International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 7, July - 2013

Aerodynamic Analysis Of Sport Utility Vehicle By Using Computational Fluid Dynamics Approach

Dinesh Dhande, Manoj Bauskar
Assistant Professor, AISSMS College of Engineering, Pune-411001, Maharashtra, India

Abstract

In an era improving the fuel economy of vehicle has become need of automobile industries to survive in the cut throat competition. As rapid and continuous increase in prizes of fuels, consumers are going for most fuel efficient vehicles. Recently stringent emission norms, fuel economy and recycling become important social concern. Fuel economy has become latest topic of discussion among not only the responsible scientists but also common citizens. The company those will cater the need of consumers will survive in the market. By aerodynamic styling of vehicle one can not only improve the fuel efficiency but also ensure better stability and good handling characteristics of vehicles at higher speed especially on highways. The paper describes assessment of drag force ($F_{d)}$ and drag coefficient ($C_{d)}$ by using computational fluid dynamics (CFD). The model of sports utility vehicle (SUV) on reduced scale 1:32 is drawn with aid of PROE software.

1. "Introduction"

Computational fluid dynamic is branch of fluid mechanics that uses numerical methods and algorithm to solve and analyze problems which constitutes involvement of fluid flows. Computers are used to simulate the interaction of air with the surfaces define by boundary conditions.

Aerodynamic styling of car is most crucial and complex phenomenon [1] .The main purpose of aerodynamic styling of vehicle is to optimize the shape of vehicle by attaining all exterior shape of vehicle closer to more streamline one.

By stream-lining the body of vehicle not only improves the fuel efficiency of vehicle but also improves the handling characteristic of vehicle at higher speeds especially on highways. In earlier models average C_d Values have improved from 0.7 to 0.3 for the more streamline vehicle [2]. Well designed aerodynamic vehicle consumes not only less fuel in overcoming the drag exerted by air while running at higher speeds but also offers good stability and handling behaviour . The brutal competition among the companies of automobile along with continuous changing need of consumers there is tremendous pressure on Engineers to design automobile which is affordable as well as appealing, safe and fuel efficient to drive. The aerodynamic drag become a critical when vehicles are driving at highways with higher speeds it has found that fuel consumption due to aerodynamic drag is more on highways as compare to vehicle those are running in urban areas. Because of uncertain future of fuel prize the world has put more focus on alternative energy. Automobile companies have been working on improving the fuel efficiency the past decades. Hybrid vehicle, human power vehicles and electric vehicle were developed to pursue a high mileage per litre. Besides finding alternative fuels for gasoline and diesel, engineers are also trying to improve vehicle efficiency by manipulating different parameters including engine parameter, aerodynamic drag weight and rolling resistance.

Drag force (Fd) can be calculated by equation (1) [6].

Where, $V_{upstream}$= Upstream Velocity,

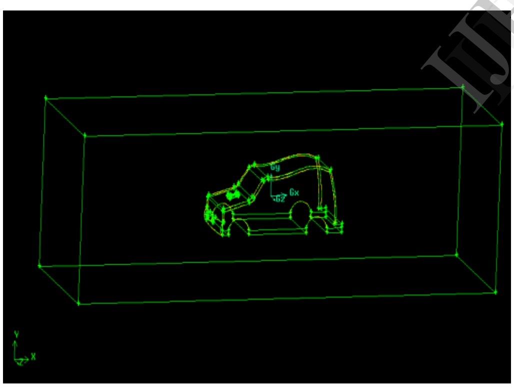
A= frontal area vehicle.

$$(F_d) = \rho.(V_{upstream}{}^2 – V_{downstream}{}^2/2).A \quad .....(1)$$

The computational fluid dynamics gives advantage to study the behaviour of fluid field of air over the exterior surface of vehicle. The aerodynamic analysis are perform over model of suv (1:32) . By optimizing exterior shape of vehicle the drag coefficient can be reduced and fuel efficiency can be improved at higher speeds [3].

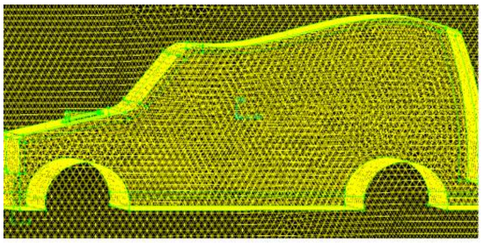
2. "Computational Approach"

In the computational approach data concerning flow field around the body of SUV is visualized by simulating the flow condition using Gambit as the pre-processing software and fluent as the solver and postprocessor.
CFD codes have three basic elements which divide the complete analysis of numerical experiment to perform on the specific geometry. The three basic elements are pre-processor, solver, postprocessor. The pre-processing stage of the CFD process involves the following:
Definition of geometry, Meshing, definition of fluid continuum and boundary conditions.
In pre-processing the geometric model of SUV is created and saved in the form of .iges file that is input to the GAMBIT by means of user friendly software (PROE) as shown in Fig. 1.



