Mohamed et al. 2016. *Int. J. Vehicle Structures & Systems*, 8(3), 157-160
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.8.3.07

International Journal of Vehicle Structures & Systems
Available online at www.maftree.org/eja

Numerical Investigation of Drag and Lift Forces on a Sedan Car using Computational Fluid Dynamics

M. Fakkir Mohamed[a,c], P.L. Madhavan[b], E. Manoj[a] and K. Sivakumar[a]

[a]*Dept. of Automobile Engg., Vel Tech University, Avadi, Chennai, India*
[b]*Manufacturing Engg., Mookambigai College of Engg., Pudukkottai, India*
[c]*Corresponding Author, Email: fakkir55@gmail.com*

ABSTRACT:

The purpose of this work is to cut back the drag, lift and aerodynamic in-stability of a sedan car at high speed levels. In early times, the cars accustomed have a flat faces, sharp edge, conjointly had higher mileage and potency. However later because of the emergence of fuel crisis, scientists improved the model of cars with regard to dynamics of the fluid around the body. Thus, it changes the structure of cars with respect to aeromechanics. Simulation of a vehicle had been done using computational fluid dynamics to obtain the coefficient of drag and coefficient of lift. Finally, these coefficients from computational fluid dynamics are compared wind tunnel simulation.

KEYWORDS:

Computational fluid dynamics; Sedan car; Drag; Lift; Aerodynamics; Wind tunnel; Simulation

CITATION:

M.F. Mohamed, P.L. Madhavan, E. Manoj and K. Sivakumar. 2016. Numerical Investigation of Drag and Lift Forces on a Sedan Car using Computational Fluid Dynamics, *Int. J. Vehicle Structures & Systems*, 8(3), 157-160. doi:10.4273/ijvss.8.3.07.

1. Introduction

The study of flow behaviour of air inside or over a body is termed as aerodynamics. The flow of fluids can be classified in the terms of the structure in or on which they flow as internal and external flow. The external aerodynamics has a vast role to play in the fields of automobiles, aerospace, architecture etc. When the fluid velocity is less than the velocity of sound then is termed as subsonic flow (Ma<1), if it is equal to velocity of sound then is called sonic flow (Ma=1), and if it is more than the velocity of sound it is called supersonic (Ma>1) or hypersonic (Ma>>1) flows. The fluid flow is said to be compressible when it has high velocity (Ma>0.3) and is incompressible at lower velocities. As fluid dynamics is a complex yet theoretically strong field of research, great effort has gone into understanding the governing loss and the nature of fluids.

In mid-1970's, general purpose computational fluid dynamics (CFD) solvers were developed. These began to materialise within the early 1980's and necessitated powerful computers. Recent advances, in computing power, alongside powerful graphics and interactive 3D manipulation of models, have created the method of building a CFD model and analyzing the results with less labour intensive, reducing time and thus overall cost of product development. CFD plays an essential role in aerodynamic design of car. The wake regions create drag vigorously leading to vibrations while driving. Xingjun [1] studied the influence of different diffuser angle on the steering stability. The drag reduces while diffuser angle increases. The relation between the outer shape and aerodynamic characteristics are described by Chacko et al [2]. Ghani [3 found another possible way to reduce the drag by redesigning the rear part of a car for drag reduction. The importance of wind tunnel and CFD methodology and how it is helpful to obtain better car body design are addressed by many researchers [4-7]. Sharma et al [8] introduced the tail pipes in car, for reducing drag up to 3.87% and the lift upto 16.62%. One of the most difficulties happened while driving is vehicle overturning or wheel unloading due to strong cross wind and the critical wind speed. Kikuchi et al [9] found the aerodynamic force coefficients at the strong cross wind and the critical wind speed. Guimineau [10] compared validation between Ahmed car body with the varied rear slant angle model and the tangible model.

