International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Impact Factor (2012): 3.358

Ansys-CFX Analysis on a Simple Automobile Design

Roopsandeep Bammidi[1], B. V. Ramana Murty[2]

[1]M. Tech (CAAD), Gayatri Vidya Parishad College of Engineering (A), Madhurawada, Visakhapatnam, India

[2]Professor, Gayatri Vidya Parishad College of Engineering (A), Madhurawada, Visakhapatnam, India

Abstract: ***Aerodynamics is the study of the effects of air when in motion. The vehicle aerodynamics have become crucial since have realized its importance. The physical appearance of the car has been given keen significance while designing the external devices. Aerodynamics drag was considered for giving a better efficiency on the road. The analysis was done for the simple automobile design at a speed of 60km/h. The technique to estimate the drag is by using input of Catia part modeling and exporting it to Ansys CFX to find the value of aerodynamics drag in terms drag coefficient. 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. All the analysis and modifications have been carried out computationally in the CFD software "ANSYS Fluent 15" and the part modeling in "CATIA V5R19".***

Keywords: modeling, Ansys-cfx, FEM, contours, vectors and drag coefficient.

1. Introduction

This study can be regarded as independent study to get the aerodynamics characteristics of the simple automobile design and involves the drag force that opposes the motion of a body. The coefficient of drag varies from body to body; hence manufacturers allot big proportion of their attention to aerodynamics. A taper, boxed shape has been designed for the analysis. The model of the body has been designed using Catia V5R19 and undergoes CFX analysis to have contours, vectors and drag coefficient which it involves for better fuel performance. Domain is a requirement for external flows analysis. It is a box filled with fluid and has boundaries. These boundaries are given conditions and the fluid is given a motion. The designed part model is as follows:

Figure 1: Catia Modeling of a simple Automobile Design

2. CFD SETUP

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

Table 1:

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

The geometry of the model is given in the following table:

Table 2

Length	4138.48 mm
Width	**1500 mm**
Height	**1364.95 mm**
Slant Angle	**50 degrees**

The drag coefficient found for this model through Ansys-CFX analysis is as follows:

Table 3

Drag Coefficient	0.65901

3. Analysis of the Designed Model

The first approach for the analysis involves the detail study of the basic model. And to read the flow and encounter the regions responsible for high drag is the main purpose of this step. The analysis of the basic shape presents the value of drag coefficient as 0.65901. The coefficient of drag obtained gives an idea about the behavior of car at the speed of 16.67 m/sec. The image following show the flow of air around the model and further gives the detailed analysis of the regions strengthening the drag coefficient. The front part of the model stagnates the air hence increases the Pressure and this causes the good proportion of high pressure region. At the inclined front windscreen due to sudden high angle the pressure also seems to be increased to the level of yellow region that is to be considered as high pressure. Across the windscreen at the top the velocity increases which causes reduction in pressure. And at immediate end of the rear declination which angle is such like to be blunt body n which the pressure increases due to flow separation occurring. In order to decrease the drag coefficient these high regions are to be considered for better drag coefficient.

The pressure contours and velocity contours are as follows:

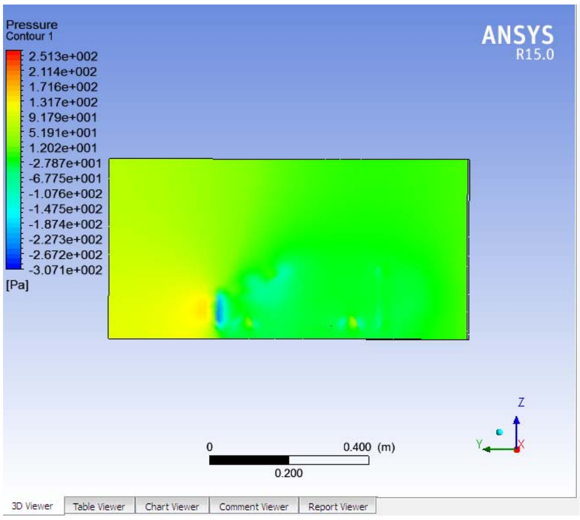


Figure 2: Pressure Contour around the model

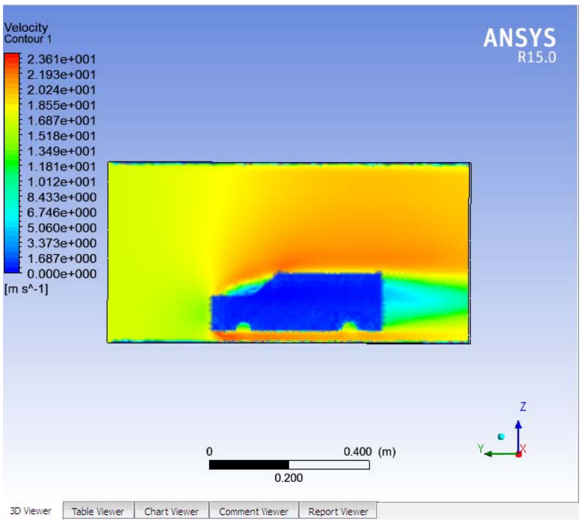


Figure 3: Velocity Contour around the model

The pressure and velocity vectors are as follows:

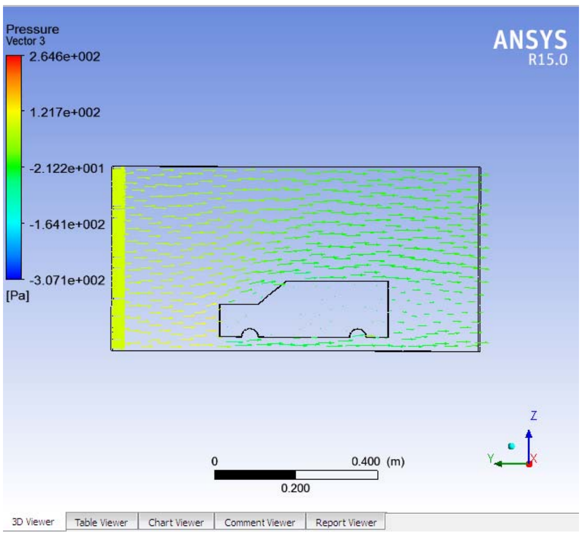


Figure 4: Pressure Vectors over the model

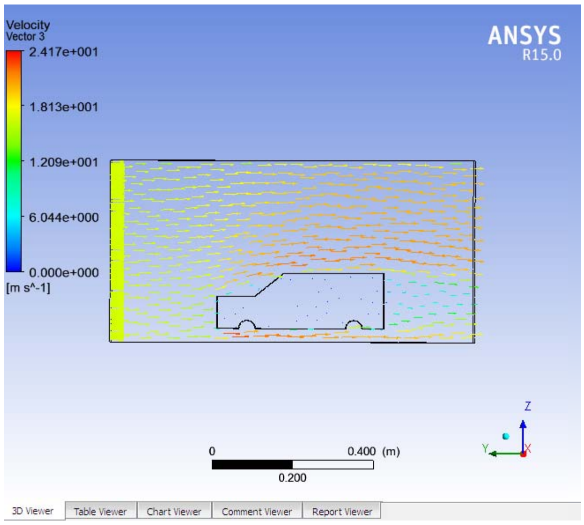


Figure 6: Velocity Vectors over the model

Turbulent 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. Due to the high mass flow rate and complex flow patterns associated with platoon study, the flow is said to be turbulent.

