© 2016 IJEDR | Volume 4, Issue 2 | ISSN: 2321-9939

Experimental and numerical aerodynamics investigation of car

[1]Sagar Bhamre, [2]Mayur Makwana,
[1]PG student, [2]Assistant professor,
[1]Department of mechanical engineering,
[1]S.V.M.I.T, Bharuch, India

***Abstract*- This project work is carried out for the aerodynamic analysis of a conceptual car. There are two methods to analyze the aerodynamic features of vehicles and importantly the turbulence: the wind tunnel and computational fluid dynamics (CFD). Modeling of the car is done in CATIA V5 R19 software. Most of the concentration is given to the exterior design of the car, while the interior is not modeled. Further, using the software ANSYS Fluent, simulation of the airflow around the sides of the vehicle is made in purpose of making changes in the outer body of the vehicle to improve design in terms of reducing air resistance and improving aerodynamics. Most of the concentration is given by changing the value of angle between the hood and front windshield of the car, and analyzing the back of the car with and without rear spoiler. Continuing with the obtained simulation and leading with the modification of initial model, an existing model is redesigned. Further, simulation is done on the modified model. The results obtained by simulation in software are compared and analyzed. According to my assumption, redesigned model will have better aerodynamic properties.**

***Index Terms* – Automobile, Computational fluid dynamics, Vehicle.**

I. INTRODUCTION

Aerodynamics is the study of solid body moving through the atmosphere and interaction which takes place between body surface and surrounding air with varying relative speeds and wind direction. Aerodynamic drag is not applicable at low vehicle speed but magnitude of air resistance become considerable with increase in speed. Aerodynamics causes greater impact on cars and trucks through its contribution to road load. Vehicle aerodynamics has greater impact on fuel economy, vehicle performance, reduction in wind noise level, improved road holding and stability of vehicle. The aerodynamic forces produced on a vehicle arise from two sources drag and viscous friction. The main concern of automotive aerodynamics are reducing drag, and preventing undesired lift forces at high speed. So in this project, simulation of the airflow around the sides of the vehicle was made in purpose of making changes in the outer body of the vehicle to improve design in terms of reducing air resistance and improving aerodynamics. For this study a sports car concept has been model in modeling software and this model was simulated in ANSYS Fluent. The model simulated will give the values of various aerodynamics loads and properties. From the above obtained values changes in the geometry will be made by using various aerodynamic aids and drag reduction properties.

II. OBJECTIVE

The objective of the present work is to design, analyze, and compare the results which are obtained by simulation in CFD software and testing the prototype in wind tunnel. Modeling of the car is done in CATIA V5 and this model is analyzed in ANSYS Fluent. The result obtained by simulation helps to know the Drag Coefficient of the model. Further, modifications are made in the geometry by using various aerodynamic aids to reduce air resistance and improving aerodynamics.

III. METHEDOLOGY AND VARIOUS STEPS

Following are the steps taken for setting up the solving techniques in solver for 3D, steady state, incompressible flow using double precision and serial processing technique.

- Imports the meshed model created in Workbench and check the mesh.
- Scale the mesh as per the dimensions specified with appropriate scaling factor.
- The solver specifications such as pressure based, transient, planner and absolute velocity formulation are set.
- Defining models such as viscous model (standard k-ε model) is done.
- The model constants in k-ε equation are:
- C_μ=0.09 (model constant for Turbulent viscosity)
- $C_{2\varepsilon}$ = 1.92 (model constant for transport equation)
- σ_k = 1.0 (Turbulent kinetic energy Prandtl number)
- σ_ε = 1.3 (Turbulent dissipation rate Prandtl number)

IV. SPECIFYING PROPERTIES

- Fluid: Air
- Density = 1.225 kg/m^3
- Viscosity = 1.7894e-05 kg/ms

- Solid: aluminum
- Density = 2719 kg/m^3
- Pressure- Velocity Coupling
- Scheme- Coupled
- Spatial Discretization
- Gradient- Least square cell based
- Pressure- Standard
- Momentum- second order upwind
- Turbulent kinetic energy- second order upwind
- Turbulent dissipation rate- second order upwind

V. SETTING UP SOLUTION CONTROLS AND UNDER RELAXATION FACTORS

Under Relaxation Factors:

- Density= 1.0
- Body Force= 1.0
- Turbulent Kinetic energy = 0.8
- Turbulent dissipation rate = 0.8
- Turbulent viscosity = 1

VI. INITIALIZE SOLUTION

Solution is initialized from inlet region with set initial values.

