

A CFD Study of Pickup Truck Aerodynamics

Julie Kim^{*}, Justin Pattermann^{*}, Varun Menon^{*} and Wael Mokhtar^{**}

Department of Mechanical Engineering

Grand Valley State University

Grand Rapids, Michigan 49504

Email: meceng44@gmail.com, pattermj@mail.gvsu.edu, menonv@mail.gvsu.edu and mokhtarw@gvsu.edu

Abstract

Due to the considerable popularity in pickup trucks, aerodynamic analysis is of interest. The advantages of pickup trucks include towing capability, durability and carrying capacity. One of the disadvantages of trucks is their fuel economy. The fuel economy of a pickup truck is directly related to the drag force created from the air flow over it. Computational Fluid Dynamics (CFD) allows flow visualization and force calculations to suggest improvements on the fuel economy. A simplified truck model was analyzed using commercial CFD software (Star CCM+) from CD-Adapco Inc.

The first step in the project was to create a CAD model for an actual 6.5 ft. pickup truck. The CAD design was modified to incorporate an 8 ft. bed instead of a conventional 6.5 ft. bed. This was done in order to study the effect of the extra 1.5 ft. bed length on the aerodynamic efficiency of the truck. The CAD model was also rather simplified in order to ease out the analysis. The model was then imported into the CFD software in the form of an IGES file. A numerical mesh was generated, boundary conditions were defined and fitting physical models were selected. A mesh dependency study was performed to select the suitable mesh and domain size for reliable results. In the first set of runs, the base model was studied for three wind speeds (30 mph, 50 mph and 70 mph) that simulate city and highway driving. Two improvements were studied to alleviate the aerodynamic drag force. The first one involved the addition of a spoiler on the tailgate of the truck and the second enhancement involved tapering half of the cab towards the open truck bed. Important results were recorded and a flow field analysis was performed in order to quantify them.

Introduction

Pick-up trucks have been a very distinct entity in the US automotive market. For the past 22 years, America's best-selling vehicle has been the pickup truck¹. According to *The Star Online*, "some Americans will drive trucks, no matter cost."² Owing to the global fuel crises that have

* Mechanical Engineering Student

** PhD, Assistant Professor of Mechanical Engineering, ASEE member.

become very evident in today's society, it has become imperative for automobile manufacturers to increase the fuel economy of their trucks while still maintaining their utility and aesthetics. Though pickup trucks are one of the most popular vehicle geometries in use today yet it has received very little attention in aerodynamic literature³. The aerodynamics of pickup trucks is more complex to analyze than other open ended trucks due to their shorter beds, which can result in an interaction of the bed walls and tailgate with the separated shear layer formed at the edge of the cab³. The large frontal area and empty truck bed creates a very conducive environment for large vortices to form in the wake region of the truck. This phenomenon causes a drag force on the pickup truck, which has a direct correlation to its fuel economy⁴.

Literature Review

The overall drag characteristics of a pickup truck have a heavy dependence on the fluid flow in the truck bed and the wake region behind the truck. In this project we attempted to characterize the air flow in the truck bed and alleviate the drag force caused by the same by the addition of aerodynamic design enhancements. Two aerodynamic enhancements were suggested: A rear spoiler on the closed tailgate and a tapered cab roof.

Tailgate Spoiler –

A 'spoiler' is an aerodynamic add-on device that is generally used to provide added stability to a vehicle by streamlining turbulent airflow around it, thereby improving its drag characteristics. The present study focuses on the advantage of installing a spoiler on the tailgate of a pickup truck. The addition of such a spoiler delays the airflow separation at the rear end of the pickup truck, as a result of which the exiting air becomes less turbulent⁵. This reduces the drag force due to the wake on the pickup truck⁵.

Tapered Roof –

The air pressure within the open box of a truck is a major factor in determining the wake drag on the pickup truck⁵. An increased pressure within the box supports drag reduction and thereby improves the vehicle's fuel economy. In vehicle aerodynamics, the main idea is to always keep the pressure at the rear greater than in the vehicle's front⁵. By tapering the roof of the truck's cab, airflow is driven more downwards than it normally is, and it was found to increase box pressure, reducing the resultant drag force⁵.

