Effects of Highway Slipstreaming on California Gas Consumption

Kevin Duan
Chris McDaniel
Amanda Muller
Brett Yokota

MAE 171B- Mechanical Lab II
University of California- San Diego
Jan Kleissl

June 13, 2007

Abstract

The goal of this report is to find the effects slipstreaming will have on California's gasoline consumption. Two methods were used to obtain results, an experimental and modeling approach. Model cars were set up in a wind tunnel to simulate slipstreaming and the drag of the 2^{nd} vehicle was recorded. Due to the restrictions of the wind tunnel, there was a limited amount of variance between the setups. Cosmos Floworks was used to simulate vehicle scenarios on the computer. From the experimental approach, there was a 61% drag reduction due to drafting and the modeling approach showed a 40-60% drag reduction, it translates into 45-67% fuel savings. However, if everyone were to follow at the legal following distance of 288 feet, at 65 mph, the drag reduction would be negligible and show no increase in fuel savings.

Table of Contents

List of Figures

List of Tables

Introduction

When a vehicle moves, it generates high air pressure in front and low pressure behind. This difference in pressure creates a drag force, also known as pressure drag, which accounts for a large part in automotive fuel consumption at high speeds. The concept of slipstreaming is to utilize the regions of reduced pressure behind moving vehicles to lessen the oncoming pressure drag, thus lowering fuel consumption. For example, take two cars driving one behind the other, the combined slipstream between the two cars will benefit both cars in terms of reducing the pressure gradient across each car. The main objective of this experiment is to estimate the reduction in individual and statewide fuel consumption due to the effects of slipstreaming.

The main parameters that can alter the effects of slipstreaming are the speed, the spacing, and the size and shape of the vehicles. In order to analyze this phenomenon and take each of these independent variables into account, two different methods were used; a modeling and an experimental approach. First, COSMOS FloWorks, an additional SolidWorks feature program, allowed for the study of the drag forces acting on a system of vehicles. This system consisted of two or more vehicles ranging in size and separated by a variable distance. One key part of this modeling was validation. Because FloWorks is typically used to analyze internal flow, and this experiment involved an external flow, a determination of the drag coefficient of a cylinder with a known value was computed for comparison. After this was performed, tests were run in the wind tunnel located in the MAE 171A lab, EBUII, UCSD. The conditions of the wind tunnel were reproduced in FloWorks for further support, and the results for the drag coefficient were compared. Finally, by researching and using highway transportation data for California, fuel savings for the individual and major freeways were estimated.

Theory

This experiment is based on the theory of external incompressible flow; the flow is characterized as incompressible because the considered ambient air has a constant density. The Reynolds number, Re, is the ratio of inertial forces ($v_s\rho$) to viscous forces (μ/L) and is an important parameter on smooth simple geometric objects. Laminar flow usually occurs at Re numbers less than 10^5, and turbulent above 10^5. Surface roughness and geometry play important roles in laminar to turbulent transition points. In this experiment, all flows are considered turbulent in our quest to find drag.

Drag is the force exerted on a body by an oncoming fluid. There are two types of forces that create drag; one is the skin friction force and the second is the pressure force. In turbulent flow conditions on a blunt object, skin friction drag is reduced, and pressure drag takes over almost 95% of the total drag. The pressure drag is caused by the pressure differential over surface area. The easiest application to analyze drag force is on a sphere or cylinder.

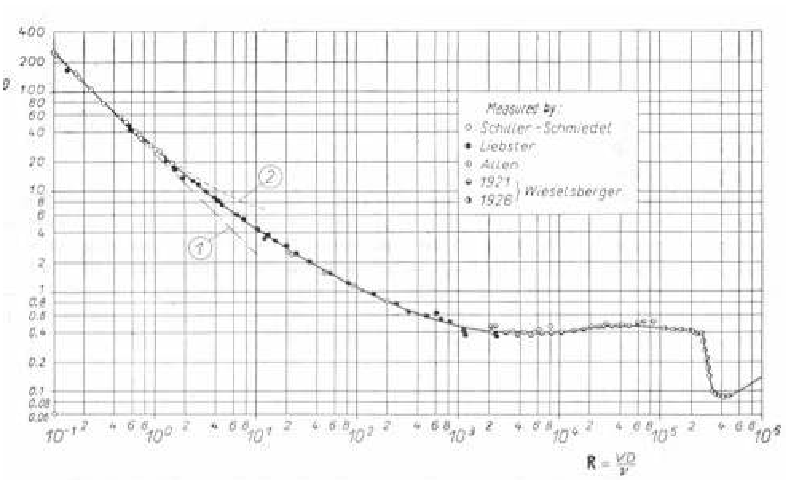

Figure 1: Re vs. Cd of a Sphere[1]

The drag force of a sphere is a function of the Reynolds number. Figure 1 shows the relationship between the Reynold's number and drag coefficient, C_d, of a sphere. Increasing the Reynolds number causes the drag coefficient to decrease. After a Re of 1000, the pressure drag completely takes over, shown by the flat portion of the plot. Increasing the Re from 1000 and higher has little effect on the C_d, up to a Re between 10^5

and 10^6 where the flow becomes turbulent. The Reynolds numbers for this experiment will range from roughly 180,000 to 320,000 signifying turbulent flow conditions.

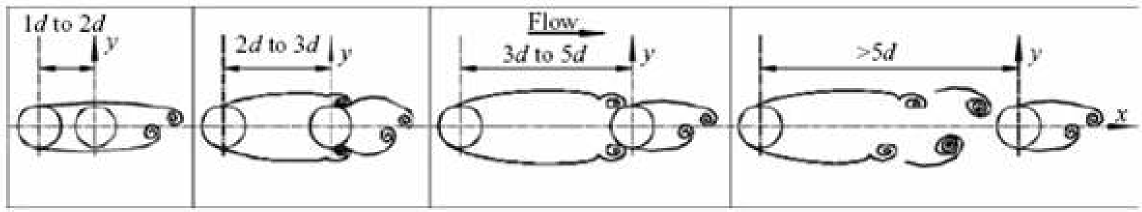

Figure 2: Flow Separation of a Sphere[1]

Directly behind the sphere the flow separates and forms an area of reduced pressure. If a second sphere is placed in this area, it experiences a reduced oncoming pressure and correspondingly less pressure drag, shown in Figure 3. Figure 4 presents numerical data showing the drag coefficients of two cylinders in the described setup.

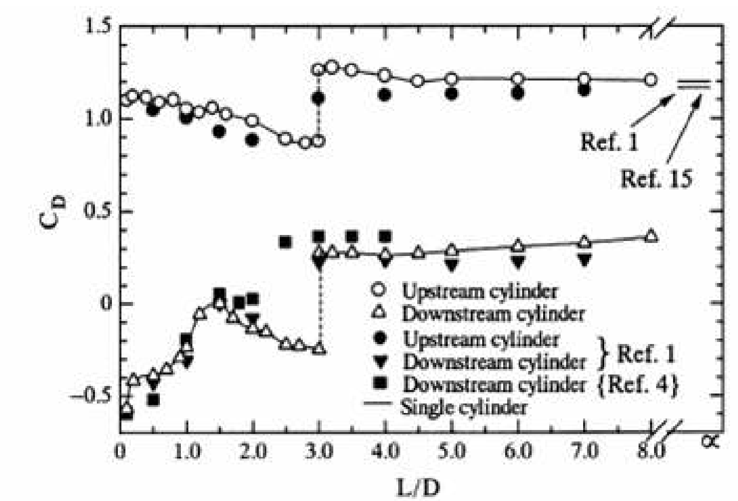

Figure 3: C_d of an Upstream and a Downstream Cylinder; L=Length, D=Diameter[1]

The downstream cylinder has a significantly smaller drag coefficient signifying a smaller drag force. The drag coefficient and force are related by equation 5.1.

$$F_D = \frac{1}{2}\rho v^2 A C_D \qquad \text{Eq. 5.1}$$

In the experimental portion of this project, the forces applied to the follower vehicles in the wind tunnel are measured with a strain gauge. The major component of a strain gauge system is a Wheatstone bridge circuit, shown in Figure 4.

