SAE INTERNATIONAL®

An Evaluation of the Fuel Economy Benefits of a Driver Assistive Truck Platooning Prototype Using Simulation

2016-01-0167

Published 04/05/2016

Hugh Luke Humphreys, Joshua Batterson, and David Bevly
Auburn University

Raymond Schubert
Peloton Technology

CITATION: Humphreys, H., Batterson, J., Bevly, D., and Schubert, R., "An Evaluation of the Fuel Economy Benefits of a Driver Assistive Truck Platooning Prototype Using Simulation," SAE Technical Paper 2016-01-0167, 2016, doi:10.4271/2016-01-0167.

Abstract

The fuel efficiency improvement of a prototype Driver-Assistive-Truck-Platooning (DATP) system was evaluated using Computational Fluid Dynamics (CFD). The DATP system uses a combination of radar and GPS, integrated active safety systems, and V2V communications to enable regulation of the longitudinal distance between pairs of trucks without acceleration input from the driver in the following truck(s). The V2V linking of active safety systems and synchronized braking promotes increased safety of close following trucks while improving their fuel economy. Vehicle configuration, speed, and separation distance are considered. The objectives of the CFD analysis are to optimize the target separation distance and to determine the overall drag reduction of the platoon. This reduction directly results in fuel economy gains for all cooperating vehicles.

In order to correlate the computational studies, fuel consumption tests were conducted conforming to the (1986) SAE J1321 Type II - Fuel Consumption standard using a pair of trucks. Testing was performed using the DATP system at separation distances of: 30ft, 40ft, 50ft, 75ft, and 150ft. These distances were chosen to validate the predicted trend between vehicle separation and drag reduction. Preliminary results from the CFD analysis are presented in this paper. Initial findings suggest that the fuel economy of vehicles significantly improves at diminishing separation distances. Effects at larger distances, as well as the effect of lateral offset, are also presented.

Introduction

In the United States, the trucking industry represents approximately 80% of a $1.33 trillion-dollar shipping and logistics industry, according to the American Trucking Association's 2015 report. [1] In order to remain competitive in such a massive industry, shipping companies and fleet owners must continually find ways to decrease their operating expenses.

Chief among these expenses is the cost of fuel. With this in mind, reducing fuel consumption even a small amount can have tremendous impact across even a single shipping fleet, let alone the entire industry. According to the American Transportation Research Institute's (ATRI's) 2014 report fuel costs represent the largest non-employee expense per mile of a heavy vehicle. [2] With most of the factors affecting the price of fuel outside of trucking companies' control, the most effective way to reduce the associated costs is to improve fuel economy.

However, improvements in fuel economy yield more than just cost savings. With rising concerns about the environmental impacts of vehicles, the reduction in fuel consumption also represents a reduction in emissions. Therefore, it can be concluded that seeking innovative techniques to improve fuel mileage has far-reaching ramifications beyond simply an economic impact.

Fundamental changes in standard operational logistics emphasizing multi-truck cooperation, or platooning, can directly contribute to improved fuel economy for all participants. Platooning accomplishes fuel reduction by organizing vehicles in a leader/follower scenario thus lowering the amount of aerodynamic drag realized on each vehicle. At present, platooning - especially involving heavy vehicles at extremely short following distances - spurs questions of public safety when the drivers are solely responsible for controlling separation distance. This is doubly important when drivers are faced with emergency braking situations. In such cases, the overall safety of the truck platoon, as well as commuters in their vicinity, is subject to the limited reaction time of the drivers. Eliminating human error and drastically reducing the reaction time is essential to make heavy vehicle platooning at close distances viable.

Utilizing GPS, radar, and Dedicated Short Range Communication (DSRC) for Vehicle-to-Vehicle (V2V) communications the DATP system addresses this issue by monitoring relative positions to achieve a world-view sufficient to control the longitudinal distance between vehicles. Under nominal cruising conditions, DATP will maintain a fixed separation distance by accelerating and decelerating the following vehicles as needed. In emergency braking situations, DATP can react faster than the driver alone thus reducing the total time between an emergency arising and the truck coming to a complete stop. With this system, the separation distance between vehicles can be reduced beyond current safe operating practices to minimize the aerodynamic drag forces and improve fuel economy.

Previous work with platooning vehicles, operating under automated separation distance control systems, indicates that decreased separation distances, result in improved fuel economy. [3]-[5] Their findings have shown that the lead truck's fuel economy improves continuously at successively closer spacings, while the trailing vehicle will see an improvement with a local maximum between 25 to 75 feet depending on the type of vehicles in the platoon and the operating conditions. DATP systems continue to push this concept to the extreme by allowing safe operation at ever decreasing separation distances. Thus further investigation into this trend is warranted. [6] This study seeks to identify potential aerodynamic effects that would strongly correlate with trends observed in measured fuel consumption data for two-truck platoons.

CFD Formulation

In order to investigate the aerodynamic drag reduction evident from platooning, a CFD study was proposed in order to determine the optimal spacing for the system. To perform this investigation, FLUENT, part of the ANSYS analysis package, was used to perform the CFD. [7]

From thermodynamic analysis, the energy consumed by the truck to overcome drag while traveling over a known distance is defined by

$$E = Q - W \tag{1}$$

where E is the energy required to be supplied by the engine, Q is energy gained or lost due to heat transfer, and W is the work imposed by aerodynamic forces. For this system, Q can be considered negligible compared to the work. The work is defined as the aerodynamic force integrated over the distance traveled:

$$W = \int_S \vec{F} \cdot d\vec{s} \tag{2}$$

where the aerodynamic force is given as

$$F = \frac{1}{2}\rho v^2 A C_d \tag{3}$$

In Eq. (3), ρ represents the density of air, v is the traveling speed, A is the cross-sectional area, and C_d is the drag coefficient. These fluid dynamic variables can be obtained computationally using ANSYS FLUENT.

