INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact factor: 4.295
(Volume 5, Issue 2)
Available online at: www.ijariit.com

Aerodynamic optimization of passenger vehicle

Hussain Ezzy
hmezzy53@gmail.com
Hindustan Institute of Technology and Science, Chennai, Tamil Nadu

Mohammed Zain M.
mohammedzain22@gmail.com
Hindustan Institute of Technology and Science, Chennai, Tamil Nadu

Dr. N. Prakash
nprakrao1@gmail.com
Hindustan Institute of Technology and Science, Chennai, Tamil Nadu

ABSTRACT

The growth in computational fluid dynamics has brought about a rapid increase in the study of vehicle aerodynamics aiding to meet the stringent fuel economic and pollution laws. Though there has been a lot of research in the segment of sedans and sports vehicles, there has been considerably less amount of research on the hatchback segment of vehicles. This paper aims at a study in the computational fluid dynamic simulations of a widely used passenger hatchback vehicle and a proposed model to obtain the reduced coefficient of drag value for the proposed model based on the initial widely used model and to understand the effect of the shape of the vehicle on the drag value of the vehicle. The widely used hatchback passenger vehicle (initial model) is designed based on a body, similar to that of a typical hatchback model widely sold in the Indian market. The drawbacks of the initial model are obtained via the CFD simulations and the proposed model is designed to overcome those drawbacks and offer minimal resistance to air thereby reducing the Coefficient of Drag value.

Keywords— *Drag Optimization, Computational Fluid Dynamics, Vehicle Aerodynamics, Wake Effect, Hatchback*

1. INTRODUCTION

Aerodynamics deals with the forces that are produced due to gasses that resist the motion of any moving body relative to its speeds and acts in the direction opposite to the direction of motion. In-ground vehicles, the biggest factor affecting the body from achieving high speeds is the aerodynamic drag on the surface of the vehicle. This drag can also produce lift forces which can cause rolling or pitching moments in the vehicle leading to fatal accidents. In this study we deal with commercially sold passenger vehicle with a hatchback model which is widely sold in the Indian markets due to its compact sizes and nifty manoeuvrability and also comparatively lower pricing, making it a very good bargain in Indian markets. But one of the major drawbacks of any hatchback model is its low drag resistance i.e. high coefficient of drag (Cd) as compared to standard sedan models, mainly due to the high wake created which accounts for 80 - 90% of drag while the rest is due to surface friction [1]. This increase in Cd value means higher fuel consumption costs and reduced performance. The freestream velocity was taken as 25m/s (90 km/h) as 80% of total drag is observed at cruising speeds greater than 80km/h [2]. There has been a significant amount of research done in the testing and analysis of sedans and SUVs [6-10] but relatively much less research has been done in the hatchback segment, hence this paper deals to propose a model that is a hatchback but has a significant reduction in its Cd value thereby decreasing fuel consumption and increasing performance.

2. MODELLING

The modelling was carried out using Autodesk Alias Software to create both models accurately (Fig 1.). The initial model was based on the Maruti Suzuki hatchback model - Swift. This model was chosen as it is one of the top most sold hatchback model in India, and so has been used as a benchmark for a standard hatchback model's Coefficient of drag value.

The proposed model was designed with the intention of getting the coefficient of drag value lesser than that for the initial model.

Both the models were designed in the scale ratio of 1:5. A computational domain of height $H = 7.3\times l$ with a width of $W = 4\times w$ and a length $L = 13.7\times l$ where l is the length and w is the width of the vehicle. The scale model was placed at a distance of $P = 5.8\times l$ inside the domain which was given wall thickness of 10 mm to idealize wind tunnel test section, as shown in Fig 2.