An input voltage, denoted by V_{EX}, goes through the series of resistors, R, and provides an output voltage V_o. The circuit shown above is completely balance, meaning the output

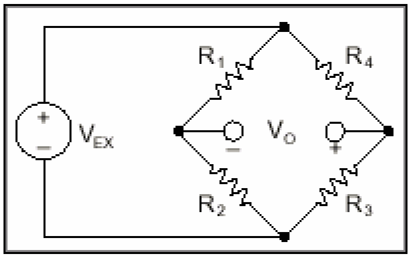

Figure 4: Wheatstone Bridge

voltage is zero. In this experiment, one of the resistors has been replaced by a strain gauge. Initially, the circuit is balanced; however, if forces are applied to the strain gauge its resistance changes resulting in a measurable output voltage. This voltage allows for the measurement of both normal and axial forces on the follower vehicle.

Experimental Procedure

A computational fluid dynamic, CFD, analysis was used to create full scale models, and predict the reduction in drag. FloWorks, a counterpart of SolidWorks, analyzed the flow. First FloWorks needed to be validated, and ensure correct setup of the model. A sphere was constructed and FloWorks analyzed the drag force and drag coefficient of an air stream with a Reynolds number of 10^5. After, a second sphere was added to show the basic advantages of the low pressure zone behind the first sphere.

Vehicle scenarios were then created. Realistic models of different vehicles types were drawn and extruded. Two different vehicles were positioned in line at variable distances from each other. The scenario was analyzed with FloWorks at different Reynold's numbers to equate the horizontal force. An equation goal was created to find the coefficient of drag over the entire model. The difference in the drag coefficient between slipstreaming and non-slipstreaming indicated the reduction of drag for the first vehicle in slipstreaming mode. The drag coefficient of the second vehicle was be determined by visual inspection of the pressure difference between the front and back of the second vehicle, using the flow trajectory graphs in FloWorks.

A wind tunnel was used to experimentally determine the drag force on the second vehicle. This was accomplished by using scaled models of a car, a truck, and a semi. The scaled models require custom-made mounting brackets to place the models in the wind tunnel. The car and truck were each mounted on the strain gauge, whereas the semi was mounted on a simple platform in the front portion of the wind tunnel test section. The experiment was set up with the same scenarios as the modeled in FloWorks, experimenting with speeds ranging from 50-150mph and variable vehicle configurations; there was not room for variable separation distances. The follower vehicle was placed on the strain gauge to measure the normal, N, and axial, A, forces that the vehicle was experiencing. These forces along with the angle of attack, α, were used to calculate the drag force as shown by equation 7.1.

$$F_D = N \sin\alpha + A \cos\alpha \qquad \text{Eq. 7.1}$$

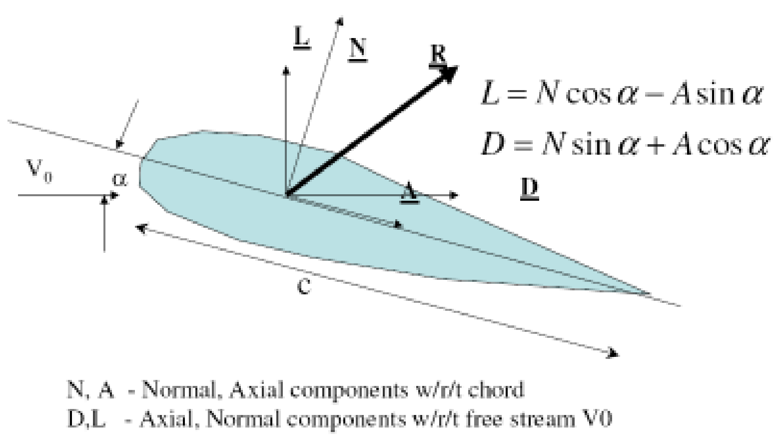

Figure 5: Normal and Axial components[4]

The values obtained from this method were compared to those using the modeling method to reduce error. Also for visual comparison, colored smoke was released into the wind tunnel to observe flow movement.

Data and Results

Finding the drag reduction of vehicles is one of the primary objectives of this experiment, so it is crucial to be able to determine accurate drag coefficients. By modeling some basic cylinders, the results can be compared with known cylinder drag coefficient that can be found in fluid texts. The results obtained, however, were off as seen in Figure 6:

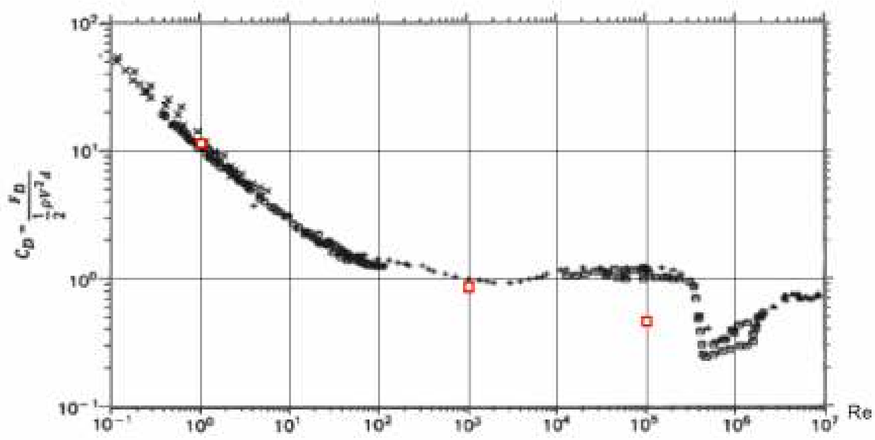

Figure 6: Floworks Cylinder Drag Coefficient compared with known values.

This does not mean that Floworks will be useless for this experiment; it just means that a different approach must be taken to obtain reliable results. Although it may not be able to calculate the exact value of drag coefficients accurately, it may still be able to calculate drag reduction accurately through relative difference.

From theory, drag reduction was demonstrated with the use of two cylinders.

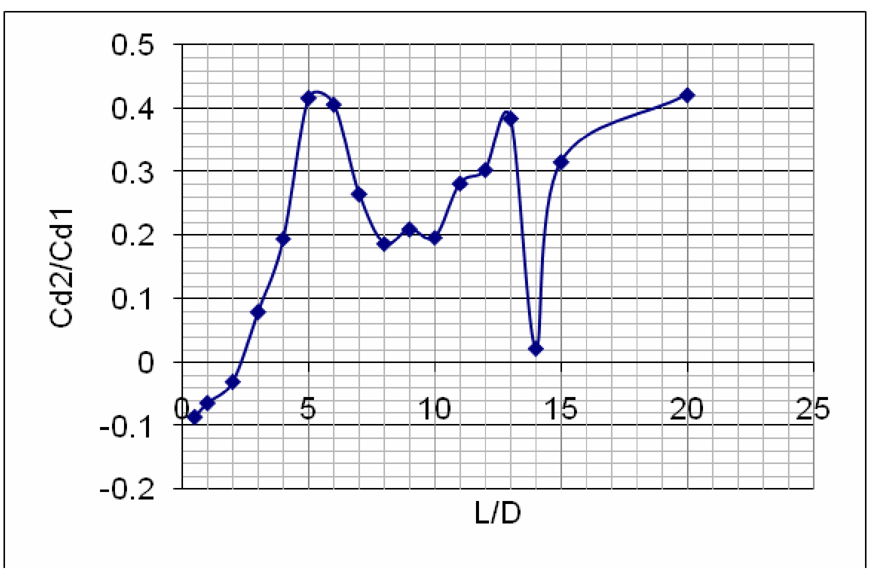

Figure 7: Two Cylinder Validations - Percent Reduction in Drag Coefficient

With Floworks, it would only be appropriate to reproduce those experiments to validate the accuracy of Floworks in a relative difference approach. Simulation results are shown in Figure 8. The plot shows the drag coefficient of the trailing cylinder divided by the original drag coefficient as if it were alone in free stream. The results were rather sporadic but the general range of results fit the theoretical shown in Figure 3. The erratic nature and error in this validation is acceptable for this experiment and will be discussed in detail in the discussion and error analysis section.

