A Parametric CFD Study of a Generic Pickup Truck and Rear Box Modifications

Wael Mokhtar; Md Maruf Hossain, and Samira Ishrat Jahan,
School of Engineering, Grand valley State University,
Grand Rapids, Michigan, USA.

Abstract

Aerodynamics of ground vehicles is becoming a very important concern for all the car manufacturers to produce heavy duty vehicles with better gas mileage. Fuel efficiency of pickup truck is poor in general due to the geometric structure. Lower drag provides better performances such as higher fuel efficiency and better stability. In this study the air-flow pattern inside the rear box of a generic pickup truck and corresponding aerodynamic drag have been analyzed. In addition, a comparison in between few market available add-on devices aimed to improve flow conditions was carried out.

It has been proven that the negative pressure zone induced in the rear box of the pickup trucks creates flow separation and strong re-circulating vortices and thus produces higher drag values. The focus of the current study was to analyze the flow topology using different add-on devices for diverting the flow and to compare their effects on the flow patterns contributing towards drag and lift.

In present study, more detailed model was used for the generic pickup truck, which includes side mirrors, front grill, wheel cavity, rims and small underbody details to have better representation of the case in focus. Numeric simulations for CFD were performed on STAR CCM+ developed by CD-ADAPCO Inc. Symmetrical models was used to achieve better accuracy with less computational burden.

Standard post-processing tools were merged with unit-less parameters such as pressure coefficient and total pressure iso-surface to understand and compare between the effects of pressure distribution for different add-on devices. By analyzing the streamlines and velocity contours, comparison of effectiveness for different modifications was concluded.

Introduction

Once used only as a work tool with few passenger comfort features, in the 1950s' consumers began purchasing pickups for lifestyle reasons and by the 1990s less than 15 percent of owners reported use in work as the pickup truck's primary purpose. Today in North America, the pickup is mostly used like a passenger car and accounts for about 18% of total vehicles sold in the US.[8] In the United States and Canada most pickup trucks are used primarily for passenger transport, agriculture, and commercial uses.

Pickup truck is a representation a multi-purpose vehicle in American culture that can be used for passenger transport, agriculture, also in law enforcement, the military, fire services since it offers a better towing capacity. Moreover, the body of most the pickup trucks now a days is made with strong metal that ensures passengers' safety in the event of an accident. The frames that are used in the boxed style of the truck give extra protection.

Pickup trucks are more durable than most other vehicle types. It is easy to transport just about anything with a pickup truck due to its rear bed which has made it unique, compared to other road vehicles. The rear beds are separated from the cab, used for carrying and pulling heavier loads than any other type of passenger vehicle.

While the rear box has the advantage of giving space for transporting goods and stuffs, at the same time it is responsible for the low fuel efficiency of the pickup trucks. The recirculation flow over the bed and the reverse flow in the wake have significant effects on the aerodynamic drag. The vortex inside the bed is a recirculation flow, which is detached from the rear edge of the roof and enters into the rear box cavity at the tailgate[4].

In this study three modification tools have been applied to observe the effects on flow structures. Also the effects of the changing flow pattern on the aerodynamic drag and lift have also been observed. There are features that have made this study unique from other studies done so far. The addition of side mirrors which has significant impact on flow pattern is one of them. Most of the studies found in literature review have been done on 6-6.5 feet rear box but this study has been performed with an 8 feet rear box. Some other additional details like foot stand, front grill pattern etc. have been included to make the model more realistic.

Literature Reviews

In a CFD study on the box configuration of a 6.5 feet box pickup truck done by Mokhtar et al[2] shows the effect of open box, a box with a tonneau cover and a pickup truck with a flat cap on the aerodynamic drag of pickup truck. It has been found from this study that the tonneau cover case has the lowest drag (around 0.297) and the cap cover has the highest drag (around 0.324). It was expected that the drag would be lower than that of open box configuration (around 0.315) in the cap configuration but due to increased surface the skin-friction component of drag also increased. So the C_d for the cap configuration is the highest.

The equation for drag co-efficient is,

$$c_d = \frac{2F_d}{\rho u^2 A} \quad \dots \dots \dots \dots \dots (1)$$

Where,
C_d = Co-efficient of drag
F_d = Drag force
ρ = Fluid density
u = Fluid velocity
A = Frontal area

In another study done by Mokhtar et al[1] described the variation of aerodynamic drag for different configurations of tailgate of the pickup truck with an overall length of 5.3 m. In this study the