VII. RUN CALCULATION

Number of iterations is set to 500 with reporting interval of 50.

VIII. GRID INDEPENDENCE TEST

Mesh was locally refined in regions that are important and coarser mesh was used at less relevant places to reduce the computational expense with sufficient number of grid needed to be solving the physics accurately. For grid generation Ansys meshing is used. In this work a tetrahedral mesh is used. Mesh independence analysis is usually conducted by considering that finer meshes produce better results due to the discretization error, which decreases with cell size. The selection of element is based on the grid independence test. The convergence criteria of the solution is 10^3. To ensure that the numerical results were independent of the mesh density, mesh independence tests were performed using three computational grids. Three computational grids with different element sizes in the mesh refinement region as shown Figure were used to estimate the drag coefficient, Cd. Coarse mesh of approximately 4.5 million elements, medium mesh of 7.5 million elements and fine mesh of 10 million elements were used.

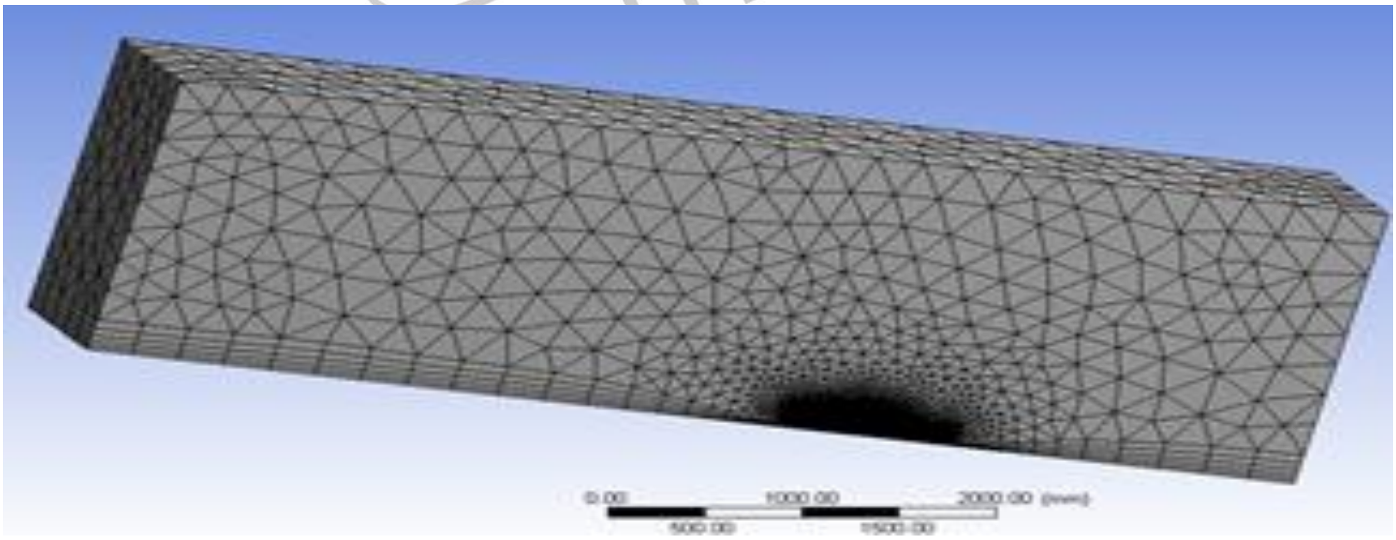

Fig 1- Discretization of car

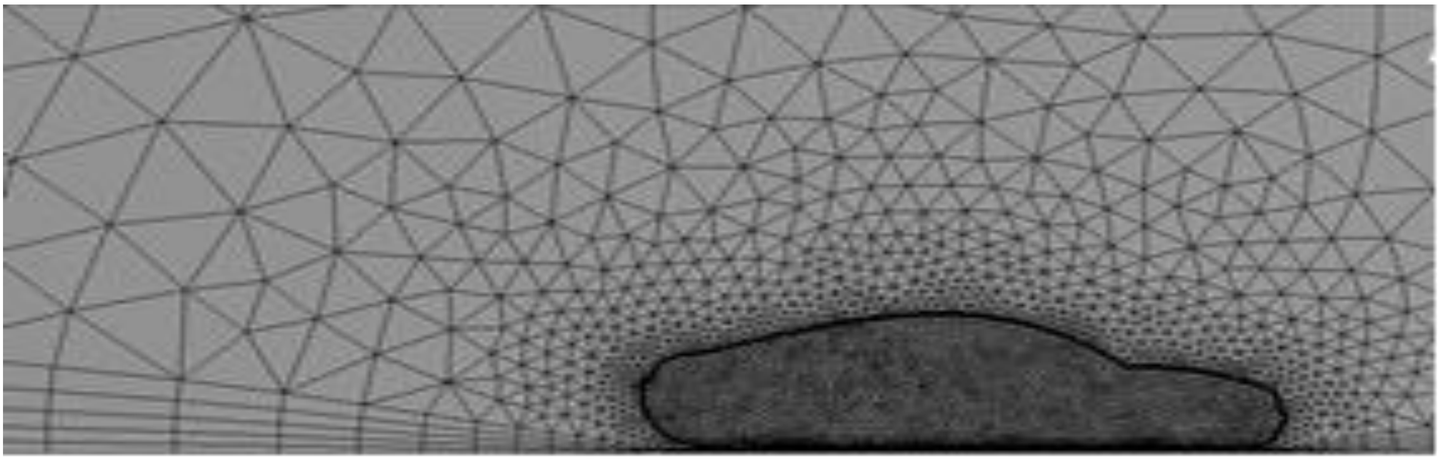

Fig 2- Grid refinement region

Table 1- Results of mesh independence test.

Mesh	Element size	Total elements	C_d
Coarse	10 mm	5 millions	0.426303
Medium	5 mm	7 millions	0.426790
Fine	4 mm	10 millions	0.426137

Based on the results of mesh independence test, a medium grid with approximately 7 million elements was selected for present study to computational time.

IX. RESULT AND DISCUSSION