The turbulent kinetic energy over the surface of the model is as follows:

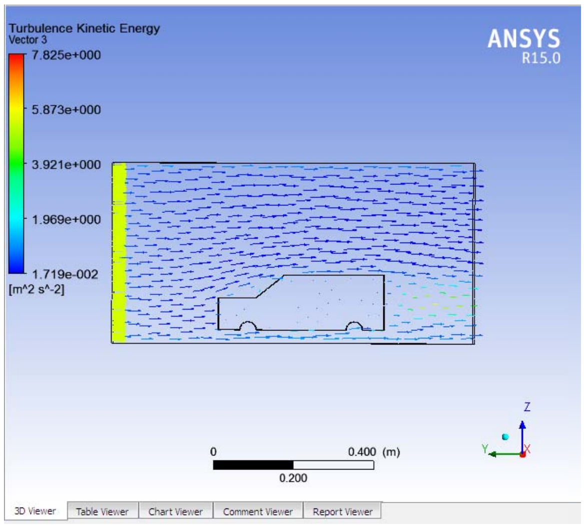
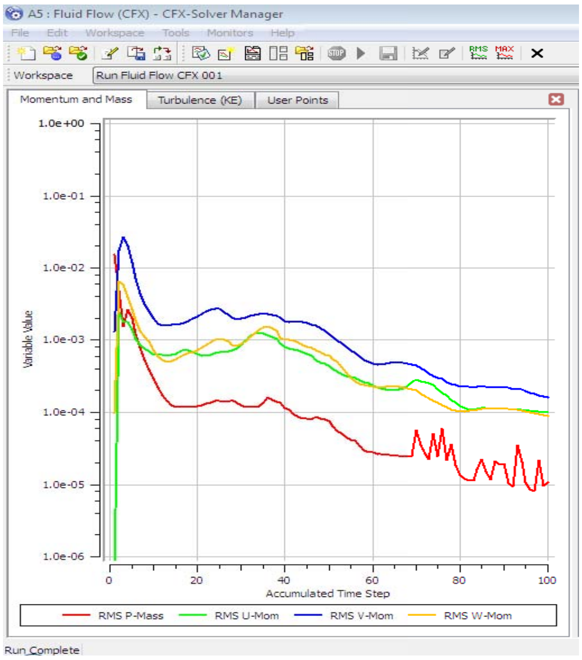
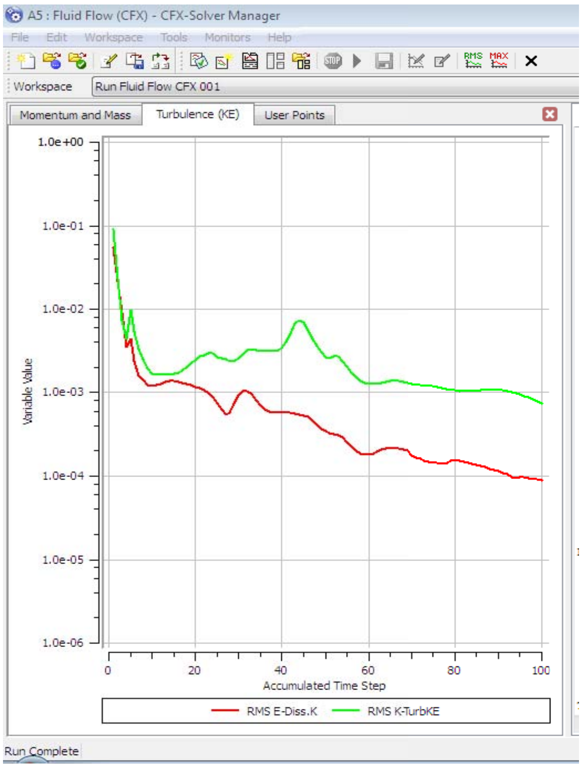