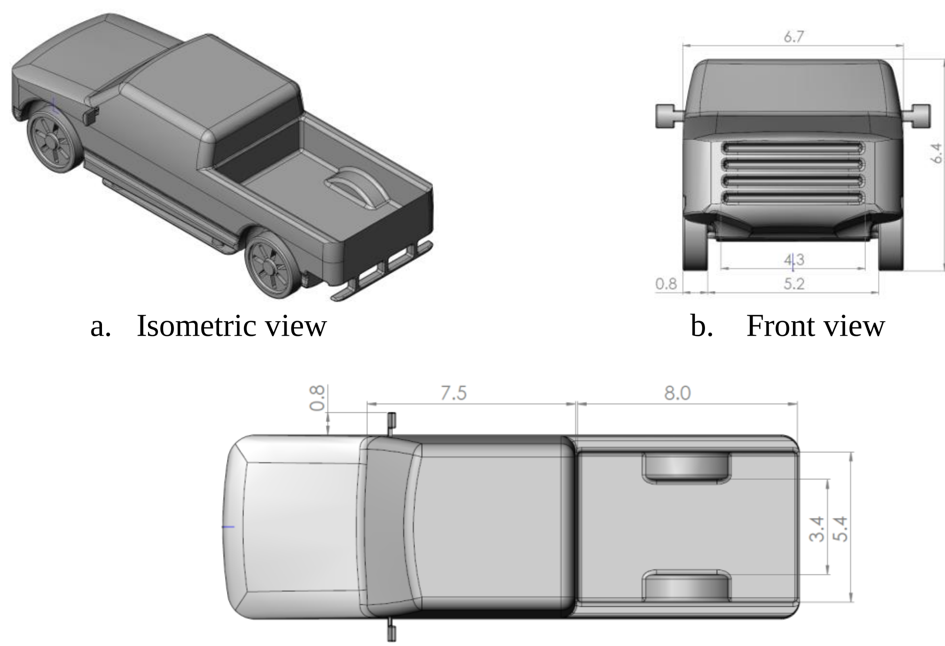
tailgate raised configuration exhibited the lowest drag (around 0.455) of the three, tailgate down (around 0.470) had higher drag and tailgate off (around 0.480) had the highest drag. One additional bed configuration, covered bed was tested which was quite effective in reducing drag.
Ha et al.[4] used both CFD simulation and wind tunnel on a 1/10th scale generic pickup truck to show that the drag coefficient was reduced with increasing flap length and downward angle despite the enlarged reverse flow in the wake. The drag coefficient decreased with the increase in the downward angle up to 12° for all length of flap. As the angle was further increased, C_d started to increase again.

Maxwell et al.[7] used wing structure mounted behind the top rear of the cab and a cover over the rear portion of the pickup bed which resulted in 5% to 6% drag reductions respectively.

With the use of full-scale wind tunnel testing and Computational Fluid Dynamics (CFD) simulations Taniguchi et al[6] showed the effects of tailgate spoiler, front spoiler, frame side deflectors and rear wheel-house covers on aerodynamic drag of pickup truck. As a result of adopting these devices there was 12% improvement on aerodynamic drag over the baseline model.

Model Design

This study focuses a generalized model for a pickup truck with 8 feet rear box. Unlike most of the models found in the literature, the geometric model includes side mirror, foot support and front details to obtain better. The major dimensions (unit feet) are presented in the figure 01.



a. Isometric view

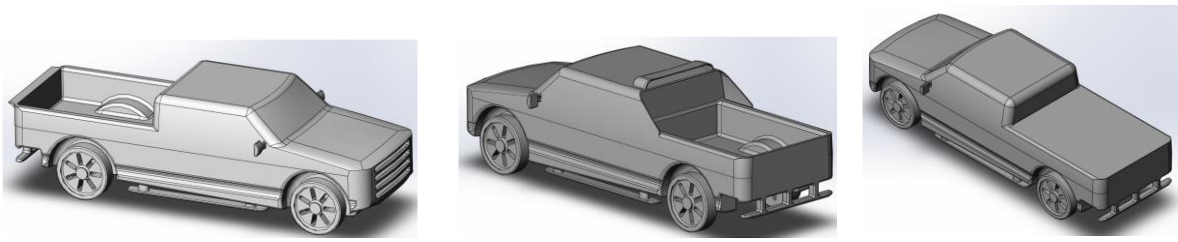
b. Front view

c. Top view

Figure 1: Major Dimensions of the baseline model (unit - feet)

Modifications considered in this study revolved around the rear box of the pickup truck, more specifically the ones that reshape the flow structure of the wake zone of the cab. Focus was on the

market available modification tools. Another interest was to observe the pressure distribution of the rear bed of the pickup trucks by geometric modifications directly effecting the flow structure in the volume of interest. The three modifications that have been done in the rear box are:
Modification 1: Wider tailgate top
Modification 2: Cab roof spoiler with back cabin screen
Modification 3: Tonneau cover



a . b . c .

Figure 2: a. Wider tailgate top, b. cab roof spoiler with back cabin screen, c. tonneau cover

CFD Study

The domain size was 7 times the length of the vehicle; three times length in front and three times long after the tailgate. Domain height was four times the height of the pickup model when the pickup was in touch of the ground. And width of the domain was six times when it was centered in the middle.

I. Mesh Model

One of the major challenges of this study was to develop a mesh model without sacrificing any detail of the model with a minimum number of cells to reduce computational power. The following mesh models have been used.

a. Surface remesher to omit specific surfaces or boundaries and preserve the original triangulation from the imported mesh.
b. Polyhedral mesher in order to get numerically more stable, less diffusive, and more accurate solution
c. Prism layer mesher to capture boundary conditions properly.

For capturing flow patterns around pickup truck and wake zone in detail with the least possible number of cells, volume control has been used. So the overall number of cells was only 4.8 million for base model with a reasonably refined mesh around the pickup truck.

