Aerodynamic Optimization of a Solar – Bio-Diesel Hybrid Vehicle

Neal A. Allgood*
Auburn University, Auburn, AL, 36830

Energy efficiency is the most critical aspect for a successful solar powered automobile and much can be gained from the reduction of aerodynamic drag on such a vehicle. Yet, for a solar car to be practical to the everyday driver, it has to be ergonomically feasible, financially sensible, and aesthetically pleasing. This research compares aerodynamic drag calculations produced by both computational fluid dynamics (CFD) and classical wind tunnel tests for a solar – bio-diesel hybrid concept car. CFD was used to develop a 2-dimensional shape with minimal drag characteristics yet still contained the qualities required to be practical. Many sections, such as the underside of the rear end of the car, were targeted as potential areas of flow separation, and much work was done to minimize this source of drag. From these findings a 3-dimensional model was created for testing in a sub-sonic wind tunnel to estimate a more accurate drag coefficient for use in power consumption and range calculations. These tests prove that the designed vehicle is aerodynamically sound with drag coefficients 12.8% less than that of the most aerodynamic hybrid vehicle currently in use. They are also a testament to the power of CFD as a tool for vehicle development and design. The solar-bio-diesel hybrid vehicle modeled from this research is currently under construction and will participate in the North American Solar Car challenge in July of 2008.

Nomenclature

C_d	=	drag coefficient
V_∞	=	free stream velocity
K	=	turbulent kinetic energy
ω	=	specific dissipation
ρ	=	density
μ	=	dynamic viscosity

I. Introduction

The SOL of Auburn has been Auburn University's solar car team since 1990 and has built a total of five solar race cars since the project's inception. Each car has focused on minimizing the effects of aerodynamic drag on the vehicle, sacrificing practicality in an effort to do so. Fig. 1 is an example of a past solar car built by Auburn University students.

Previous versions of Auburn's solar cars typically feature highly streamlined bodies, optimizing aerodynamics while integrating solar arrays and chassis enclosure. To further streamline the car, fairings house the wheels and suspension components and the trailing edge is tapered to a point to minimize air flow separation. This method was standard for typical solar race vehicles; it optimized both the performance and aerodynamic aspects, and met a perfect balance between vehicle components and aerodynamic configurations. Yet practicality was sacrificed as the driver was required to lie down in order to steer, and the lack of standard automotive doors made ingress and egress particularly difficult.

In an effort to create a more practical car for the everyday driver, the team developed **TIGER** (**T**ransporter **I**ntegrating **G**reen **E**nergy **R**esources); the future of solar car transportation. TIGER involves a combination of solar power and bio-fuel used to optimize performance, as well as, reduce environmental emissions. This two-passenger

* Aerodynamics Team Leader, Auburn University Solar Car Team, 159 Burton St. Apt #2, AIAA Student Member.

hybrid vehicle utilizes two renewable resources yet uses a standard driver interface. The car was designed so that solar power could be used for the daily commute in typical daily driving conditions while bio-fuel acts as a reserved energy source, used to extend the driving range through the use of an internal combustion engine (ICE). This is done by monitoring the vehicles battery conditions. If the battery charge drops too low, the back-up ICE can provide drive power utilizing a cleaner burning, renewable bio-fuel. The car can recharge in a driveway or parking lot with deployable solar arrays. With this versatility, it is a viable option for future alternatively powered cars and the idea is currently patent pending.

Figure 1. 2005 Auburn Solar Car.

The team encountered design challenges in trying to make the car practical to the ordinary driver. As a vehicle with the stated goals, TIGER not only had to be stylish and appeasing to potential buyers, but it was imperative that the drag be low enough to achieve the desired efficiency goals (using solar power for commuting). Instead of starting from scratch, TIGER took influence from many current hybrid cars, yet has remained unique and elegant.

II. Initial Concept

The project started by researching what was already on the market in terms of aerodynamic vehicles. The Toyota Prius and the Honda Insight are among the most aerodynamic production vehicles and they gave the team an understanding as to what is necessary to have a car with low drag. Fig. 2 and Fig. 3 show the Toyota Prius and the Honda Insight respectively.

Influences taken from these cars include the flat transition from hood to windshield, the bluntness of the front end, and wheel fairings. These aspects helped to guide the team to the initial concept. The project began with a rough design as seen in Fig. 4.

The first concept resembled both the Insight and the Prius, while allowing for a large area for solar cells. There is a very good transition from hood to windshield and rear wheel fairings are employed to reduce drag around the rear wheel wells. The car is kept low to the ground in an effort to reducing aerodynamic drag. Yet there were many places for improvement. The car was rather boxy and lacked aesthetic appeal. Though somewhat rounded, the front edge is rather vertical, which would theoretically create air separation and thus drag. The windshield is very large due to the flat transition from hood to windshield and the requirements for driver seeing distance, which in turn, could create structural problems. The entire design is untested and could benefit from drag calculations.

Figure 2. The Toyota Prius.

Figure 3. The Honda Insight.

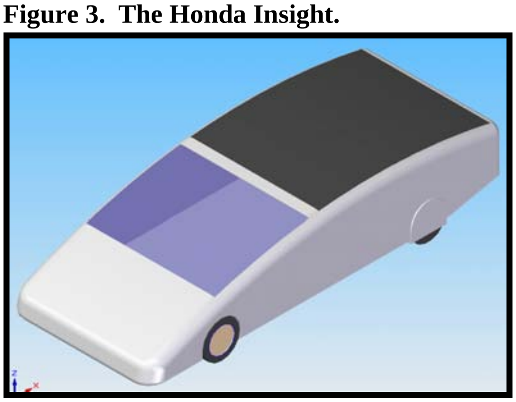

Figure 4. Initial Concept for TIGER.

III. Testing Process

Computational Fluid Dynamic (CFD) analysis was chosen instead of conventional wind tunnel testing because of CFD's ability to comparatively quickly and inexpensively model simple shapes, such as a two dimensional version of the vehicle. CFD also had the ability to more aptly measure the effects of a ground plane on such a vehicle. *GAMBIT* was chosen as the meshing software because it is more user friendly than other softwares and ultimately can be learned more rapidly. *Fluent* was used as the actual fluid analysis program. To test in two dimensions, a centerline cut is made from a 3-D model of the car so the wheels are not tested. This is important, because in 2D the wheels would stop air from flowing under the car.

This is not a step by step instruction manual on how to use *GAMBIT* and *Fluent*, so many steps in the actual meshing of the vehicle and the steps in running the fluid analysis have been omitted. However, the equations and sample calculations can be found in the appendix.

Meshing occurred in *GAMBIT* using an unstructured grid. The mesh was then exported into Fluent in the two dimension, double precision (2d dp) setting. It is important to note that *Fluent* converts all English units into SI units. Therefore, even though the grid is created using inches, the results are given in Newtons, Pascals, and meters.

Each model was run in Fluent using Wilcox's K-Omega turbulence model.[†] The first few models were tested at both 40 mi/hr and 70 mi/hr but later models were only tested at 40 mi/hr in an effort to stream line the process and receive results faster. For the sake of this report, only the plots of the vehicles at 40 mi/hr will be considered because the tests at 70 mi/hr differ only slightly. However, the values for drag and the coefficient of drag of the 70 mi/hr tests will be included. Much thought was given on how to most correctly model the road surface. Like a wind tunnel, this CFD process models the air flowing over the vehicle rather than the vehicle traveling through the air. In actuality air does not travel laterally over a road surface, yet this occurs in both wind tunnel and CFD testing. So, to best model what actually occurs, the road is modeled as a slip-wall, that is, a wall that creates no friction. The values that are needed in the K-Omega model are listed in Table 1.

Table 1. Fluent Values.

Variables	Values based on Velocity
40 mi/hr	
V_∞	17.8816 m/s
K	0.0191817 m^2/s^2
ω	122.784 s^{-1}
70 mi/hr	
V_∞	31.2928 m/s
K	0.0587544 m^2/s^2
ω	376.026 s^{-1}

Most of the solutions converged in between 5000 and 8000 iterations. Once the residuals converged, a drag value was \reported. The case and data files were then saved. These files were loaded into a program called *Tecplot* where further analysis was done such as stream tracing, which aided in locating vortices.

IV. Design Optimization

Testing started with two two-dimensional versions of vehicles to compare the differences in drag due to varying nose shapes. These two shapes were generated from the same ideas that led to the initial body design. The sharp corners were used because it made modeling easier and produces more accurate results due to better meshing in GAMBIT. The two shapes that were first modeled are shown in Fig. 5 and Fig. 6.

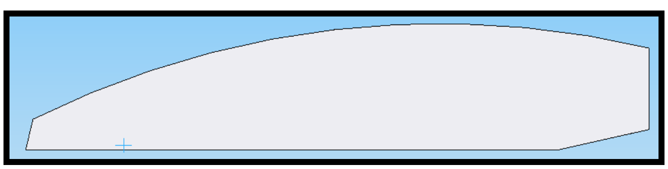

Figure 5. CFD Test Profile 1.

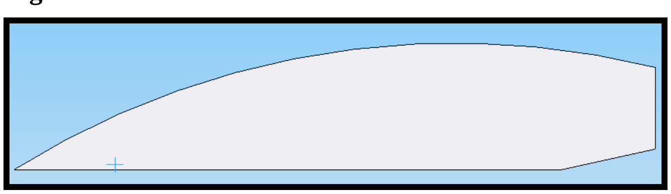

Figure 6. CFD Test Profile 2.

Both of these models were tested at 4 inches off the ground. They were both tested at 40 miles per hour and 70 miles per hour, the two speeds used by the team for power consumption calculations.

The first task was to compare the two models with differing frontal noses to see which was more aerodynamic. The tests were

[†] The K-Omega model was used on the recommendation of Dr. Christopher Roy.

run using the above mentioned process and the results were found.

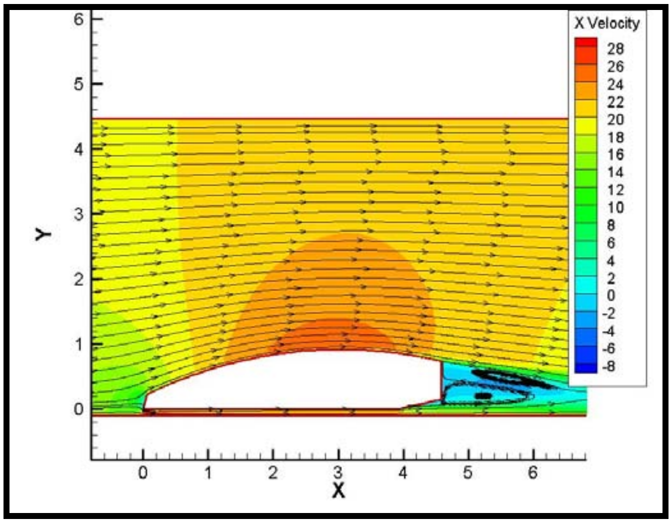

Figure 7. Profile 1 X-Velocity Plot at 40 mi/hr.

Fig. 7 shows the values and trace lines of the airflow velocity in the X direction for the entire vehicle at 40 mi/hr. Trace lines are the arrowed lines shown in Fig. 7-11 and Fig. 13. They follow the flow of the air and help visualize the fluid flow It is apparent that there are two vortices at the rear of the car. Upon closer inspection, the vortices become more prominent as seen in Fig. 8. It is apparent that there is a great deal of airflow separation even before the flow reaches the trailing edge. In the trace lines, it can be seen that the actual vortex sweeps up under the rear of the car. A small amount of separation can be seen on the top edge by the reduction in velocity of the air, yet this is not very significant.

Fig. 9 is the nose section of the profile. A stagnation point, where the velocity drops to zero, can be seen in front of the nose, and the sharp corners, though they make the meshing easier, create separation and vortices. The coefficient of drag is calculated from the reported drag value, the free stream velocity and the cross sectional area of the vehicle. To find the cross sectional area vehicle, the maximum height was multiplied by the width of the car (72 inches). This allows for a rough approximation for the coefficient of drag. The values for the first profile are reported in Table 2.

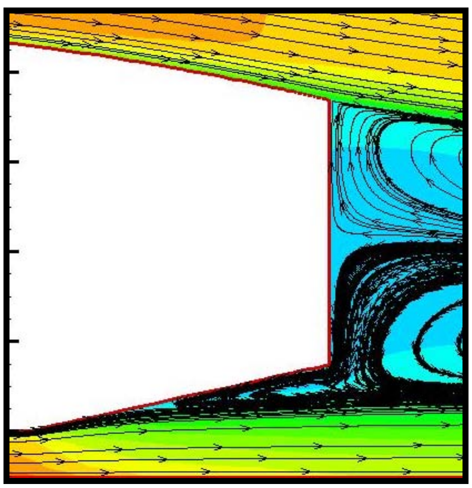

Figure 8. Trailing edge of Profile 1 X-Velocity Plot at 40 mi/hr.

Table 2. Profile 1 Drag and C_d Values.

Velocity	Drag	Coefficient of Drag
40 mi/hr	62.475 N	0.172
70 mi/hr	205.744 N	0.185

Fig. 10 is an x-velocity plot of the second profile at 40 mi/hr. Vortices can again be seen behind the car. The problem with separation causing recirculation under the back end of the car is again present in this test. This is to be expected as the geometries of the end of the vehicles are the same.

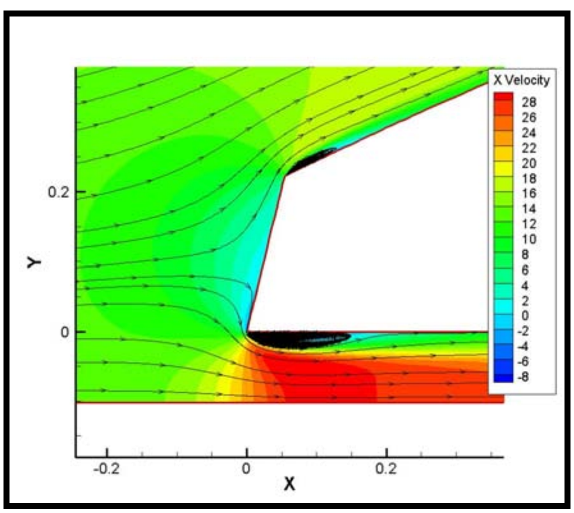

Figure 9. Profile 1 X-Velocity plot at front at 40 mi/hr.

The frontal nose of Profile 2 is pictured in Fig. 11. A stagnation point is marked by the blue area at the tip. It is apparent that the velocity under the car dramatically increases due to the sharp edge, yet there is not a vortex generated by flow separation. The drag and C_d values are shown in Table 3.

Table 3. Profile 2 Drag and C_d Values.

Velocity	Drag	Coefficient of Drag
40 mi/hr	60.804 N	0.167
70 mi/hr	198.457 N	0.178

Thus, it is shown that the pointier Profile 2 is more aerodynamic because the coefficients of drag are smaller than the corresponding values for Profile 1. Yet both cases had large amounts of separation near the back end and it seems that both designs could benefit from rounding and curves. These two profiles served their purpose as a basis for this project. As more aesthetic features were added, these profiles served as a reference.

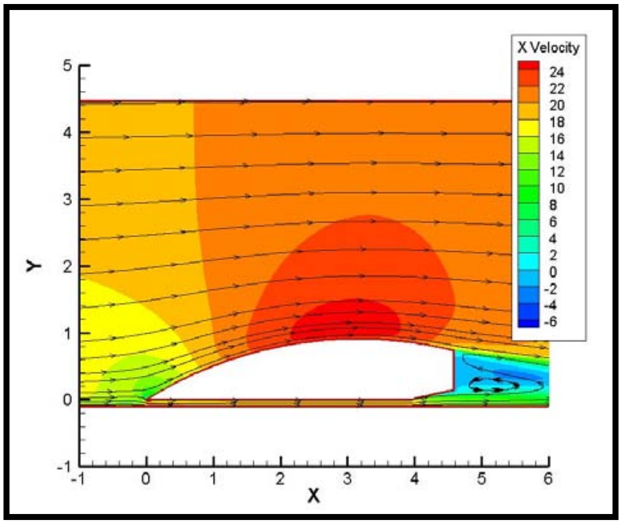

Figure 10. Profile 2 X-Velocity plot at 40 mi/hr.

The third profile was generated in an effort to eliminate the amount of separation near the back end of the two previous models. The last two profiles used a rear angle of approximately 12 degrees, as shown in Fig. 12.

Research suggested that an angle of 5 degrees or less would drastically decrease the amount of airflow separation.[1] A model was creating using the exact Profile 1, except the rear angle was decrease from 12 degrees to 5 degrees. Everything else remained precisely the same. The results are shown in Fig. 13.

It can be seen that at 5 degrees, the separation is reduced by a great deal and the vortex no longer reaches under the back end of the car. The drag and C_d values changed as well as seen in Table 4.

Table 4. Profile 3 Drag and C_d Values.

Velocity	Drag	Coefficient of Drag
40 mi/hr	59.972 N	0.165
70 mi/hr	197.313 N	0.177

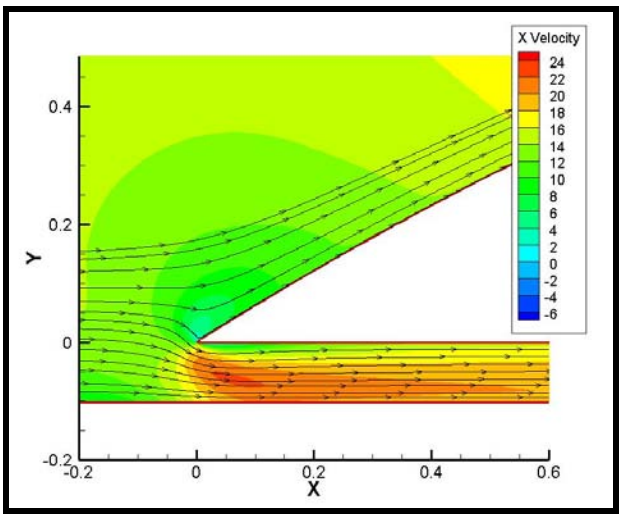

Figure 11. Profile 2 X-Velocity plot at front at 40 mi/hr.

There is a significant decrease in drag by simply changing the rear angle. In fact, changing the angle from 12 to 5 degrees created a 2.67% decrease in drag at 40 mi/hr and a 3.54% decrease in drag at 70 mi/hr.

Development therefore became an iterative process in which key areas were targeted, a new profile was created, and then tested. In an effort to stream line the process, each profile was only tested at 40 mi/hr after the first few profiles. After several cycles of development a profile was decided upon, shown in Figure 14. This final model yielded values that can be seen in Table 5.

Table 5. Drag and C_d value for final Profile.

Velocity	Drag	Coefficient of Drag
40 mi/hr	54.65 N	0.149

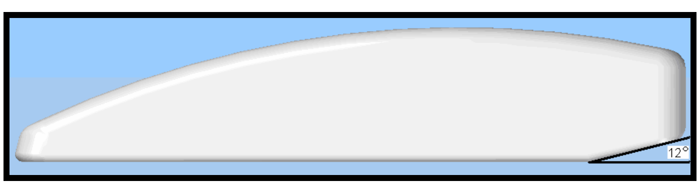

Figure 12. Profile 1 rear angle.

The final profile marks an improvement of 13.37% over Profile 1. Though this profile could still be aerodynamically improved upon, it is limited by the projects goal of being practical and stylish. From this profile, a 3-dimensional model was created from which the master mold of the car was build. Figure 15 shows an artist's rendering of the final car.‡

A 1:15 scale 3-D model was manufactured using a process called rapid prototyping in which a computer model is physically built by a machine by applying thin layers of plastic, which correspond to the size and shape of the corresponding cross section of the car. The model created was used as a wind tunnel model to find more accurate

‡ Artist's rendering created by Steven Guice.

drag values for a 3-D model. The model was tested in Auburn University's 3ft x 4ft closed circuit sub-sonic wind tunnel at speeds of 40 mph, 70 mph, and 80 mph. The preliminary results are shown in Table 6.

These preliminary values have a 5% error adjustment for associated errors in the wind tunnel modeling. Though these numbers are only preliminary results, they are quite promising. The listed C_d value for the Toyota Prius is 0.26[2] and the Honda Insight is 0.25[3]. Thus, using the largest average C_d (0.218) there is a 12.8 % reduction in drag from the Honda Insight and a 16.2 % reduction in drag from the Toyota Prius.

More wind tunnel testing is planned for the future in an effort to find more precise and accurate values for the coefficient of drag.

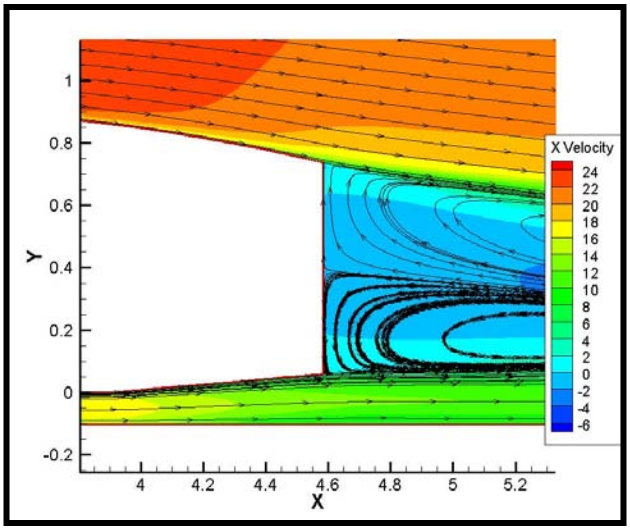

Figure 13. Profile 3 X-Velocity plot at rear at 40 mi/hr.

Table 6. Average C_d for 3-D wind tunnel model.

Velocity	Average C_d
40 mph	0.205
70 mph	0.218
80 mph	0.215

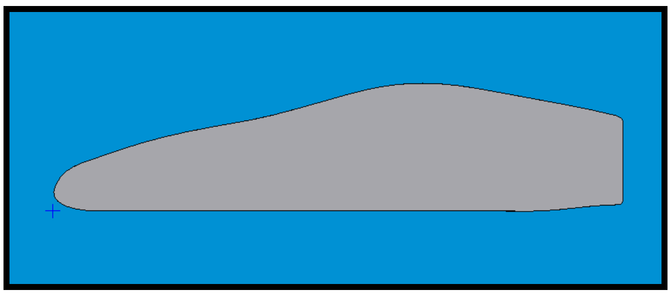

Figure 14. Final Profile.

Figure 15. Artist's rendering of final car.

V. Conclusion

It can be concluded that CFD is a powerful tool for vehicle design optimization when being applied to the development of a solar - bio-diesel hybrid powered car. With the results from the preliminary wind tunnel testing, it is apparent that CFD is a useful and powerful tool for relatively quick and comparable results. This process is particularly useful as alternatively powered vehicles seek to be more efficient and aerodynamic. The iterative process of this testing marked an estimated 13.37% improvement in aerodynamic drag from the first tested profile to the final profile. This improved model was used to make a 3-dimensional model which had an estimated improvement of 12.8% and 16.2% drag reduction over the most aerodynamic commercial vehicles.

TIGER is now under construction and will undergo many tests once completed in order to compare actual C_d values to that of the wind tunnel model and CFD analysis. Once the drag characteristics for TIGER are understood, areas of improvement can be found in an effort to advance future alternatively powered vehicles. TIGER will participate in the North American Solar Car Challenge in July of 2008.

Appendix

Sample Calculations for K-Omega model:
All calculations are done assuming sea level atmospheric conditions.

$V_\infty = $ 40 mi/hr = 17.8816 m/s

$$K = \frac{1.2}{2}\left[\left(\frac{Tu}{100}\right)V_\infty\right]^2$$

$$Tu = 1$$

Substituting:

$$K = \frac{1.2}{2}\left[\left(\frac{1}{100}\right)17.8816\frac{m}{s}\right]^2 = 0.019181664\frac{m^2}{s^2}$$

$$\omega = \frac{\frac{\rho K}{\mu}}{10}$$

$$\rho = 1.184\ \frac{kg}{m^3}$$

$$\mu = 1.85\times10^{-5}\ \frac{N\cdot s}{m^2}$$

Substituting:

$$\omega = \frac{\dfrac{(1.184\frac{kg}{m^3}\cdot 0.019181664\frac{m^2}{s^2})}{1.85\times10^{-5}\ \frac{N\cdot s}{m^2}}}{10} = 122.784 s^{-1}$$

Acknowledgments

I would like to thank the many people whom, without their help, this paper would not be possible. Dr. Christopher Roy and his graduate students, Pavan Veluri, and Eric Tendean taught and guided me through the CFD process. Dr. Anwar Ahmed and his graduate students Chris Worley and Brian Recktenwald aided with the wind tunnel testing and analysis. George Vallone helped with all parts of this research. I would also like to thank Steven Guice for his rendering of the final car.

References

[1]Carroll, D. R., *The Winning Solar Car: A Design Guide for Solar Race Car Teams*, SAE International, Warrendale, PA, 2003, