DESIGN MODIFICATION OF CAR BONNET TO REDUCE THE FRICTIONAL DRAG

Ganesh Badiger[1], Aniket Dindore[2]

[1]B.E Student, Department of Mechanical Engineering, SCOE, Pune, Maharashtra, India.
[2]B.E Student, Department of Mechanical Engineering, SCOE, Pune, Maharashtra, India.

---***---

Abstract - *Requirement of High speed transportation have led to increasing capacity of the vehicles like cars. Though achievement of top speed is possible by increase in the vehicle power and transmission, majority of Engine power is lost due frictional drag force on the car applied by air. 50-60% of available energy is lost in overcoming this drag force. This paper focuses on reducing the frictional drag coefficient on the car body by modifying the bonnet design of the car. Maximum frictional drag force is found to occur at the speed of 85 – 110 kmph. In the following paper, the car bonnet area is modified such that the frontal area is reduced by 0.054 m^2 and the attack angle the bonnet has been decreased by an angle of 2.36°, resulting in reduction in friction drag. Governing equations used for the analysis is standard k- Ω .The initial car model is based on the design of Honda Amaze and was designed in Siemens NX 12.0. CFD analysis is performed on initial and final design of the cars in FLUENT 18.1 to determine the frictional drag coefficient and pressure difference between front and rear ends were analysed.*

Key Words: Geometry modification, Ansys Fluent, NX, drag force, pressure difference, CFD, aerodynamic drag.

1.INTRODUCTION

Aerodynamic forces are one of the most crucial parts of the automobile body design and is a keen topic for researchers. Many researchers and engineers have suggested different techniques for reduction of the Aerodynamic drag force occurring on the automobile body. Rigorous research and development in the field has resulted in optimised aerodynamic design of the body. Competition in high speed, manoeuvrability and control, fuel efficiency and safety have led to optimization of car body.

2. LITERATURE REVIEW

Numerous researchers, engineers and scholars have contributed their valuable work in the field of modification in automobile bodies. A literature review was conducted on the same.

Hassan [1] research was based on reduction of frictional drag by modification in diffuser angle of racing car. Diffuser angle is modified with the ground to get less pressure difference and hence drag reduction. Another method suggested by Hassan is of redirection of exhaust gases which lead to high acceleration of the racing car due to low wake formation.
Vignesh [2] made modification in windscreen inclination and hood angle to reduce the frictional drag coefficient of the car body. It was found that the hood angle needs to be less and windscreen need to be at an acute angle preferably less than 45° to obtain less drag force.

Gunpinar [3] stated a mathematical formulation for creating silhouette edges curving of the body for obtaining reduced frictional drag for a given wheelbase and track width with required top speed. Mathematical modelings for required specifications were obtained along with reduction in the frictional drag force.

Eric [4] studied the effects of exit angle in NASCAR Xfinity cars and hence suggested the changes needed to be made in the vehicle exit angle of spoiler and its result wake formation which leads to less aerodynamic drag force of the trailing car.
Corrado Groth [5] performed a numerical simulation on a Le Mans prototype car by Dallara (LMP1) 2001 model. The frontal wing was analysed with frequency and speed change for stability. Further the whole body was analysed using shell structures for vibration and flow characteristics.

Krzysztof Kurec [6] experimentally and numerically investigated slow control for an active rear mounting on a race car. Experimental model with 1:2.5 was studied using Wind Tunnel and CFD analysis was performed using the same wind tunnel configuration. Different spoiler angles resulted in variation in drag force, downforce and pressure difference.
W. Kieffer [7] performed turbulence k-ε CFD analysis on front wing of Mazda Formula One race car. He suggested that an angle below 12° leads to stalling condition due high drag force. CFD analysis was done on STAR-CCD using a 2D model.

3. GEOMETRY MODELLING

The car body is created using SIEMENS NX 12.0. The model of the car is derived from the Honda Amaze 2017 model. Geometry was created using solid modelling by symmetric properties. Silhouette edges were obtained by providing parabolic fillets to the edges. Below shown is the car Geometry.

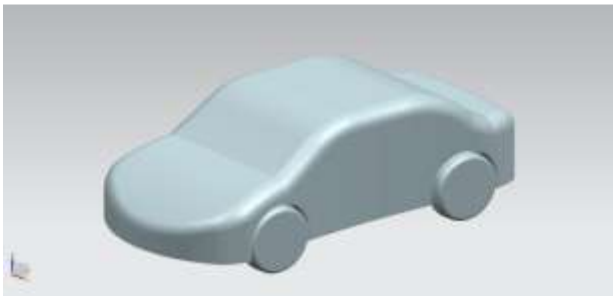


Fig -1 Fig -2

1. Fig 1 shows the initial model of the car without any modifications in the geometry. The bonnet is modelled as per the Mass produced model of Honda Amaze
2. Fig 2 shows the modified design model of the car. The bonnet area is modified which eventually provides a lesser area as compared to the initial design.