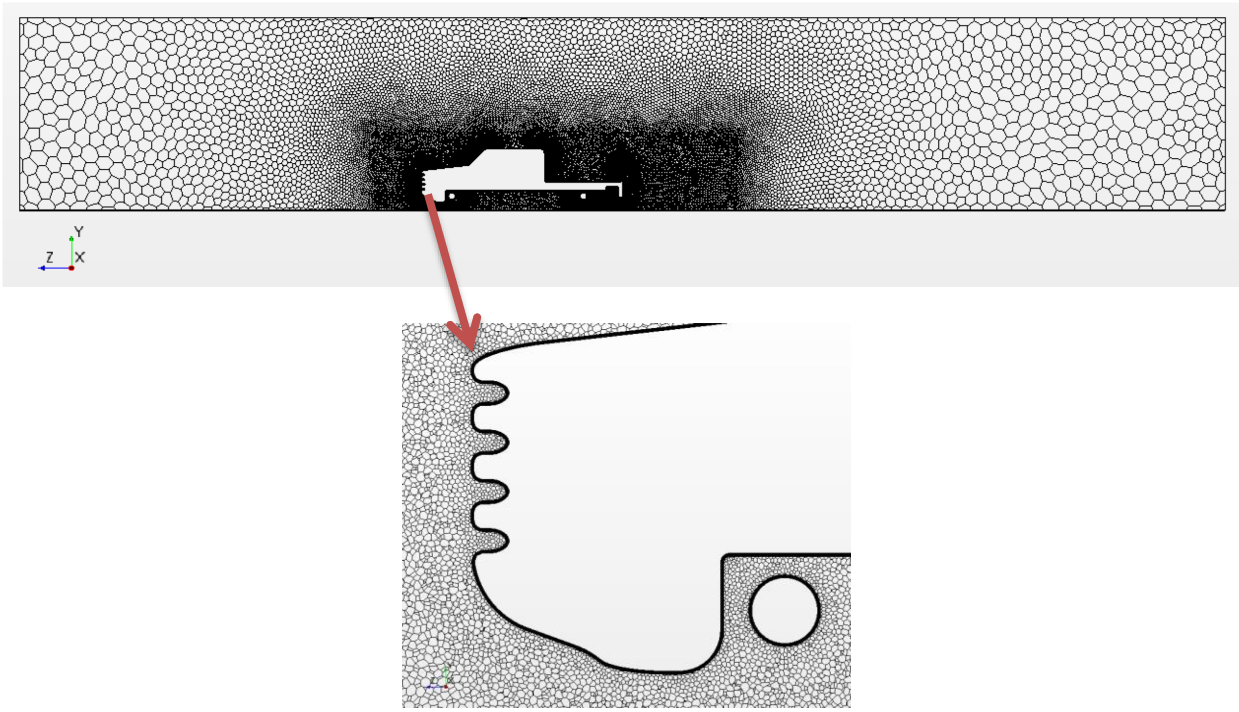


Figure 3: Volume mesh with prism layers around pickup truck

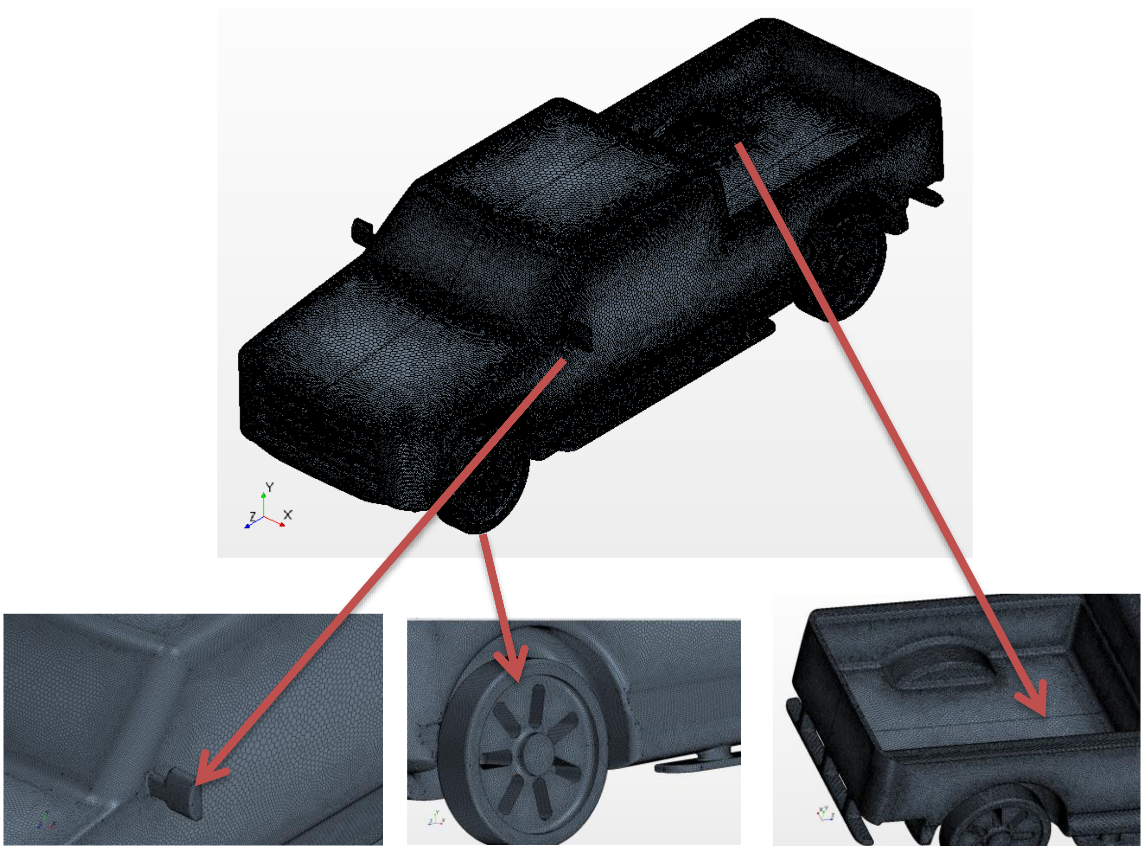
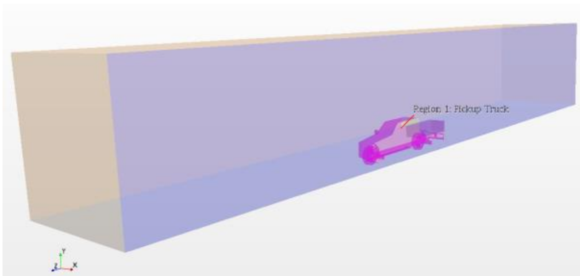


