



## DRAG REDUCTION FOR A FAST BACK PASSENGER CAR (LOGAN)

Suliman Mohammed Mohammed Ali<sup>1</sup>, Jalal Mohammed Zayan<sup>2</sup>

Department Of Mechanical Engineering, Kulliyah Of Engineering , IIUM

\*Corresponding author email: [solli22@yahoo.com](mailto:solli22@yahoo.com)<sup>1</sup>, [zayan\\_mohammed@yahoo.co.in](mailto:zayan_mohammed@yahoo.co.in)<sup>2</sup>

### ARTICLE DETAILS

#### Article History:

Received 3 July 2017

Accepted 3 October 2017

Available online 5 December 2017

#### Keywords:

component, formatting, style,  
styling, insert.

### ABSTRACT

The car manufacturers all over the globe are intensively competing among each other in terms of the design and fuel consumption factor. Nowadays the reduction of drag is becoming a very important challenge for all the car manufacturers as they are producing powerful cars with better fuel consumption in the market regulated with law reinforcement on fuel emissions and consumers. Lower drag provides better performances such as higher top speed and better stability. It also often lowers aerodynamic noise and greenhouse gas emission above all decreases in fuel consumption. However, modern designs of cars tend to go higher and wider and thus they have higher frontal areas due to the functional, economic and aesthetic requirements. Increasing frontal area of the vehicle tend to increase the drag force acting on the vehicle which is proportional to the dimensionless drag coefficient and the projected area of the vehicle. Consequently, to hold or even decrease the drag on a car that has a larger frontal area, tremendous effort has to be made. In this research both experimental and numerical analysis have been done to an existing model of a car with some various aerodynamic add-on devices that can be attached to car and reduce aerodynamic drag of the vehicle without comprising on its main design features.

## 1. INTRODUCTION

The drastic and continuous increase in the price of the automotive fuel and along with the uncertainty of future supply has created widespread interest in vehicles with high efficiency including passenger cars, pickup trucks and commercial vehicles all over the world. The passenger cars account for about approximately 74% of the total motor vehicle annual production in the world [1]. In 2012, for the first time in history, over 60 million cars passenger cars were produced in a single year (or 165,000 new cars produced every day). These manufacturers facing intense competitiveness in terms of higher fuel efficiency demands by the buyers. Therefore, improving the fuel economy of cars will have tremendous impact on energy security, emission of greenhouse gas and cost of fueling when gasoline price rises. Today, auto manufacturers are competing intensely to produce powerful passenger cars with better fuel efficiency in the market regulated with law reinforcement on fuel emissions and consumers' need for bigger size cars with more horse powers and seating capacity. The Energy efficiency of any road vehicle can be improved by reducing the total structural mass, using engine with higher thermally efficiency, or altering the exterior body shape to reduce the aerodynamic drag. According to US department of energy, in urban driving aerodynamic drag accounts for 2.6% of the 12.6% of fuel energy being used to propel the car [2]. Since the aerodynamic drag increases at higher speeds, the aerodynamic drag on a highway driving accounts for 11% of 20% fuel energy needed to propel the vehicle. Therefore, improving vehicle aerodynamics is one of the factors that play crucial role for getting better mileage and better performance including the handling of the vehicle especially at high speeds. The body shapes of passenger cars are primarily designed to meet the functional, economic and aesthetic requirements along with the comfort ability factor while travelling long distances. Aerodynamic drag is often the consequence of the body shape designed to meet the functional, economic and aesthetic design constraints. The use of add-on device enables us to reduce the aerodynamic drag of the vehicle without compromising on its main design features. Studying flow over a passenger car with add-on devices is costly in wind tunnel due to cost for the setup as well as number of runs required for successful drag reduction and optimization of the add on devices. With the use of CFD these costs are avoided, and multiple runs can be set up at the same time for comparison and optimization. It is motivated for this research by using a CFD approach to analyze the flow over a passenger car with add-on device such as horizontal grooves on the roof cover, spikes at the rear ends for drag reduction.

## 2. MODEL PREPARATION

The model of the fast back passenger car (Renault Logan) was prepared by scaling down the actual size of the car taken from its catalogue. The numerical model was scaled down to 1:15. And for the actual physical model which was used for the flow visualization was about 1:30 of the actual size of the car.

### 2.1 Numerical model

The geometry was created in CATIA figure 1, and later transformed into Initial Graphics Exchange Specification (IGES) file format. The neutral data format is imported to Star Design for preliminary setup. A computational grid of C-H type was generated from this boundary region. Sufficient spacing of 15 diameters between the car model and the boundary region was selected. The boundary region was defined as free-stream control volume. The setup was converted into Parasolid Transmit Files and was exported to STAR CCM+ for meshing and further surface treatment, refinement, modeling and running the simulation.

