International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 4, Issue 11
November 2017

Flow Analysis on 2-Dimensional Sports Car Model using OpenFOAM

Dr.P.V.Selvan
Associate Professor
Department of Aeronautical Engineering
Sri Ramakrishna Engineering College,Coimbatore-22

Abstract

Aerodynamics plays an important role in designing any automotive. Because of aerodynamic properties, the entire performance will be altered. In this project, a car Model is took into concern and it is external design is modelled using CATIA V5R19 modelling software. The modelled car is considered as a 2D model for future analysis using OpenFOAM to determine the aerodynamic characteristics like Pressure, Velocity, Down-Force and Drag. Objective of this project is to Study the aerodynamic properties of the car and become familiarise with the Software.

Keywords- OpenFOAM, CATIA V5R19, 2D Sports car, Drag, Vortex

INTRODUCTION

Aerodynamics is a vast field in any core of engineering. As the model to be analysed gets complex, the computation process increases. Manual computation gets difficult when its complexity increases. Therefore, we are in need to move to machine computing as a result of analysis. All the mechanical industries study the aerodynamic loads on their vehicles by analysing computationally and experimentally.

Fuel economy has become latest topic of discussion among the owners of the respective vehicle. Here we need to ensure the fuel stability and also to ensure the stability with good handling characteristics of vehicle with higher speed is needed.

Drag

It is common experience that a body meets some resistance when it is forced to move through a fluid, especially a liquid. Drag is usually an undesirable effect, like friction. A dimensionless quantity that is used to quantify drag of an object in a fluid environment is called “Drag coefficient”.

Drag: $D = 0.5\ \rho * V2 * A * C_d$

V---Velocity of air in m/s **ρ ---Density of Air in Kg/m^3**

A---Area of the Object in m^2 **C_d ---Coefficient of Drag (No units)**

Vorticity

Vorticity is defined not simply as a scalar quantity, but as a vector. It is a tendency of the fluid particle at particular velocity. A **vortex** is a region in a fluid in which the flow rotates around an axis line, which may be straight or curved .The plural of *vortex* is either **vortices** or **vortexes**

International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 4, Issue 11
November 2017

OpenFOAM

OpenFOAM (Open source Field Operation and Manipulation) is a kind of CFD software embedded with C++ program for solving practical applications, and pre/post processing utilities for the solution of continuum mechanics problems, including Computational Fluid Dynamics (CFD). The code is released as free and open source software under the GNU General Public License

Uniqueness of OpenFOAM

- No license costs
- Fully documented source code
- Highly secured as its working platform is in LINUX

Analysis Procedures

Fluid: The working fluid is Air and it has a Density of 1.225 kg/m^3 and Dynamic Viscosity is 1.983e-5pa.s.

Pre-Processing:

The modelling of the car is done in CATIA V5R19 and the concern file is then converted to a mesh file in Salome MECA software. Then the mesh is imported by introducing a command "fluentMeshtoFoam car.msh"

Processing and Analysing:

- ✓ Change the Initial and Boundary Conditions based on patch type
- ✓ Create a New "polyMesh"
- ✓ In Velocity- Wall & Inlet has Fixed Value, remaining all are Zero Gradient
- ✓ In Pressure-Outlet is Fixed Value, remaining all are zero Gradient
- ✓ Here Top Surface of domain under "out flow" boundary condition so all are zero Gradient
- ✓ To run the Solver command is Solver name i.e. simple

Post processing:

- ❖ To open the Paraview, command paraFoam -builtin

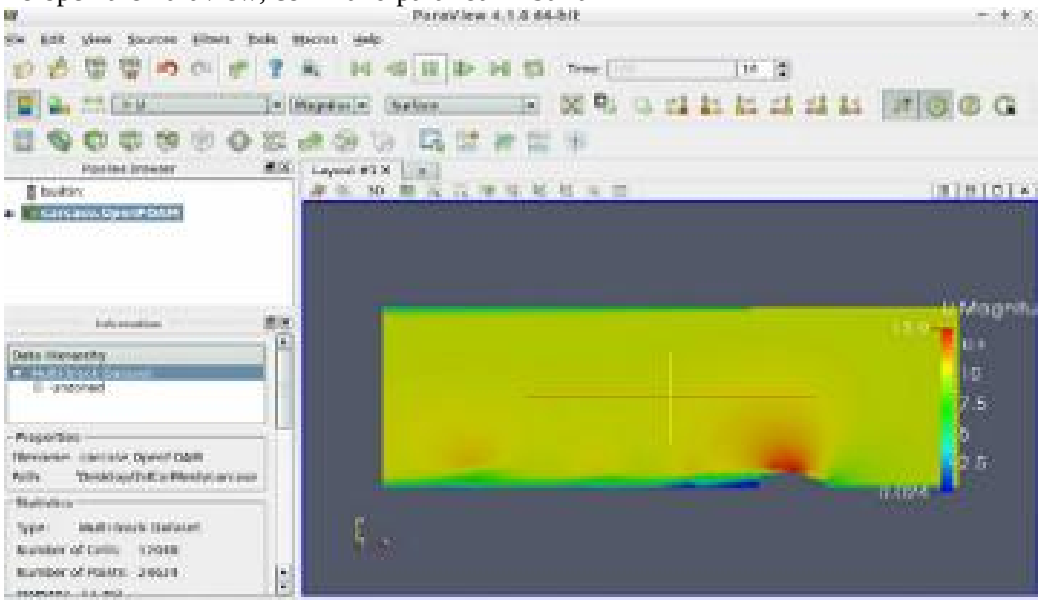

Fig 1. Pressure Distribution Over 2D Car Mesh

International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 4, Issue 11
November 2017