The car geometry is modified such that the difference in the attack angle is 2.63. The initial design has a bonnet area of 0.728 m2 and the modified design has a bonnet area of 0.674 m2.

Table -1: Specifications of the designed car

Parameter	Dimensions
Length	4.113 m
Width	1.963 m
Height	1.216 m
Surface Roughness	20 microns

4. ANALYSIS IN ANSYS FLUENT

4.1 CAD importing and clean-up

Finite Volume Method has been utilized in analysing the model. Hence an enclosure has been created for the car model with dimensions of 12600 x 3600 x 3435 mm. The car model was then created as a cavity in the enclosure by Boolean operations. Fig 3. Represents the car geometry in enclosure

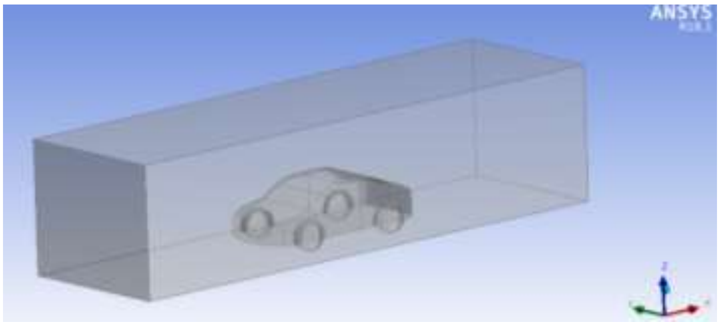


Fig -3

4.2 PRE-PROCESSING

Meshing was performed using with target element size of 12 mm. Tetrahedral elements were generated since they offer required skewness for air flow. Fig 4 represents the meshed model of the car

The meshing parameters are as follows:
Element type: Tetrahedral
Element order: Linear
Element target size: 12 mm
Number of Nodes: 86051
Number of Elements: 453905

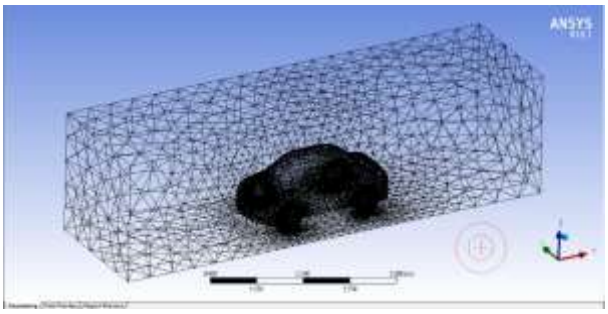


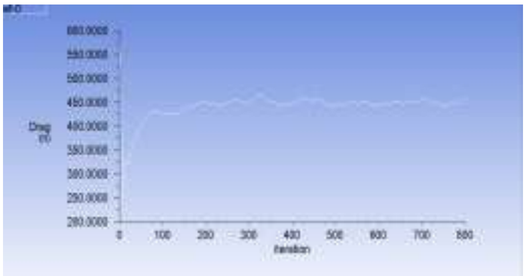
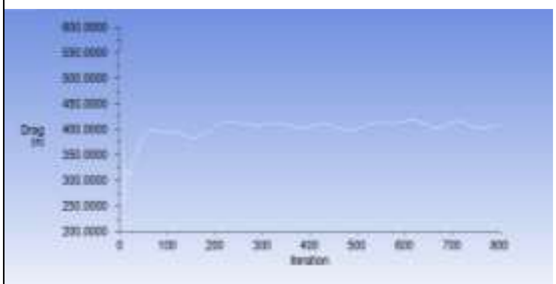
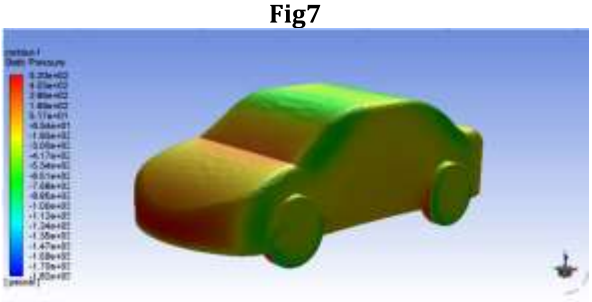
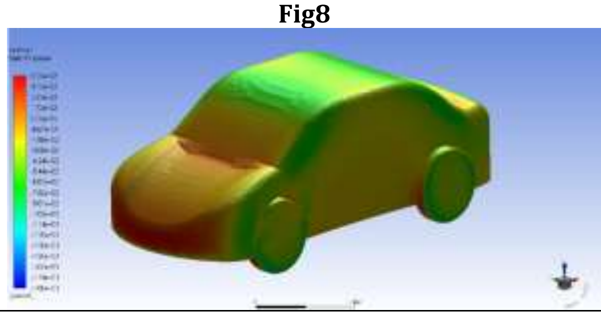
Fig -3