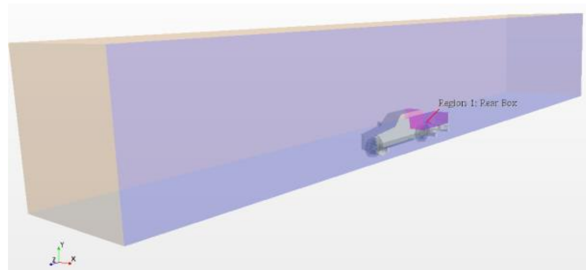
Figure 4: Volume Mesh with close view of side mirror, wheel, and rear box

II. Boundary Conditions

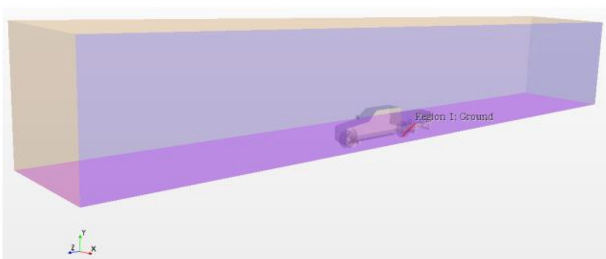
The study has been performed with half symmetric model. The entire domain has been divided into five regions: a. Pickup truck, b. Rear box, c. Ground, d. Air stream , e. Symmetry plane. The ground has been considered as moving with highway speed and there is no rotation in wheels.



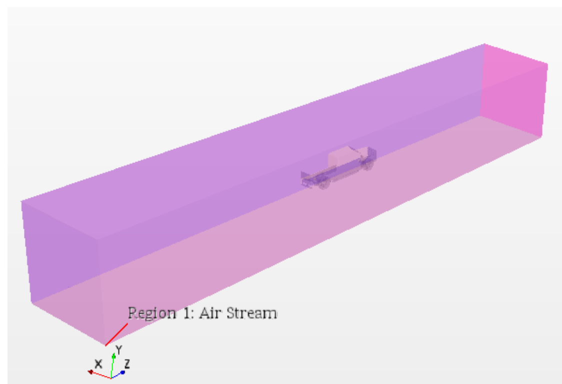
a. Pickup truck



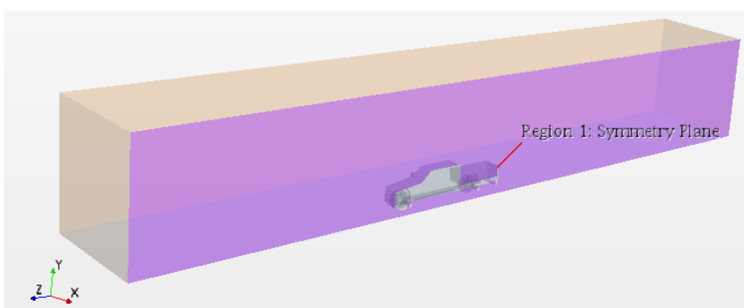
b. Rear box



c. Ground



d. Air stream



e. Symmetry plane

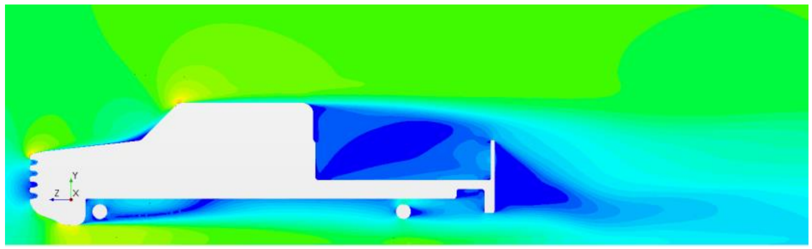
Figure 5: Regions

a. Solver settings

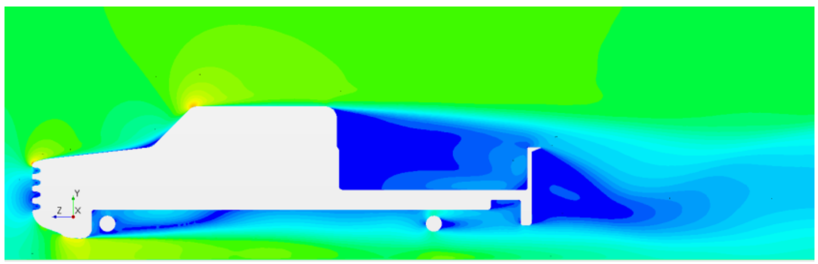
For the present study, the flow has been considered as segregated and K- ε model has been used since it predicts well far from the boundaries (wall). Specially for aerodynamic blunt bodies K- ε turbulence model is more preferable. As use of RANS (Reynolds Averaged Navier Stokes) makes it possible to simulate practical engineering flows and it has reduced computational requirements, RANS is considered here.