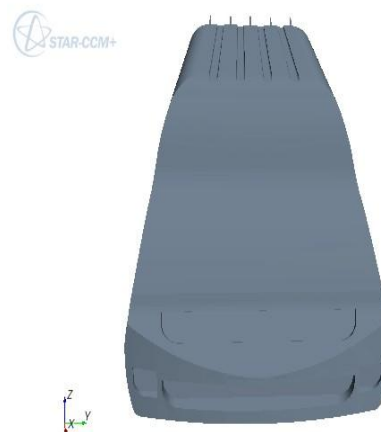


Figure 1: Modified model

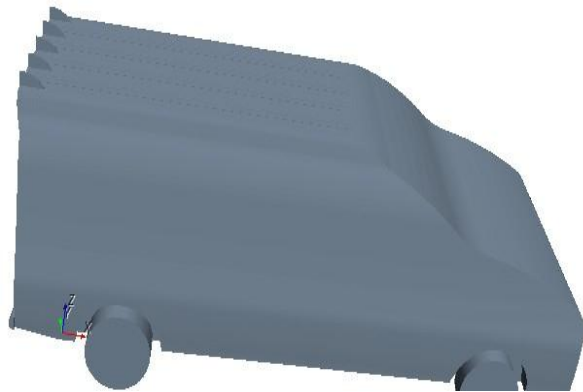


Figure 2: Modified model



Figure 3: Modified model.

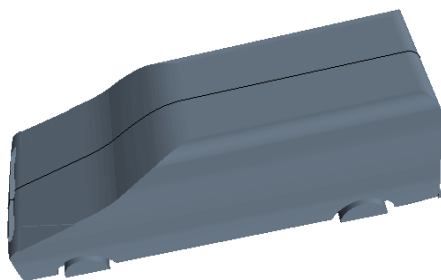


Figure 4: Original model.

### 2.2 Physical Model

The physical model for the experiments conducted were made of a wooden structure which was very finely hand crafted to match the exact features of the actual car. The model was scaled down to a ratio of 1:30.



Figure 5

### 3. EXPERIMENTAL SETUP

The experiments were conducted using an open loop low speed wind tunnel. The car model was bolted from the bottom. The velocity of the flow was then matched to that of the numerical setup and the air flow was switched on. Smoke generator was used at the front end of the model and a stream of continuous white smoke was made to flow around the car. This was to visualize and to study the vortex pattern on all sides of the car model.



Figure 6



Figure 7



Figure 8

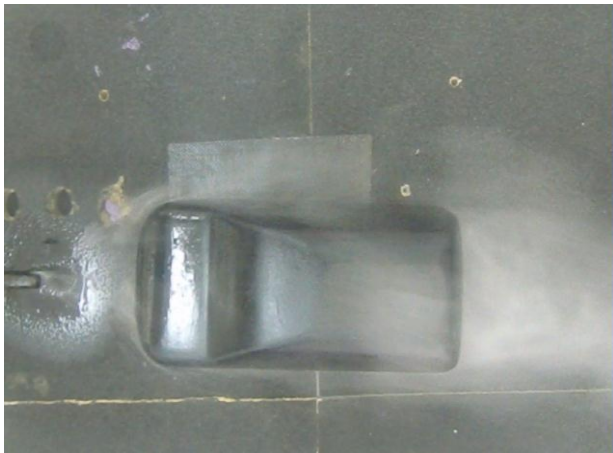


Figure 9



Figure 10

4. RESULTS AND DISCUSSION

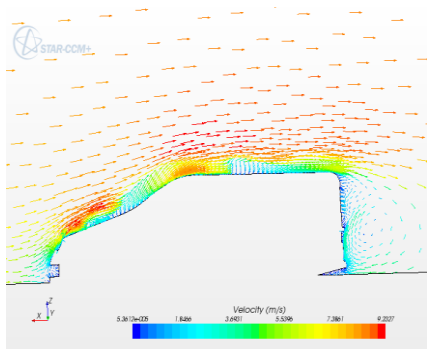


Figure 11: Velocity

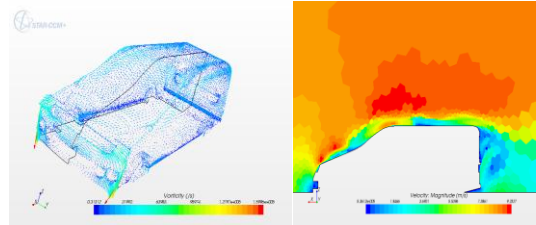


Figure 12: Vorticity contour.

