The Greening Brown Team

Aerodynamics of a UPS Delivery Truck

Final Report

December 15, 2008

Sponsored By:
Environmental Protection Agency

In Collaboration With:
Morgan Olson

The Greening Brown Team

Haoyun Fu
Suzanne Lessack
Willie Luckett
Andrew McArthur
Jin Yan

Table of Contents

List of Figures

List of Tables

1.0 Executive Summary

United Parcel Services, UPS, a multi-billion dollar package delivery company, operates a fleet of over 100,000 ground vehicles. Fuel costs for this fleet contribute to over $3 billion in expenditures annually. With the upward pressure on fuel prices over recent years, and the expectation of high prices in the future, UPS is investing in alternative fuel-efficient vehicles. As a part of this endeavor, UPS has partnered with the Environmental Protection Agency, EPA, to investigate several fuel-efficient vehicles. Our sponsor, EPA, has hired us to investigate new aerodynamic designs and modifications and to determine their effect on vehicle drag. As a part of our investigation, our team has partnered with Morgan Olson, an aluminum truck supplier for UPS.

In order to reduce the drag on the baseline P50 truck, two new designs were proposed: a modified version and a completely redesigned truck. The Modified vehicle was created in order to study a truck that would reduce drag, as well as be cost-efficient. This concept features a shallow windshield, a filleted top, and a slanted back. The Redesigned model was designed to reduce the drag on the truck by as much as possible. While the vehicle may not be entirely feasible or inexpensive, it was created to study the drag on a streamlined design.

Computational Fluid Dynamics (CFD) and a 5' x 7' wind tunnel were used to determine the effects the new designs had on drag. The models were tested in CFD in both 2D and 3D at full-scale without wheels. The 3D CFD models were designed to more accurately represent a real truck by filleting the sides of the truck nose and the top of the vehicle. The wind tunnel tests, however, were performed on 1/9th scaled foam models. To reduce imperfections on the surface of the wind tunnel models, the designs were kept very simple and involved no changes in the 3rd dimension.

After investigating these three primary designs, we recommend that our Modified design be used in place of the current P50 UPS truck model. Both of the new designs show improvement in drag values; however, the CFD and wind tunnel results differ on which design offers more improvement. Wind tunnel results show the Modified design has a 30% reduction in drag, compared to 20% for the Redesigned model, whereas, three-dimensional CFD results show a 26% improvement in the Redesigned model, and only 3% for the Modified design. A smoke visualization test performed in the wind tunnel showed a smooth, even flow over the top of the truck, consistent with the CFD 2D results. Since the 3D CFD results do not closely match the wind tunnel results, no numerical data can be reported. However, due to the large size and impracticality of the Redesigned model, the Modified model provides a feasible solution.

Further analysis needs to be done on the Modified body in order to determine a more accurate percentage drag reduction. Also, more precisely manufactured models should be used to complete the wind tunnel testing, as our foam scaled models were imperfect. Since the CFD simulations did not include tires, future CFD analysis should be performed including these features.

2.0 Project Introduction

The industry as a whole has been effected by an increase in expenditures due to escalating fuel costs. Industries dependent on motor vehicles, as well as several government agencies, have expanded their interest in research to improve fuel economy.

2.1 History

As fuel prices remain a national concern, industries reliant on the use of delivery vehicles have become conscience of the need to design more fuel-efficient delivery vehicles. EPA has teamed with UPS in designing more fuel-efficient delivery vehicles. Previous studies have centered on the concepts of hydraulics and engine modifications to improve fuel efficiency. However, recent interest by both EPA and UPS led to an investigation on the effect that aerodynamic modifications can have on fuel-efficiency. Our studies involved the first wind tunnel testing on models of the current UPS delivery vehicle by Morgan Olson.

2.2 Scope

In order to support the high demand for delivery service, UPS operates a global fleet of over 100,000 ground vehicles. In 2007, UPS spent just under $3 billion in fuel, and with continued upward pressure on fuel costs worldwide, this number is expected to increase further. It is evident that expenditures on fuel are a concern, and essential that UPS decrease its fuel costs by improving fuel economy.

2.3 Impact

Managing fuel consumption and greenhouse gas emissions is paramount in the reduction of the effects of human-induced climate change. In accordance with the UPS's ongoing initiative to "implement operational technologies that improve efficiency and reduce miles driven," our focus was on evaluating the drag characteristics and optimizing the aerodynamics of UPS delivery vehicles. Preliminary calculations showed that for a full-sized truck, a reduction in the drag coefficient by 0.01, on average, results in an improvement of the fuel economy by 0.1 miles per gallon. Our findings show a reduction in drag, which results in a decrease in fuel consumption and emissions.

3.0 Project Definition

There has been limited examination into the benefits of vehicular aerodynamic modifications. Along with alternative-fuel, aerodynamics appears to have much promise in increasing fuel efficiency for larger motor vehicles. Thorough aerodynamic research suggests both theoretical and physical examination of motor vehicle bodies.

3.1 Thesis

UPS is investing in several alternative-fuel fleets including hybrid electric, electric, compressed natural gas, liquefied natural gas, and propane. Our investigation, in close collaboration with

EPA, and Morgan Olson, consisted of researching the aerodynamics of the P50 UPS ground-fleet body.

3.2 Investigation

Our team used a two-fold approach in investigating the UPS truck aerodynamics. We used computer software, specifically, CFD, to analyze potential designs for aerodynamic improvements. We initially used Computer Aided Design (CAD) programs to visualize our new designs. Morgan Olson also provided us with their CAD files for the P50 design. We performed CFD analysis on the Modified and Redesigned models, using the Baseline model as a benchmark. The CFD analysis assisted in determining the effectiveness of our design modifications. To validate our design changes and the CFD analysis, we performed wind tunnel testing. The wind tunnel testing consisted of our new designs with variations of our aerodynamic modifications. The baseline Morgan Olson model was once again used as the benchmark. Based on the results we were able to determine the optimal design changes.

4.0 Project Methodology

To determine the effects of the design changes for the Modified and Redesigned models, CFD and wind tunnel analyses were performed. Both 2D and 3D CFD tests were completed to obtain a visual and quantitative understanding of the impact of the alterations. Wind tunnel testing was performed to validate the CFD results. Testing was executed on all three primary model configurations, as well as two other concepts in order to determine the effects of certain aspects of the new designs. Once the analyses were completed for CFD and the wind tunnel testing, the results were compared and a recommendation as to the optimal design was made.

4.1 Computational Fluid Dynamics Simulations

Computational Fluid Dynamics was used to explore and optimize the external aerodynamics of the UPS delivery truck models.

4.1.1 Facilities

CAD software, CATIA, was used to create both 2D and 3D truck models. A CFD software package containing Gambit and Fluent available through the Computer Aided Engineering Network (CAEN) at the University of Michigan was used for all Computational Fluid Dynamic simulations.

4.1.2 Models

CATIA part files of the current P50 UPS truck model and two optimized designs were created for analysis in CFD. The two aerodynamically improved models were intended to reduce the overall drag on the trucks based on our knowledge of aerodynamics and related reference materials.

4.1.3 Procedure

There are two types of drag forces on bluff bodies: friction drag and pressure drag. Computational studies show that about 80% of total drag of a truck is from pressure drag, and the rest is from friction. Therefore, we are more concerned about the pressure drag part of the truck models in CFD simulations. We know that the maximum pressure difference is observed at the back surface of the truck, where complex flow phenomena, such as separation, reattachment and vortices are found. CFD software such as Fluent provided us with visualizations of these flow patterns near the truck models. The visualizations assisted us in understanding the complex physical processes involved, and complement our wind tunnel tests. Both 2D truck profiles and 3D truck models are studied in our project. Results obtained by CFD analyses were one of the major components to drive and verify our final design down-selection.

The whole CFD analysis procedure includes pre-processing, solving and post-processing, which is shown in Figure 4.1. In pre-processing, we used CATIA for CAD modeling and Gambit for model meshing. Fluent was employed for both solving and post-processing stages. Numerical solving and automatic grid adaption were iterative processes since trade-off between accuracy and simulation time needed to be balanced. Visualization and data analyses were the major results based on which we could perfect the aerodynamics of the designs.

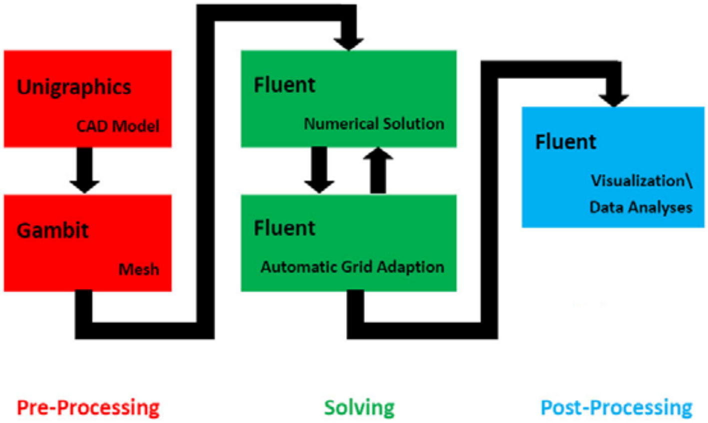


Figure 4.1: Flow Chart of CFD Simulations

4.2 Wind Tunnel Testing

To validate the CFD models of our truck designs, we performed wind tunnel tests and calculated the drag on the truck. We used the 5- by 7-foot wind tunnel, a sting-moment balance, a ground plane, and several scaled models of our design. We used a Computer Numerical Control (CNC) to construct a modular foam model of the three designs. The models were placed in the wind tunnel and three data samples were taken of each. The data was analyzed to determine the drag on each of the trucks, and this analysis was compared to that of the CFD. Based on our analysis, we made a recommendation of the ideal design.

4.2.1 Facilities

In order to physically test our experiment, we used the subsonic, low turbulence 5- by 7-foot wind tunnel located in the Engineering Programs Building (EPB) at the University of Michigan.

We tested at speeds of 60 feet per second (ft/s), 95 ft/s and 120 ft/s. These speeds correspond to driving speeds of 41 miles per hour (mph), 65 mph, and 82 mph, respectively.

4.2.2 Instrumentation

For our experiment, we used a sting-moment balance to simultaneously measure all forces, most important of which was drag. To determine the testing conditions, we used a barometer to measure the ambient pressure and a thermometer to measure the room temperature. We also used a smoke generator in order to visualize the flow.

4.2.3 Truck Models

Five different models were used in the wind tunnel: the Baseline model, the Modified model, the Redesigned model, the Redesigned with Baseline Top model, and the Modified with Redesigned Back model. The designs are shown in Figure 4.2 through Figure 4.6.

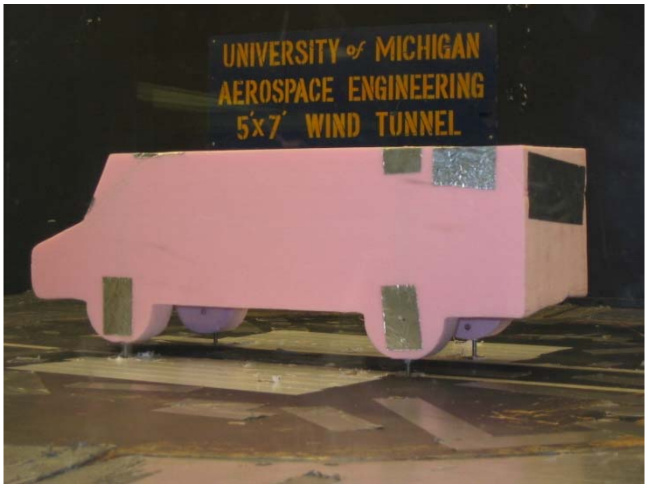


Figure 4.2: Baseline Model

Figure 4.3: Modified Model

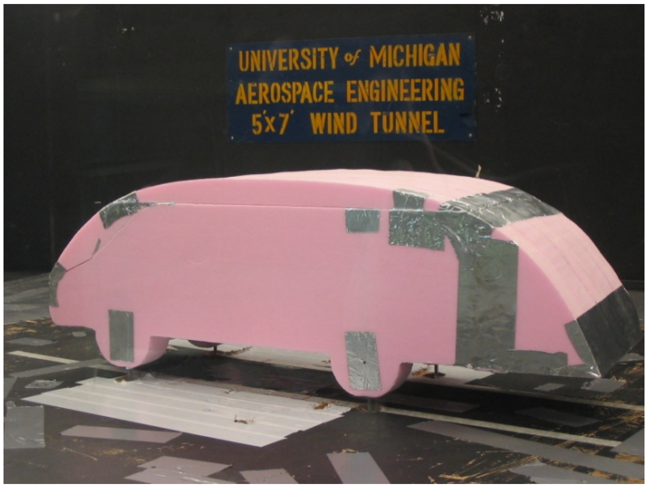


Figure 4.4: Redesigned Model

Figure 4.5: Redesigned with Baseline Top Model

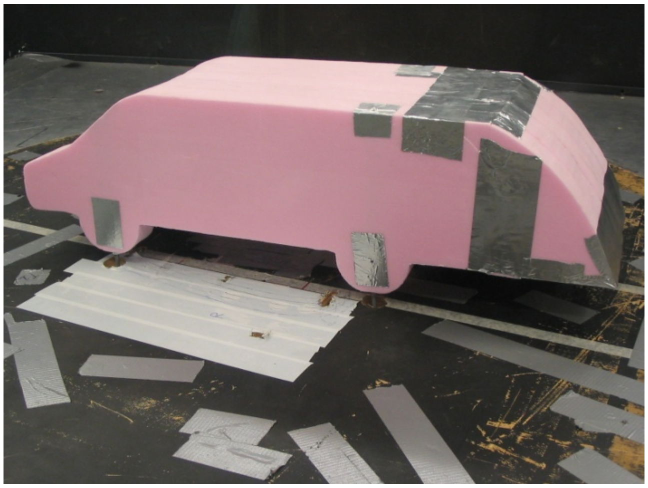


Figure 4.6: Modified with Redesigned Back Model

4.2.4 Non-Dimensional Scaling

Dimensional analysis and similarity are useful tools for analyzing lift and drag forces measured in wind tunnels because they allow full-scale forces to be predicted from tests involving small, inexpensive models.

Non-dimensional scaling was employed in building our test models. For aerodynamic configurations, wind tunnel restrictions require that test must be conducted on a geometrically smaller model under conditions that simultaneously match the Reynolds number, Re. However, it was not possible to match the Reynolds number to the actual values of our UPS truck because Reynolds number, Re, is given by

$$\mathrm{Re}_{\mathrm{actual}} = \frac{(\rho VL)_{\mathrm{actual}}}{\mu_{\mathrm{actual}}} = \mathrm{Re}_{\mathrm{model}} = \frac{(\rho VL)_{\mathrm{model}}}{\mu_{\mathrm{model}}}, \tag{4.1}$$

where ρ is density, L is the length factor, and μ is the viscosity of air.

Thus in order to test at a 1/9th scale, the test speed must be 9 times that of the actual speed. Since there are no wind tunnels at the University of Michigan that can test at these speeds while still being large enough to contain our model, we could not match the Reynolds number. This was not an issue in determining the ideal model, however, since the baseline model was scaled down as well. The Reynolds number of the baseline scaled model and our improved scaled models was the same. Thus the change in drag between the models is still valid, and is not dependent on the Reynolds number.

4.2.5 Construction

To create the truck models, we first created CAD models using CATIA. Morgan Olson provided our team with a CAD drawing of the current truck, which was used as the baseline to create our improved designs. These models were made from foam with a CNC router. Once the pieces were cut, they were glued together.

To avoid having to remount each model in the wind tunnel, a "jigsaw puzzle" approach was used. Since the designs all had the same wheelbase, pieces of the truck were either added or subtracted in order to build a different model. In order to cut out each "puzzle piece," plastic profiles were cut with the laser cutter to use as guides for the hot wire. Once all of the pieces were cut, the entire model was sanded to eliminate any bumps or inconsistencies in the material.

4.2.6 Procedure

The wind tunnel experiment was used to physically validate the CFD drag analysis. The experiment was designed to analyze the drag force component acting on the UPS truck while in operation. Five sets of wind tunnel tests were conducted in order to determine the drag on the Baseline body style, the Modified design, Redesigned concept, the Redesigned with Baseline Top model, and the Modified with Redesigned Back concept. In addition, a smoke test was performed on the Modified design.

The wind tunnel was operated at a free stream speed of 60 ft/s, 95 ft/s and 120 ft/s to simulate the conditions of highway driving. The data acquisition system obtained three different output instances to reduce the measurement error. A smoke generator was used during the last test to assist in visualizing the flow around the Modified test model.

5.0 Computational Fluid Dynamics Analysis

Based on CFD results, we compared the total force exerting on the three truck models, explained the total force differences by looking at the flow pattern around the bodies, and proposed the best designs.

5.1 Two-Dimensional Simulations

Three 2D, full scale, simplified UPS truck models (Figure 5.1) were generated in CATIA for CFD analysis. The Baseline model is the current UPS truck model, the Modified one is cost-effective design and the Redesigned one is our idealized streamline design. For simplicity, we didn't include the wheels when the three models were exported as STEP files for meshing. This is a commonly practiced engineering simplification in almost all initial automobile estimations.

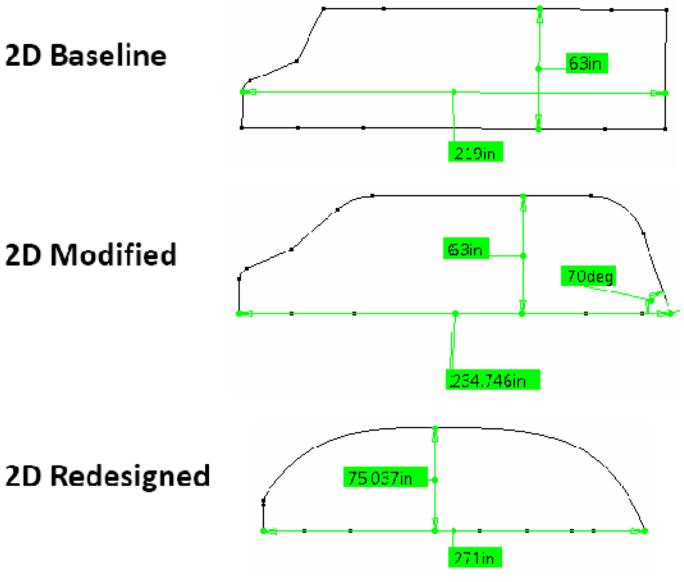


Figure 5.1: 2D Full Scale Truck Models in CATIA

Using Fluent 5/6 as solver in Gambit, we created a 25- by 8-meter (m) rectangular control surface around all the 2D profiles. Next, we meshed the edges of the control surfaces and then the face as a whole. We used a smaller spacing (0.02 m) along the truck profiles than the edges of rectangular control surface (0.2 m) since we are more interested in the flow patterns close to the profiles. Triangle mesh was preferred in 2D meshing since they are generally used for 2D surface meshing. The meshed Baseline model surface is shown in Figure 5.2. Once this was completed, we applied specific boundary conditions to all of the edges. Table 5.1 shows the name and boundary conditions assigned to each edge for the Baseline design. The meshed files were finally exported as MSH files for Fluent simulation.

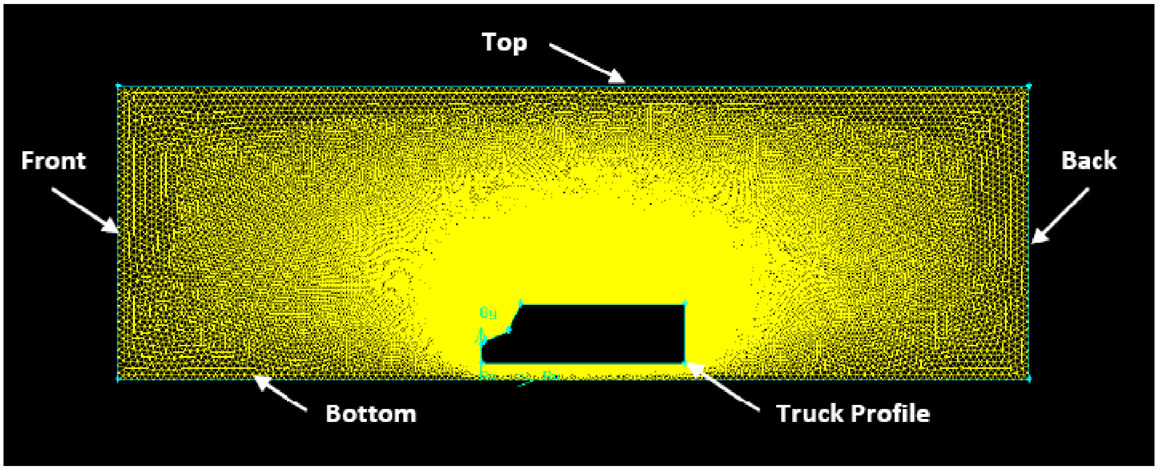


Figure 5.2: Meshing 2D Baseline Truck Model in Gambit

Face	Name	Boundary Condition
Edge of the Truck	Truck Profile	Wall
Bottom Edge	Ground	Wall
Left Edge and Top Edge	Inlet	Velocity Inlet
Right Edge	Outlet	Pressure Outlet

Table 5.1: Boundary Conditions for 2D Simulations in Gambit

Once the file was imported into Fluent, settings were modified from several menus. Table 5.2 lists the settings that were used to perform the simulations.

Window	Settings
Solver Model	Pressure Based, 2D, Steady,
Viscous Model	Standard $k-\varepsilon$ Model
Material	Air with Constant Density
Operating Conditions	101325 Pa
Boundary Conditions	Velocity Inlet: 95 ft/s in x direction
Initialize	Velocity Inlet: 95 ft/s
Residual Monitor	All Residuals=10^{-5}

Table 5.2: 2D Simulation Settings in Fluent

A pressure based solver model was preferred in our case because this solver is applicable to low speeds (95 ft/s) and incompressible flow. For the viscous model, we selected the Standard $k-\varepsilon$ Model since it is valid for fully turbulent flows, which is the case around our truck bodies at highway speeds. This model is a semi-empirical model based on model transport equations for the turbulent kinetic energy (k) and its dissipation rate (ε). Air with constant density was chosen based on the assumption that we could treat ambient air flow as incompressible when the truck is operating on the highway. Air velocity was chosen to be 95 ft/s in the horizontal direction.

After the solution convergence requirement was met, the visualization of flow patterns near the truck models were available for analysis.

Figure 5.3 shows pressure contours on three truck profiles. Maximum pressure occurs in the frontal region of the models and the two improved designs have lower frontal area pressure than the current model.

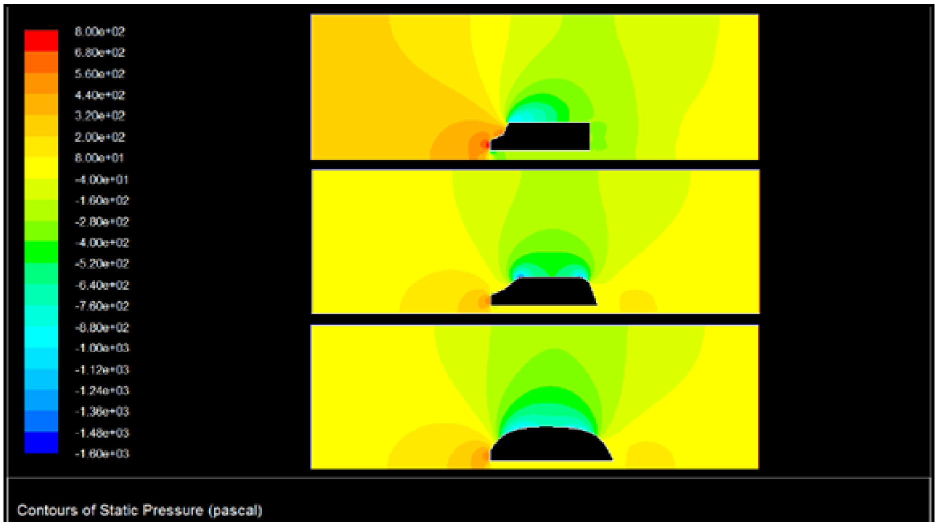


Figure 5.3: 2D Static Pressure Contours

Although the highest pressure takes place on the front surfaces, most of the drag occurs at the back and top of the truck models due to the separation of the flow. Figure 5.4 shows distributions of local circulation for the three models. Circulation caused by flow separation is a major source of aerodynamics drag. For the Baseline model, there are three circulation regions around the truck model compared to only one for the two optimized models. This leads to higher local drags contributed by these regions for the Baseline model.

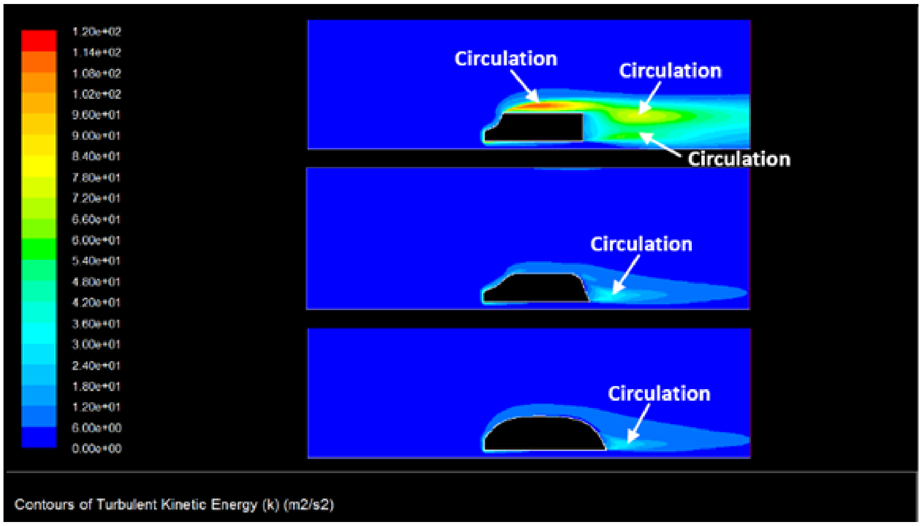


Figure 5.4: 2D Turbulent Kinetic Energy Contours

Finally, we obtained the total drag exerted on the three models. Table 5.3 shows the values of both pressure forces and viscous forces per unit distance perpendicular to the truck profile. From these values, the pressure force per unit distance of the Modified model has 80% less than that of the Baseline model and the Redesigned model has 11% less pressure drag than the Modified model. We concluded from 2D simulations that our Redesigned model is the most aerodynamic model of the three.

	Pressure Unit Force (N/m)	Viscous Unit Force (N/m)	Total Force (N/m)
Baseline	1150	7.7	1160
Modified	210	22.3	230
Redesigned	190	25.7	220

Table 5.3: 2D Drag Forces

5.2 Three-Dimensional Simulations

Similar to the 2D analysis, three 3D full-scale, simplified UPS truck models (Figure 5.5) were generated in CATIA for CFD analysis. The 3D models were essentially the extrusions of the 2D models with fillets and chamfers. Again for simplicity, we eliminated the wheels when the three models were exported as STEP files for meshing.

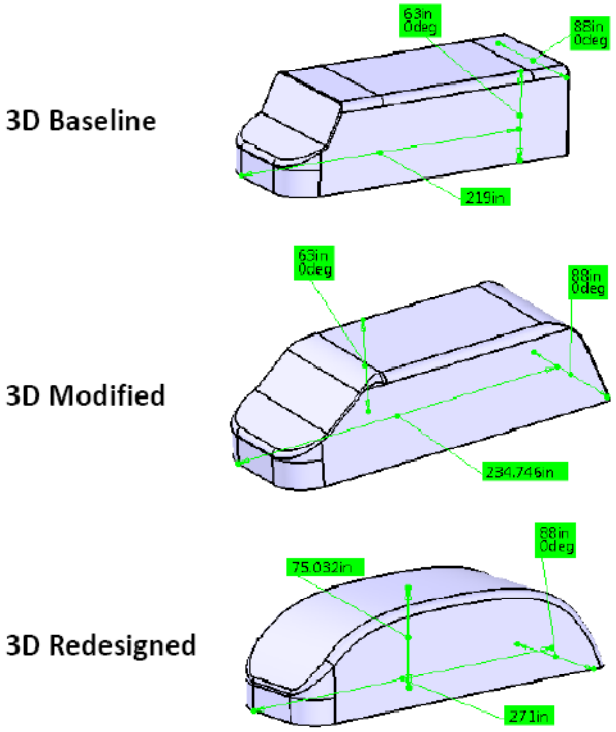


Figure 5.5: 3D Full Scale Truck Models in CATIA

Using Fluent 5/6 as solver in Gambit, we created a 25- by 8- by 10-m control volume around all the 3D profiles. Next, we meshed the faces of the control volumes as well as the truck model surfaces. We then meshed the remaining volume as a whole. A smaller spacing (5 decimeter^2, dm^2) was used along the truck model faces than the faces of the control volume (20dm^2) since we are more interested in the 3D flow patterns close to the models. The spacing used in 3D was relatively large due to constraint of both time and computer memory. Triangular mesh was used for faces, while tetrahedral mesh was used in volume. The meshed Baseline model is shown in Figure 5.6. We applied specific boundary conditions to all of the faces. Table 5.4 shows the name and boundary conditions assigned to each surface of the control volume for the Baseline design. The meshed files were finally exported as MSH file for Fluent simulation.

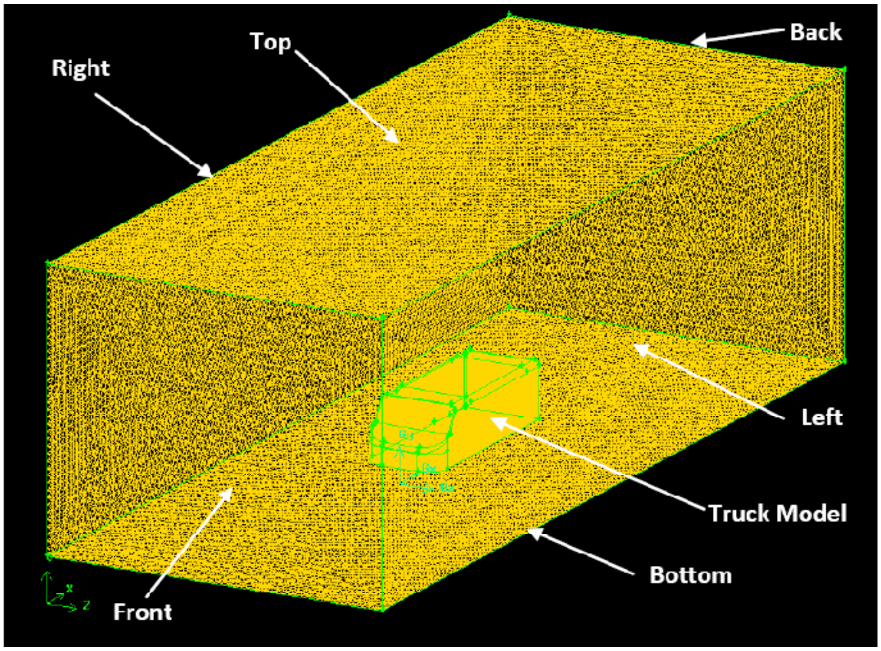


Figure 5.6: Meshing 3-D Baseline Model in Gambit

Face	Name	Boundary Condition
Faces of Truck Model	Truck Surfaces	Wall
Bottom Face	Ground	Wall
Top, Left, Right and Front Faces	Inlet	Velocity Inlet
Back Face	Outlet	Pressure Outlet

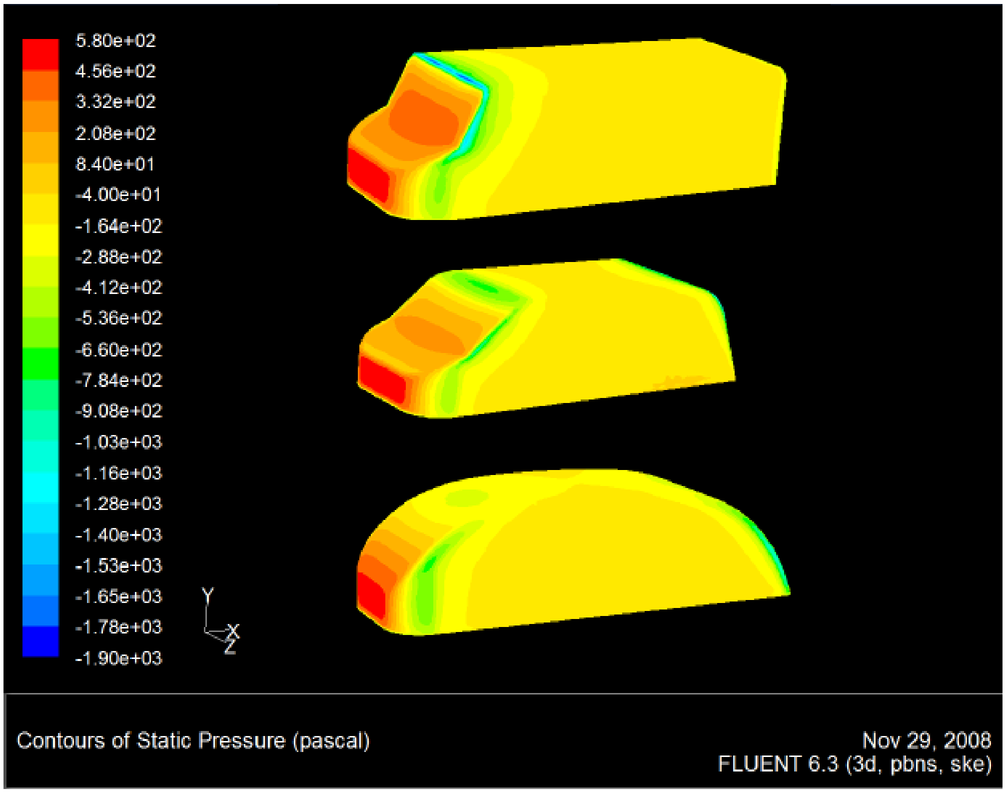
Table 5.4: Boundary Conditions for 3D Simulations in Gambit

Table 5.5 lists the Fluent settings used to perform 3D simulations, which are similar to the 2D simulation settings except one minor change from 2D to 3D in solver model. Air velocity was again chosen to be 95 ft/s along horizontal direction.

Window	Settings
Solver Model	Pressure Based, 3D, Steady,
Viscous Model	Standard $k-\varepsilon$ Model
Material	Air with Constant Density
Operating Conditions	101325 Pa
Boundary Conditions	Velocity Inlet: 95 ft/s in x direction
Initialize	Velocity Inlet: 95 ft/s
Residual Monitor	All Residuals=10^{-5}

Table 5.5: 3D Simulation Settings in Fluent

After the solution convergence requirement was met, visualization of 3D flow patterns near the truck models was available for analysis. Figure 5.7 indicates the static pressure distributions on the three truck models. The frontal pressure of the Baseline model has a larger high-pressure region than the other two, which means the pressure force generated on the front surface of the Baseline model is the largest.



Contours of Static Pressure (pascal)

Nov 29, 2008
FLUENT 6.3 (3d, pbns, ske)

Figure 5.7: 3D Static Pressure Contours

Figure 5.8 shows path lines along the truck bodies of three designs. At the rear of the truck bodies, the Baseline model has a larger separation area than the other two models. These separation areas are the major source of aerodynamic drag in the rear region of the trucks.

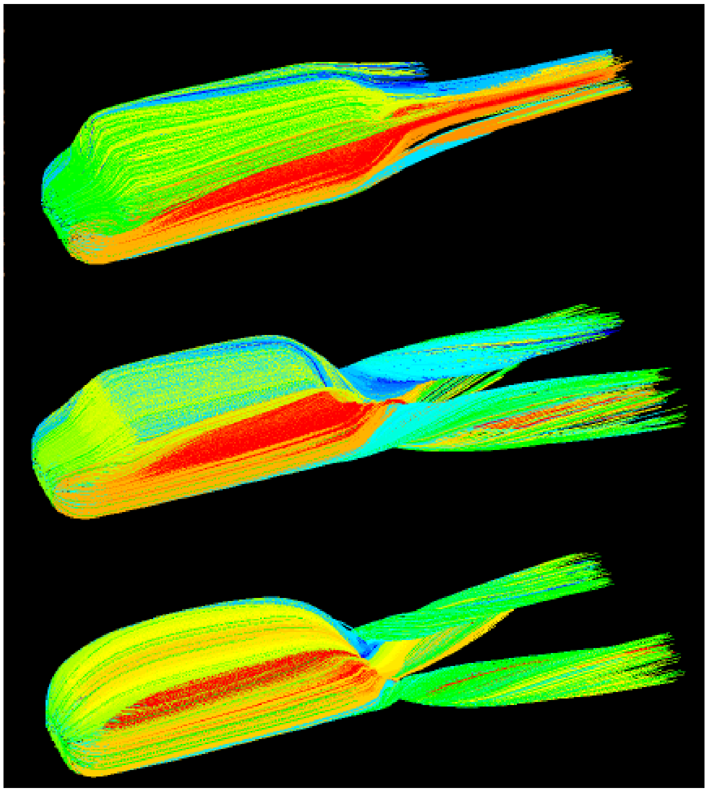


Figure 5.8: 3D Flow Path Line around Truck Bodies

Pressure and total forces exerted on the three models and their corresponding drag coefficients are presented in Table 5.6. Based on our data, the Modified design is 2.9% more aerodynamic efficient than Baseline design while Redesigned is 26% more efficient than the Baseline design. The total force differences between the Modified and Redesigned models are not as great as in 2D due to 3D effects including fillets and chamfers on the models. The aerodynamic forces in 3D are caused by various viscous flow phenomena, such as the 3D turbulent boundary layers on the body surfaces, longitudinal vortices induced by 3D separations, recirculating flows caused by separations, and the ground-plane boundary layers. These phenomena can interact with each other, which makes the aerodynamic drag components even more complex. Based on 3D simulation results, we concluded that Redesigned model is again the most aerodynamically efficient based on total force comparison, followed by Modified and Baseline designs. This conclusion is consistent with the 2D conclusions made earlier.

	Pressure Force (N)	Viscous Force (N)	Total Force (N)	Drag Coefficient
Baseline	1510	55.4	1560	0.80
Modified	1450	63.2	1520	0.78
Redesigned	1300	80.6	1380	0.60

Table 5.6: 3D Drag Forces

In both simulations, vehicle tires were removed, which is a commonly practiced engineering simplification in initial estimation. However, computational studies show that tires of a vehicle would contribute at least 10% of its overall aerodynamic drag, which is significant. Therefore our future work on CFD would focus on studying the effect of tires as well as undercarriage on the overall drag of our truck models.

5.3 Recommendations

From both 2D and 3D CFD conclusions, Redesigned model has the smallest drag and shows significant improvement over the Baseline one. Therefore we recommend that this model be the best option for the purpose of drag reduction.

6.0 Wind Tunnel Testing Analysis

The wind tunnel testing was performed on five different models at various speeds. Data analysis showed an improvement in drag for all of the models. The Modified design (Figure 4.3), however, showed the most drastic improvement with a drag savings of 30%. Thus the modified design is the best option to reduce drag and overall cost.

6.1 Data Analysis

By testing various models on the balance in the 5- by 7-ft wind tunnel, we were able to directly obtain the drag on each of the models. Since the drag is dependent on velocity and the models were tested at several wind speeds, the coefficient of drag could be determined. This method allows each model variation to be easily compared. Equation 6.1 was used to find the coefficient of drag, C_d. In this equation, D is the drag, ρ is the air density, A is the cross-sectional area of the model, and V is the velocity of the air.

$$C_d = \frac{2D}{\rho A V^2} \tag{6.1}$$

The air density (Equation 6.2) was determined from the temperature, T, the pressure, p, and the specific gas constant, R.

$$\rho = \frac{p}{RT} \tag{6.2}$$

For each model, the average coefficient of drag was calculated and the results are compiled in Table 6.1. Detailed tables of the wind tunnel data and analysis can be found in Appendix A.

	Coefficient of Drag	% Drag Savings
Baseline	0.19	N/A
Modified	0.13	30%
Redesigned	0.15	20%
Redesigned with Baseline Top	0.15	20%
Modified with Redesigned Back	0.14	25%

Table 6.1: Coefficient of Drag Results for Wind Tunnel Models

In addition to numerical analysis, a smoke test was performed at various speeds. The Modified model was used for this demonstration, as shown in Figure 6.1.

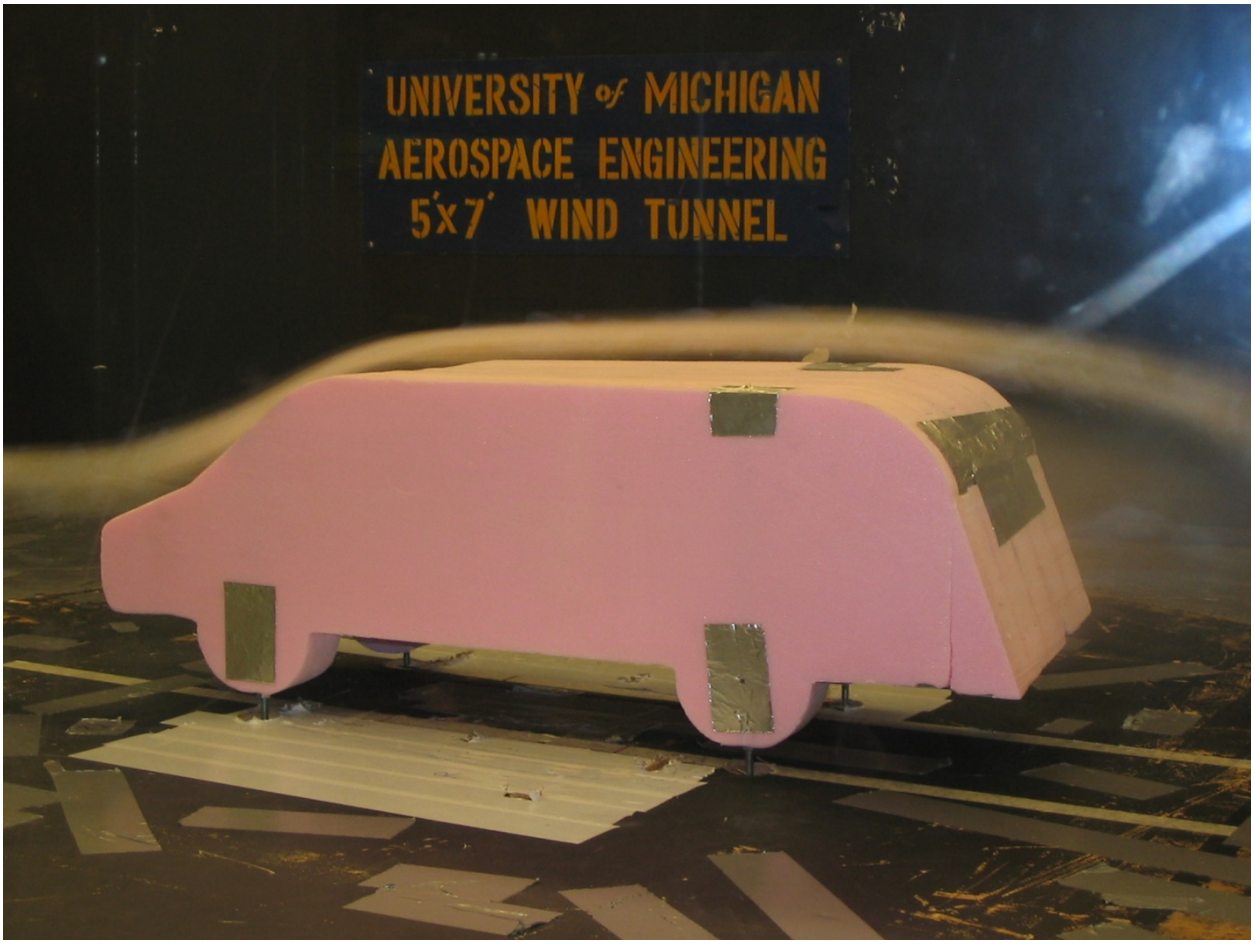


Figure 6.1: Smoke Test with Modified Model

From this image, we can see that the air flow is smooth over the front and top of the model, and over the back. The baseline truck is prone to turbulence behind the truck due to the abruptness of the back end. Thus the Modified concept eliminated this turbulent area as seen from the wind tunnel smoke test.

6.1.1 Component Effects

By measuring the drag of models based on variations of the three main designs, we were able to isolate the effect of certain changes on the truck. For instance, the "Redesigned with Baseline Top" model allowed us to determine the effects of the rounded top on the drag by comparing it with the Redesigned model. Since the coefficient of drag was the same for these models, there is no advantage of using a rounded top versus the flat baseline top.

Likewise, the "Modified with Redesigned Back" model can be compared with the Modified model to see the effects of the Redesigned rear piece. Since the Modified model showed a 30%

drag savings and the “Modified with Redesigned Back” model showed a drag savings of 25%, the extended back had a negative effect on the drag. Thus the extended back should not be used.

6.1.2 Future Work

While these model modifications provided more insight into the effect of a few of the design changes, more testing should be done to determine the effect of the other alterations. We do not have enough data to single out the effects the Modified and Redesigned windshield, the Modified rear fillet, and the Modified back had on the truck. Thus the Baseline model should be tested with the Modified back, with the rear fillet and again with just the Modified windshield. These additional tests would provide sufficient information to determine the effect of all of the components on the drag.

In addition, more testing should be performed on the Modified model to determine the optimal angle for the back piece. Since the slant angle was chosen arbitrarily (10 degrees from vertical), there is room for improvement. The back end should be tested at angles varying from 0 degrees to 45 degrees in order optimize the design, while remaining feasible.

6.2 Recommendations

Based on our wind tunnel analysis, we recommend that the Modified design be used in place of the current P50 model. This new design resulted in a 30% improvement in drag from the Baseline model, which will increase fuel efficiency, thus reducing the overall cost per vehicle. Since a reduction in the drag coefficient for a full-sized truck of 0.01 results in an improvement of the fuel economy by 0.1 miles per gallon, the Modified design will result in 0.6 miles per gallon savings. An initial estimate of the cost savings is 30%, which corresponds to an annual savings of approximately $900 million. This is based on the assumption that the drag savings are equal to the overall cost reduction.

Should the Modified design be implemented, UPS may be able to use a lighter engine on their trucks since they will not have to provide as much power. Thus a decrease in drag on the vehicle not only affects the cost in terms of the fuel efficiency, but it can dramatically affect the cost via the weight and engine choice as well. Since there are so many variables in the vehicle production, further study will need to be conducted to more precisely determine the impact of a 30% drag reduction on cost.

Furthermore, the Modified design was selected for its feasibility. While the other designs significantly reduced the drag as well, none of them were realistic. The Redesigned model was too long to be easily maneuverable, and would not be capable of parking to make deliveries. In addition, there would have been a lot of wasted space inside the truck since the roof was curved and the packages are typically rectangular. The other designs tested would have similar problems with the Redesigned truck.

The Modified design, however, is very feasible. It is approximately the same size as the current P50 model, and is similar to postal trucks. While the slanted back imparts slightly more complexity, the cargo door can still operate in the same fashion, and the loading and unloading process can remain the same.

The Modified design is recommended for further investigation because of the potential for cost savings and because of its feasibility.

7.0 Project Schedule

The time to complete this project was 88 days. We began gathering information on September 18, 2008 and completed the Final Report on December 15, 2008. The key dates were as follows:

- November 2, 2008: Completed all CAD models
- November 8, 2008: 2D CFD calculations completed
- November 12, 2008: Wind tunnel models finished
- November 13, 2008: Wind tunnel testing
- November 16, 2008: 3D CFD calculations completed
- November 21, 2008: Wind tunnel analysis completed
- November 21, 2008: CFD analysis completed
- November 24, 2008: Project results compilation
- December 9, 2008: Poster Session
- December 15, 2008: Final Report

A more detailed schedule including reports and presentations can be found in Appendix B.

8.0 Project Cost

Labor was the most significant cost associated with the investigation described by the proposal and carried out by our team. The labor costs were attributed to two sources – engineering work completed by our team, and outsourced work completed by others. Over the project span of 12 weeks, we accrued 383 hours of engineering work by our team. This compares favorably to the 480 hours originally budgeted in our proposal. This results in a cost savings of approximately $4,000. Additionally, our project required significantly less consultation time, only 2 hours compared to the 12 hours budgeted. Machinist and technician time was accurately estimated at 10 hours each. In total, our labor cost was $16,225 versus the original estimate of $21,700.

The second component to our project cost was the facilities and materials costs. For our wind tunnel testing, we used the more expensive 5- by 7 foot wind tunnel versus the 2- by 2-foot. The smaller wind tunnel would have cost $100/hour but would have required 10 hours of testing time resulting in $1,000. The larger wind tunnel is charged on a per day basis, at $800 per day. Our team was able to complete our testing in one day, providing $200 of savings. The manufacturing of our wind tunnel models was significantly cheaper because our team used the CNC router and foam, versus the polystyrene plastic sheets originally planned. In total, our models cost $174 in materials and manufacturing costs compared to $600. Finally, some costs associated with the publication of the final report and the poster presentation, totaling $144, was required. Facilities and materials costs totaled $1,118 versus the proposal estimate of $1,714.

The subtotal of all these costs is $17,343. Our team is subject to a 55% overhead, resulting in a grand total of nearly $26,900. Our proposal estimated a grand total, including a 15% overhead, of $41,700. Our project was completed under budget, with nearly 36% of cost savings. Our finalized project budget is given as Appendix C.

9.0 Project Conclusions and Future Work

After comparing the CFD and wind tunnel analyses, a recommendation of the ideal design for the current model was made. Since the percentage drag reduction on the models is inconclusive, future research will need to be conducted.

9.1 Summary

The Greening Brown Team recommends the Modified design be used in place of the current P50 UPS truck. After analyzing the Computational Fluid Dynamics and wind tunnel results, we concluded that both the Modified design and Redesigned model succeeded in reducing drag. The 3D CFD analysis, however, shows that the Redesigned model reduces drag by 26% from the Baseline model. Theoretical analysis alone is insufficient to characterize the aerodynamic performance because it is difficult to simulate the complicated conditions of the truck. In addition, the vehicle wheels were removed in our CFD simulations, which would give a smaller aerodynamic drag than expected.

On the other hand, the wind tunnel experimental results show that the Modified design is the most efficient in drag reduction as the Modified model has a 30% drag savings from the Baseline model. Although quantitative analysis determines the drag savings, qualitative analysis determines the feasibility of the design. Since UPS requires a truck that meets their delivery specifications, a new design should be practical, feasible, and economical. The Redesigned model will increase the cost for manufacturing because of its large body, and will also cause inconvenience in parking and loading or unloading goods. The Modified design, however, is very feasible and is approximately the same size as the current UPS P50 model.

9.2 Future Work

In the future, CFD analysis should focus on studying the effect of the whole UPS truck (including wheels). In addition more advanced numerical methods should be used to improve the calculation speed when employing more meshes to obtain the aerodynamic analysis of the truck.

Further wind tunnel studies need to be conducted to determine the effect each component has on the overall drag. Specifically, more research must be done on the windshield, the rear fillet, and the back of the truck.

Finally, we recommend that further experiments be conducted on both the wind tunnel and CFD tests. Since no precise quantitative data was obtained from these tests, additional research may provide more useful data. This will result in more information of the Modified design, which will assist in further improvement of the Modified design.

Appendix A: Wind Tunnel Raw Data

Baseline Design Data and Analysis:

Drag (lbf)	Drag (N)	Velocity (ft/s)	Velocity (m/s)	Temperature (F)	Temperature (K)	Density (kg/m^3)	Coefficient of Drag
2.27	10.10	59.80	18.23	56.50	286.8	1.19	0.18
2.27	10.09	58.10	17.71	56.50	286.8	1.19	0.19
2.27	10.08	60.90	18.56	56.50	286.8	1.19	0.18
5.86	26.08	95.40	29.08	57.60	287.4	1.19	0.19
5.84	25.97	96.00	29.26	57.60	287.4	1.19	0.18
5.86	26.08	94.60	28.83	57.60	287.4	1.19	0.19
8.75	38.91	117.80	35.91	57.50	287.3	1.19	0.18
8.74	38.87	118.50	36.12	57.60	287.4	1.19	0.18
8.74	38.89	117.90	35.94	57.60	287.4	1.19	0.18
8.69	38.67	117.10	35.69	57.60	287.4	1.19	0.18
8.68	38.60	117.10	35.69	57.60	287.4	1.19	0.18
8.71	38.72	117.90	35.94	58.60	287.9	1.18	0.18
5.82	25.88	94.40	28.77	58.60	287.9	1.18	0.19
5.81	25.83	95.10	28.99	58.70	288.0	1.18	0.19
5.81	25.83	94.50	28.80	58.60	287.9	1.18	0.19
2.28	10.12	57.60	17.56	57.50	287.3	1.19	0.20
2.25	10.01	58.10	17.71	57.50	287.3	1.19	0.19
2.23	9.92	57.50	17.53	57.50	287.3	1.19	0.19

Average Coefficient of Drag 0.19

Modified Design Data and Analysis:

Drag (lbf)	Drag (N)	Velocity (ft/s)	Velocity (m/s)	Temperature (F)	Temperature (K)	Density (kg/m^3)	Coefficient of Drag
1.62	7.19	57.90	17.65	56.50	286.8	1.19	0.14
1.62	7.19	58.70	17.89	56.50	286.8	1.19	0.14
1.61	7.17	57.90	17.65	56.50	286.8	1.19	0.14
4.14	18.43	95.70	29.17	56.60	286.8	1.19	0.13
4.13	18.35	95.70	29.17	56.50	286.8	1.19	0.13
4.12	18.34	95.70	29.17	57.50	287.3	1.19	0.13
6.18	27.50	117.90	35.94	57.50	287.3	1.19	0.13
6.13	27.28	117.90	35.94	57.50	287.3	1.19	0.13
6.13	27.27	117.30	35.75	57.60	287.4	1.19	0.13
6.07	27.01	117.60	35.84	57.50	287.3	1.19	0.13
6.07	27.00	117.00	35.66	57.60	287.4	1.19	0.13
6.10	27.14	117.70	35.87	57.60	287.4	1.19	0.13
4.08	18.15	96.20	29.32	57.50	287.3	1.19	0.13
4.10	18.22	95.80	29.20	57.50	287.3	1.19	0.13
4.11	18.26	95.90	29.23	57.50	287.3	1.19	0.13
1.55	6.87	59.20	18.04	57.50	287.3	1.19	0.13
1.56	6.93	57.90	17.65	57.50	287.3	1.19	0.13
1.56	6.92	57.90	17.65	57.50	287.3	1.19	0.13

Average Coefficient of Drag 0.13

Redesigned Design Data and Analysis:

Drag (lbf)	Drag (N)	Velocity (ft/s)	Velocity (m/s)	Temperature (F)	Temperature (K)	Density (kg/m^3)	Coefficient of Drag
1.92	8.54	59.40	18.11	56.50	286.8	1.19	0.15
1.91	8.51	59.50	18.14	56.50	286.8	1.19	0.15
1.90	8.46	58.10	17.71	56.50	286.8	1.19	0.15
4.94	21.98	94.90	28.93	57.50	287.3	1.19	0.15
4.97	22.09	95.10	28.99	57.60	287.4	1.19	0.15
4.94	21.99	96.00	29.26	57.60	287.4	1.19	0.15
7.36	32.74	117.10	35.69	57.50	287.3	1.19	0.15
7.35	32.69	117.80	35.91	57.50	287.3	1.19	0.15
7.35	32.68	117.20	35.72	57.50	287.3	1.19	0.15
7.32	32.57	117.30	35.75	58.60	287.9	1.18	0.15
7.35	32.68	118.10	36.00	58.70	288.0	1.18	0.14
7.35	32.70	117.20	35.72	58.60	287.9	1.18	0.15
4.89	21.75	95.50	29.11	58.60	287.9	1.18	0.15
4.90	21.79	95.90	29.23	58.60	287.9	1.18	0.15
4.90	21.80	94.30	28.74	58.60	287.9	1.18	0.15
1.92	8.55	59.60	18.17	58.70	288.0	1.18	0.15
1.94	8.64	60.40	18.41	58.70	288.0	1.18	0.15
1.96	8.72	58.80	17.92	57.70	287.4	1.19	0.16

Redesigned with Baseline Top Design Data and Analysis:

Drag (lbf)	Drag (N)	Velocity (ft/s)	Velocity (m/s)	Temperature (F)	Temperature (K)	Density (kg/m^3)	Coefficient of Drag
1.81	8.07	58.30	17.77	56.50	286.8	1.19	0.15
1.80	8.02	58.40	17.80	56.50	286.8	1.19	0.15
1.80	8.01	58.10	17.71	56.50	286.8	1.19	0.15
4.78	21.24	96.50	29.41	57.60	287.4	1.19	0.15
4.84	21.53	96.40	29.38	57.60	287.4	1.19	0.15
4.81	21.38	96.40	29.38	57.60	287.4	1.19	0.15
7.03	31.29	117.10	35.69	58.60	287.9	1.18	0.15
7.01	31.19	117.20	35.72	58.70	288.0	1.18	0.15
6.96	30.97	117.30	35.75	58.70	288.0	1.18	0.15
7.04	31.30	117.50	35.81	58.70	288.0	1.18	0.15
7.00	31.15	117.50	35.81	58.70	288.0	1.18	0.15
6.97	31.00	117.70	35.87	58.70	288.0	1.18	0.15
7.00	31.14	117.00	35.66	58.70	288.0	1.18	0.15
4.69	20.85	95.20	29.02	58.60	287.9	1.18	0.15
4.64	20.64	95.00	28.96	58.60	287.9	1.18	0.15
4.63	20.57	95.20	29.02	58.60	287.9	1.18	0.15
1.75	7.79	60.30	18.38	58.60	287.9	1.18	0.14
1.75	7.79	58.10	17.71	58.60	287.9	1.18	0.15
1.72	7.67	57.40	17.50	58.60	287.9	1.18	0.15

Average Coefficient of Drag 0.15

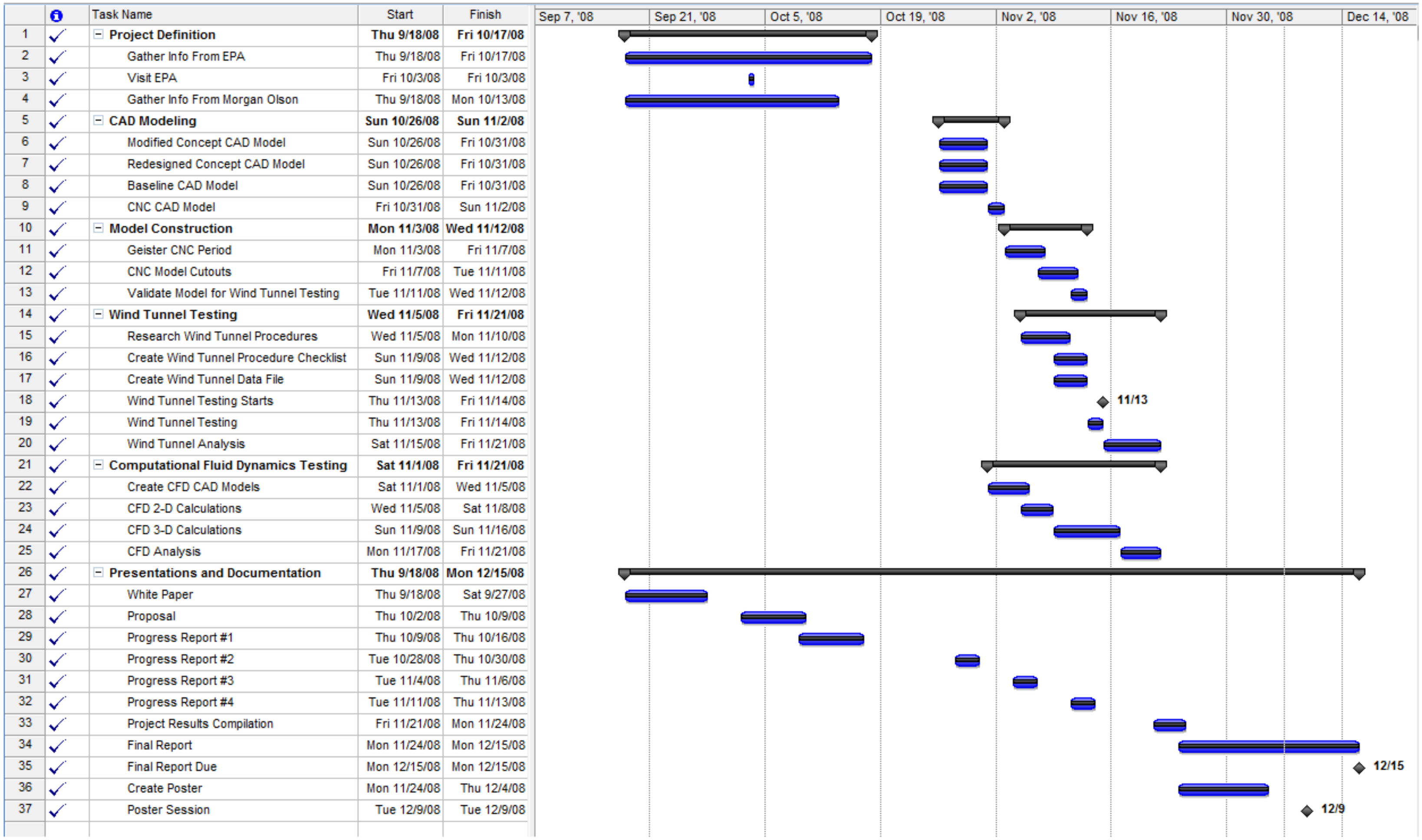
Modified with Redesigned Back Design Data and Analysis:

Drag (lbf)	Drag (N)	Velocity (ft/s)	Velocity (m/s)	Temperature (F)	Temperature (K)	Density (kg/m^3)	Coefficient of Drag
1.70	7.54	59.20	18.04	57.60	287.4	1.19	0.14
1.66	7.40	56.60	17.25	57.60	287.4	1.19	0.15
1.66	7.40	59.20	18.04	57.60	287.4	1.19	0.14
4.46	19.84	95.30	29.05	57.50	287.3	1.19	0.14
4.47	19.87	95.40	29.08	57.50	287.3	1.19	0.14
4.43	19.68	95.30	29.05	57.50	287.3	1.19	0.14
6.61	29.39	116.70	35.57	58.60	287.9	1.18	0.14
6.70	29.79	117.70	35.87	58.70	288.0	1.18	0.14
6.59	29.31	117.40	35.78	58.60	287.9	1.18	0.14
6.51	28.96	117.50	35.81	58.70	288.0	1.18	0.14
6.60	29.37	117.30	35.75	58.60	287.9	1.18	0.14
6.61	29.38	117.30	35.75	58.60	287.9	1.18	0.14
4.40	19.55	94.90	28.93	58.70	288.0	1.18	0.14
4.42	19.66	95.60	29.14	58.60	287.9	1.18	0.14
4.41	19.61	95.50	29.11	58.60	287.9	1.18	0.14
1.69	7.53	60.00	18.29	58.60	287.9	1.18	0.14
1.69	7.52	57.90	17.65	58.60	287.9	1.18	0.15
1.67	7.41	58.40	17.80	58.60	287.9	1.18	0.14
					Average Coefficient of Drag		0.14

Wind Tunnel Analysis:

	Coefficient of Drag	% Drag Savings
Baseline	0.19	N/A
Modified	0.13	0.30
Redesigned	0.15	0.20
Redesigned with Baseline Top	0.15	0.20
Modified with Redesigned Back	0.14	0.25

Appendix B: Greening Brown Team Project Schedule



	ℹ	Task Name	Start	Finish
1	✓	⊟ **Project Definition**	**Thu 9/18/08**	**Fri 10/17/08**
2	✓	Gather Info From EPA	Thu 9/18/08	Fri 10/17/08
3	✓	Visit EPA	Fri 10/3/08	Fri 10/3/08
4	✓	Gather Info From Morgan Olson	Thu 9/18/08	Mon 10/13/08
5	✓	⊟ **CAD Modeling**	**Sun 10/26/08**	**Sun 11/2/08**
6	✓	Modified Concept CAD Model	Sun 10/26/08	Fri 10/31/08
7	✓	Redesigned Concept CAD Model	Sun 10/26/08	Fri 10/31/08
8	✓	Baseline CAD Model	Sun 10/26/08	Fri 10/31/08
9	✓	CNC CAD Model	Fri 10/31/08	Sun 11/2/08
10	✓	⊟ **Model Construction**	**Mon 11/3/08**	**Wed 11/12/08**
11	✓	Geister CNC Period	Mon 11/3/08	Fri 11/7/08
12	✓	CNC Model Cutouts	Fri 11/7/08	Tue 11/11/08
13	✓	Validate Model for Wind Tunnel Testing	Tue 11/11/08	Wed 11/12/08
14	✓	⊟ **Wind Tunnel Testing**	**Wed 11/5/08**	**Fri 11/21/08**
15	✓	Research Wind Tunnel Procedures	Wed 11/5/08	Mon 11/10/08
16	✓	Create Wind Tunnel Procedure Checklist	Sun 11/9/08	Wed 11/12/08
17	✓	Create Wind Tunnel Data File	Sun 11/9/08	Wed 11/12/08
18	✓	Wind Tunnel Testing Starts	Thu 11/13/08	Fri 11/14/08
19	✓	Wind Tunnel Testing	Thu 11/13/08	Fri 11/14/08
20	✓	Wind Tunnel Analysis	Sat 11/15/08	Fri 11/21/08
21	✓	⊟ **Computational Fluid Dynamics Testing**	**Sat 11/1/08**	**Fri 11/21/08**
22	✓	Create CFD CAD Models	Sat 11/1/08	Wed 11/5/08
23	✓	CFD 2-D Calculations	Wed 11/5/08	Sat 11/8/08
24	✓	CFD 3-D Calculations	Sun 11/9/08	Sun 11/16/08
25	✓	CFD Analysis	Mon 11/17/08	Fri 11/21/08
26	✓	⊟ **Presentations and Documentation**	**Thu 9/18/08**	**Mon 12/15/08**
27	✓	White Paper	Thu 9/18/08	Sat 9/27/08
28	✓	Proposal	Thu 10/2/08	Thu 10/9/08
29	✓	Progress Report #1	Thu 10/9/08	Thu 10/16/08
30	✓	Progress Report #2	Tue 10/28/08	Thu 10/30/08
31	✓	Progress Report #3	Tue 11/4/08	Thu 11/6/08
32	✓	Progress Report #4	Tue 11/11/08	Thu 11/13/08
33	✓	Project Results Compilation	Fri 11/21/08	Mon 11/24/08
34	✓	Final Report	Mon 11/24/08	Mon 12/15/08
35	✓	Final Report Due	Mon 12/15/08	Mon 12/15/08
36	✓	Create Poster	Mon 11/24/08	Thu 12/4/08
37	✓	Poster Session	Tue 12/9/08	Tue 12/9/08

Appendix C: Project Cost Table

Category	Item	Unit Cost	Unit	Cost
Labor	Engineer	$40/hour	383	$15,320.00
	Consultant	$150/hour	3	$ 450.00
	Machinist	$35/hour	3	$ 105.00
	Technician	$35/hour	10	$ 350.00
Facilities	Wind Tunnel	$800/day	1	$ 800.00
	CNC Router	$60/hour	2	$ 120.00
Materials	Foam Material	$7.50/sheet	5	$ 37.50
	Toothpicks	$2.50/package	1	$ 2.50
	Duct Tape	$7/roll	2	$ 14.00
	Final Report	$8/copy	3	$ 24.00
	Poster Presentation	$60/poster	2	$ 120.00
Subtotal				$17,343.00
Overhead			55%	$ 9,538.65
GRAND TOTAL				$26,881.65