body	Drag Coefficient
Model-1	0.659010
Model-2	0.551017

The Velocity Streamlines of both these models are shown below:



Velocity Streamlines of Model-1

By these streamlines around the model gives the formation of high pressures on the surface of the model and the wakes formation at the rear end of the model.



Velocity Streamlines of Model-2

CONCLUSION

The Aerodynamic study on these two Ahmed bodies of simple automobile designs with different geometrical dimensions. The slant angle of Model-1 and Model-2 are 50 and 65 degrees respectively. The best lowest drag coefficient is 0.551017 of Model-2 from Ansys-CFX analysis gives better fuel improvement and protects the global environment and also Model-2 provides less wake formation at rear end when compared with Model-1.

Further Work: As Aerodynamically Designed bodies have less drag, faster and more fuel efficient and globally protect high. The models are to be further designed which includes roundness at front and rear ends to gain aerofoil nature and would be to achieve more Aesthetic Design and to have highly handling performance.

REFERENCES

1) Wolf-Heinrich Hucho.(1998), Aerodynamics of road vehicles, IV edition, Publisher- SAE International.
2) Mohd Aliff Bin Mohd Nor and Aswatha Narayana, CFD Study of Rear Spoiler Fitted to Sedan Car, Technical report.
3) Bhagirath zala M.E. thesis, "Aerodynamic performance of Sedan and hatchback car by experimental method and simulation by computational fluids dynamics" Gujarat Technical University 2011 .
4) http://www.autozine.org/technical_school/aero/tech_aero.html.
5) http://www.up22.com/Aerodynamics.html

AUTHORS

<u>Roopsandeep Bammidi</u>

Pursuing M-Tech in Computer Aided Analysis and Design, Department of Mechanical Engineering, GVP College of Engineering(A), Madhurawada, Visakhapatnam.

<u>Dr.B.V.Ramana Murty</u>

Professor, Department of Mechanical Engineering, GVP College of Engineering(A),Madhurawada, Visakhapatnam.