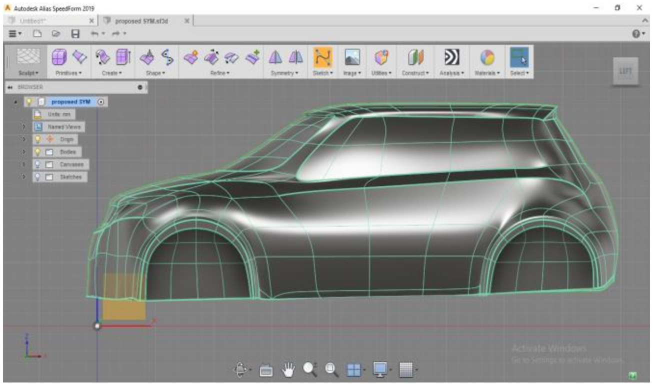

Fig. 1: Modelling of initial model

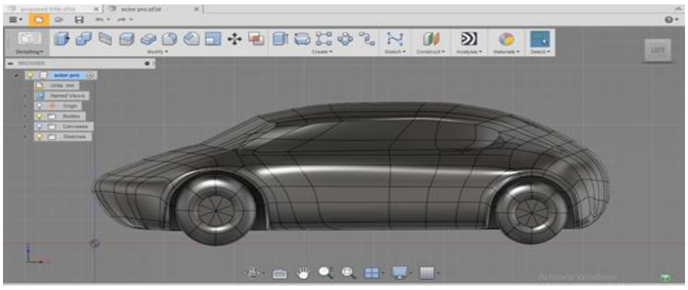

Fig. 2: Modelling of proposed model

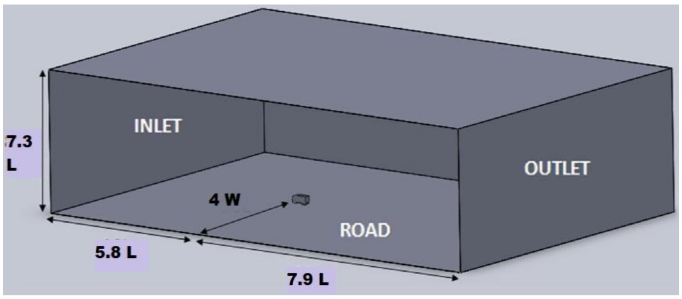

Fig. 3: Computational domain

3. CFD SIMULATIONS

The RANS simulation setup consists of three major subdivisions [3]

- The preprocessor is used to handle the geometry modelling process and grid generation.
- The flow solver which consists of various other turbulence models used to solve the flows around the vehicle.
- The post processor is used to produce the flows on the surface of the model.

A steady-state compressible flow Reynolds averaged Navier Stokes equation with the Menter's SST k omega turbulence model was taken. Menter's SST K-Omega Turbulence model was selected because it uses insensitivity to freestream conditions of K-Epsilon in the far field while retaining the advantages of standard K-Omega Turbulence Model [4].

3.1 Mesh generation

To obtain accurate results it is important to choose the right mesh. We selected the polyhedral mesh as it has the better-attained accuracy and efficiency of numerical computations as compared to tetrahedral mesh [4].

A Volume of refinement (VOR) was created around the body of the vehicle and stretched across the rear end of the vehicle to obtain more precise results and for a finer mesh near the body so as to get accurate results and also reduce computational power and time by minimizing the number of cells in the entire domain. The domain was given a much coarser mesh size (10 cm) whereas the VOR was given a finer mesh size (1.5cm). The volume mesh generated is shown in Fig 4.

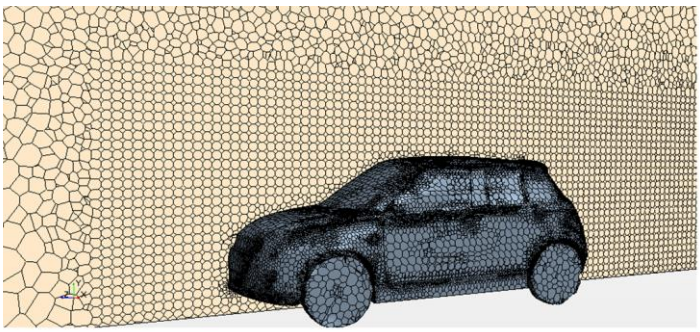

Fig. 4: Volume mesh of initial model,

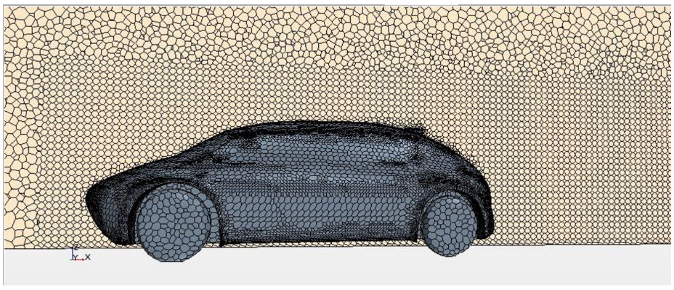

Fig. 5: Volume mesh of proposed model

3.2 Simulation setup

The Models were imported into STAR CCM+ software and were converted into a single region with different boundaries with each surface having a specific boundary type. The body and the wheels of the vehicle having wall boundary type, the domain inlet and side walls were given velocity inlet and the domain outlet was set as pressure outlet.