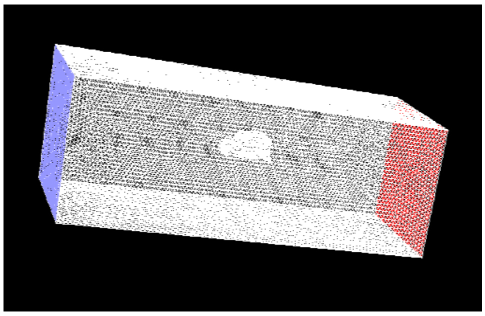
"Figure 1. Model of SUV"

In meshing, the region of interest need to be divided into several structure elements using Gambit as software & grid is generated automatically in 3 dimensional domain with tetrahedral mesh [4,5]. This is very important stage in CFD as it affects to the accuracy of results Fig. 2 shows meshed model.



"Figure 2. Grid Generation."

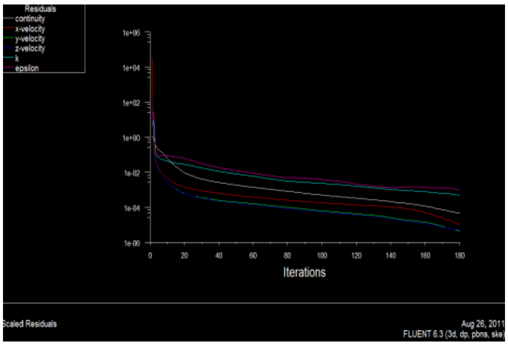
Fluid properties are defined. Boundary conditions are specified (Fig.3) as inlet velocity on the nearest edge and exit pressure at the farthest edge for visualization of flow.



"Figure 3. Boundary Condition Specification"

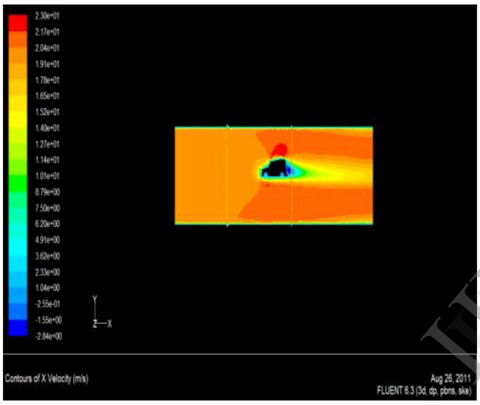
Numerical solver: The fluent which is numerical solver is key element of CFD process. The steps involved in this solving process are usually carried out in following sequence. Formal integration of governing equation of fluid flow over all the control volume, conversion of integrated forms of equations by algebraic equations and then calculations of algebraic equations by an iterative method.

A 3ddp steady state, incompressible solution of the Navier-stokes equations is obtained by implementing turbulent modelling with standard k-Ɛ model using standard wall functions and second order upwind discretization scheme. The free stream air velocities for series of test are varied from 5m/s to 21 m/s, while the exit pressure is set to atmospheric pressure.i.e. 1.013bar.[6]

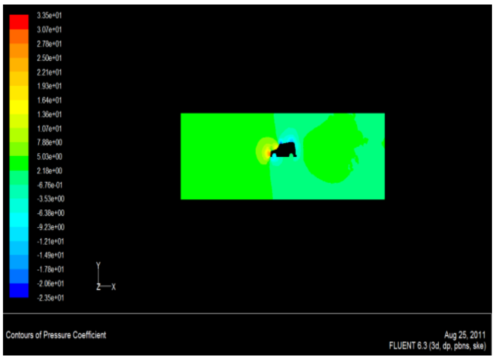


"Figure 4. Processing"

Post processing is the last phase of CFD i.e. results and simulation. Contours of velocity and pressure coefficient can be obtained from display option. (Fig.5,6) shows it respectively for judging aerodynamic performance of vehicle.



"Figure . 5 Contours of Velocity"



"Figure 6. Contours of Pressure coefficient"

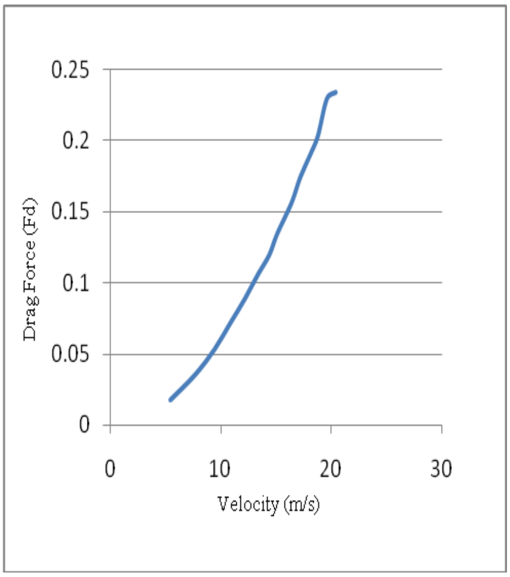
3. "Results"

From Fig. 7, it appears that as air velocity increases the drag force increases. Fig. 8 revels that initially there is decrease of drag coefficient and then attainment of constant value with further increase in Reynolds number.