b. Simulation Results and Analysis

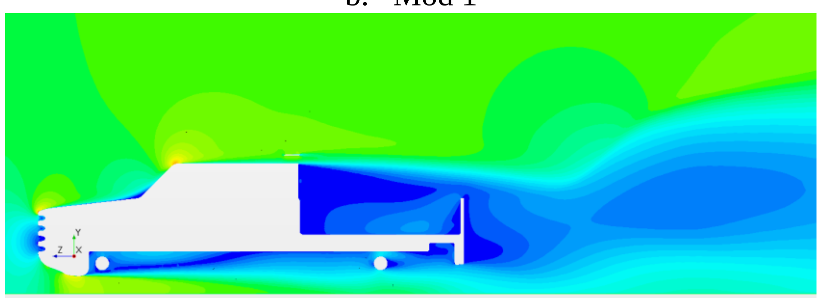
To observe the flow patterns in the wake zone several plots are taken in the planes parallel to the symmetry plane. Vortex formation are clearly visible in these velocity plots and some significant differences are marked which are presented in the figure 6.



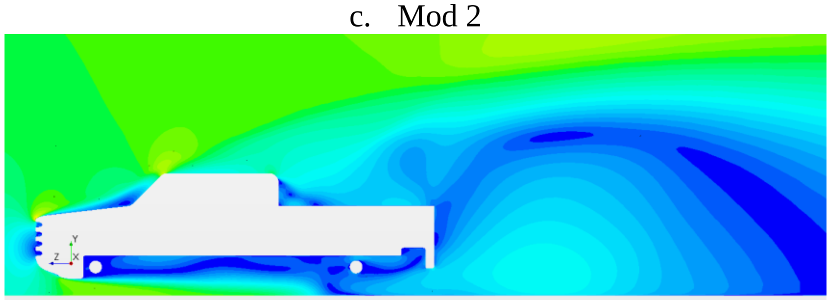
a. Baseline



b. Mod 1



c. Mod 2



d. Mod 3

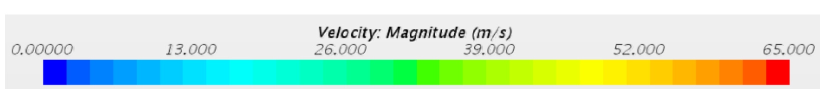


Figure 6: Velocity distribution in plane parallel to symmetry plane

Modification 3 presents the most interesting pattern as the vortex zones are shifted significantly further away from the pickup truck rear box. Because of the closed rear box geometry, the cabin downstream flow slams on top of the tonneau cover and blends with the flow from the underbody; in turn forms the vortex further down the stream than the other modifications and the baseline model. As a result, it has the least drag value.

On the other hand, modification 1 presents the largest vortex formation in the wake zone of the pickup but moderate vortex in the rear box. The baseline has the highest vortex formation inside the rear box yielding the highest drag value in the group.

Shifting core of the vortex is another phenomena which marks each model's flow characteristics. This is very important in the resulting drag values as it defines when the stream will hit the tailgate at what frequency shaping the wake zone flow characteristics.

Velocity contours in the plane parallel to ground reveal unique signatures of each modification where the effects of side mirrors in the wake zone can be identified clearly. (figure 8) The vector plots of the flow structure in the close up views show the vortices created in the wake zone of the side mirror. And for each simulation, these effects are noticeable and influencing the downstream flow.

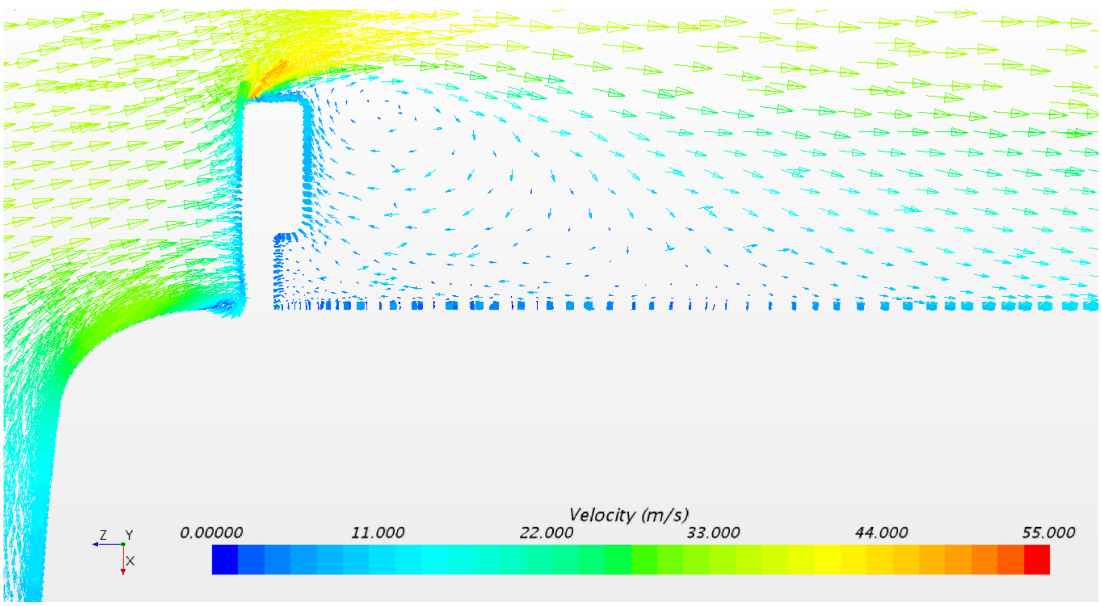


Figure 7: Flow around side mirrors

The scalar plots in the figure 8 displays another phenomena unique to each modification. For baseline, the flow pattern defined by the side mirror is conical shaped in the wake zone. For Modification 1 this pattern stays somewhat similar but in modification 2 it takes a sharp converging pattern followed by a diverging pattern. For modification 3 this layer is forming a diverging pattern. These patterns are captured at a height of 1.45 meter from the ground.