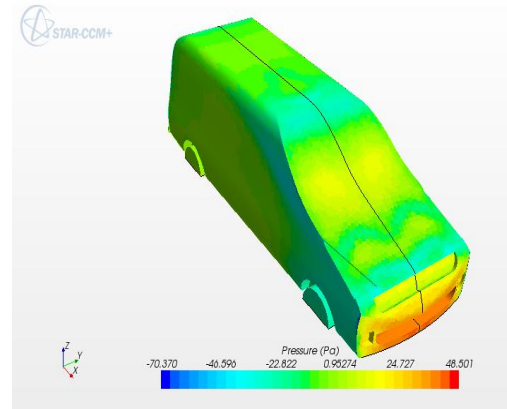


Figure 13: Pressure

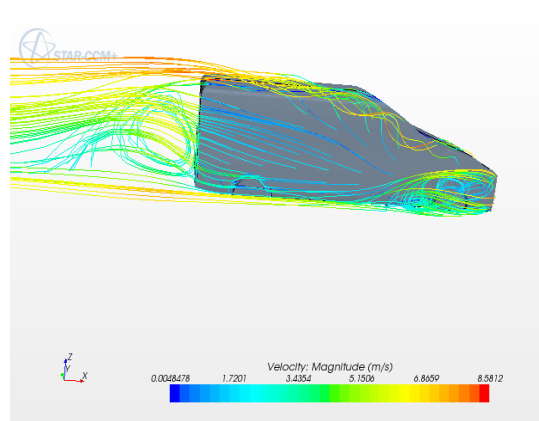


Figure 14: Streamlines

Figure 13 shows static pressure distribution over the car surfaces, indicating that pressure was very high on the grill of the vehicle where the velocity of the flow becomes zero and stagnation point was created. Figure 13 also shows relatively high static pressure created at the junction of the windshield with the hood of the vehicle [3]. Both front and rear tires also experience high static pressure, but the front wheels were subjected to slightly higher static pressure than the rear. On the sharp edges of the vehicle with the A-pillar, the edges of the hood, grill junctions with side-frame and edges of the wind shield, flow separation was expected to occur, and the static pressure was low. The pressure difference created between the front and rear end of the vehicle causes the net aerodynamic force acting on the vehicle to generate a drag against the motion of the vehicle. Figure 11 shows the wake profile for the car (velocity vector on iso-velocity surface at 8.33m/s), indicating that turbulent wake was formed behind the car. The vectors figures 5, 6 7 and 9 indicate the flow separation occurring at the rear edge of the car and the vortex created in the rear back of the car. It also indicates the downwash created at the outer edge of the tailgate behind the car. The aerodynamic drag and lift coefficients computed from the simulation were  $c_d = 0.36$  and  $c_l = 0.554$  respectively.

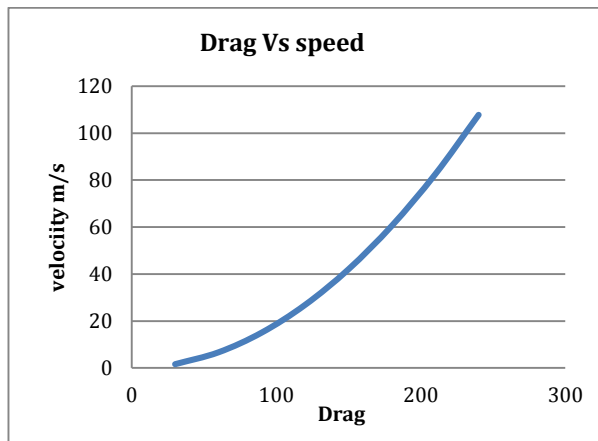


Figure 15: Drag Vs. velocity (m/s)

## 5. CONCLUSION

The drag coefficient from CFD simulation was predicted less than the real life drags coefficient of passenger car. This phenomenon was also observed by Mukhtar, Britcher and Camp, when they conducted CFD simulation on generic model of the passenger car used in their experimental investigation. These might be due to the fact that the generic car model lacks accessories such as side mirror and windshield wipers. Also in the case of the generic passenger car model there were no exposed axles, underbody, radiator cooling vents and many cavities on the surface of the vehicle that connects the inside of the vehicle to the flow.

## REFERENCES

- [1] <http://www.worldometers.info/cars/>
- [2] U.S. Department of Energy Energy Efficiency and Renewable Energy <http://www.fueleconomy.gov/feg/atv.shtml>
- [3] Mokhtar, W., Britcher, C., Camp, R. 2009. Further Analysis of Pickup Trucks Aerodynamics. SAE Paper, 01,1161.