Present Work

For the present study, CFD simulations were used to characterize the flow field around an 8-ft bed pickup truck. The main objective of these simulations was to better understand the flow around the pickup truck and to attempt to find different methods of reducing aerodynamic drag. A baseline truck model with a reference frontal area of 4753 in² was used as a control for this

analysis. Two design improvements were considered and compared to a baseline model. These improvements were the addition of a rear spoiler on the tailgate, and a tapered roof on the cab of the truck. The CAD software SolidWorks was used to model each configuration and the farfield geometries. Figure 1 shows the different truck configurations that were studied.

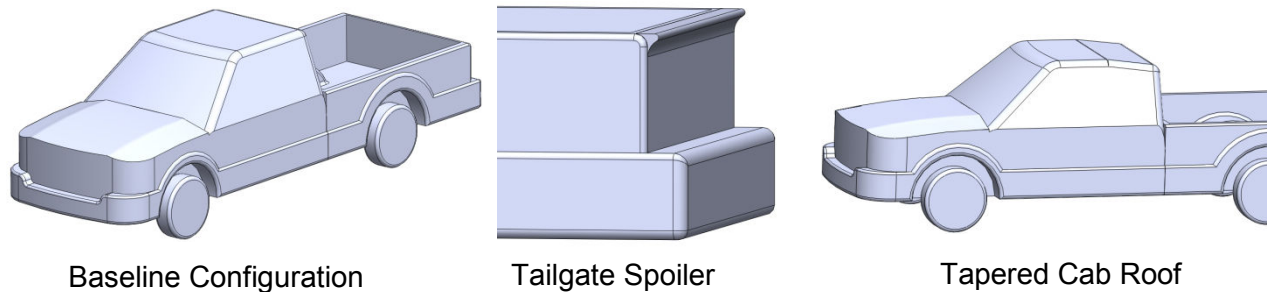


Figure 1: Pickup Truck Configurations

Simulations of the baseline truck with no modifications were run using a 30, 50, and 70 mph headwind. The same wind velocities were used to simulate the flow around the truck with each of the design improvements studied. Table 1 displays a summary of the different simulations run.

Table 1: Simulations Studied

Vehicle Configuration	Yaw Angle	Speed (mph)
Baseline	0°	30, 50, 70
Tapered Cab	0°	30, 50, 70
Tailgate Spoiler	0°	30, 50, 70

Computational Method

This study was performed using STAR CCM+ commercial software. The software is developed by CD-Adapco Inc. and uses a computational finite volume method. For the current study a three dimensional analysis was performed using a segregated flow solver. Turbulence was modeled using the two equation SST $k-\omega$ model.

The trimmer model was used to create the volume mesh around the truck. Cells were clustered most densely close to the surface of the truck and in the prism layer. Volumetric controls were utilized on the hood, in the bed and behind the truck to capture specific separation regions around the truck and the wake. For the final mesh, an average of around 850,000 to 950,000 cells was used in the domain. Figure 2 displays the volume mesh used on the truck, as well as the growth rate of the mesh as it moves outward in the domain.