Figure 6: Turbulent Kinetic Energy

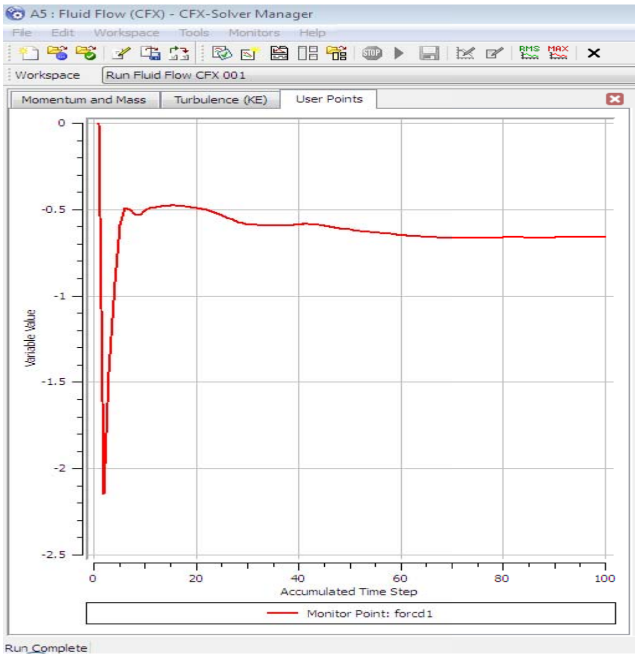
The characterized study of CFX Analysis of the simple automobile design which the graphs associated with it such as Momentum and Mass, Turbulence, and Drag Coefficient are as follows:



Graph 1: Momentum and Mass



Graph 2: Turbulence



Graph 3: Drag Coefficient

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 this simple model is 0.65901.

The streamlines presented following shows the Working of the tool, which reduces the high pressure region as the drag coefficient obtained is 0.65901. 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.

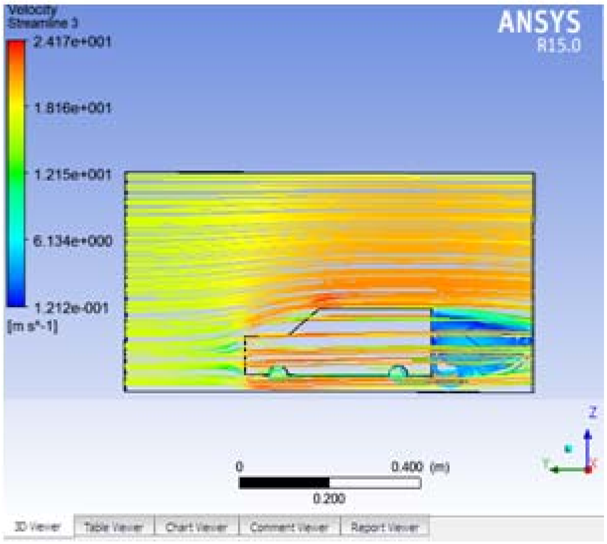


Figure 7: Streamlines around the surface of the Model

4. Conclusion

The Aerodynamics study of a simple automobile design results in efficient drag coefficient which the fuel consumption can be improved as there would be less opposing force acting on this simple blunt body. The drag coefficient for a standard design of a blunt body would be between 0.5-0.8. For this simple design seems to be like a blunt body of small truck shaped got a drag coefficient of 0.65901 which is in between the standard value and can say this blunt model gives a better fuel consumption.

References

[1] Mohd Aliff Bin Mohd Nor and Aswatha Narayana, CFD Study of Rear Spoiler Fitted to Sedan Car, Technical report.
[2] Wolf-Heinrich Hucho.(1998), Aerodynamics of road vehicles, IV edition, Publisher- SAE International.
[3] Masaru Koike; Tsunehisa Nagayoshi and Naoki Hamamoto. (2004) No. 16, Research on Aerodynamic Drag Reduction by Vortex Generators, technical papers.
[4] http://www.up22.com/Aerodynamics.html.
[5] http://www.autozine.org/technical_school/aero/tech_aero.html.

Author Profile

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

Dr. B. V. Ramana Murty is working as Professor, Department of Mechanical Engineering, GVP College of Engineering (A), Madhurawada, Visakhapatnam, India