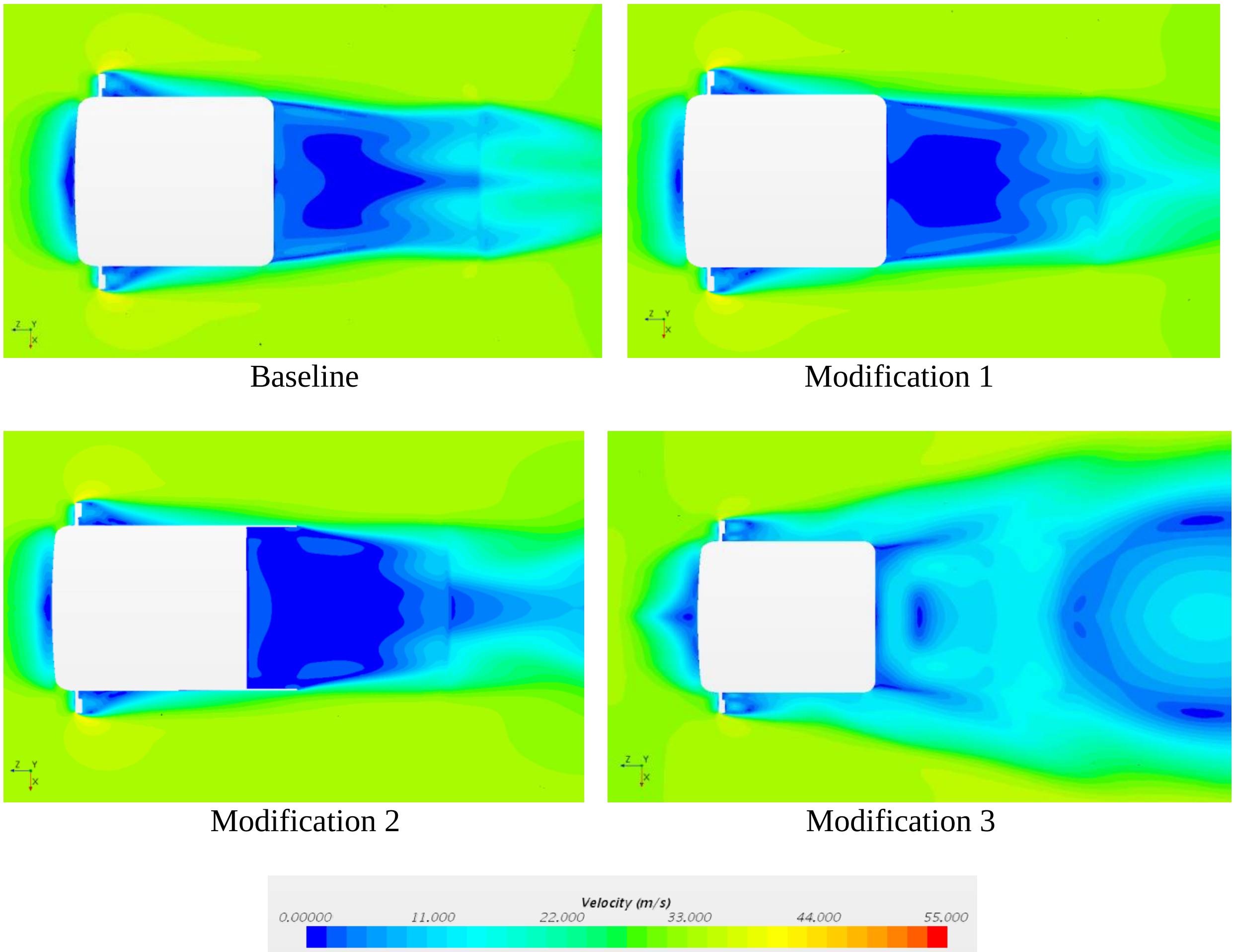
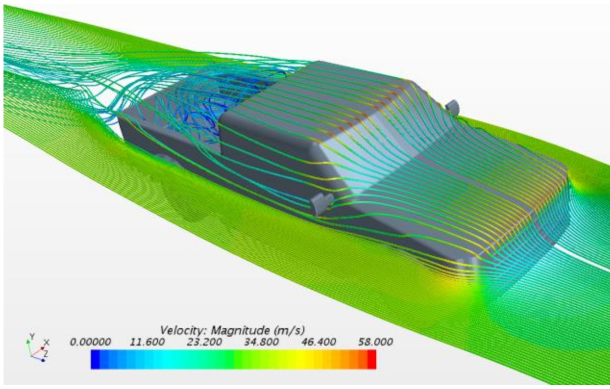


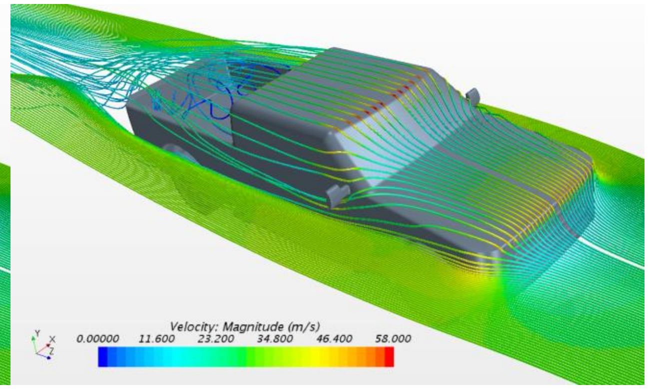
Figure 8: Velocity contours in plane parallel to ground at y = 1.45 m (At ground, y = 0 m)

As it can be noticed from the plots, (figure 8) that the vortex is shifted in the wake zone of the car in the case of modification 3, the overall drag value is the smallest in the group. The other modifications along with the baseline shows the streams at the level 1.45 m from ground are collapsing onto the tailgate inside the rear box at different levels which in turn directly affect the drag values. This hypothesis can further be supported by observing the streamlines later in the result section. (figure 9)

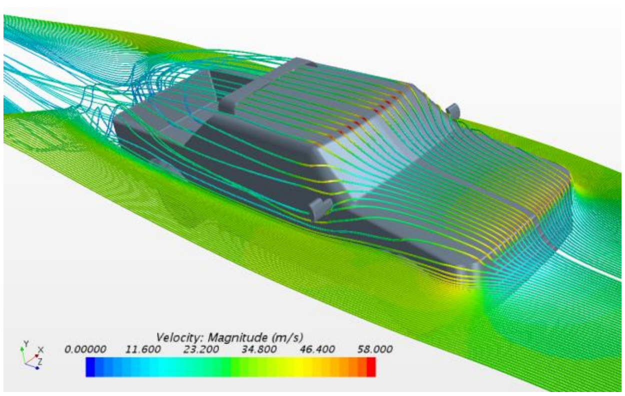
Plots taken in the plane parallel to the front, shows the flow effected by the wheel cavity and underbody geometry. It can also be noted that the side mirror geometry leaves a signature in the plots as the flow develops along the car body. This effect is also visible in the iso-surface plot for total pressure in forms of cones and streamline plots.



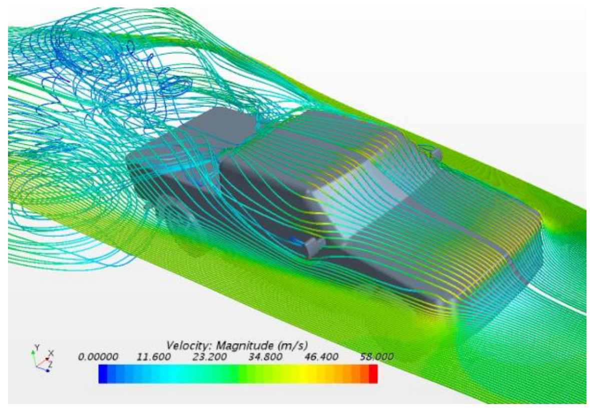
Baseline



Modification 1

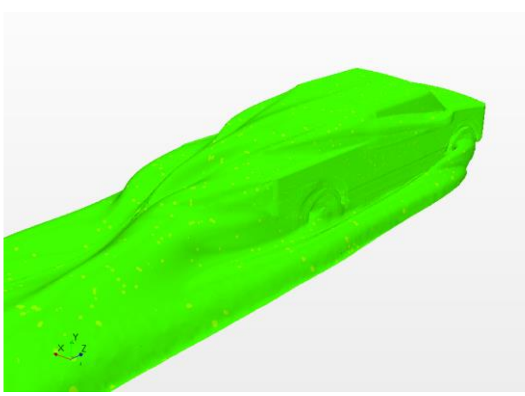


Modification 2



Modification 3

Figure 9: Velocity streamlines



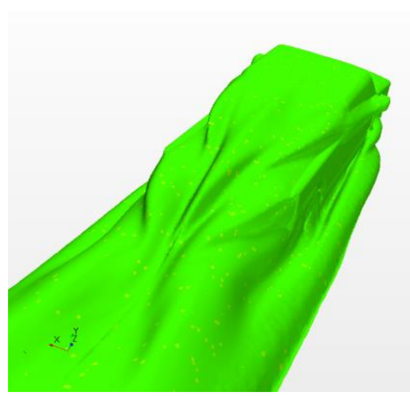


Figure 10: Total pressure iso-surface

The value of pressure coefficient is 1.00 at the front of the pickup where a stagnation point is created (figure 11).