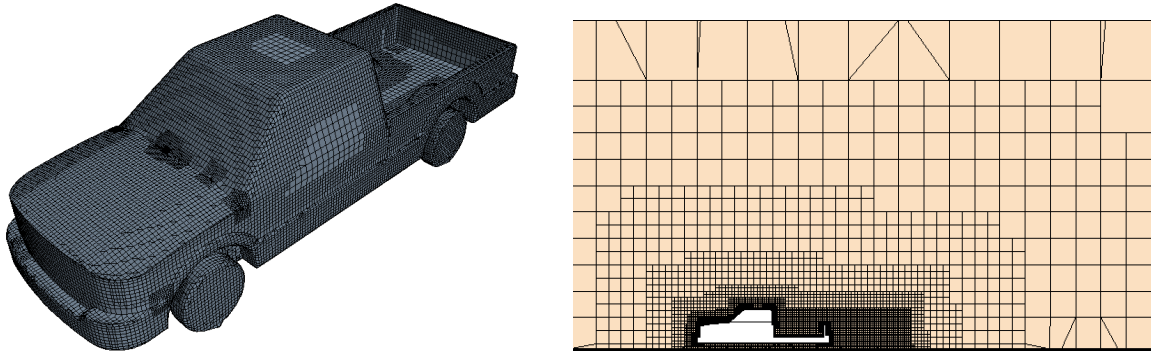


Figure 2: Generated Volume Mesh on the Truck and Growth Rate through the Domain

CFD Results And Discussions

One of the objectives for this study was to better understand the flow field around the pickup truck. In a headwind, the fluid will flow around the body of the pickup truck and will separate in several regions. One separation region occurs above the hood, and the other separation regions occurred in the bed and immediately behind the tailgate. It is important to understand the locations of these separation regions because of the high amount of form drag that they create. Figure 3 shows streamlines around the pickup truck which detail the separation regions.

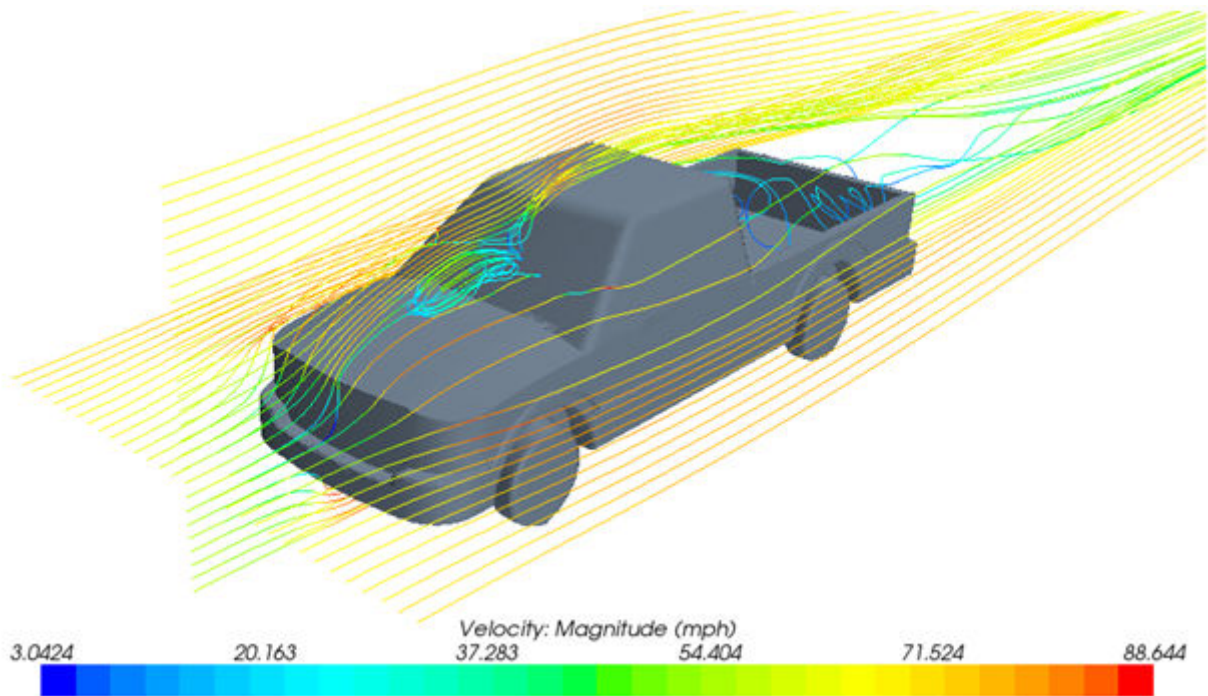


Figure 3: Streamlines of Flow around Pickup Truck

Another interesting characteristic about the flow field is the wake that is generated as the fluid flows around the truck. It can be seen that immediately behind the tailgate, the flow circulates forming two distinct vortices behind the truck. These vortices can be viewed in a velocity vector plot one meter above the ground plane, as shown below in Figure 4.

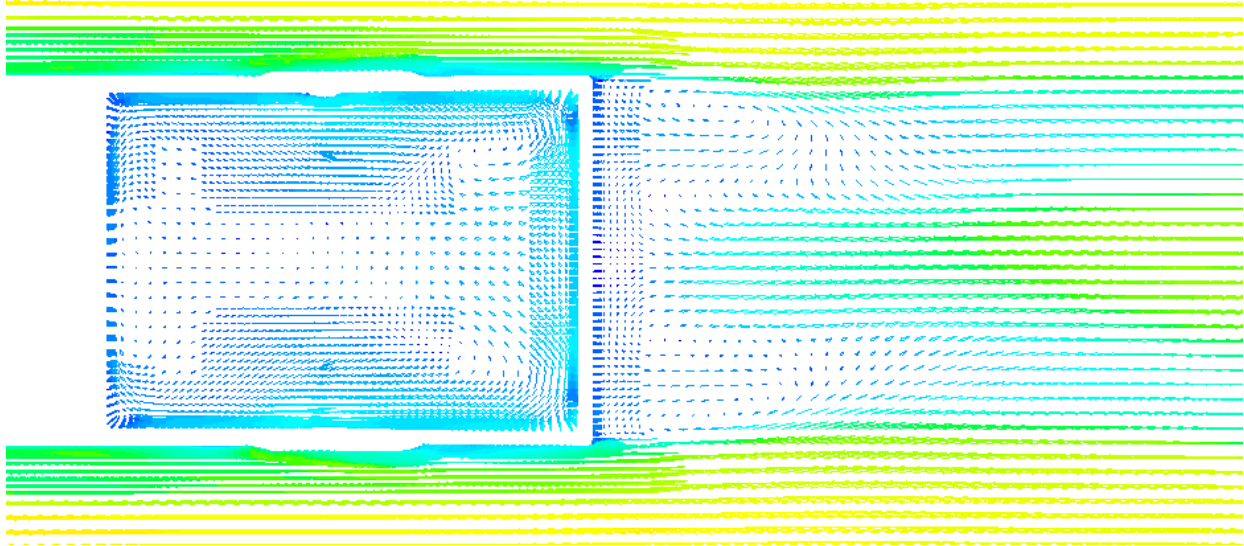


Figure 4: Detailed View of Wake Generated by Pickup Truck

After examining the separation regions, improvements were suggested to help streamline the flow around the truck. The effects of these improvements are examined below.

Tailgate Spoiler –

The tailgate spoiler proved to help reduce the drag on the truck at higher speeds. The spoiler modified the flow over the tailgate causing a visible difference in the separation region behind the tailgate. Figure 5 shows a center section cut comparison of the baseline truck simulation to the simulation with the tailgate spoiler added.

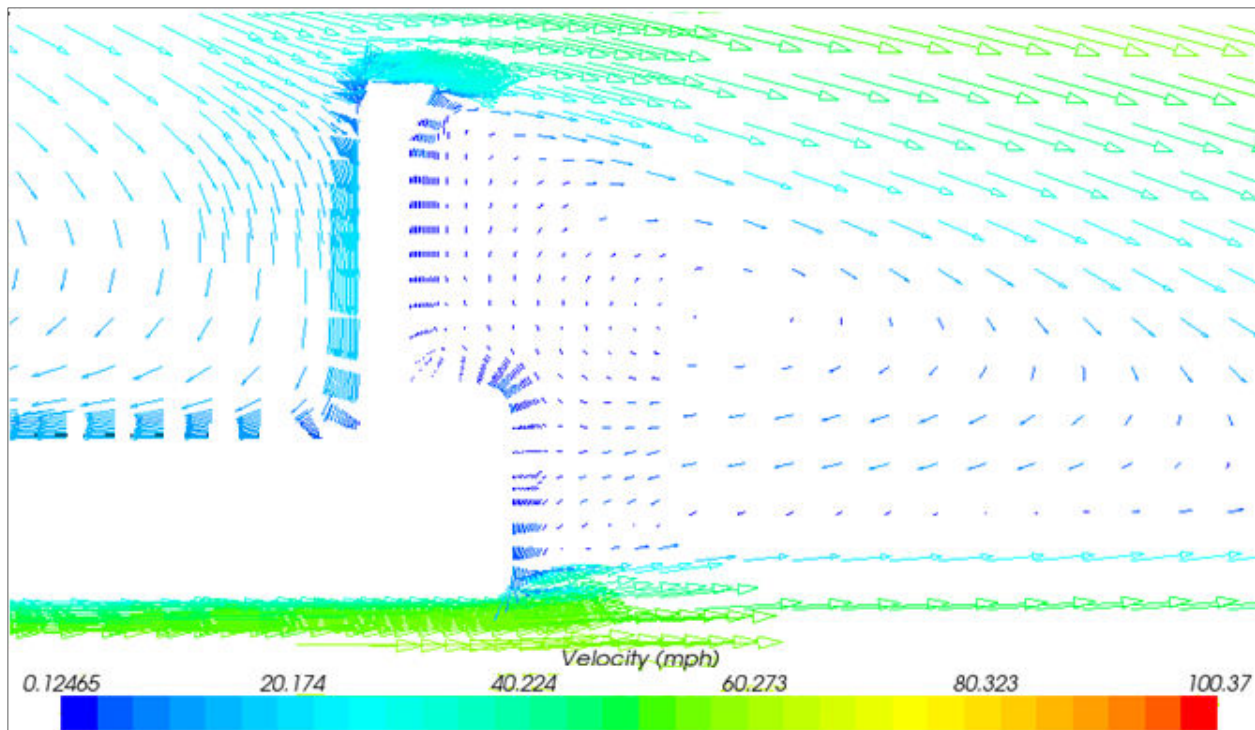
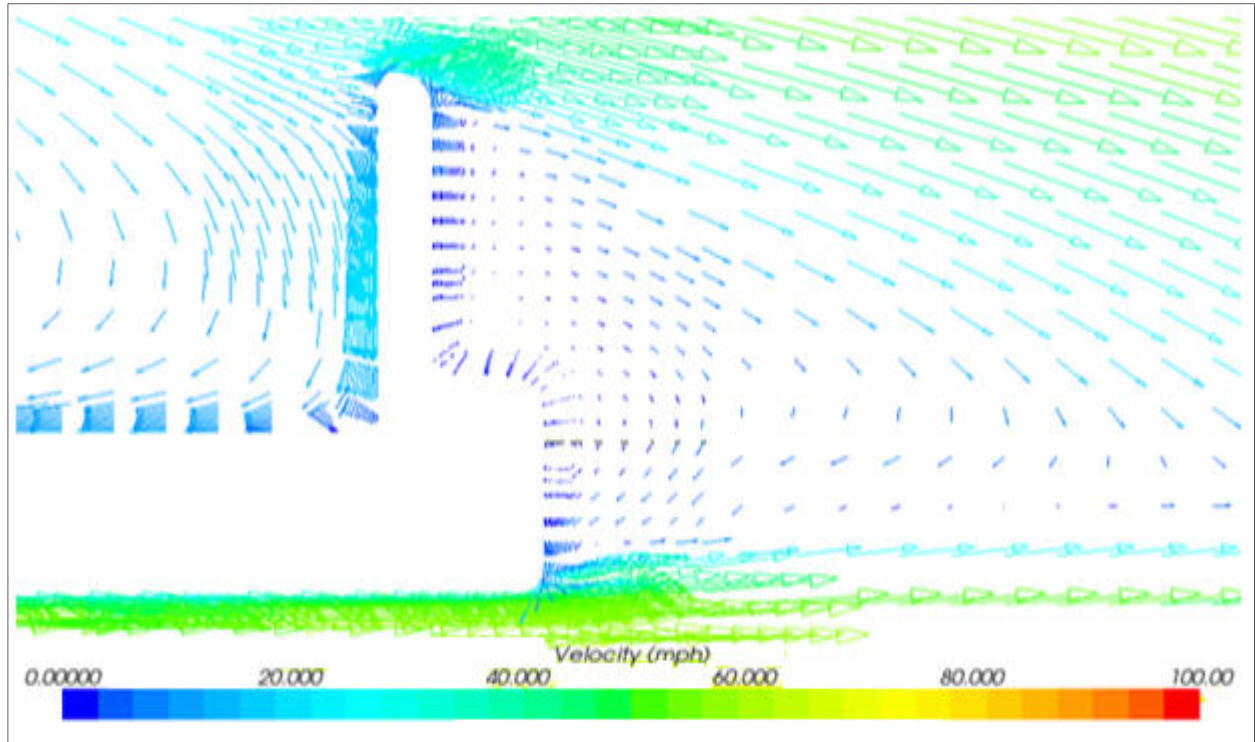


Figure 5: Comparison of Flow Behind the Standard Tailgate (Top) and Modified Tailgate (Bottom)

It can be seen in Figure 5 that the tailgate spoiler delays the separation of the flow creating a more clearly defined flow circulation behind the tailgate. This circulating cushion of air works to direct the flow over the tailgate creating a more streamlined flow pattern, and thus decreasing the drag.

Tapered Cab Roof –

The second modification that was simulated was a tapered cab roof. When the roof of the cab was tapered down, it modified the flow around the top of the cab and into the bed of the pickup truck. Figure 6 compares the fluid velocity around the truck for the baseline and the tapered cab in the center plane.

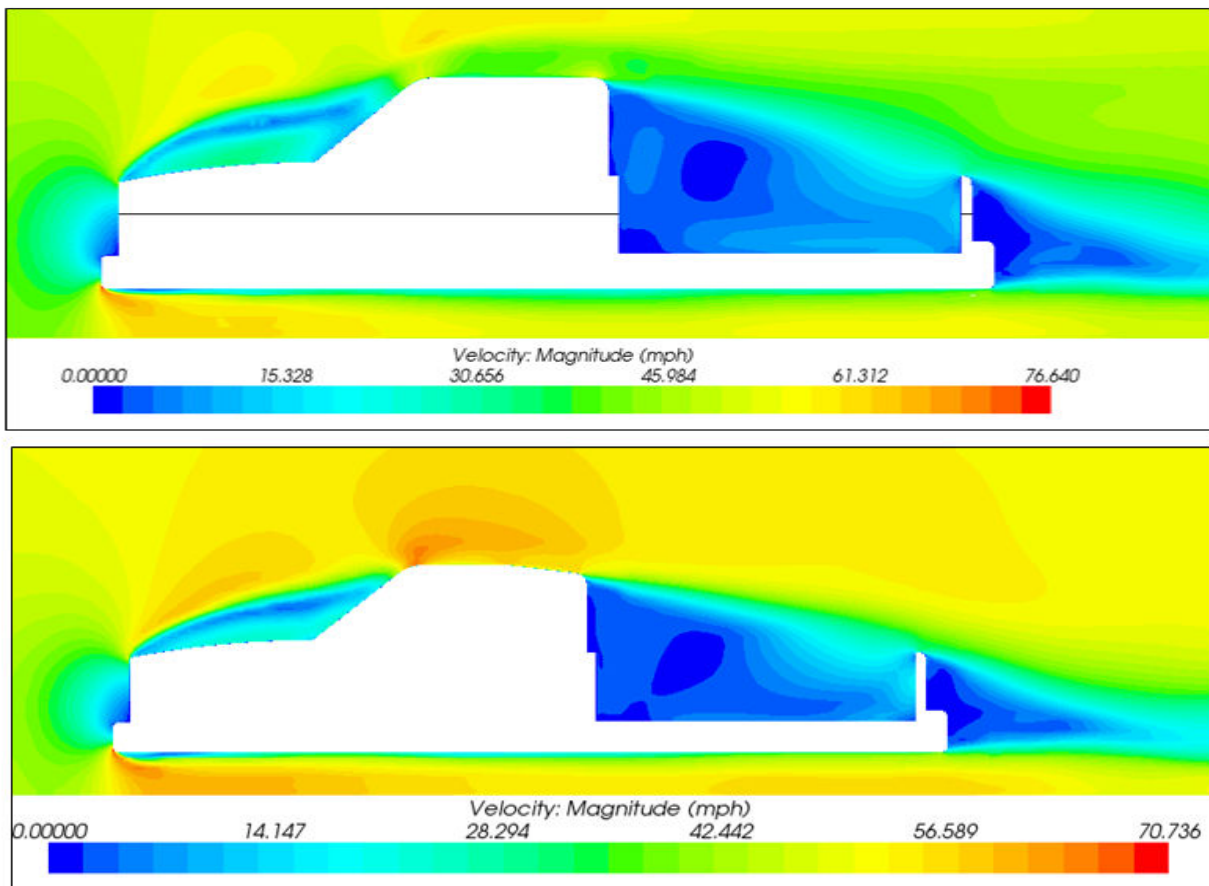


Figure 6: Comparison of Velocity Scalar Plots for Standard (Top) and Tapered Cab (Bottom)

From Figure 6, it can be seen that with the tapered cab, there is a much larger region of stagnant air in the bed of the pickup truck. This allows for a more streamlined fluid flow over the bed because it provides an effective cushion for the fluid to flow over. The streamlined flow over the bed results in a much lower drag force acting on the truck especially at higher speeds.

The main indicator that the modifications simulated improved the aerodynamics of the truck is the drag coefficient. By comparing the drag coefficient of the modified truck to the baseline configuration, the overall effect of the modifications can be clearly seen. Table 2 presents a summary of the results from the nine simulations that were run for this study. The values of the drag coefficient represent both viscous and form drag. It was found that the tapered cab roof was most effective in reducing the drag coefficient of the pickup truck across all the speeds simulated. The tailgate spoiler was found to help reduce the drag coefficient of the truck at higher speeds however it resulted in a higher drag coefficient at lower speeds.

Table 2: Summary of Drag Coefficients for the Different Configurations