In this paper, the baseline design of Sedan car has been simulated using CFD. Based on the simulations, the baseline design has been improved through provision of rear spoiler and re-shaping the car body. The improved design has been verified through CFD simulation for the same boundary conditions as those used in the baseline simulations. The coefficients of lift and drag were compared to assess the design improvement from aerodynamics perspectives of the considered Sedan car.

2. CFD simulation models

In order to obtain a better design using CFD, first task to accomplish in a numerical flow simulation is to extract the fluid domain or the region where the fluid flow is occurring. Using the CFD tools, the geometrical components which are all not needed for the simulation is removed thus only the fluid domain is prepared for the next process. One of the foremost cumbersome and time intense a part of the CFD is that the mesh generation.

Volumetric meshes are a polygonal representation of the interior volume of an object. Unlike polygon meshes, which represent only the surface as polygons, volumetric meshes also discretize the interior structure of the object. In this analysis, a procedure known as the reference Jacobean has been developed for the improvement of 3D mesh quality by node placement. This general procedure consists of iterations involving node placement on the boundary followed by node placement within the interior (with boundary nodes fixed). The procedure has been tested as very effective in rising mesh quality of multi-material tetrahedral and hexahedral meshes whilst minimizing the changes to the mesh characteristics by separating the boundary surfaces. The Sedan car size is considered as 4665mm*1760mm*1445mm for the simulations. The wind tunnel aerodynamic flow domain is considered as 46650mm*10560mm*7156mm. The fluid and wind tunnel conditions are given in Table 1.

Table 1: Fluid and wind tunnel conditions

Fluid conditions	
Speed of air	16.66 m/s
Operating pressure	1 Atm
Density of air	1.225 kg/m^3
Wind tunnel conditions	
Bottom surface (road)	0.22 Roughness constant
Sides	Symmetry boundary condition
Top	Symmetry boundary condition
Inlet	Velocity Inlet - 16.66 m/s
Outlet	Pressure outlet - 0 Pa

2.1. CAD model

The baseline design model of Sedan car considered for the CFD simulation is shown in Fig. 1. For the modified Sedan car design, the rear spoiler is attached to the baseline design car.

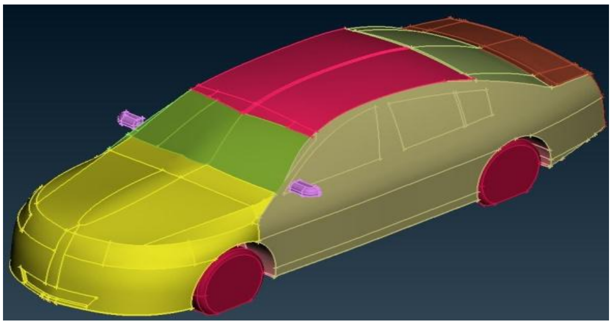

Fig. 1: Baseline design model

2.2. Clean up and domain formation

The CAD models of the baseline and modified designs are exported to ANSA software to repair various surface edges. The model is surrounded with an artificial domain that represents the original atmospheric or wind tunnel conditions as shown in Figs. 2 and 3 for baseline design and modified design respectively. This volumetric domain is considered as the total domain for the simulations where the boundary conditions are applied.

2.3. Surface mesh

Using ANSA, the surface mesh is generated in triangular shapes such that it could idealise any intricate geometry in the CAD model. These meshes are of equilateral triangular character with maintained quality of 0.4. The surface mesh of baseline and modified designs are shown in Fig. 4 and Fig. 5 respectively.