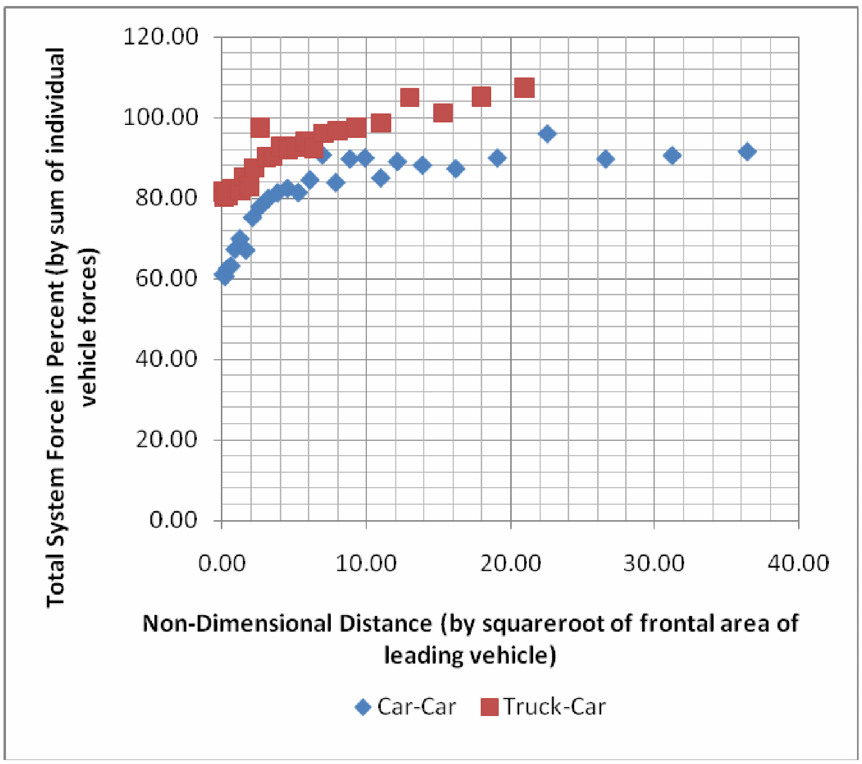

Figure 8: Floworks Simulation of total drag reduction for the cases of truck-car and car-carDrag Coefficient of trailing cylinder Divided by Original Drag

Next, drag reduction simulations were performed using Solidworks-Floworks models in the cases of semi-truck in front of a car and also a car in front of another car at the typical freeway speed limit of 65 mph. Because the individual drag force values obtained in Floworks may be inaccurate, the relative difference between the reduced drag cases (vehicles in slipstream) will be compared to the drag forces of each vehicle as if it were by itself (no slipstream). The results can be seen in Figure 9, where the x-axis is a non-dimensional length obtained from dividing the spacing length by the square-root of the leading vehicle's frontal area. It is clear that slipstreaming does indeed reduce drag force at small spacing; however, this advantage vanishes as the spacing increases. It is also worthy to note that the total force should theoretically reach 100% as the distance between the vehicles become infinity; however, due to the resolution of Floworks simulation, the accuracy diminishes with large computational domains. But nevertheless, the results from these simulations are acceptable and they show that drag reduction is definitely possible and significant at small vehicle spacings.

At the same time, wind tunnel experimentations of a car, a truck, and a semi-truck were conducted at different speeds. The results can be seen in Figure10, where the cases of: car alone, truck alone, truck-car, and truck-truck were conducted at 3 different percentages of the 150 mph full speed. As expected, the drag force of the second vehicle in slipstream was significantly smaller compared to its original drag force as if it were alone. The car experienced a 61.8% reduction in drag coefficient at full speed beginning with a value of 0.089 and ending with a much improved value of 0.034. The truck experienced a 58.4% reduction; its drag coefficient of 0.137 reduced to 0.057.

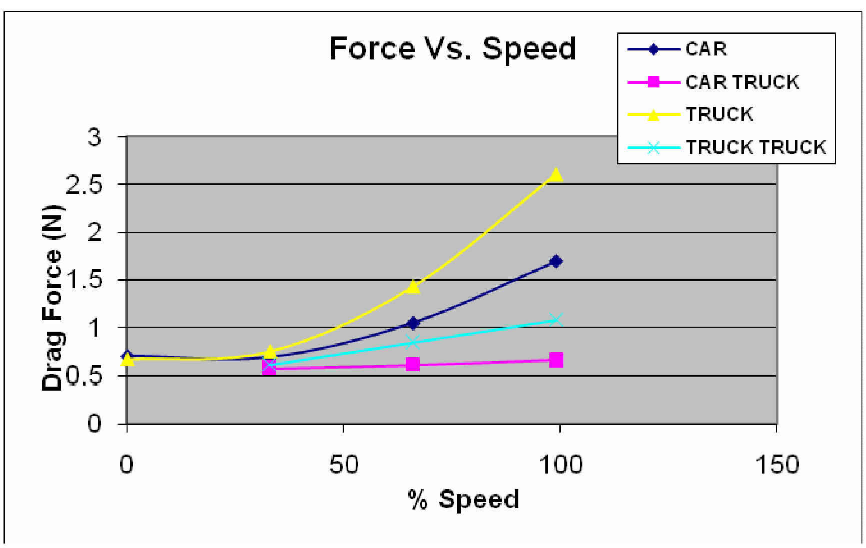

Figure 9: Wind tunnel Experiment Results

The results from this experimental procedure were prone to few errors which are discussed in the error analysis sections. Therefore, a Floworks simulation of this wind tunnel experiment was also performed.

The validation of the wind tunnel experiments with Floworks can be seen in Figure 11. Although the exact values maybe off, the relative difference is still somewhat accurate. It is worthy to note that at higher speeds, the difference between Floworks and experimental results increase due to the phenomena shown in Figure7, which is that Floworks generally under estimates drag at higher speeds. The reduction results can be seen in Figure 6 where the results are somewhat consistent with the results shown in Figure 9 considering the close spacing in the wind tunnel experiment. A 20-40% reduction in drag force can be concluded for vehicles at extreme close spacing of about 0.5 m in actual scale. The anomaly for the truck-truck case in the wind tunnel is due to the space limitation in the wind tunnel which will be further elaborated in the discussion section.

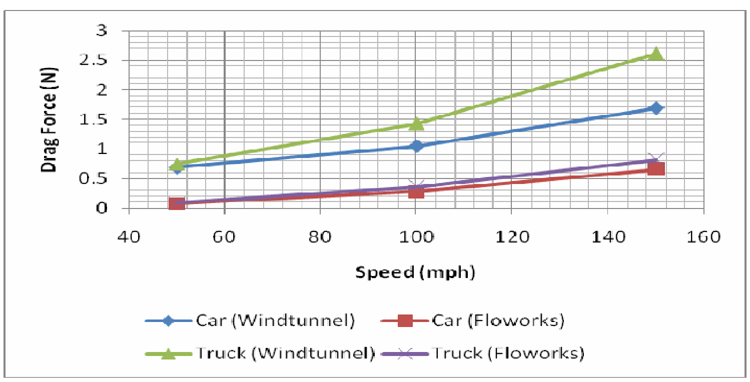

Figure 10: Wind tunnel experiment validation with Floworks models

Figure 11: Wind tunnel experiment validation of drag reduction

The average fuel consumption for California is 40 million gallons of gasoline daily. From Mythbusters, a plot was obtained on percentage of mpg saved verses drag force reduction.