Truck Configuration	Speed (mph)	Drag Coefficient	Percent Change
Baseline Truck	30	0.549	N/A
	50	0.508	N/A
	70	0.474	N/A
Improvement 1 - Tailgate Spoiler	30	0.556	1.28%
	50	0.493	-2.95%
	70	0.467	-1.48%
Improvement 2 - Tapered Cab Roof	30	0.525	-4.37%
	50	0.441	-13.19%
	70	0.406	-14.35%

Conclusions

CFD is an effective tool to help visualize the fluid flow around a pickup truck in the attempt to help reduce the aerodynamic drag. This study compared a baseline truck model to two different modified truck models in order to see if the modifications reduced the drag coefficient. The two modifications studied were a spoiler on the tailgate of the truck and a tapered roof on the cab of the truck. When the analysis was completed, it was found that the tapered roof resulted in the greatest percentage decrease in drag coefficient on the truck. The tailgate spoiler benefitted the truck at high speeds however the drag coefficient was slightly higher at lower speeds. These modifications reduced the drag coefficient by modifying the separation regions in the bed and behind the tailgate.

References

1. Sovani, S., Lokhande, B., *Streamlining Pickup Trucks*, <http://www.fluent.com/about/news/newsletters/04v13i2/a14.htm>, Fall 2004, Page 1.
2. Hopkins, A., “Some Americans will drive trucks, no matter costs”, http://thestar.com.my/news/story.asp?file=/2008/7/2/worldupdates/2008-07-02T100031Z_01_NOOTR_RTRMDNC_0_-343289-1&sec=Worldupdates, July 2nd, 2010, Page 1.
3. Al-Garni, M., Bernal, L.P., Khalighi, B., “Experimental Investigation of the Near-Wake of a pickup truck”, SAE paper no. 2003-01-0651, January 2003, Page 1-11.
4. Busa, K., Depue, M., Ingraham, H., Kozak, S., Naccarato, A., “Drag Reduction of Your Regular Everyday Pickup”, Syracuse University - L.C. Smith College of Engineering and Computer Science, May 5th, 2009, Pages 1-19.
5. Massarotti, M.R., Valarelli, I.D., “CFD Assessment of Aerodynamic Enablers for Pickup Trucks”, Universidade Paulista, November 2009, Page 1- 10.