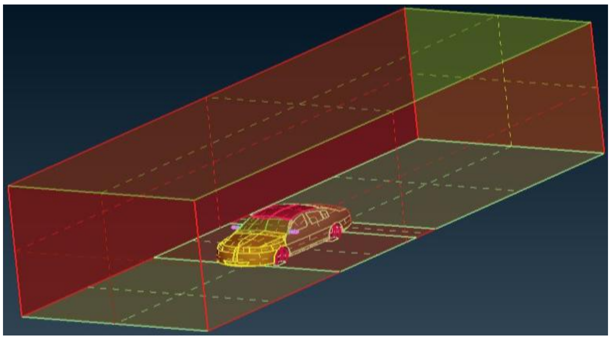

Fig. 2: Clean up and domain formation for baseline design

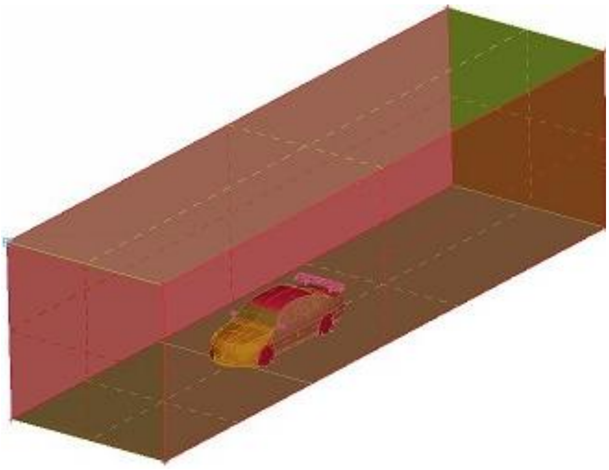

Fig. 3: Clean up and domain formation for modified design

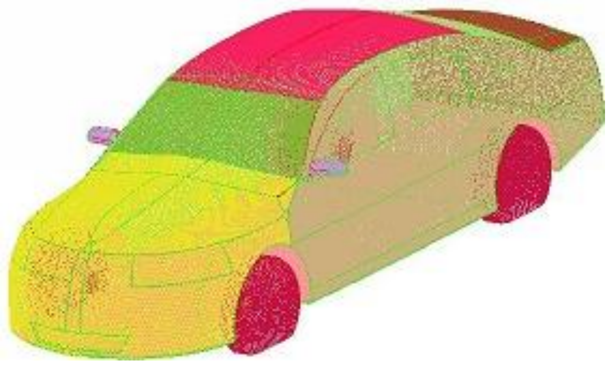

Fig. 4: Surface mesh of baseline design

Fig. 5: Surface mesh of modified design

2.4. Volume mesh

The surface mesh is transferred to TGrid and the volume is meshed in the shapes of tetrahedral elements. These volume meshes are produced all over the closed domains of the model. Higher the mesh count, the need for computational power is also increases proportionally. Hence, the volume meshes are required only in the zones where the fluid flow is significant. The volume inside the car is not required for external aerodynamic tests, thus they are removed from the simulation. Figs. 6 and 7 show the volume mesh of baseline and modified design model respectively.

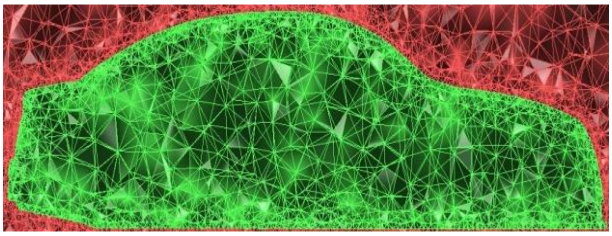

Fig. 6: Volume mesh of baseline design

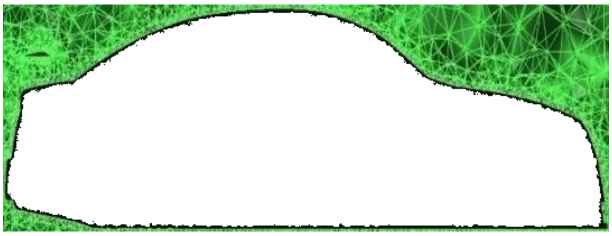

Fig. 7: Volume mesh of modified design with rear spoiler

2.5. Mesh grid independence

Since no experimental values are available to validate the grid count, convergence and resulting coefficient drag values are compared in Table 2 to finalize the grid count. Tria surface mesh and tetrahedral elements are used with volumetric of 0.86 is maintained on all grid counts for base model. Grind count of 1.2 million elements is considered independent and is maintained on all cases of CFD simulation. A summary of surface and volume mesh details is presented in Table 3.