At a minimum, FLUENT solves two governing equations: conservation of mass and conservation of momentum:

$$\text{Continuity:} \quad \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \tag{4}$$

$$\text{Momentum:} \quad \frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v})\vec{v} = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \tag{5}$$

When combined, Equations (4) and (5) are referred to as the Navier-Stokes equations and fully characterizes fluid motion for an incompressible, non-reacting flow. FLUENT resolves fluid flows by numerically discretizing and integrating these equations. This is necessary in most realistic flow regimes where closed-form solutions do not exist.

Also relevant is the way turbulence is handled within FLUENT'S solvers. Turbulence, characterized by chaotic, random, complex motion, is a well-known phenomenon to occur within realistic fluid flows. For a numerical solver, it is one of the most difficult tasks to achieve realistic modeling. For the purposes of this investigation, two different methods provided within FLUENT were looked at to resolve the turbulence within the fluid flows, realizable k-ε (RKE) and Detached Eddy Simulation (DES).

The RKE method is a Reynold's Averaged Navier-Stokes (RANS) technique developed by Jones and Launder in 1972. [8] RANS-based models typically begin their analysis by performing a Reynolds decomposition for any general field variable, ϕ, consisting of an average term and a fluctuating term:

$$\phi = \bar{\phi} + \phi' \tag{6}$$

This can then be applied to both the conservation of momentum and conservation of mass equations by assuming that each will have an average value. The turbulence provides the fluctuation upon that mean value. By decomposing the field variables and averaging yields the averaged conservation of mass equation:

$$\frac{\partial \bar{v}_j}{\partial x_j} = 0 \tag{7}$$

and the averaged conservation of momentum:

$$\rho\left[\frac{\partial \bar{v}_i}{\partial t} + \bar{v}_j \frac{\partial \bar{v}_i}{\partial x_j}\right] = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\bar{\tau}_{ij} - \rho\left\langle v_i' v_j' \right\rangle\right] \tag{8}$$

Equations (7) and (8), when combined, are known as the Reynolds Averaged Navier-Stokes equations. These equations represent the traditional method for turbulence modeling. However, now that these equations have been developed, two stress terms arise: one from the fluid's viscosity, and one from non-linear acceleration term. The latter is not a physical stress but rather a source of turbulence. This unfortunately complicates the problem by adding unknowns without providing equations in which to close the system. The difference in the handling of this extra stress term is how the various RANS stress models differentiate themselves.

Realizable k- ϵ Formulation

For the realizable k - ε model, the RANS system is closed by defining two new quantities: the turbulent kinetic energy (TKE), k , and turbulent dissipation, ε where the TKE is:

$$k = \frac{1}{2}\left(\overline{u'u'} + \overline{v'v'} + \overline{w'w'}\right) \tag{9}$$

and ε is:

$$\varepsilon = \nu \overline{\frac{\partial v_i'}{\partial x_k}\frac{\partial v_i'}{\partial x_k}} \tag{10}$$

The transport equations can be generated as:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k v_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\varepsilon - Y_M + S_k \tag{11}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho\varepsilon v_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \rho C_1 S\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu\varepsilon}} + C_{1\varepsilon}C_{3\varepsilon}\frac{\varepsilon}{k}P_b + S_\varepsilon \tag{12}$$

The dynamic and eddy viscosities are represented by μ and μ_t , respectively. From here, experimentally-derived constants are substituted to close the model. [9] This provides the two transport equations needed to solve for newly introduced quantities, k and ε .

The last element of the realizable k - ε model is the near-wall treatment. As the origin of turbulence, the wall-treatment is a critical component of any turbulence modeling. One of the primary weaknesses of the RKE model is its inability to capture the non-trivial effects present near the wall. To remedy this, FLUENT imposes a secondary treatment to the wall in order to capture these effects.

DES Formulation

Detached eddy simulation is a hybrid turbulence modeling technique that combines a RANS model with a new form of turbulence model known as Large Eddy Simulation (LES). LES differs fundamentally from RANS models in its formulation of turbulence. For the LES, both the temporal and spatial domains are filtered, yielding a turbulence function of the form:

$$\overline{\phi}(\vec{x}.t) = \int_{-\infty}^{\infty}\int_{-\infty}^{\infty}\phi(r,t')G(\vec{x}-\vec{r},t-t')dt'd\vec{r} \tag{13}$$

Where G represents a filter convolution kernel designed to separate the gridded cells into sub-gridded scale (SGS) cells, and others left in their original gridded state. This represents a low-pass filter following the form:

$$\overline{\phi} = G * \phi \tag{14}$$

which is similar to the RKE formulation. However, for the elements that are applied in the SGS scale, the element is then further re-gridded into smaller elements and then resolved, while elements that do not pass the filter are solved normally. When properly refined, this provides a much more accurate solution at the expense of a potentially prohibitive increase in computational time. In addition, while not inherent to the formulation, the LES formulation for FLUENT was designed exclusively for transient analysis. This makes steady state solutions with DES unreliable. [7]

For DES, near the wall, RKE formulations are used to model the wall effects sufficiently, where the grid should be even further refined. Away from the wall, where the grid is less refined, the LES model is applied. This allows for a highly accurate turbulence model in the far-field where the flow is no longer attached to the model, while not exponentially increasing the computationa load in the already refined regions. This saves the requirement of even further regridding an extremely small mesh.

Considering the problem at hand, a steady-state RKE model of solving the fundamental equations was chosen. With the flow around the platooning system being a low-speed, extremely low heat transfer system, the RANS model suffices to provide accurate results, without the computational expense of a DES model. Furthermore, modeling the tractor-trailers as a steady-state solution should not incur significant loss of accuracy in examining the characteristics of the flow around the platoon.

Comparison to Previous Works

Initially, a literature search was conducted to find the most recent work on the platooning of bluff bodies, with a focus on tractor-trailer configurations. Computational studies require validation to be effective analysis tools. Experimental data is absolutely necessary to prove effective turbulence modeling and simplification techniques.

While significant work has been done in simplified bluff bodies a comprehensive CFD study for platooning of various sizes and spacings had not been conducted yet. One of the more common models that is frequently studied is the Ahmed body shown in Figure 1. Originally developed by Ahmed in 1984, they are designed to be a simplified representation of a car to model the wake behind vehicles. [10] In his work, he experimentally tested the simplified bodies in a wind-tunnel to examine the drag coefficient for various back-slant angles at various Reynolds numbers. Since then, they have become a common standard for validation of numerical simulations.

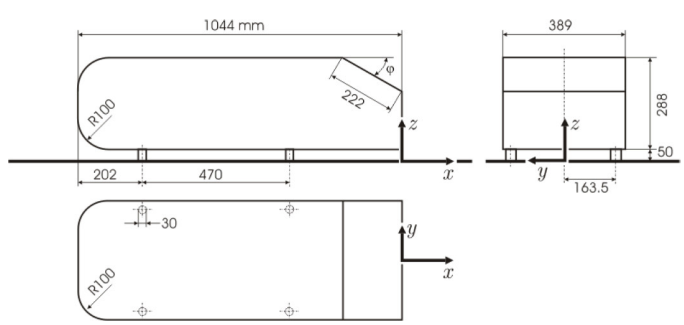

Figure 1. Simple Ahmed Body 2-D Drawings

Additionally, while single bluff bodies are fairly common in the simulation and experimental work, multiple bodies are fairly uncommon. Despite this, Pagliarella's SAE Technical Paper published in 2007 investigates Ahmed bodies in series and thus proves applicable to the present study. [11] In their work, they investigated the effect of the tilt-slant of multiple Ahmed bluff bodies. While not directly relating to platooning systems, they provided extremely well-documented coefficient of drag results from experimental data for various spacings of platooned Ahmed bodies.

By modeling the Ahmed bluff bodies in platoon, the models can be refined until the numerical simulations match the experimental data. Pagliarella's experiment are extremely valuable since they also vary the spacings as one of the independent variables considered in the study. This provides insight into how the bodies should behave in platoon. Given that Ahmed bodies are already intended to model vehicles; it can be expected that the simulation of the tractor-trailer vehicles will exhibit similar large-scale phenomena to that of the Ahmed bluff bodies. Thus, this work proved invaluable for validating turbulence modeling in the tractor-trailer platoon.

To this end, it can be verified that the turbulence models and solution methods are accurate and appropriate. In addition, with Pagliarella's well-documented coefficient of drag and visualization studies, the numerical simulations can be visualized to ensure that effects that are anticipated to appear are resolved by the simulation. With the flow around the back of the Ahmed body being turbulent, and thus highly complex, having a validated baseline reference is necessary for this simulation process.

Tractor-Trailer Mesh Generation

Once the modeling scheme is proven valid for simplified bodies, it can then be extended to the complicated geometries of the tractor-trailers with some assurance that the computational model will translate to realistic systems. With this in mind however, even utilizing an unstructured solver, the computational domain can also be simplified in order to achieve a model that can be solved within an acceptable computational time.

The key geometric features defining the fluid flow consist of the cab including the hood, grill, fenders, and fairings; the trailer incorporating the box, and side-skirts; and the undercarriage with the wheels, frame, and separation between the cab and the box. Because these features all exist on large length scales comparable to the overall length of the vehicle, their influence to the aerodynamic forces overshadows those of smaller features. In this vein, the contribution of mirrors, exhaust pipes, steps into the cab, and other small-scale details are assumed to be inconsequential to the problem at large. Their inclusion would lead only to complicated computational meshes and increased computational time with very little benefit.

Thus, as shown in Figure 2, features such as the rear view mirrors were eliminated in order to greatly simplify the model for simulation. This improves the overall mesh quality by minimizing skewed elements as well as removing areas that would require additional mesh refinements to accurately model the flow around the complex shape. This proves important for generating a high quality mesh around solid boundaries, which are in turn are necessary for accurately capturing the boundary layer about the vehicle's surfaces accurately.

Figure 2. Photo of Peterbilt 579 Cab (provided by Peterbilt) vs. Defeatured Cab and Trailer for Simulation

Additionally, features that would create problems for the mesher must also be eliminated. For example, where the wheel meets the road is a physical boundary that causes a singularity in the mesh where the two surfaces meet. This is caused by the mesher attempting to discretize an ever-smaller volumetric region. While these boundaries could all be resolved using the ANSYS mesher, each case of non-singularity would have to be individually addressed. It is far simpler to alter the geometry in trivial ways to eliminate these singularities without appreciably changing the flow characteristics. An example of such a modification is shown in Figure 3. For this example, the singularity is eliminated by adding a small 1-inch stair step to the wheel. In the analysis of a vehicle with a reference length of 840 inches, this change negligibly affects the solution.

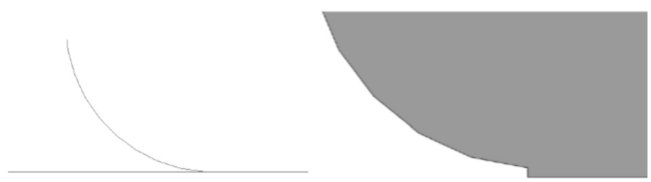

Figure 3. Example of Singularity Changed by Trivial Modification of Geometry

Once the model is defeatured, the mesh can be generated. Next, the mesh is refined to ensure grid independence. For this purpose, the ANSYS vehicular modeling best practices was consulted. [12] For the tractor-trailer defeatured model, this entails adding five volumetric refinement regions, along with surface refinements. This generates an acceptable grid necessary to obtain results that would accurately exhibit all the anticipated flow features.

Meshing a model represents a balancing act of achieving high-fidelity results while minimizing computational time. Refinements to the mesh offer diminishing returns in terms of the computational accuracy, while exponentially increasing the amount of time required to converge to a solution. This tendency is demonstrated in Figure 4 which depicts the solution accuracy for the Ahmed body vs. number of elements being considered by the solver.

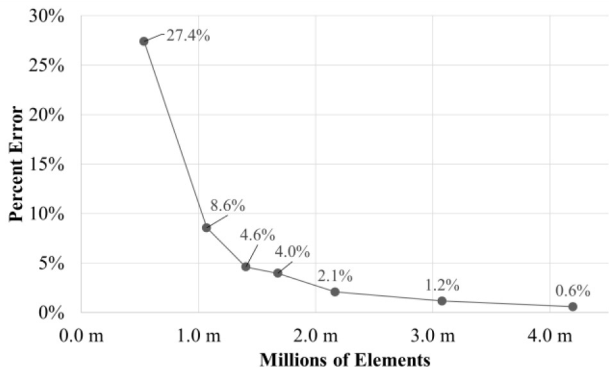

Figure 4. Accuracy vs. Millions of Elements for Ahmed Body

In addition to refining the mesh in each region, the overall quality of the mesh is considered. Since FLUENT is an unstructured solver, the shape of the computational element can vary from cell to cell. This means that the relative quality of each element must be analyzed. Since the fluid forces are dependent on the surface of each cell, a highly skewed element presents problems for the solver. Decreasing the size of the mesh increases the amount of elements, and therefore forces the mesher to sometimes use a higher-than-optimal skewness for some elements. Therefore, one of the goals of a successfully meshed model should be that of low element skewness.

Volume Refinements

Figure 5 shows the corresponding volume refinement regions in the mesh. Each of these volumetric refinement regions represent a maximum element sizing restriction that must be maintained within that volume.

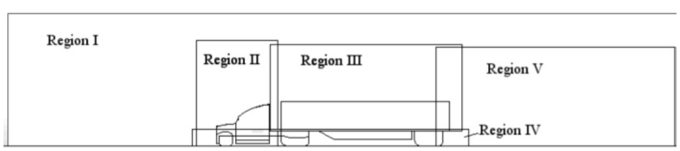

Figure 5. Diagram of Volumetric Refinement Regions

Region I represents the intermediary region of the domain where the elements are less coarse than the far field but not nearly as fine as those nearer to the vehicle. This serves as a smooth transition region into the far field.

Region II and III encompass the top of the vehicle. Region II contains significantly more elements than Region III because the flow around the tractor is much more complex than the region around the top of the trailer, which is similar to that of a flat plate. Despite this both regions require much finer gridding than the far-field.

Region IV defines the underbody of the vehicles. This contains a particularly small maximum element size since the flow under the trucks is important to the development of the wake and changes dramatically in platoon compared to a single Tractor-Trailer.

Finally, Region V defines the wake region of each vehicle. With the highly turbulent flow behind the vehicles, extra cells should be defined in order to adequately capture the effects of the vortex shedding behind the trucks.

Surface Refinements

In addition to the volume refinements, surface refinements were also generated for the mesh in order to accurately model the geometry. For the surface refinements, there were two primary forms of refinement applied to the model. First, a minimum element size grid refinement was applied to areas where the model's geometry changed relatively rapidly, such as the top of the cab, where the vehicle's profile slopes. In contrast, along the trailer, the surfaces are similar to that of a flat plate, and thus a coarse grid suffices to capture the small changes evident along their surfaces.

Next, we define an inflation layer around the surfaces. Inflation layers are a series of surface-fitted, pseudo-structured elements. This ensures an accurate depiction of the boundary layer along each surface. Without an inflation layer around the surfaces, it is possible that the boundary layer around each surface would be misrepresented in the solution, and an inaccurate solution would be calculated.

Including the volume and surface refinements, Figure 6 depicts the local grid around a single truck.

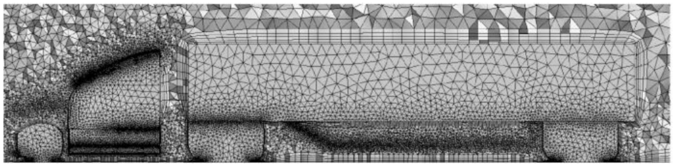

Figure 6. Slice of Model Demonstrating Surface Meshing Techniques

Initial Computational Results

First, a single truck was modeled at 65 mph. This was then used as the baseline to provide a reference drag each truck experiences while not platooned. The coefficient of drag was calculated for every surface as well as the overall drag. The overall drag coefficient was computed to be 0.5721 for a single truck.

Once the single-truck baseline was established, a series of simulations were executed for trucks in platoon at various separation distances. This coefficient of drag was then normalized by the single truck drag, where the results are displayed in Figure 7. In agreement with findings from previous studies [13] [14]: as the separation distance is reduced, the drag on the platooned bodies is also reduced. This is attributed to a mechanism similar to drafting. Namely, the rear truck sees reduced drag due to being within the slipstream of the front truck. Thus the rear truck does not experience a large unfavorable pressure gradient on the front of the cab. Simultaneously, the rear truck disrupts the formation of a low-pressure vortex structure behind the front truck ultimately reducing its wake drag.

According to these results, one could conclude that since drag reduction continues to improve as separation distance decreases, the theoretical improvement in fuel economy is limited not by the aerodynamics, but rather by the practical limits of acceptable operation. It is interesting to note, however, that the rear truck drag asymptotically approaches a minimum reduction at increased spacings. Even though the lead truck sees no appreciable reduction in drag past approximately 70 ft, the combined trucks still see an overall reduction in realized drag. This suggests that a DATP system still sees performance gain even at substantially longer following distances.

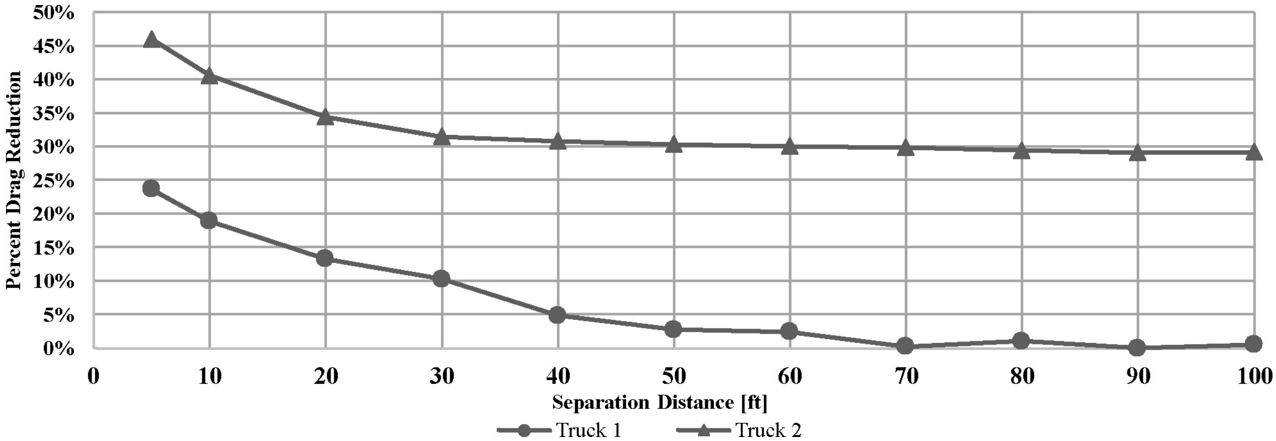

Figure 7. CFD Study Initial Results

Knowing that the RKE models typically do not capture large-scale turbulence as well as DES models, the simulations were then repeated utilizing DES. While offset, the qualitative trend was similar for a reduced set of testing. From this it was concluded that there were no significant vortical effects being misrepresented by the RKE values at practical operational limits. At excessively long distances, the RKE model breaks down, predicting an unphysical wake that extends to infinity. This is most likely due to RKE models being unable to re-laminarize the flow properly.[7]

Fuel Economy Type II Test

After completing the numerical simulation of the trucks in platoon, and attempting to validate those results with previous studies, a test was completed in order to help correlate the results with experimental data. To accomplish this, Auburn utilized Peterbilt 579 tractors leased from project partner Peterbilt Trucks along with a prototype DATP system developed by Peloton Technology. Figure 8 depicts normal operation of Peloton's DATP system.

Figure 8. Peterbilt 579 Tractors in Platoon Utilizing Peloton's Prototype DATP System

The trucks were tested for fuel economy according to the SAE Type II Fuel Test, using the 1986 standard. [15] To accomplish this rigorous testing regimen, the trucks were brought to a large test track in Ohio. Before testing, the trucks were serviced to ensure that they were operating nominally. Once the trucks were confirmed to be operating as intended, a set of trial runs were conducted in order to verify the testing procedure, which is as follows.

The trucks were warmed up for a minimum of one hour on the track at 60 mph. This ensures that the engine is operating in a fashion similar to what the truck would experience during the test throughout the test, minimizing the fluctuations in readings due to efficiency differences. Once the trucks were warmed up sufficiently, the platooning trucks were again inspected to ensure nominal operation. This included making sure the DPF system was not in need of a regeneration and that the SCR emission system was operating properly. Then, a half-lap was competed to allow proper linking between the platooning trucks. Once the half-lap was completed, the air conditioning was turned off and the fans on both trucks were turned down to match each so that no additional stress was added to the engine. At the 4.8-mile marker, the electronic fuel switch was synchronously engaged to switch between the truck's normal operating saddle and the external fuel tank. This is the point at which the trucks were considered on-test. From this point, the trucks completed 6 full laps around the track at 65 mph, in platoon at prescribed following distances.

For the control truck, a similar, but not identical procedure was used in order to maintain consistency between runs. Without a fuel switch like the platooning trucks, the control truck was required to be on-test from the initial key-on to the end of the test. To accomplish a consistent run, its tanks were weighed, then swapped onto the truck after a one hour warm-up time. Once the fuel tank was swapped, the truck was then keyed-on and the truck was on-test. After a 60 second engine idle, the control truck accelerated directly to 65 mph, and then held 65 mph for 7 laps in order for the steady-state, cruise control portion of the test to be equal to that of the test trucks. On the final lap, the truck maintained a speed of 65 mph until it reached the 7.2 mile marker. At this point, it engaged the brakes until it reached a designated point in the pit lane. Once stopped, the truck was idled for 60 seconds, then keyed off.

Each of the external fuel tanks were removed from each of the trucks. Meanwhile, a new, full, weighed external fuel tank was secured to the test and control trucks. A fuel handler on site measured the tank's weight, and compared it to its previous weight prior to the run. The amount of fuel consumed is determined by subtracting the final from the initial weight. The amount by volume of fuel burned can be determined by dividing the difference in weight by the fuel density. From this, the amount of fuel saved can be compared at each separation distance. Note that the elapsed data run times for the control truck were consistent, within 2% variance, and the effects of extra fuel burned by the control truck due to the additional lap and engine idle time is addressed via the test-to-control (T/C) ratio method recommend in the SAE Fuel Consumption Test Procedure-Type II. Also, note that the fuel ratio data is within 2% variance for each set of runs for a given separation distance.

Fuel Economy Results

The results from the fuel economy tests are still being developed, with the expectation that they will be fully complete in early 2016. In order to compare the CFD to practical results, data from the National Renewable Energy Laboratory's (NREL) fuel economy test was also analyzed. [5] Their work in Texas is complementary to the work done in Ohio, utilizing a similar testing standard and an early prototype version of Peloton's platooning system.

Some interesting trends were observed in NREL's data. Without a comprehensive engine model, the effect of aerodynamic drag on fuel consumption cannot be directly quantified. This is particularly exacerbated in DATP systems, since the engine model is tightly coupled with the DATP control algorithm. However, because aerodynamic drag is a known contributor to fuel consumption qualitative comparisons between the trends in drag reduction can be compared to improvements in fuel economy.

Figure 9 shows some of NREL's fuel economy results. The front truck behaved as expected according to the numerical results: The smaller the separation distance, the greater the gains in the fuel economy. However, the rear truck exhibited some interesting trends that are not revealed in the initial computational analysis. For the rear truck, a peak efficiency is observed to occur at a separation distance between 50 and 75 feet when operating at 65 mph. Despite the rear truck's downward inflection at spacings below 40 feet, the combined economy of the platoon still exceeds a 4% improvement over a single truck. Not shown in the figure presented is evidence for fuel economy gains beyond 75 feet. During NREL's testing small fuel economy gains for the front truck were obtained even beyond the predicted 70 feet, suggesting possible fuel gains beyond the projected threshold.

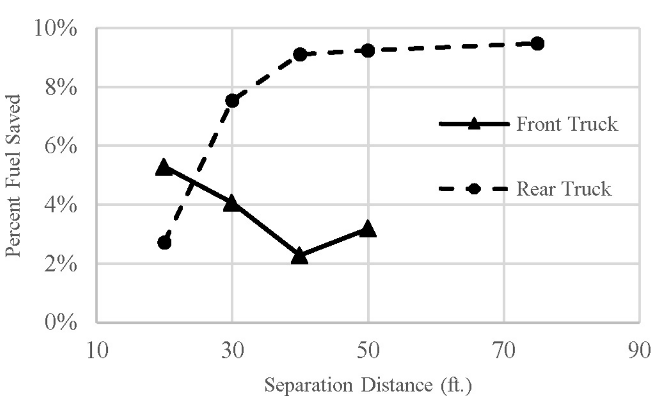

Figure 9. Percentage Fuel Saved for NREL Tests, 65 mph, 65K lbs/Loaded Weight [5]

In terms of validating the CFD study, fuel economy tests represent a significant challenge due to the lack of direct correlation between the predicted drag force on the vehicle and the fuel consumed. Practical factors also play a role in the performance of the platoon, which are not easily isolated during operation. These include changes in engine control due to the DATP system, engine temperature and flow conditions, platoon spacing gap control, lateral offset, and environmental factors like heat, humidity, and wind. These factors make it extremely difficult to separate out differences in modeled aerodynamic effects from these real-world factors to explain the rear truck's unexpected trend. However, none of these explanations alone adequately clarifies the large difference between the expected trend and the realized trend in the fuel economy.

There is also evidence that fuel economy improvements extend beyond the previously predicted 70-foot following distance for both the front and rear trucks. In fact, the rear truck may show significant improvements beyond the 100-foot following distance.

Expanded Computational Scope

To better understand the apparent discrepancy between the percent fuel saved and the predicted aerodynamic drag, further computational studies were undertaken. Of particular interest was the effect of relative lateral position between the lead and follow vehicles. Figure 10 depicts the percent drag reduction versus separation distance for a relative lateral offset of two feet. This offset distance was selected as a representative of a practical maximum offset during nominal DATP operation.

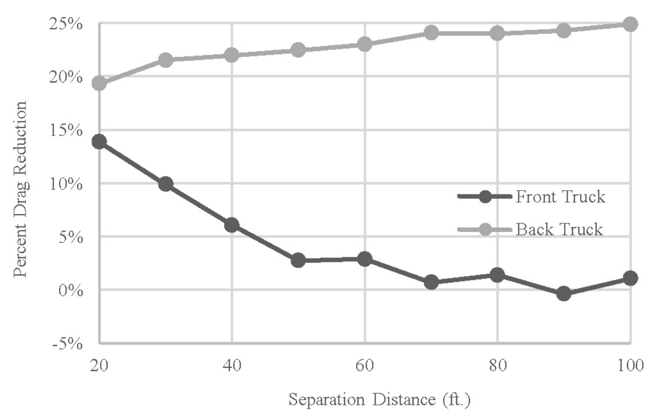

Figure 10. Percent Drag Reduction vs. Separation Distance at a Lateral Offset of Two Feet

Once again, it is impossible to make fully assertive statements regarding the rear truck fuel consumption trend. However, the drag reduction trend for the lateral offset case qualitatively appears to track the measured trends in fuel saved. Particularly promising is the changed inflection in the rear truck's trend in conjunction with little observable change in the front truck's profile. This suggests that the front truck is affected by lateral offset far less than the rear truck, possibly explaining why the rear truck's inflection point is masked by the ideal case of perfectly centered platooning. This difference is shown in Figure 11.

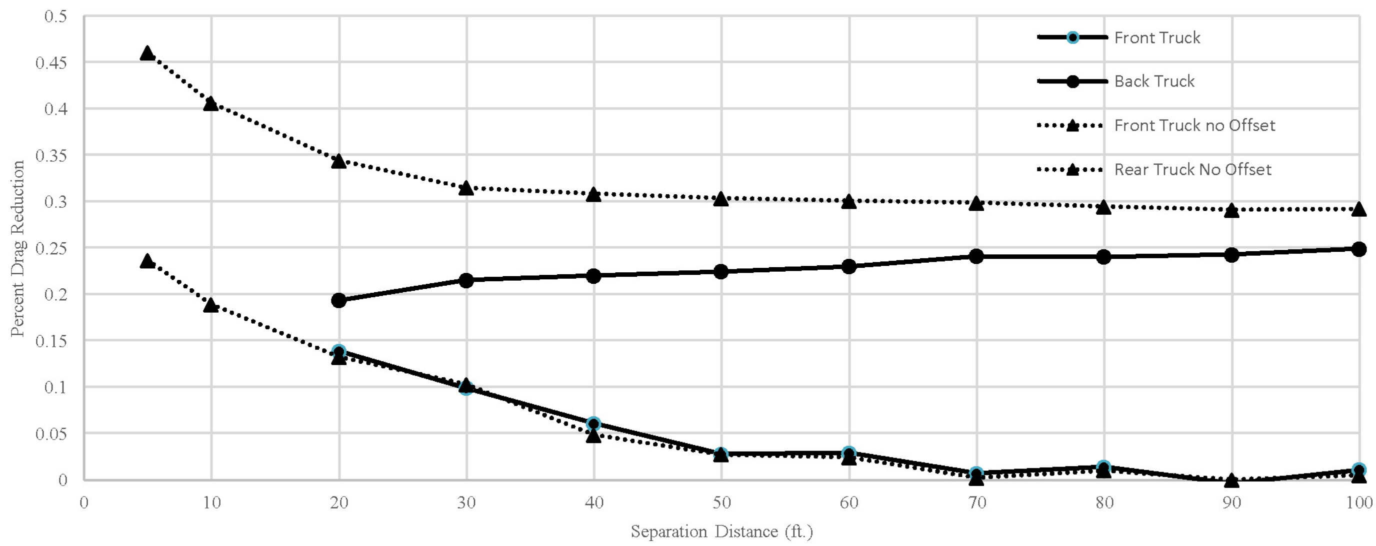

Figure 11. CFD Results including 2 ft. Lateral Offset

This trend can be explained by considering the dynamics of the wake shedding off the front truck. With an offset, a portion of the rear truck is exposed to the undisturbed free-stream flow. This creates an asymmetrical pressure distribution on the front of that truck, as evidenced in Figure 12. This is most likely due to vortex shedding off of the front trailer working in tandem with the portion of the rear truck exposed to the free-stream flow. As the separation distance decreases, the width of the wake also decreases, since the wake does not have time to dissipate in the longitudinal direction. Thus, at close spacings, the effect of being offset seems to be exacerbated.

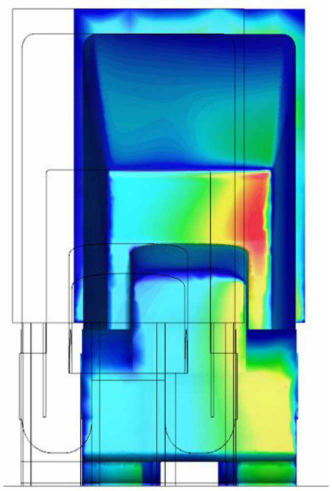

Figure 12. Pressure Contour on the Front Surface of the Rear Truck with Outline of Front Truck

Thus, the margin of error for remaining inside the wake likely becomes slimmer at small separation distances. In addition, at close spacings, vortices shed off the front truck have less time to dissipate, leading to buffeting of the rear truck. This in turn causes the rear truck to be offset more often and more drastically at close spacings. It is therefore plausible that lateral offset can be attributed, at least in part, to the reduced performance at very tight spacings, where its impact is greatest. This aerodynamic effect due to the lateral offset, along with other effects stemming from overall vehicle control, provide possible explanations for the measured trend.

Summary/Conclusions

Throughout this work, a relationship between the longitudinal separation distance and the fuel consumption of a platooning system was sought. Utilizing CFD as a simulation tool, a trend demonstrating that the fuel consumption increases as a function of the separation distance was discovered. In general, it was believed that the smaller the separation distance, the greater the gains in performance for the platoon.

After looking at a series of fuel economy tests, this prediction was shown to differ for the rear truck at spacings less than 50 ft. At these spacings, the fuel consumption benefit for the rear truck degrades. This spurred a second set of numerical simulations in which the lateral offset between the lead and following trucks was carefully examined in an attempt to explain this trend. After the secondary round of numerical simulations of the platoon, the model suggests that the lateral offset of the vehicles is related to the fuel consumption of the platoon, with lateral offset being an even more critical component of the performance of the platoon at closer spacings. It is suggested that both the effect of lateral offset and its likelihood are inflated at closer spacings. This is due to the strength of the vortex shed from the front truck causing buffeting of the second truck coupled with the effect of the unabated free stream now directly impacting the rear vehicle.

This suggests that further investigation is warranted to fully characterize the impact of lateral offset in real-world platooning scenarios. This could inform design of future systems regarding the addition of lateral control assistance, particularly at closer following distances. In general, this work shows that high fidelity aerodynamics simulations can play an important role in the design and evaluation of Driver Assistive Truck Platooning, a system which holds great promise for gaining fuel economy and safety benefits in trucking operations.

References

1. *ATA American Trucking Trends 2015*. American Trucking Association, 2015.
2. Torrey W. F. (ATRI) IV and Murray D. (ATRI), “An Analysis of the Operational Costs of Trucking: A 2014 Update,” Atlanta, Georgia, 2014.

3. Bonnet C. and Fritz H., “Fuel Consumption Reduction Experienced by Two Promote-Chauffeur Trucks in Electronic Towbar Operation,” in Proceedings of the 7th World Congress Conference on Intelligent Systems, 2000.
4. Bonnet, C. and Fritz, H., "Fuel Consumption Reduction in a Platoon: Experimental Results with two Electronically Coupled Trucks at Close Spacing," SAE Technical Paper 2000-01-3056, 2000, doi:10.4271/2000-01-3056.
5. Lammert, M., Duran, A., Diez, J., Burton, K. et al., "Effect of Platooning on Fuel Consumption of Class 8 Vehicles Over a Range of Speeds, Following Distances, and Mass," *SAE Int. J. Commer. Veh.* 7(2):626-639, 2014, doi:10.4271/2014-01-2438.
6. SAE International Surface Vehicle Information Report, “Taxonomy and Definitions for Terms Related to On-Road Motor Vehicle Automated Driving Systems,” SAE Standard J3016, Rev. Jan. 2014.
7. *ANSYS FLUENT Theory Guide*. ANSYS Inc., 2009.
8. Jones W. . and Launder B. ., “The prediction of laminarization with a two-equation model of turbulence,” *Int. J. Heat Mass Transf.*, vol. 15, no. 2, pp. 301-314, Feb. 1972.
9. Wilcox D. C., *Turbulence Modeling for CFD*. DCW Industries, Inc., 1994.
10. Ahmed, S., Ramm, G., and Faltin, G., "Some Salient Features Of The Time-Averaged Ground Vehicle Wake," SAE Technical Paper 840300, 1984, doi:10.4271/840300.
11. Pagliarella, R., Watkins, S., and Tempia, A., "Aerodynamic Performance of Vehicles in Platoons: The Influence of Backlight Angles," SAE Technical Paper 2007-01-1547, 2007, doi:10.4271/2007-01-1547.
12. Lanfrit M., “Best practice guidelines for handling Automotive External Aerodynamics with FLUENT,” 2005.
13. Davila A., “Report on Fuel Consumption.” SARTRE, 2013.
14. Browand F., McArthur J., and Radovich C., “Fuel Saving Achieved in the Field Test of Two Tandem Trucks,” 2004.
15. SAE International Surface Vehicle Recommended Practice, "Joint TMC/SAE Fuel Consumption Test Procedure-Type II," SAE Standard J1321, Rev. Oct. 1986, p. 29.

Contact Information

The main authors can be contacted at the following:

Hugh (Luke) Humphreys
hnh0004@auburn.edu

Dr. Joshua W. Batterson
jbatters@gmail.com

Acknowledgments

This work was funded under the Federal Highway Administration Contract DTFH61-13-R-00011. Auburn University is leading this research under a Federal Highway Administration Exploratory Advanced Research project to investigate technical and business aspects of DATP. The project is led by Auburn University with their partners Peloton Technology, Peterbilt Trucks, Meritor WABCO, and the American Transportation Research Institute.

Definitions/Abbreviations

DATP - driver assisted truck platooning

CFD - computational fluid dynamics

V2V - vehicle to vehicle

ATRI - American Transportation and Research Institute

DSRC - dedicated short range communication

RKE - realizable k -epsilon

DES - detached eddy simulation

RANS - Reynolds averaged Navier-Stokes

TKE - turbulent kinetic energy

LES - large eddy simulation

SGS - sub gridded scale

NREL - National Renewable Energy Laboratory

The Engineering Meetings Board has approved this paper for publication. It has successfully completed SAE’s peer review process under the supervision of the session organizer. The process requires a minimum of three (3) reviews by industry experts.

ISSN 0148-7191

http://papers.sae.org/2016-01-0167