CFD analysis is done for all four inlet parameter, and results are obtained through it. For this case input temperature 25^{0}C, air flow velocity 22.8 m/s in which outlet pressure 0 Pa for simulation solution is initialized from velocity.

Table 2- Drag force at different velocities

Velocity (m/s)	C_D
27.7	0.4558
30	0.4944
33	0.5238

Fig 3- velocity contour for 33 m/s

Above figure shows that the velocity result on a car body and it indicates that velocity is low on the back side of the car due to the suction in back side of car which creates wake region at the rear end of the vehicle.

Table 3- values at different ground clearance

Ground clearance	C_D	C_L	C_M
120 mm	0.436125	0.978759	0.223859
123 mm	0.441165	1.15217	0.247230
130 mm	0.461888	1.860869	0.181278

Table 4- values at different front angle

Front angle	C_D	C_L	C_M
130^0	0.434456	1.09672	0.235705
136^0	0.426886	1.18629	0.232690
140^0	0.426303	1.14478	0.265133

Table 5- values at different rear angle

Rear angle	C_D	C_L	C_M
120^0	0.444025	1.06121	0.393575
130^0	0.545122	1.28486	0.390387
135^0	0.563587	1.56237	0.39027

Leading to the obtained results, redesign of the car geometry will be made. Redesign in terms of increasing the angle between the hood and front windshield of car, keeping minimum ground clearance, optimizing the rear angle of car and adding rear wing. These changes will result with better airflow around the car, and producing more down - force using the rear wing. Bigger amount of down force will result with better stability of the car and increasing traction. It is assumed that redesigned car geometry there will be less turbulence behind the car and turbulent zone will be cleaner.

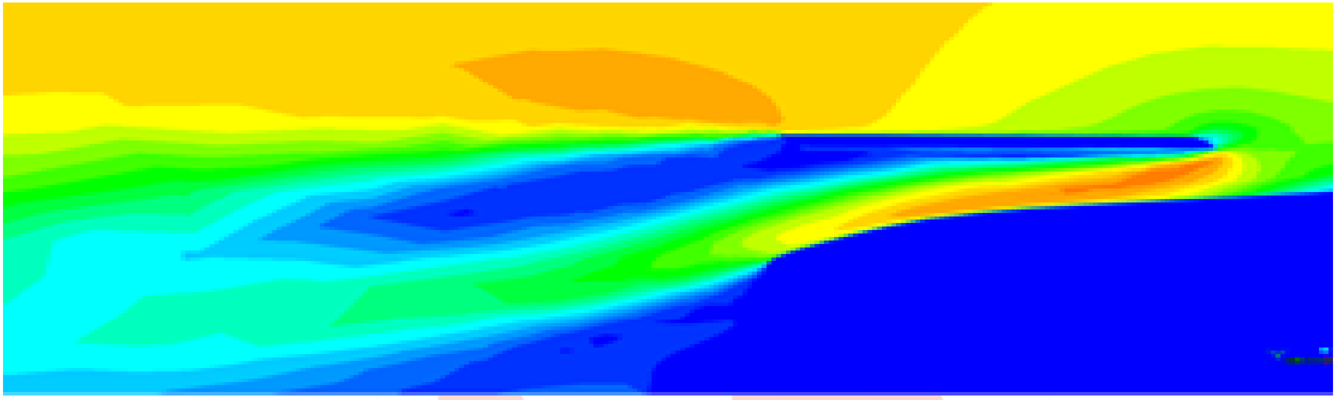

Fig 4- velocity contour of car with spoiler

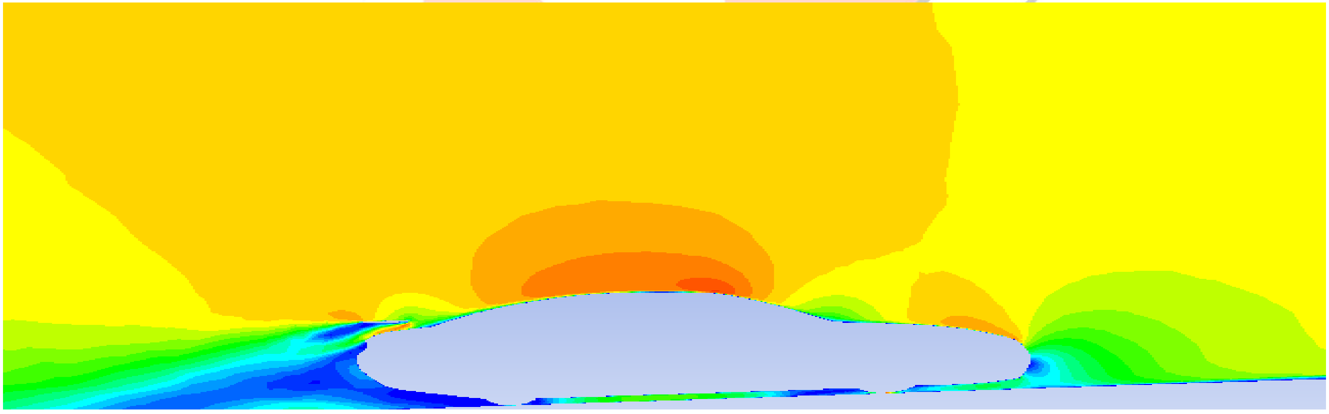

Fig 5- velocity contour of final model

Table 6- values of redesigned final model

C_D	C_L	C_M
0.3676	1.9653	0.5589

From the above obtained values, the value of drag co-efficient is decreasing because of the addition of rear spoiler which separates the air flow from the surface of the car and also increases the down force which gives better stability to car.

X. CONCLUSION

Leading to the obtained 2D simulation and leading to the modification of an existing 2D model in terms of redesigned side contour of the car, the existing 3D model is redesigned. Redesign in terms of increasing the angle between hood and front windshield of the car, and adding the spoiler. Further the 3D analysis of airflow around the redesigned car geometry was achieved. With the obtained 2D and 3D results, it is concluded that the mentioned changes in the geometry of redesign car are resulting in better airflow around the car, and producing more down force using the rear wing. Higher amount of down force resulting in better stability of the car while increasing traction.

XI. REFRENCES

[1] Chien-Hsiung Tsai, Lung-Ming Fu, Chang-Hsien Tai, Yen-Loung Huang Computational Aero-Acoustic Analysis Of A Passenger Car With A Rear Spoiler,2008.

[2] Emmanuel Guilmineau , Oussama Chikhaoui, Ganbo Deng, Michel Visonneauequipe Modélisation Numérique, Laboratoire De Mécanique Des Fluides, Cross Wind Effects On A Simplified Car Model By A Des Approach, 2011

[3] Bahram Khalighi, Shailesh Jindal. General Motors Global R&D, Aerodynamic flow Around A Sport Utility Vehicle—Computational And Experimental Investigation, 2012.

[4] W. Kieffera, S. Moujaesb, N. Armbyab Msme Unlv , Cfd Study Of Section Characteristics Of Formula Mazda Race Car Wings, 2005

[5] R. Himeno, K. Fujitani Numerical Analysis and Visualization Of Flow In Automobile Aerodynamics Development Senior Researcher, Nissan Motor Co., Ltd, Natsushima-Cho, Yokosuka, Japan Research Engineer, Nissan Motor Co., Ltd, Natsushima-Cho, Yokosuka 237, Japan

[6] Hitoshi Fukuda, Kazuo Yanagimoto, Hiroshi China, Kunio Nakagawa.Improvement of Vehicle Aerodynamics By Wake Control, 1994

[7] Damjanović, Darko, Kozak, Dražan, Ivandić, Željko, Kokanović, Mato Josip Juraj Strossmayer Car Design As A New Conceptual Solution And Cfd

[8] M. M. Islam and M. Mamun Computational Drag Analysis Over A Car Body Department Of Mechanical Engineering,2010.

[9] W. Angelis, D. Drikakis, F. Durst, W. Khier Numerical and Experimental Study Of The Flow Over A Two-Dimensional Car Model, 1996

[10] D.E. Aljure, O. Lehmkuhl,I. Rodríguez,A. Oliva. Flow and Turbulent Structures around Simplified Car Models, 2014

[11] Makoto Tsubokura, Toshio Kobayashi, Takuji Nakashima, Takahide Nouzawa, Huilai Zhang, Keiji Onishi, Nobuyuki Oshima, Takaki Nakamura,computational visualisation of unsteady flow around vehicles using high performance computing,2008.

[12] H. Zaidi, S. Fohanno, R.Taiar, G. Polidori. Turbulence model choice for the calculation of drag forces when using the CFD method, 2009.

[13] Rizal E. M. Nasir, Firdaus Mohamad, Ramlan Kasiran, M. Shahriman Adenan, M. Faizal, Mohamed, M. Hanif, Mat, Amir R. A. Ghani. Aerodynamics of ARTeC's PEC 2011 EMo-C Car.

[14] AHMED, H[aseeb] & CHACKO, S[ibi]. Computational Optimization Of Vehicle Aerodynamics, 2012.

[15] R.J. Corin, L. He,R.G. Dominy. A CFD investigation into the transient aerodynamic forces on overtaking road vehicle models, 2008.