Table 2: Coefficient of drag for various element counts

No. of elements	Coefficient of drag
612,568	0.682
856,221	0.655
1,086,358	0.632
1,242,553	0.611
1,541,789	0.610

Table 3: Mesh summary for baseline & modified Sedan car models

Parameter	Baseline design	Modified design
Surface mesh count	2L	2L
Surface mesh quality	0.4	0.4
Volume mesh count	12L	13L
Volume mesh quality	0.86	0.9

3. Results and discussion

Baseline and modified design CFD models are analyzed for the variation of coefficient of lift with same boundary conditions. The CFD solver has been set with the following parameters:

- Steady state.
- Incompressible.
- 2[nd] order discretization.
- Standard solution initialization from 'Inlet'.

The physical modelling has used viscous, K-epsilon, standard equation model with standard wall functions and the intensity of turbulence as 5%. The solver convergence criteria were set as follows:

- Root Mean Square (RMS) value of 1×10^5 is maintained for X, Y, Z momentum.
- RMS of 1×10^4 is achieved for K and Epsilon.
- No initial solution imbalances were seen.

The coefficients of drag and lift for the base model are shown in Figs. 9 and 10. As observed from Fig. 11, the coefficient of lift for the modified design with rear spoiler provision is -0.36 (negative sign indicates upward suction due to low pressure). As observed from Fig. 12, the coefficient of lift for the modified Sedan car model is -0.04 (negative sign indicates upward suction due to low pressure). In the modified design, the expected drag and lift values are lower compared to those from the baseline simulation model. Thus the aerodynamic properties are enhanced by providing the rear spoiler in vehicle.

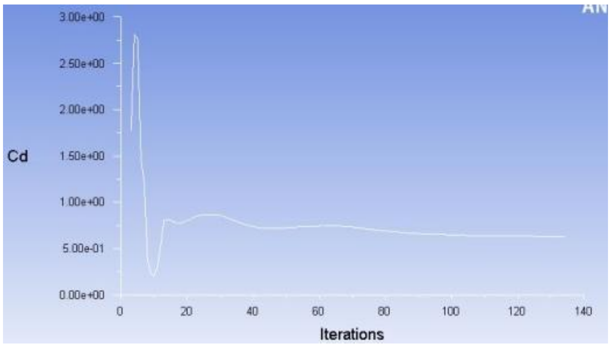

Fig. 9: Coefficient of drag (baseline design)

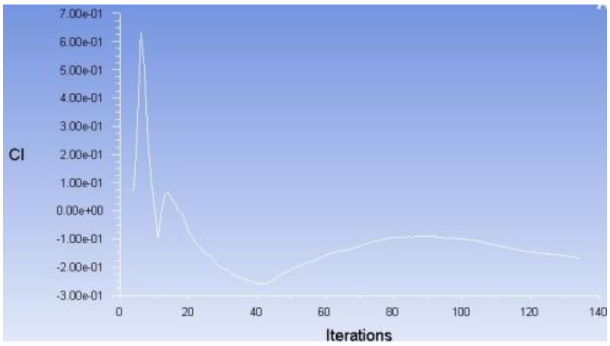

Fig. 10: Coefficient of lift (baseline design)

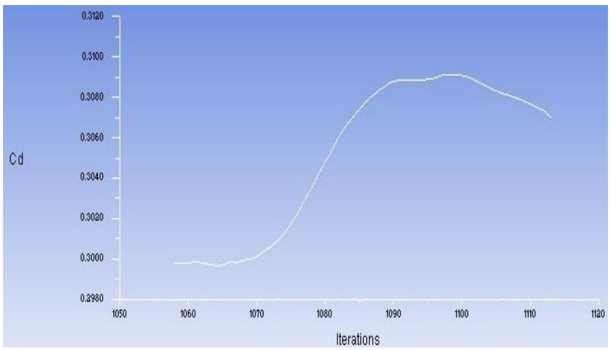

Fig. 11: Coefficient of drag (modified design)

