© 2016 IJSRSET | Volume 2 | Issue 6 | Print ISSN: 2395-1990 | Online ISSN : 2394-4099
Themed Section: Engineering and Technology

Visualisation of Flow Pattern around a Car Model Using Wind Tunnel and CFD Simulation

Chandrashekar M

Department of Mechanical Engineering, Dr Ambedkar Institute of Technology, Bengaluru, Karnataka, India

ABSTRACT

To analyse the behaviour of air flow around a car model under varying conditions for the purpose of finding the different forces acting on the model, definitely wind tunnel act as a tool. It is very difficult to determine the various forces in the real car moving with certain speed through the air, due to lack of technical feasibility and uneconomical also. Wind tunnel is a device to create real situation in the laboratory i.e. relative motion between a car model and air flowing around it by keeping the model as stationary object and blowing the air through the model in the wind tunnel. This is as good as a car moving with certain speed on the road. The visualisation of air pattern and analysis of the test results will considerably reduce cost and time of producing the prototype.
Keywords : CFD, Wind Tunnel, CATIA, LED, ABAQUS, Fluid Dynamics Analysis

I. INTRODUCTION

Wind tunnel enable researchers to study the flow pattern over the objects of interest so that forces acting on them and interaction with flow can be easily studied. Controllable varying conditions in the test chamber of wind tunnel are flow speed, flow uniformity and turbulence level. For example, requirements of these conditions are very much appreciated when laminar boundary layers have to be investigated with low turbulence and high uniformity in the flow.

CATIA is used for modelling and analysis of given car model, with the help of various tools in it and thus car is designed part by part and is assembled finally for CFD analysis. Computational Fluid Dynamics is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyse problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of wind with surfaces defined by boundary conditions.

II. METHODS AND MATERIAL

Experimental Procedure:

Open circuit subsonic wind tunnel is used in this experiment, it consist of settling chamber with contraction cone, test section of size (240×120×120) mm, diffuser and sucking fan as shown in the Fig 1.

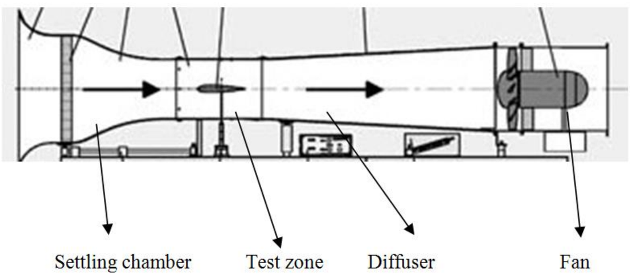

Figure 1

A required model of car with various instruments and sensors kept in the test zone and is freely suspended so that flow around is not disturbed. Fan sucks the air from the atmosphere, honey comb structure in the settling chamber straightens the air flow and as it flows through the conical section of it, flow velocity increases (max wind velocity is 12.5 m/sec). Stream of air with minimum turbulence enters the test chamber with inside pressure of 110 Pa, where scaled down test model is placed as the air flows around it sensors record the data and visual observations are made. Subsequently air flows through diffuser in order to exit with low velocity without causing turbulence in the test zone. A fan is mounted in the downstream of the test zone; it sucks the air rather than blowing the air into the test zone to avoid possible turbulence. Flow visualisation is done by observing the pattern of smoky wind flowing over the required model with the help of LED fixed in the test zone and flow pattern is recorded for further analysis.

To compare flow pattern around the model of car, Computational Fluid Dynamics analysis is done using ABAQUS by importing the car model on to the software generated by CATEIA. Fig 2 shows flow pattern around the car model kept in the test zone. Fig 3 shows flow velocity using CFD. Fig 4 shows flow pressure around the car model using CFD.

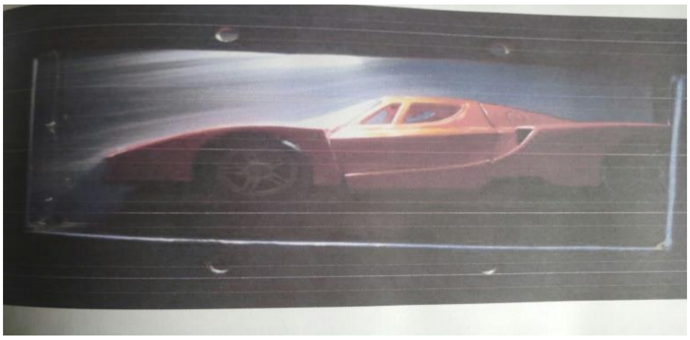

Figure 2

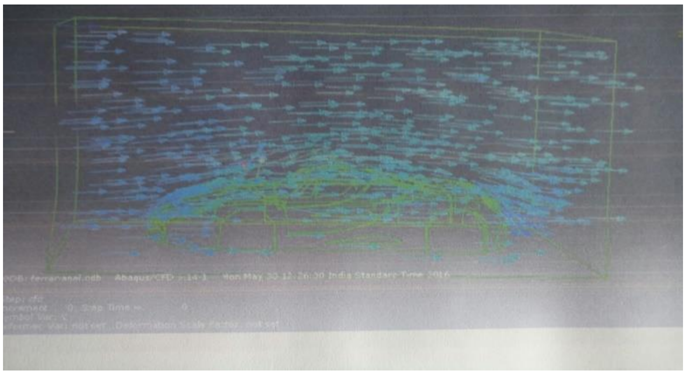

Figure 3

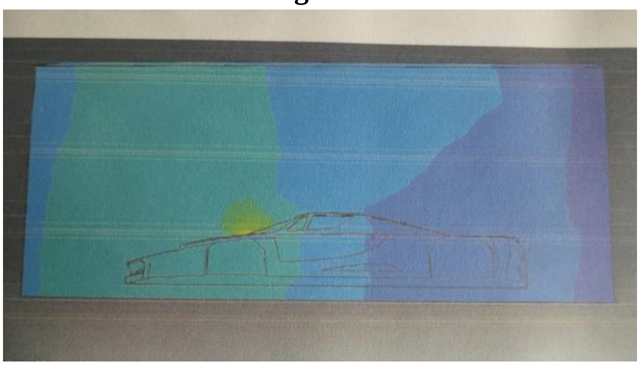

Figure 4

III. RESULTS AND DISCUSSION

On comparison, Flow visualisation (Fig 2) and Velocity and pressure distribution (Fig 3 & 4) obtained by CFD shows a common point of critical zone where more turbulence is observed, hence importance for design should be given before going for prototype of the given car model.

IV. REFERENCES

[1]. A Project Report on "STUDY OF WIND FLOW PATTERN USING A LOW SPEED WIND TUNNEL" submitted by Aakash Pattankar & Suhas Halemane J.

[2]. www.wikipedia.com