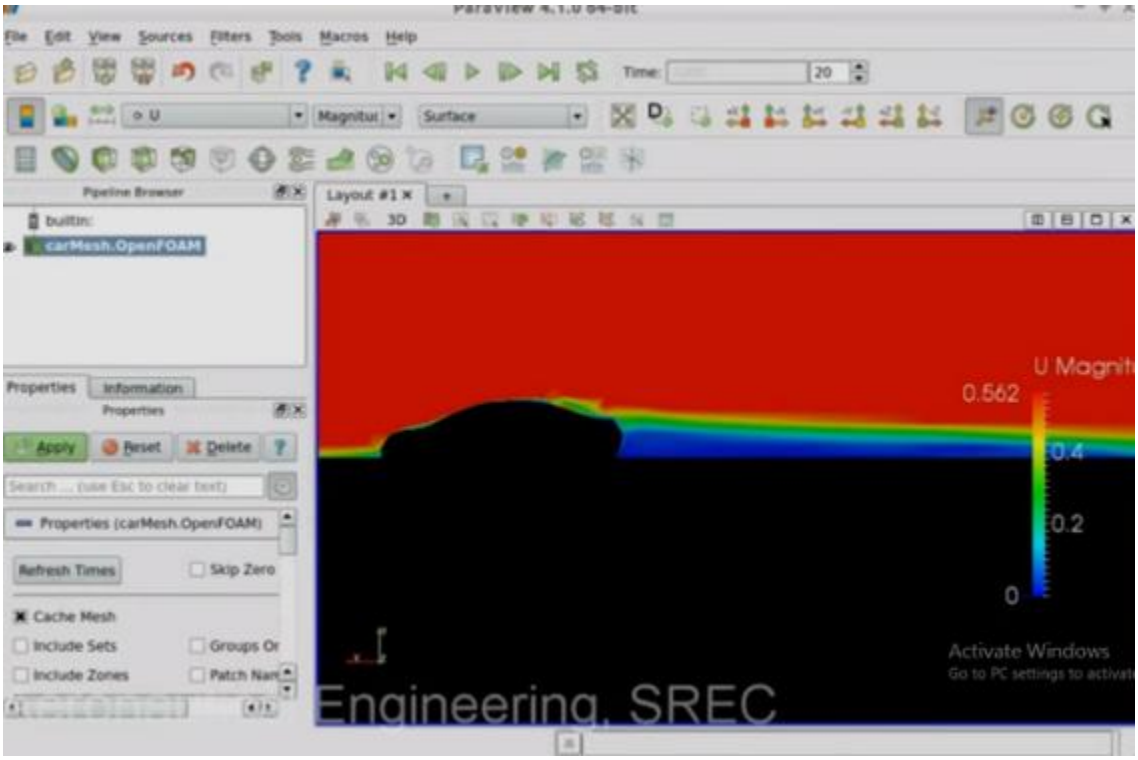

Fig 2. Velocity Distribution over 2D Car Mesh

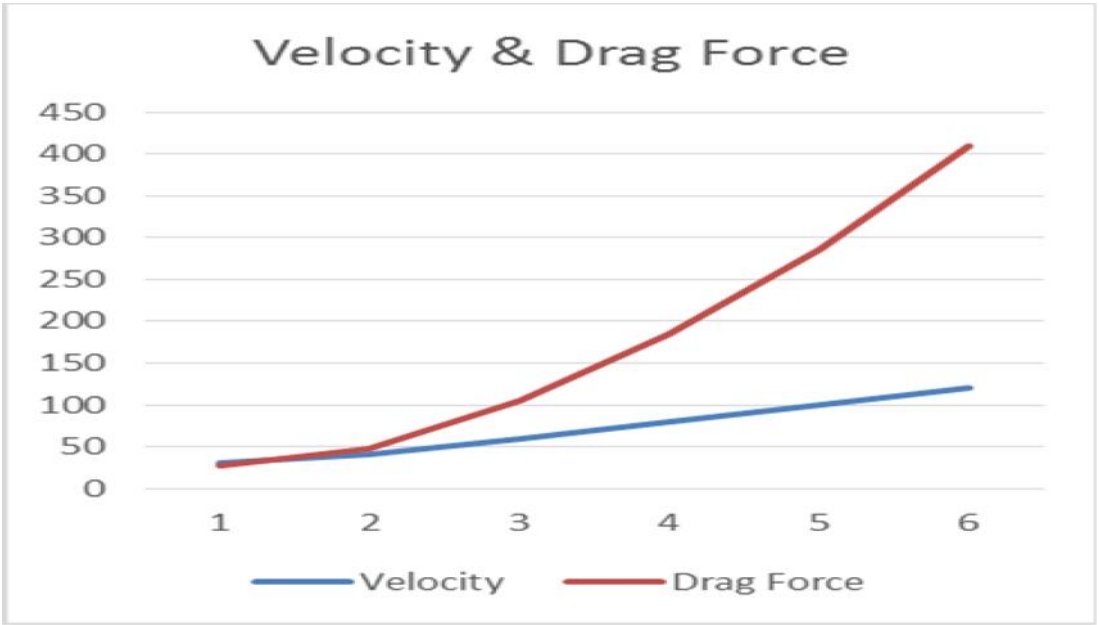

Fig 4.Graph between velocity and its Corresponding Drag

Flow Analysis on 2-Dimensional Sports Car Model using OpenFOAM

International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 4, Issue 11
November 2017

CONCLUSIONS

Thus, the effect of drag force is experimentally viewed and analysed. The very basics and some of the advanced tools in OpenFOAM are studied by this experiment. Hence forth the above formula is satisfied as the Velocity is directionally proportional to the Drag. As per the graph, the result matches with the corresponding physics

ACKNOWLEDGMENT

I would like to manifest my credit of my project to my guide Dr.R.Sundararajan, whose encouragement, counsel and reinforce me to develop an understanding of this subject.

Finally, I take this opportunity to extend my deep cherishing and treasuring to my family and friends, for all that they meant to me during the climacteric times of my project.

References

[1] Toukir Islam, S.M. Rakibul Hassan and Dr.M.Ali "Aerodynamic Drag of Racing Cars". Proceedings of the Global Engineering, Science and Technology Conference 2012 ,28-29 December 2012, Dhaka, Bangladesh.
[2] Dinesh Dhande and Manoj Bauskar, 2013. "Aerodynamic Analysis Of Sport Utility Vehicle By Using Computational Fluid Dynamics Approach" in International Journal of Engineering Research & Technology
[3] C.N.Patil, Dr.K.S.Shashishekar, K Balasubramanian and Dr.S.V.Subbaramaiah, "Aerodynamic Study and drag coefficient optimization of passenger vehicles".
[4] L J Clancy M.Sc., F.I.M.A , "Aerodynamics" Indian Edition 2006, Sterling Book House Publications
[5] Dr. R.K. Bansal B.sc Engg(Mech.), M.Tech. Hons. (IIT., Delhi), Ph.D., M.I.E.(India) "Fluid Mechanics and Hydraulic Machines"