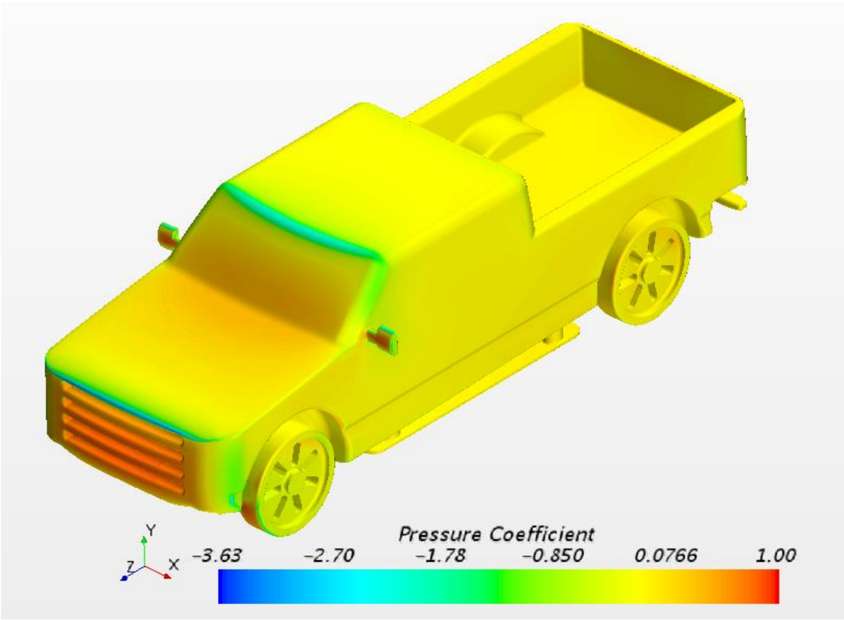


Figure 11: Pressure coefficient

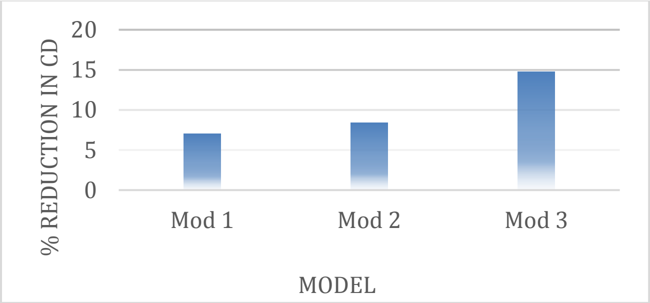


Figure 12: Percentage reduction in drag coefficient w.r to baseline

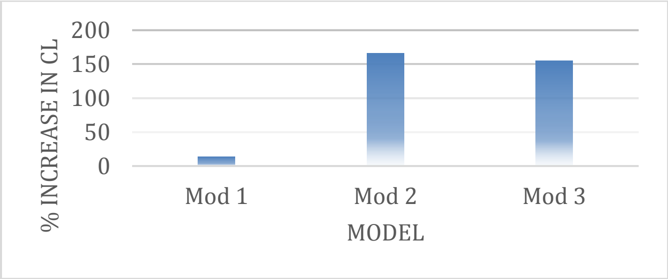


Figure 13: Percentage increase in lift coefficient w.r to baseline

Conclusion

Since flow structures around pickup trucks have been a topic of interest for the automotive industry for a long time, this study aimed to look into the flow structure with more geometric details added to the pickup truck model used here. The use of Computational Fluid Dynamics has made this process more dynamic by reducing the time to carry out in-depth analysis. This work analyzed the flow patterns around a generic pickup truck base model and three modifications inspired by market available add-on devices. All the cases discussed here are only for highway speed and the stream is at yaw angle of 0^0.

After analyzing the flow patterns in baseline model and three modifications, it was observed that the drag coefficient value of pickup trucks are usually high due to the vortex inside the rear box cavity. Improving the drag coefficient values directly affect the fuel consumption by the vehicle.

In this study, the lowest drag (15% reduction) was observed for the modification 3 which has tonneau cover in rear box. The reason for improvement was in shifting of air vortex in the downstream of cabin, especially in the rear box and wake zone. Modification 1 (wider tailgate top) has done some improvement (5% reduction) where modification 2 (cab roof spoiler with back cabin screen) has done around 8% reduction but increased the lift.

Another observation of the study is that the side mirrors should be included in the CFD analysis for any generic car. The side mirrors clearly effect the flow characteristics downstream which can be clearly seen in the velocity contours.

References

1. Mokhtar, Wael A., Colin P. Britcher, and Robert E. Camp. *Further analysis of pickup trucks aerodynamics*. No. 2009-01-1161. SAE Technical Paper, 2009. DOI: 10.4271/2009-01-1161

2. Mokhtar, Wael, and Robert Camp. "Pickup Trucks-Box Configuration and Drag Reduction." *28th AIAA Applied Aerodynamics Conference*. 2010. DOI: 10.2514/6.2010-4954

3. Mokhtar, Wael, and Nahid Pervez. "Underbody Drag for Pickup Trucks." *30th AIAA Applied Aerodynamics Conference*. 2012. DOI: 10.2514/6.2012-3210

4. Ha, Jongsoo, Shinkyu Jeong, and S. Obayashi. "Drag reduction of a pickup truck by a rear downward flap." *International Journal of Automotive Technology* 12.3 (2011): 369. DOI: 10.1007/s12239-011-0043-7

5. Taniguchi, Keiichi, et al. *A Study of Drag Reduction Devices for Production Pick-up Trucks*. No. 2017-01-1531. SAE Technical Paper, 2017. DOI: 10.4271/2017-01-1531

6. Maxwell, Timothy T., Jesse C. Jones, and William B. Jones. *Pickup Truck Drag Reduction-Devices That Reduce Drag Without Limiting Truck Utility*. No. 881874. SAE Technical Paper, 1988. DOI: 10.4271/881874

7. Yang, Zhigang, and Bahram Khalighi. *CFD simulations for flow over pickup trucks*. No. 2005-01-0547. SAE Technical Paper, 2005. DOI: 10.4271/2005-01-0547

8. Al-Garni, Abdullah M., and Luis P. Bernal. "Experimental study of a pickup truck near wake." *Journal of Wind Engineering and Industrial Aerodynamics* 98.2 (2010): 100-112. DOI: 10.1016/j.jweia.2009.10.001

9. Adem, Feysal Ahmed. *Drag reduction of pickup truck using add-on devices*. Diss. 2010. DOI: http://csus-dspace.calstate.edu/handle/10211.9/169