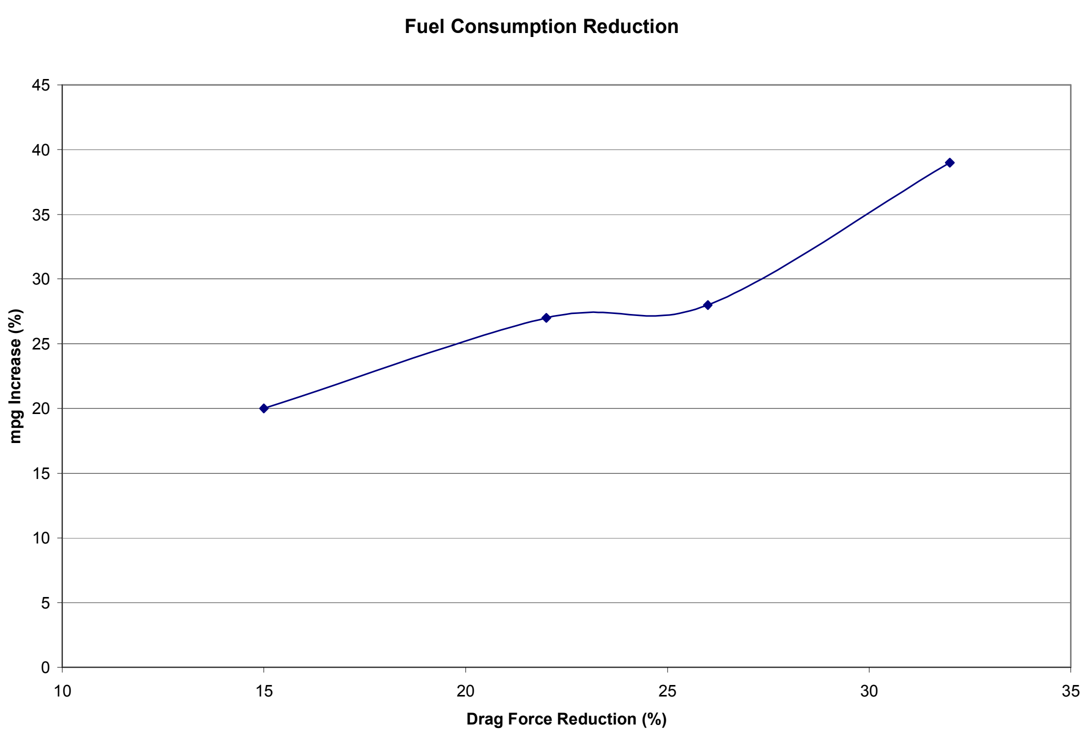

Figure 12: Fuel Consumption Reduction

This shows as the drag force reduction increases, the mpg also increases linearly.

Discussion

By comparing the known experimental drag coefficient of cylinders to Floworks simulation, the results concluded a decrease in accuracy in high Reynolds number regimes. The error at Reynold's number of 10^5 turns out to be off by almost an entire order of magnitude. This means that the drag forces calculated by Floworks to simulate highway slipstreaming will be inaccurate, and in turn the coefficient of drag will also be invalid. To compensate for these problems, additional modeling was done to validate Flowork results in terms of relative differences. The drag reduction of two cylinders has been investigated in another paper[1], and those results were compared directly with Floworks models. The results were satisfactory and that the range of the differences was quite accurate. This concludes that in the experimenting regime of $Re \sim 10^5$, the absolute values of Floworks analysis (such as the drag force and drag coefficient) cannot be trusted; however, the difference in drag will still be valid.

Next, due to the Floworks limitation, the results from the modeling of two vehicles can only be displayed in terms of difference. So from the results section, Figure 3 shows the total drag of both vehicles divided by the sum of both vehicles as individuals. What is interesting is that the drag reduction in the car-car case seems greater than that of the truck-car case. This is because the drag force of the leading truck alone is so large that the reduction of drag for the trailing car seems minimal to the entire system. In fact, the total force reduced from the truck-car case is actually greater than that of the car-car case as seen in Table 3 which makes sense. In conclusion, this method gives an accurate account of the drag reduction of slipstreaming versus driving alone. The results show a 20-40% reduction in drag force at very small spacing between the vehicles. Considering the typical safe distance between vehicles, the reduction of drag would only be around 10%.

In addition to the modeling approach, actual wind tunnel experiments were performed. The results were error prone due to the size and other limitations of the wind tunnel. Thus, the wind tunnel experiment was also modeled in Floworks for validation. The results were acceptable although large amounts of error plagued the experimental process. The results also show a reduction of drag at smaller spacing and higher speeds.

If one was to assume that half of the 40 million gallons of gas used is used up from highway usage, then that would leave a total of 20 million gallons. Using the results from the experimental approach resulting in 40-60% drag reduction, it translates into 45-67% fuel savings. However, if everyone were to follow at the limited following distance of roughly 3 seconds, or 288 feet, the drag reduction would be negligible and show no increase in fuel savings. If people were to travel at a following distance of 100 feet and at a rate of 55mph, one would save approximately 10% of fuel consumption and California would save 2 million gallons of fuel a day.

Limitations

Floworks is a engineering package that can simulate external flows over various models. However, because of the chaotic turbulence regions, Floworks simulations at high Reynolds number regimes are far from accurate. To overcome this difficulty, the relative difference approach was taken as relative error should remain unchanged.

Floworks computes the drag force on two body systems by calculating the total drag force on the entire assembly. This means that the drag force contribution of each vehicle is unknown. In other words, the drag and drag coefficient cannot be calculated individually. Instead, the drag force can be expressed as a reduction on the system, by dividing the Floworks result by the sum of the stand alone drag force from each vehicle. And in fact, calculating the reduction in the total drag of the system translates to the amount of fuel consumption reduction of the entire system.

For the wind tunnel experiment, there were many restrictions and limitations. For one, the maximum speed the wind tunnel can produce is only 150 mph, but for this experiment, over 1,920 mph was necessary. Next, the available space to put the experiment models was extremely small and therefore differently spaced runs were never conducted. Another effect of the small spacing is the problem of slight contact which could be the main error source to the results in Figure 6. Not to mention the accuracy of the strain gauges used, the wind tunnel experiments require fabrication of the vehicle mounts to place the vehicles and strain gauges.

Future Work and Recommendation

One major shortcoming in this experiment was the inability to accurately calculate the drag forces and drag coefficients in the slipstream. Although relative differences give a general insight of how closely drag force relate to vehicle spacing and speed, it fails to account for various car shapes and vehicle arrangement. Therefore, it may be wise to investigate and search for other Computational Fluid Dynamic software packages, such as Fluent, for this experiment.

Next, another source of drag reduction has been overlooked in this study, the existence of a third vehicle exerting a pressure on the back of the middle vehicle. In reality, most cars are formed in platoons where there is a chain of vehicles lined up one behind another. The vehicles in the middle would experience the greatest drag reduction due to reduced pressure in front and increased pressure from the back. So, for future work, experiments should take a platoon of vehicles into consideration.

Error Analysis - UCSD Wind tunnel Experiment

The drag force of the second vehicle was measured by a strain gauge that was directly attached to second vehicle. The LabView VI program installed on the UCSD wind tunnel workstation was configured to measure the error values of the strain gauge. The error propagation in the drag force was found by using statistical error analysis, which was applied to Equation 7.1.