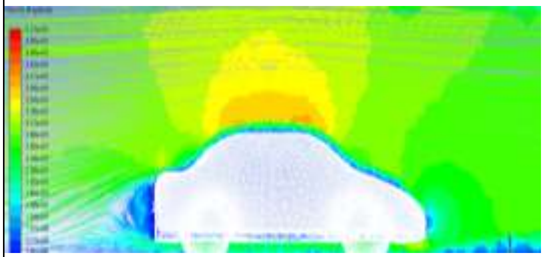
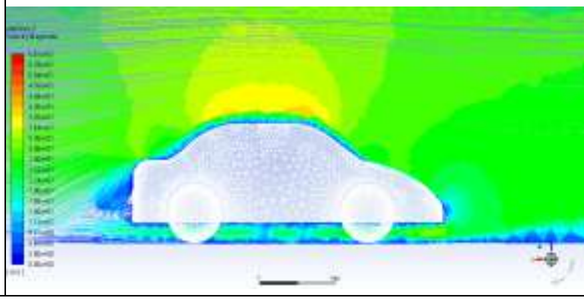
4.3 BOUNDARY CONDITIONS AND GOVERNING EQUATIONS

The meshed model is defined with inlet, outlet and wall bodies. The inlet condition is velocity of 100 kmph. The governing equation for solving used in standard- k- omega. The wall roughness of car is 20μm.

5. RESULTS AND DISCUSSIONS

Table -2: Comparison of results for the initial design of car and bonnet modified design

Parameter	Initial Design	Modified Design
Drag Force v/s Iterations	**Fig 5** 	**Fig 6** 
Pressure Contours	**Fig7** 	**Fig8** 

Velocity Profile	**Fig 9** 	**Fig 10** 

Drag force obtained for the initial design of the car is 450 N (Fig 5) and that for the modified design of the car is about 408 N (Fig 6). Reduction of 42 N of force is observed on change in the geometry of bonnet in the car.

Pressure contours observed in Fig 7 and Fig 8 for initial and modified design. Pressure obtained for initial design at frontal end is 520 Pa and at the rear end is 169 Pa. whereas for the modified design the pressure at the front is 530 Pa and at the rear end is 291 Pa.

The difference between front and rear side pressure in Fig 7 is 351 Pa and in Fig 8 is 239 Pa. The pressure difference obtained in initial design is high as compared to that of modified design of the car. The pressure difference leads to a resistive force for the motion of the car. Reduction of this pressure difference leads to less resistive force and henceforth leading to better performance of the car. Hence it can be concluded that the modified design of the car will have better performance parameters compared to that of the initial design.

Fig 9 and Fig 10 represents velocity contours of initial and modified design of the car. The maximum air velocity obtained in both the cases is situated on the top of the car. The velocity above the hood for Fig 9 is 52 m/s and for Fig 10 is 56 m/s.

Table -3: Result Table

Parameter	**Initial Design**	**Modified Design**
Drag Force	450 N	408 N
Pressure Difference	351 Pa	239 Pa
Velocity	52 m/s	56 m/s
Drag Coefficient	1.602	1.569

6. CONCLUSION

The frictional drag coefficient is reduced by 0.03 and also drag force by 42 N. the pressure difference has been reduced by 112 Pa resulting in low resistive force in opposite direction. From the results obtained it can be concluded that the modified model is optimized from the existing model. This will lead increase in efficiency and speed compared to initial model.

REFERENCES

[1] S.M. Rakibul Hassan*, Toukir Islam, Mohammad Ali, Md. Quamrul Islam, "Numerical Study on Aerodynamic Drag Reduction of Racing Cars," 10th International Conference on Mechanical Engineering, ICME 2013
[2] Vignesh S*, Vikas Shridhar Gangad, "Windscreen Angle and hood inclination optimization for drag reduction in cars," 14th Global Congress on Manufacturing and Management (GCMM-2018)
[3] Erkan Gunpinar a,*, Umut Can Coskun a, Mustafa Ozsipahi a, Serkan Gunpinar b, "A Generative Design and Drag Coefficient Prediction System for Sedan Car Side Silhouettes based on Computational Fluid Dynamics," Elsevier Journal Ltd. Computer-Aided Design 111 (2019) 65–79
[4] Eric Jacuzzi*, Kenneth Granlund "Passive flow control for drag reduction in vehicle platoons", Journal of Wind Engineering & Industrial Aerodynamics 189 (2019) 104–117, Elsevier Ltd
[5] Corrado Grotha, Andrea Chiappaa, Stefano Porziani, " Multiphysics numerical investigation on the aeroelastic stability of a Le Mans Prototype car ", AIAS 2019 International Conference on Stress Analysis

[6] Krzysztof Kurec , Micha_ Remer , Tobiasz Mayer ,Sylwester Tudruj , Janusz Piechna, “Flow control for a car-mounted rear wing”, International Journal of Mechanical Sciences
[7] W. Kieffer, S. Moujaes, N. Armbya, “CFD study of section characteristics of Formula Mazda race car Wings,” Mathematical and Computer Modelling 43 (2006) 1275–1287, 2005 Elsevier Ltd