The model was simulated at 25m/s (Re=$8.7\text{x}10^5$) [5] the total number of cells generated were close to 0.55 million with the minimum cell size of 1.5 cm (0.0015 m). The volume meshes around both the models are shown in Fig 3. The mesh and the physics parameters used in STAR CCM+ simulations are shown in Table 1. All the initial conditions and material properties were taken for simulations of both models are summarized in Table 2.

The simulations were run for 2000 convergent iterations. The resultant flow around the models is displayed in Fig 4 and Fig 5.

Simulations with varying mesh coarseness were performed on the initial model to test grid independency. Results of the grid independence test are shown in Table 3. There is a maximum Variation of only 4.5% between the coarsest and finest meshing used, and as such, the results are assumed to be largely grid independent. The medium mesh was used throughout this study because it was both computationally efficient and making it finer has little effect on the precision of C_d value.

Table 1: Mesh and physics parameters

Mesh property	Parameters
Surface mesh	Surface remesher, Surface wrapper.
Volume mesh	Polyhedral
Minimum cell size	0.0015 m
Total no. of cells	550,000

Physics properties	Parameters
Flow Characteristics	Steady, Stationary, Incompressible, Turbulent, Segregated Flow
Turbulence Model	K-Omega (SST Menter)
Flow velocity	25 m/s

Table 2. Initial conditions and material properties.

Conditions	Inputs	
	Initial model	**Proposed model**
Initial Velocity	25 m/s	
Coordinate System	Laboratory	
Static Temperature	300K	
Turbulence Specification	K + Omega	
Turbulent Kinetic Energy	0.0625 J/kg	
Specific Turbulent Dissipation Rate	2.396 1/s	1.713 1/s

Material properties	Inputs	
Air Density	1.177kg/m3	
Dynamic Viscosity	3.07x 10^{-4} Pa-s	4.295x 10^{-4} Pa-s
Reynolds Number	8.7x10^{5}	

Table 3. Grid independence.

Mesh	No. of cells in millions	C_d
Fine	0.75	0.377
Medium	0.55	0.380
Coarse	0.33	0.395