The UCSD wind tunnel experiment had many sources of error. First, the size limitation of the wind tunnel did not allow the semi truck to be attached to the trailer, or it would have experienced undeveloped flow. The trailer is blunt shaped object, and created a higher pressure region in the front, than if the semi truck was attached. Secondly, the vehicle spacing was extremely close, The vehicle spacing was only 0.1 car length for the car behind semi experiments, and 0.19 car length for the truck behind semi experiments. The size limitation of the wind tunnel did not allow for any variance in vehicle spacing, which didn't allow any comparison. Although only the maximum speed was used to determine the drag coefficient, the tests were performed and slower speeds to reduce error.

The most important source of error was due to the flow speed of the wind tunnel relative to the 1/32 model. The Buckingham Pi theorem explains that the velocity of a model is inversely proportional to the scale. This means that a 1/32 scale model requires a wind tunnel velocity of 1,920 mph. The UCSD wind tunnel is capable of a maximum velocity of only 150 mph. Moreover, a speed of 1,920 mph is faster than the speed of sound, and compressible flow effects must be considered which would not produce valid results. In reality, the wind tunnel test represents a vehicle velocity of less than 10 mph. At this velocity on the highway, pressure differential drag force is minimal.

Error Analysis - SolidWorks Cosmos FloWorks

FloWorks is a Computation Fluid Dynamics program intended for analyzing internal laminar flow. It is unable to correctly calculate the drag force on objects with external turbulent flow. A full explanation of attempting to validate FloWorks can be found in Appendix A. The drag force results from the wind tunnel experiment were compared to simulated experiments analyzed in FloWorks. The percent error is shown in Table 1.

Table 1: %Error in FloWorks drag force compared to the UCSD wind tunnel experiment results.

	Speed (mph)		
Scenario	50 mph	100 mph	150 mph
Car	88.50%	73.00%	61.50%
Truck	88.10%	74.80%	68.70%

It was first assumed FloWorks was accurate, and could calculate drag force with marginal error. The drag force of the second vehicle is never directly given by FloWorks. FloWorks can only calculate the total system drag. In an attempt to find the drag force on the second vehicle, the drag force on the first vehicle was calculated, and then subtracted from the total system drag. This presented two forms of error. Error is present in FloWorks' results because is not a sophisticated turbulent CFD program. The drag force calculated drag force is not accurate., and therefore, can not be used explicitly**.** The second form of error evolves from the assumption that the first vehicle experiences no drag reduction. The USC study clearly shows that slipstreaming at vehicle spacing from 0 to 1.5 car lengths produces considerable drag reduction on the first vehicle.[5]

The final approach to find the drag reduction did not entail a specific break down on each vehicle, but analyzed the system as a whole. FloWorks results represent the percentage of total system drag reduction compared to the sum of drag force from the individual vehicles. This approach analyzes the two vehicles as a system; when slipstreaming, and when not. The inaccuracy of FloWorks still persists, however the error resulting from first vehicle drag reduction was eliminated. See a more detailed explanation in the appendix.

Conclusion

This experiment showed, both experimentally and analytically, that driving behind a semi on the freeway will significantly reduce drag, and therefore fuel consumption. Although the estimated fuel savings ranging from 45-67% are exaggerated due to experimental limitations, the conclusion is that significant fuel and money can be saved simply by driving behind a semi, even at reasonably safe distances. Both a full scale experiment and a more advanced computational fluid dynamics analysis program are suggested for providing accurate results.

Appendix A: Solid Works Cosmos FloWorks Validation

Much effort went into validating SolidWorks Floworks drag force calculations. Floworks results were compared to published experimental data from a single cylinder, two inline cylinders experiencing tandem flow, and data acquired from UCSD wind tunnel experiments.

Single Cylinder Validation

The drag coefficient of a smooth cylinder was calculated using Floworks over a range of Reynolds numbers from 1-10^6. The calculated drag force is not accurately portrayed, but is margin on the smooth cylinder with Reynolds number up to 10^2. A turbulent boundary layer arises around a Re# of 10^5 on a smooth cylinder. Turbulent boundary layer formation is independent of the Re#, and is dependent on object shape and surface roughness, in which blunt shaped objects form turbulent boundary layers in lower Reynold's number. Floworks has a difficult time accurately solving turbulent flow with a turbulent boundary layer, and therefore presents significant error after a Re# of 10^3. Figure 2 shows the Floworks results compared to published experimental data.

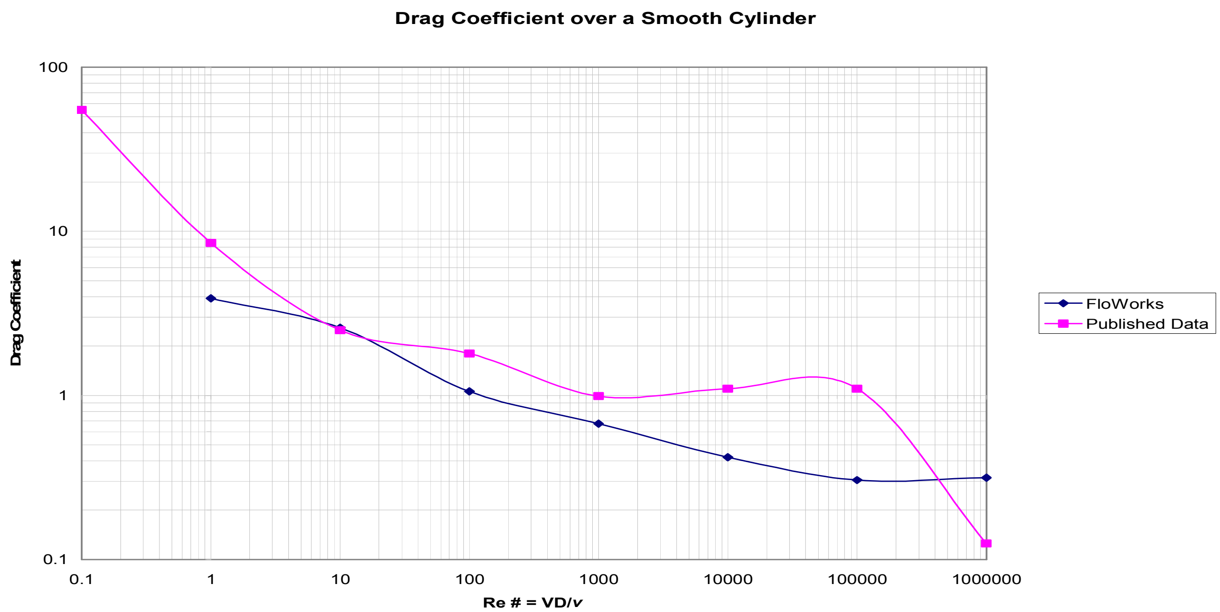

Figure 8: Comparison of Drag Coefficient of a smooth cylinder using Floworks and Published Data. (Fox, 441)