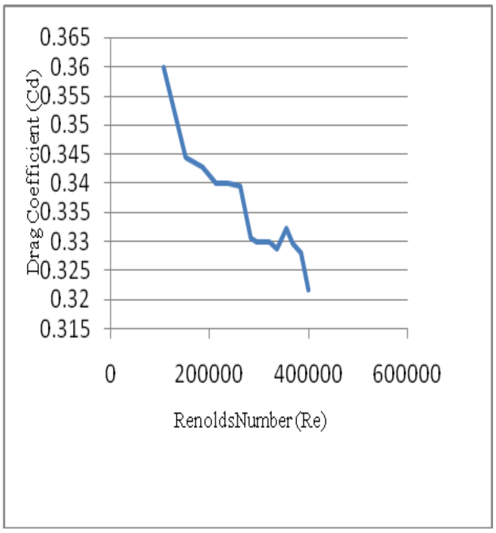
"Table 1. Calculation of F_d & C_d"

UP STREAM VELOCITY	DOWN STREAM VELOCITY	DIFFERENCE	DRAG FORCE (Fd)	DRAG COFFICIENT (Cd)	REYNOLDS NUMBER
m/s	m/s		N		
5.4600	4.42	5.1376	0.0179	0.36	107059
7.7200	6.25	10.2680	0.0358	0.34	151373
9.4500	7.66	15.3135	0.0535	0.34	185294
10.9000	8.8	20.6850	0.0722	0.34	213725
12.20	9.89	25.5140	0.0891	0.34	239216
13.4000	10.89	30.4840	0.1064	0.34	262745
14.4000	11.78	34.2958	0.1197	0.33	282353
15.1000	12.29	38.4830	0.1343	0.33	296078
16.4000	13.4	44.7000	0.1561	0.33	321569
17.2000	14	49.9200	0.1743	0.33	337255
18.1000	14.79	54.4330	0.1900	0.33	354902
18.8000	15.39	58.2940	0.2035	0.33	368627
19.6000	15.9	65.6750	0.2293	0.33	384314
20.4000	16.8	66.9600	0.2338	0.32	400000

Using equation (1) , variation of F_d with air velocity as shown in Fig.7 and C_d in relation with Reynolds number as shown in Fig.8 are plotted by using Table.1.



"Figure 7. Variation of Drag Force (Fd) with Air Velocity"



"Figure 8. Variation of Drag Coefficient (C_d) with Reynolds Number"

4. "Conclusion"

The drag coefficient acquired by computational approach gives the accurate value. Drag coefficient can be evaluated and compared for optimization of vehicle exterior surface and to improve the fuel efficiency. However as compare to experimental calculation of C_d this is less time consuming.

References

[1] H. Braess Hermann & U. Seiffert, *Handbook of Automotive Engineering*,SAE International, Warrendale,SA,2005.

[2 J Kartz, *Race Car Aerodynamics:Designinig for speed*, Bentley Publishers, Cambridge, USA,1995.

[3] W. Stapleford, Aerodynamic Improvements to the body and cooling System of a typical small saloon car, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 9, 1981 pp. 63-75.

[4] Y. Sun, G. Wu & Xieshuo, Numerical Simulation of the External Flow Field around Bluff Car, 2000.

[5] E. Nielsen & W. Anderson, Recent Improvements in Aerodynamic Design Optimization on Unstructured Meshes, *AIAA Jour*nal, vol.40, Nov. 6, 2000, pp.1155-1163

[6] Manan Desai, S.A.Channiwala, H.J. Nagarsheth, "Experimental and Computational Aerodynamic Investigation of a Car," in *Wseas Transactions on F*luid *Mechanics*, vol.3, pp 359-368, Oct. 2008.

Authors

Dinesh Dhande Obtained his bachelor's degree in Mechanical Engineering from Mumbai University in 1998 and master's degree in Design Engineering from University of Pune in 2004 and is currently pursuing PhD from University of Pune. He is presently working as Assistant Professor in AISSMS College of Engineering Pune in department of Mechanical Engineering, Pune. His areas of interest include tribology of bearings, wear & computational fluid dynamics.

Manoj Bauskar Obtained his bachelor's degree in Production Engineering from Pune University in 2004 and pursuing master's degree in Automotive Engineering from University of Pune. His areas of interest include computational fluid dynamics.