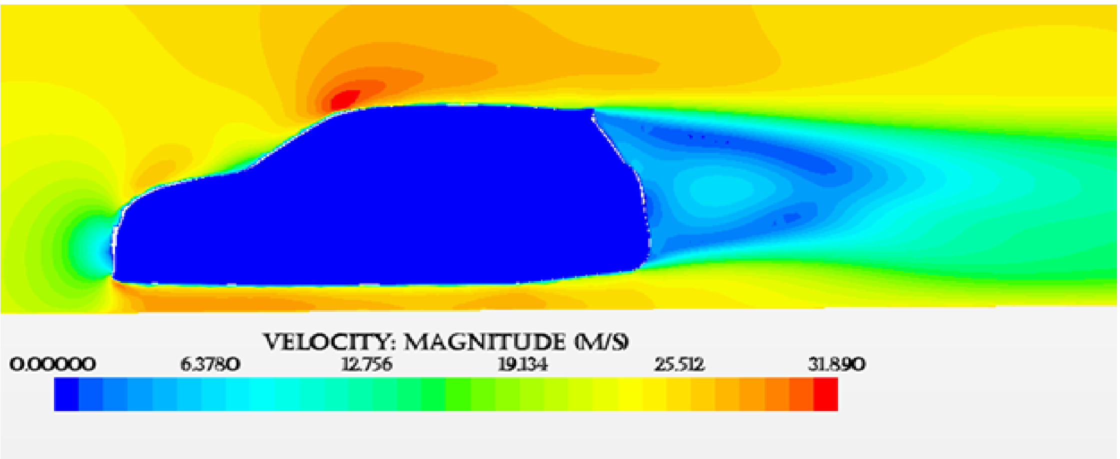

Fig. 6: Velocity Scalar contours around initial model

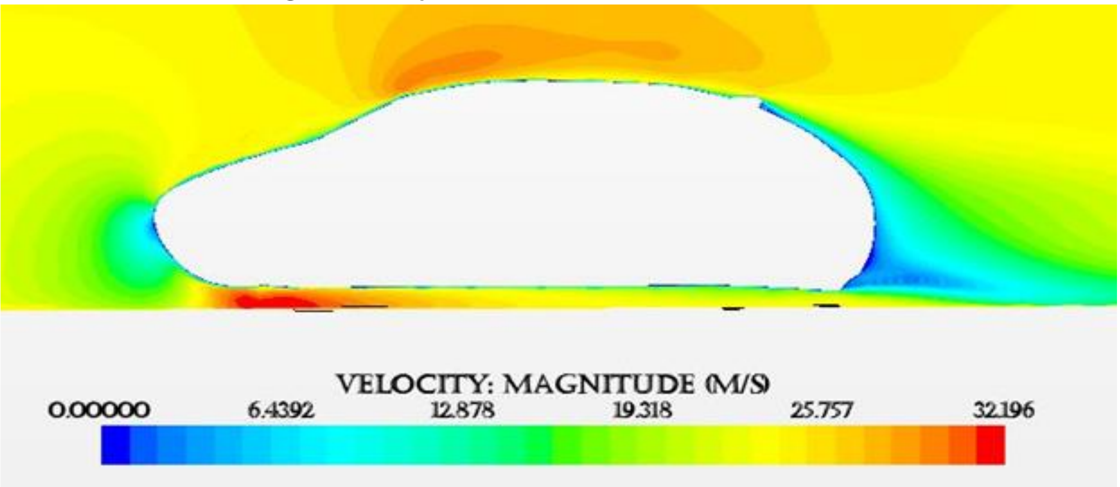

Fig. 7: Velocity scalar contours around proposed model

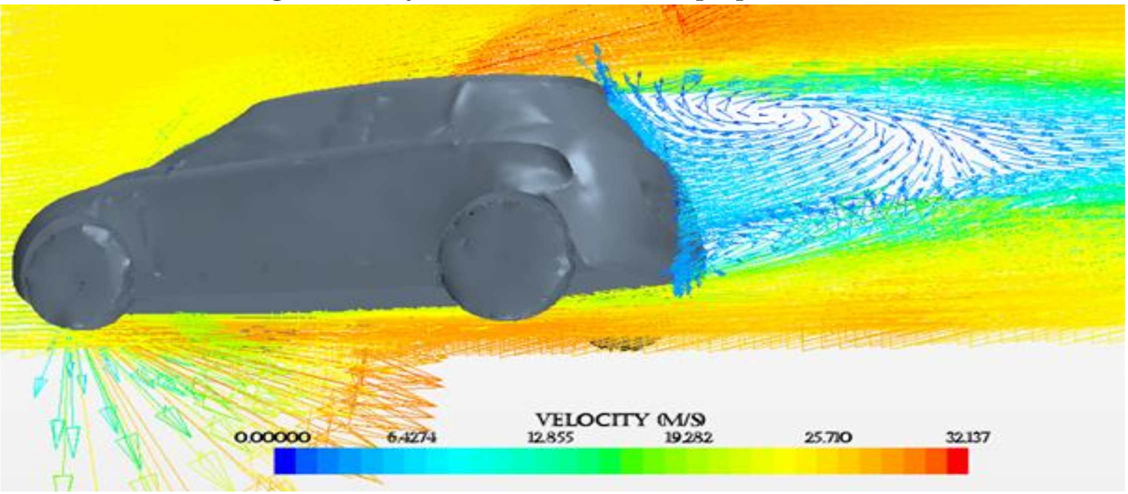

Fig. 8: Velocity Vector Contours around the rear end of initial model

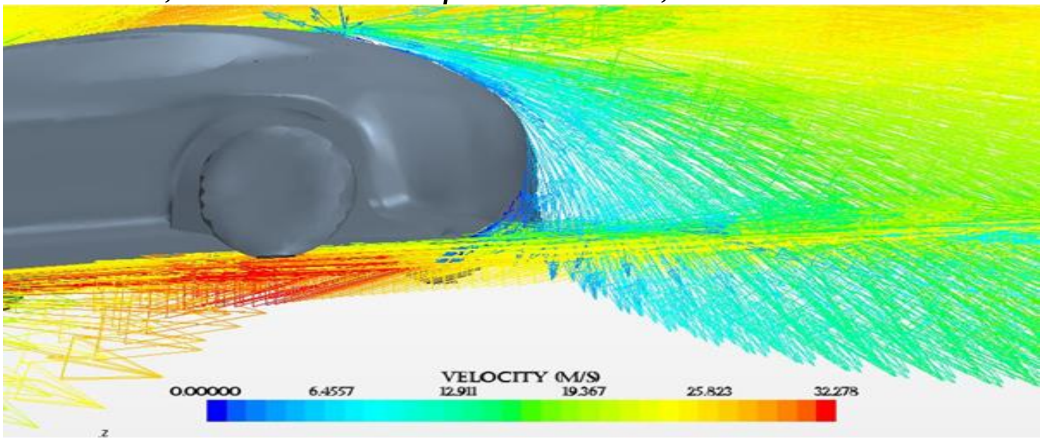

Fig. 9: Velocity Vector Contours around the rear end of proposed model.

4. RESULTS

The C_d value obtained for the proposed model was found to be 0.23 which is significantly lower than that of the initial model (0.38). This is mainly achieved due to the aerodynamic design which completely eliminates the wake created at the rear end of the vehicle and as it is evident that 80%-90% of the drag is due the rear wake that is produced [1], the proposed model has very little wake created as the air flowing over the roof follows the inclined path of the rear end, the flow separation is reduced to a high extent. The proposed model was designed to have smooth edges which helps air flow transition from the front of the vehicle to the back allowing for more air to reach the rear wake without losing much of its velocity. This helps in reduction of overall drag coefficient of the vehicle. The vortex generated at the back of the vehicle in the initial model is very prominent whereas the vortex in the proposed model is drastically reduced, also the flow separation of the boundary layer is minimized in the proposed model as is clear from the vector contours in Fig 5.

5. CONCLUSIONS

The drag characteristics of the initial model and the proposed model were studied and it was found that the proposed model has very favourable drag characteristics compared to the initial model. The proposed model displayed a favourable drag coefficient of 0.23 compared to the drag coefficient value of the initial model which was 0.38 at the speed of 25m/s, therefore indicating the decrease in the percentage of drag by 39.5% .the proposed model was found to be very efficient at minimising the wake formation at the rear of the vehicle and reducing the boundary layer separation across the body of the vehicle.

6. REFERENCES

[1] Altaf et al; Passive drag reduction of square back road vehicles, Journal of Wind Engineering & Industrial Aerodynamics 134 (2014) Pg 30–43

[2] W. Hucho et al; Aerodynamics of Road Vehicles, SAE Int 4th edition, (1993) Pg 485-537

[3] S. Maleki et al; Assessment of various turbulence models (ELES, SAS, URANS and RANS) for predicting the aerodynamics of freight train container wagons, Journal of Wind Engineering & Industrial Aerodynamics 170 (2017) Pg 68–80

[4] Jiyuan T et al; CFD A Practical Approach - Second Edition. (2012) Pg 170-190

[5] W. Hanfeng et al; Aerodynamic drag reduction of an Ahmed body based on deflectors, Journal of Wind Engineering & Industrial Aerodynamics. 148 (2016) Pg 34–44

[6] Dan Barbut et al; CFD analysis for road vehicles - a case study, Incas Bulletin, Volume 3, Issue 3, (2011) Pg. 15 – 22

[7] R. Grosche, G.E.A. et al; Research at DLR G. Ottingen on bluff body Aerodynamics, Drag Reduction by wake ventilation and active flow control, Journal of Wind Engineering & Industrial Aerodynamics 89 (2001) Pg 1201–1218

[8] B. Khalighietal et al; Aerodynamic flow around a sport utility vehicle - Computational and experimental investigation, Journal of Wind Engineering & Industrial Aerodynamics 107–108 (2012) Pg 140–148

[9] Gavin Dias et al; Aerodynamic Analysis of a Car for Reducing Drag Force, IOSR Journal of Mechanical and Civil Engineering. Volume 13, Issue 3, Ver. I (2016), Pg 114-118

[10] A.Anish et al; Modelling And Analysis Of A Car For Reducing Aerodynamic Forces, International Journal of Engineering Trends and Technology - Volume 47 Number 1 (20110 Pg 23-39