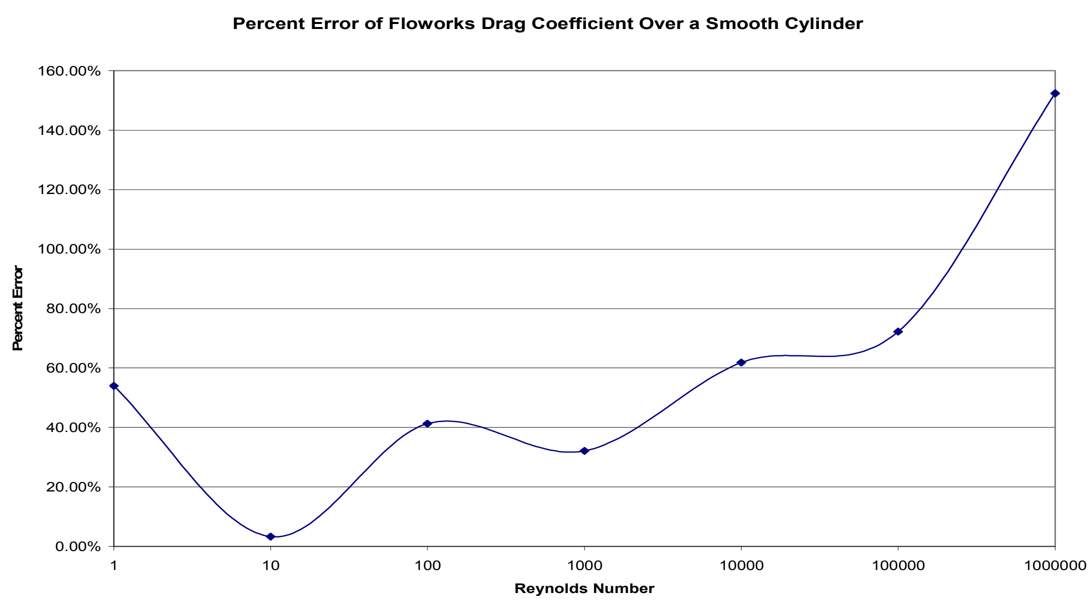

Figure 9: Percent error of Floworks drag coefficient over a smooth cylinder.

Two Cylinders in Tandem Flow Validation

In depth research has been performed on the effects of two cylinders in tandem flow. A FloWorks simulation, with parameters of prior experiments, was used to estimate the drag coefficient of the trailing cylinder. A Reynolds number of 6.5×10^4 was used. In the experiment, an interesting phenomenon occurs at an L/D spacing of 3.0, in which reattachment flow and jump flow occurs.

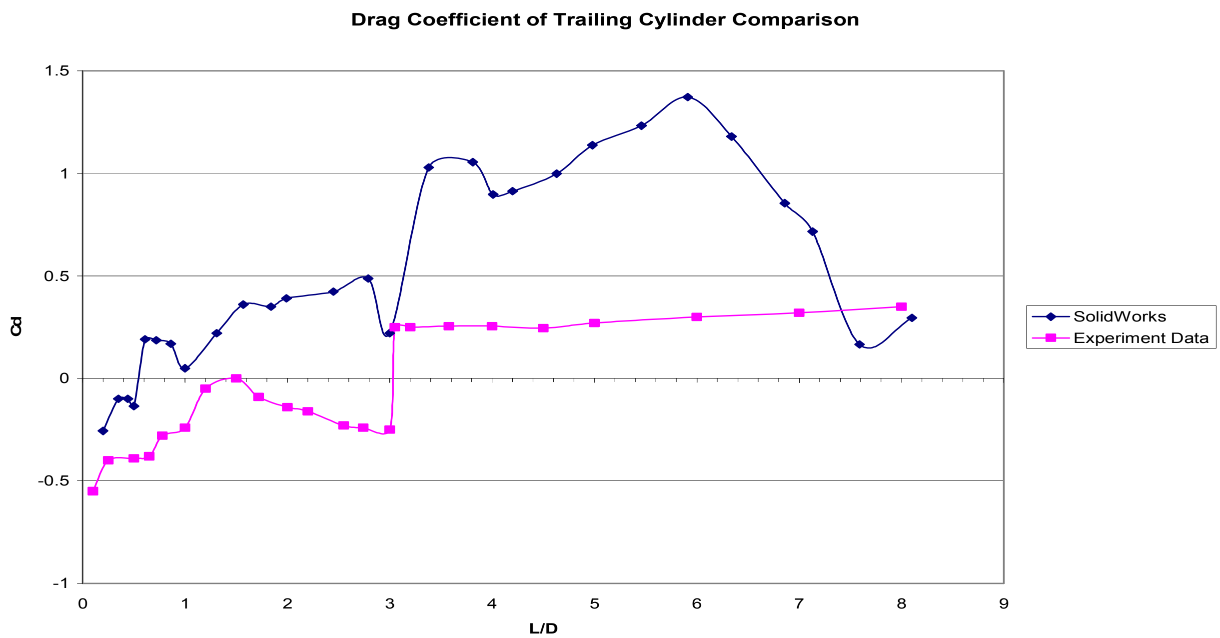

Figure 10: FloWorks comparison to experimental data of 2 cylinders in tandem flow.

UCSD Wind Tunnel Experiment Validation

The wind tunnel experiment was simulated in Floworks to see if the drag of the second vehicle could be found by subtracting the first vehicle's drag from the total drag of both vehicles calculated in Floworks. There was a large percent of error in calculating the second vehicle's drag by this method. The drag of the semi subtracted from the semi and car total drag is compared to the drag of the second vehicle from UCSD wind tunnel experiment. It is peculiar to note that this method shows a major drag force reduction of the first vehicle when the two vehicles are extremely close (less than one car length). The experimental vehicle spacing was approximately 0.1 car lengths for the semi-car scenario and 0.19 car lengths for the semi-truck scenario.

Table 2: Percent Error in FloWorks drag force compared with wind tunnel experimental data.

	Speed (mph)		
Scenario	50 mph	100 mph	150 mph
Car	88.50%	73.00%	61.50%
Truck	88.10%	74.80%	68.70%
Car behind Semi	100.70%	99.40%	105.50%
Truck behind Semi	104.90%	111.40%	125.70%

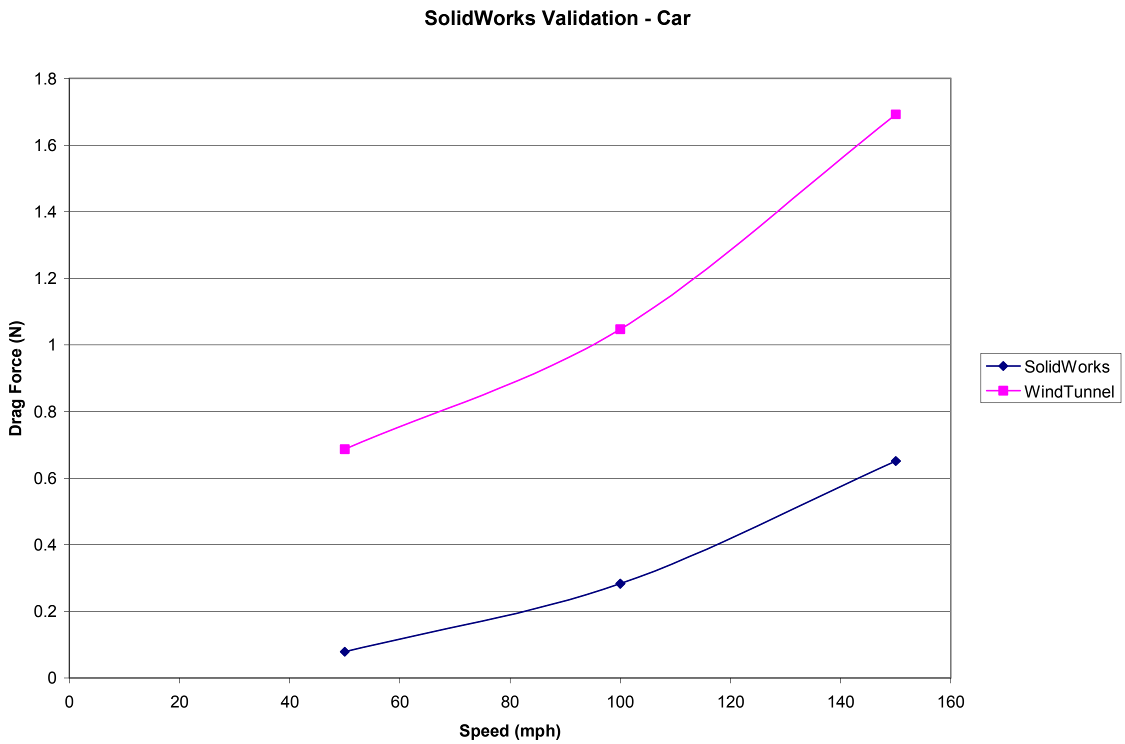

Figure 11: Drag force of car alone. FloWorks model compared to experimental data.