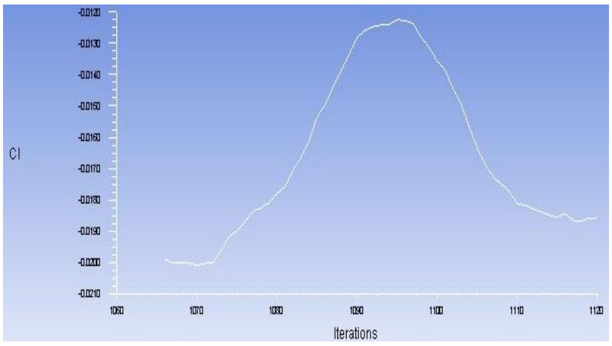

Fig. 12: Coefficient of lift (modified design)

4. Conclusion

CFD simulations were carried on the baseline design and modified design with rear spoiler for Sedan car through iteration process until convergence. The existing value of drag and lift for Sedan car is 0.6 and 0.3. The experimental value of drag and lift for the modified design are 0.3 and 0.04. The expected drag and lift values in the modified design are lower than those from the simulations of baseline design. Thus, the Sedan car design has been improved through the use of CFD simulation.

REFERENCES:

[1] H.U. Xingjun. 2011. Influence of different diffuser angle on sedan's aerodynamic characteristics, *Proc. Int. Conf. Physics Science and Technology*, Dubai, UAE, *Physics Procedia*, 22, 239-245. http://dx.doi.org/10.1016/j.phpro.2011.11.038.

[2] H. Ahmed and S. Chacko. 2012. Computational optimization of vehicle aerodynamics, *Annals of DAAAM for 2012 & Proc. 23rd Int. DAAAM Symposium*, 23(1), 313-318.

[3] O.A. Ghani. 2013. *Design Optimization of Aerodynamic Drag at the Rear of Generic Passenger Cars using Nurbs Representation*, Master Thesis, University of Ontario, Canada.

[4] Z. Mohamed, D. Ramasamy and M.R. Hanipah. 2010. Small Sedan aerodynamics analysis, *Proc. National Conf. Mech. Engg. Research and Postgraduate Students*, University Malaysia Pahang, Malaysia.

[5] C.J. Baker and N.D. Humphreys. 1996. Assessment of the adequacy of various wind tunnel techniques to obtain aerodynamic data for ground vehicles in cross wind, *J. Wind Engg. and Industrial Aerodynamics*, 60, 49-68. http://dx.doi.org/10.1016/0167-6105(96)00023-2.

[6] S. Watkins and G. Vino. 2008. The effect of vehicle spacing on the aerodynamics of a representative car shape, *J. Wind Engg. and Industrial Aerodynamics*, 96(6-7), 1232-1239. http://dx.doi.org/10.1016/j.jweia.2007.06.042.

[7] M. Khaled, H.E. Hage and F.H.H.P. Hossaini. 2012. Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body, *J. Wind Engg. and Industrial Aerodynamics*, 107-108, 36-47. http://dx.doi.org/10.1016/j.jweia.2012.03.019.

[8] R.B. Sharma and R. Bansal. 2013. CFD simulation for flow over passenger car using tail plates for aerodynamic drag reduction, *IOSR J. Mech. and Civil Engg*., 7(5), 28-35. http://dx.doi.org/10.9790/1684-0752835.

[9] K. Kikuchi and M. Suzuki. 2008. Study of aerodynamic coefficients used to estimate critical wind speed for vehicle overturning, *J. Wind Engg. and Industrial Aerodynamics*, 147, 1-17. http://dx.doi.org/10.1016/j.jweia.2015.09.003.

[10] E. Guimineau. 2015. Computational study of flow around a simplified car body, *J. Wind Engg. and Industrial Aerodynamics*, 145, 292-303. http://dx.doi.org/10.1016/j.jweia.2007.06.041.