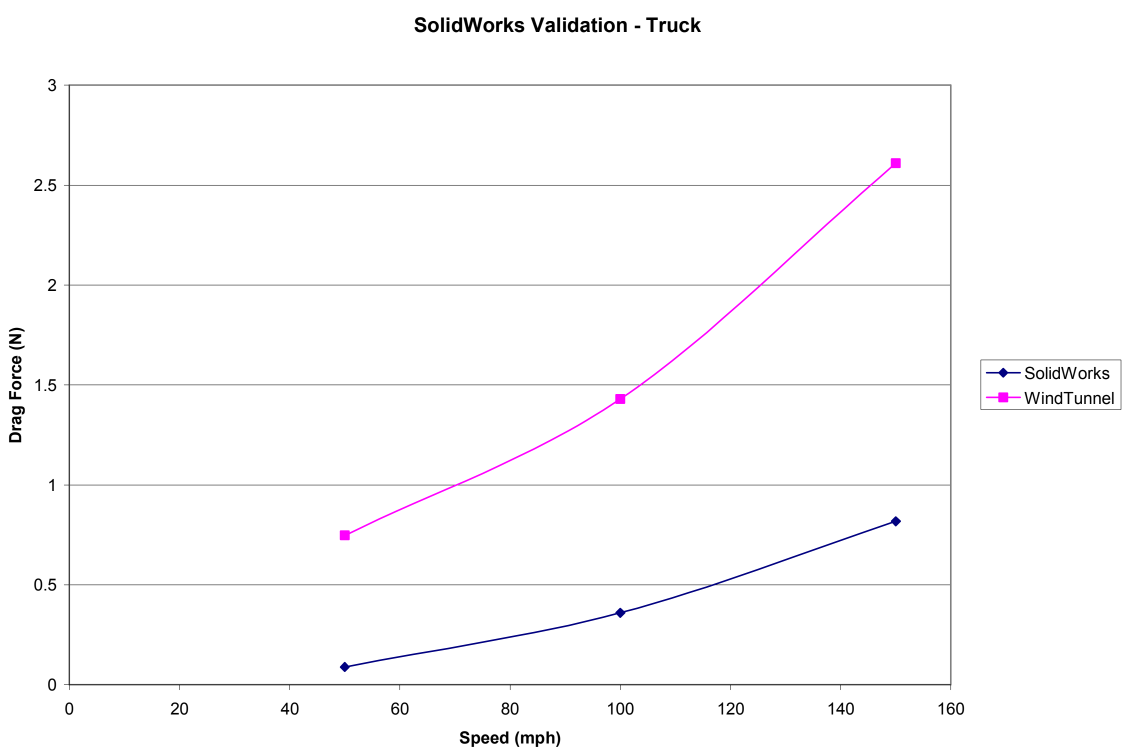

Figure 12: Drag force of truck alone. FloWorks model compared to experimental data

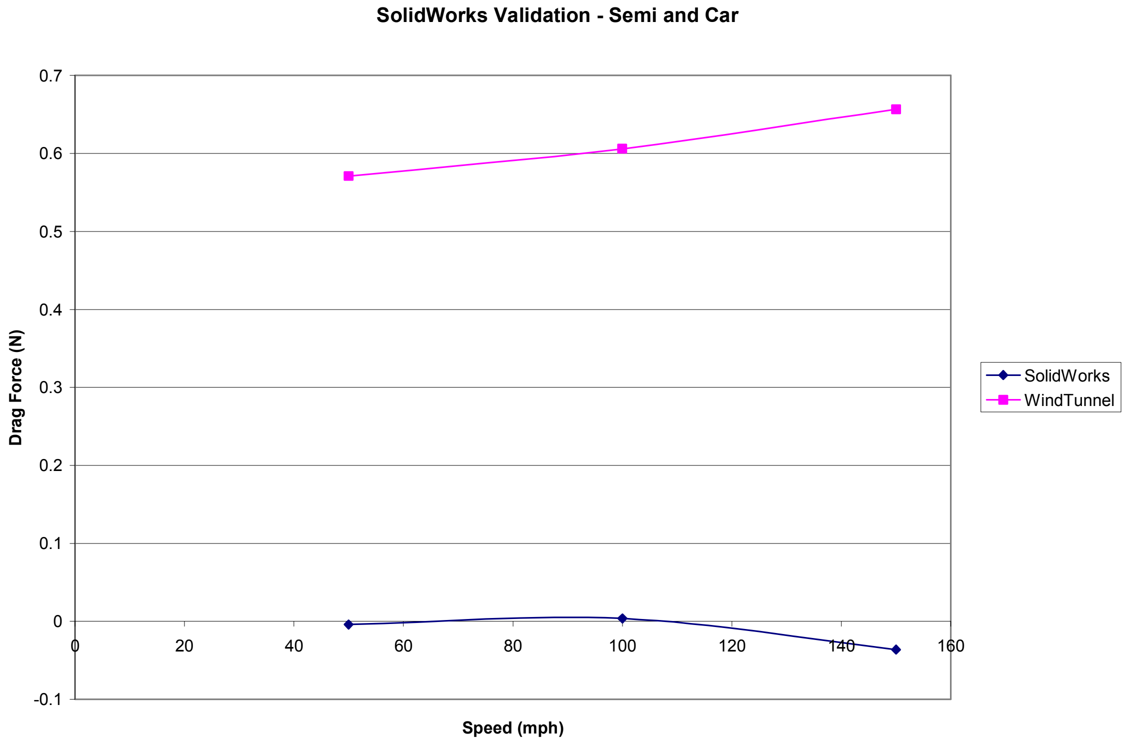

Figure 13: Drag force on car behind semi. FloWorks model compared to experimental data

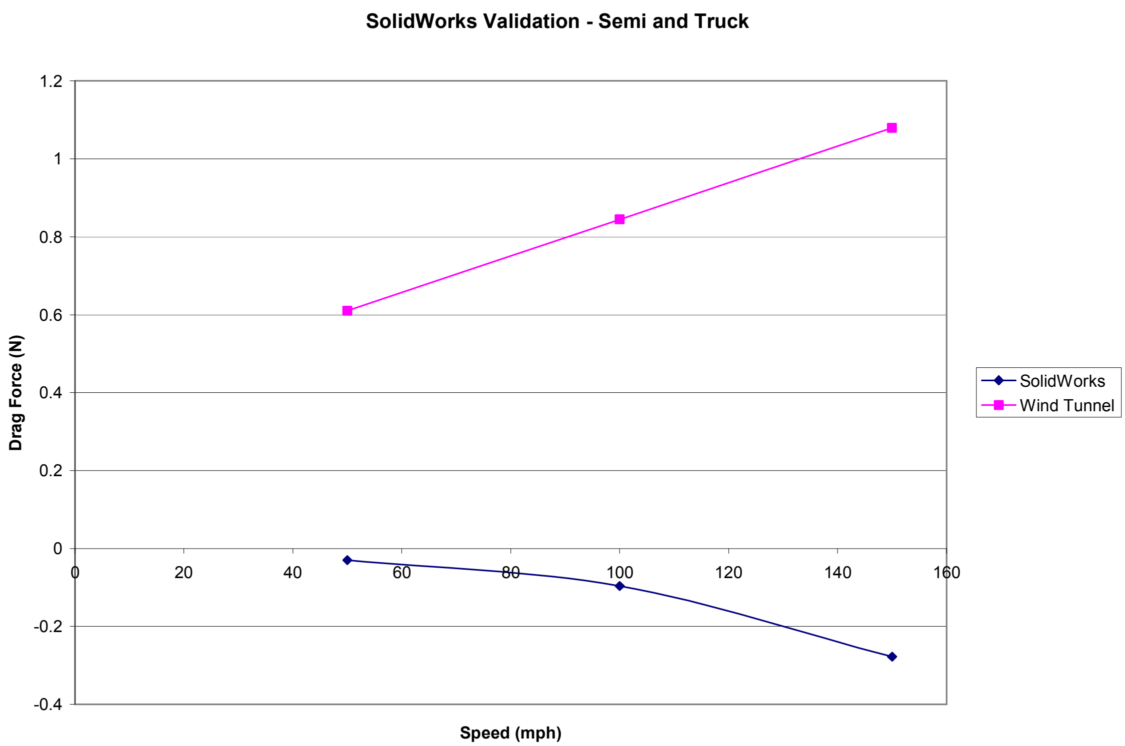

Figure 14: Drag force on truck behind semi. Calculated Floworks compared to experimental data.

Table 3: Mythbusters Drafting Test

Drafting at 55mph	Table of Results	
Distance Apart (ft)	**mpg**	**% Difference**
Control	32	-
100	35.5	11
50	38.5	20
20	40.5	27
10	44.5	39
2	41	28

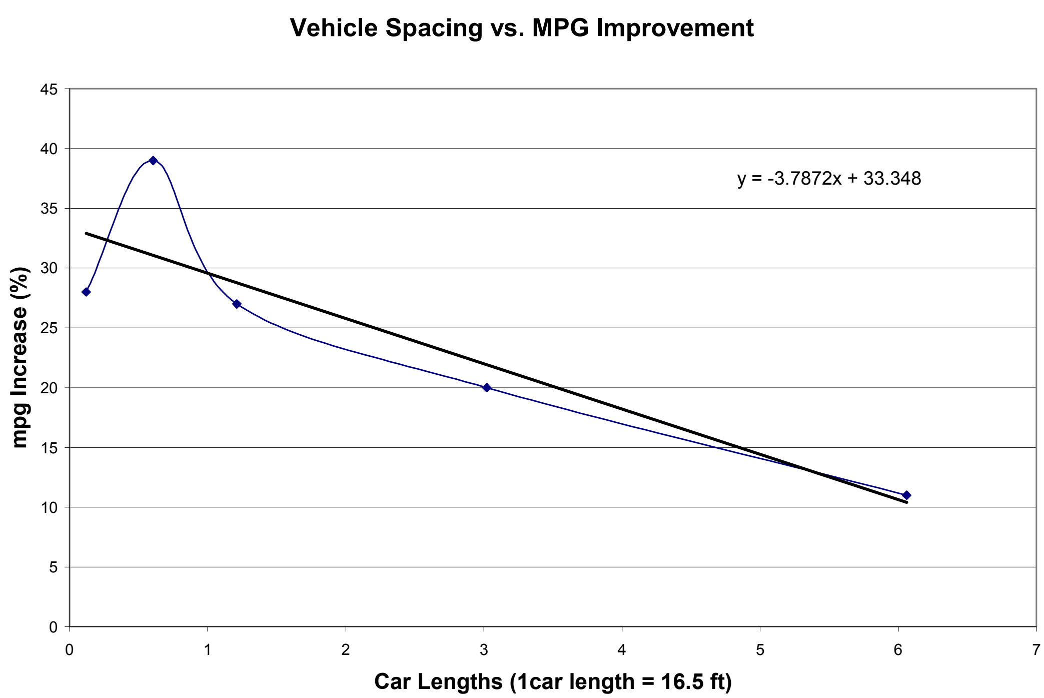

Figure 15: Vehicle Space vs. MPG Improvement

Table 4: Car-Car Results

car	213.818	car area	3	
2 cars	427.636	ND length	1.732050808	

	distance (m)	total force	ND spacing	force/total force
1	0.1	261.882	0.06	61.239
3	0.3	259.805	0.17	60.754
6	0.6	267.945	0.35	62.657
10	1.0	270.710	0.58	63.304
15	1.5	288.385	0.87	67.437
21	2.1	299.498	1.21	70.036
28	2.8	287.402	1.62	67.207
36	3.6	322.058	2.08	75.311
45	4.5	333.474	2.60	77.981
55	5.5	342.175	3.18	80.015
66	6.6	348.171	3.81	81.418
78	7.8	352.859	4.50	82.514
91	9.1	348.450	5.25	81.483
105	10.5	361.847	6.06	84.616
120	12.0	388.257	6.93	90.791
136	13.6	358.987	7.85	83.947
153	15.3	383.952	8.83	89.785
171	17.1	385.130	9.87	90.060
190	19.0	363.949	10.97	85.107
210	21.0	381.158	12.12	89.131
240	24.0	377.369	13.86	88.245
280	28.0	373.715	16.17	87.391
330	33.0	385.039	19.05	90.039
390	39.0	410.613	22.52	96.019
460	46.0	383.971	26.56	89.789
540	54.0	387.607	31.18	90.639
630	63.0	391.778	36.37	91.615
5000	500.0	383.750	288.68	89.738

Table 5: Truck-Car Results

car	213.818	car area	9
truck	947.471	ND length	3
total	1161.289		

	distance (m)	total force	ND spacing	force/total force
1	0.1	947.275	0.03	81.571
3	0.3	930.954	0.10	80.166
6	0.6	948.036	0.20	81.636
10	1.0	934.722	0.33	80.490
15	1.5	948.602	0.50	81.685
21	2.1	955.595	0.70	82.287
28	2.8	955.581	0.93	82.286
36	3.6	949.598	1.20	81.771
45	4.5	988.913	1.50	85.156
55	5.5	962.248	1.83	82.860
66	6.6	1014.900	2.20	87.394
78	7.8	1132.010	2.60	97.479
91	9.1	1047.270	3.03	90.182
105	10.5	1047.570	3.50	90.207
120	12.0	1075.530	4.00	92.615
136	13.6	1068.440	4.53	92.005
153	15.3	1076.030	5.10	92.658
171	17.1	1090.710	5.70	93.922
190	19.0	1071.650	6.33	92.281
210	21.0	1115.230	7.00	96.034
240	24.0	1122.170	8.00	96.631
280	28.0	1132.040	9.33	97.481
330	33.0	1144.600	11.00	98.563
390	39.0	1217.390	13.00	104.831
460	46.0	1174.000	15.33	101.094
540	54.0	1219.250	18.00	104.991
630	63.0	1246.500	21.00	107.338
5000	500.0	1112.470	166.67	95.796

References

1. Alam, Md. Mahbub. Moriya, M. Sakamoto, H. Takai, K. "Fluctuating fluid forces acting on two circular cylinders in a tandem arrangement at a subcritical Reynolds number." Journal of Wind Engineering and Indsutrail Aerodynamics 91 (2003) pp. 139-154.

2. Barth, Matthew. Norbeck, Joseph. Transportation Modeling For The Environment: Final Report. 1996. http://repositories.cdlib.org/its/path/reports/UCB-ITS-PRR-96-6

3. Zhou, Y. Yiu W., M. "Flow structre, momentum and heat transport in a two-tandem-cylinder wake." J. Fluid Mech (2006) vol. 548 pp. 17-48. 2006 Cambridge University Press.

4. Ioannou, Petros A. "Automated Highway Systems". University of Southern California. Pp.273. California: Kluwer Acadmeic/Plenum Publishers 1997.

5. Prof. K. Seshadri. "Wind Tunnel Experiment" Handout. MAE 171A Winter 2007.

6. Fred Browand. Reducing Aerodynamic Drag and Fuel Consumption. Global Climate and Energy Project.

7. MythBusters. "Big Rig Myths." Discovery Channel. 2007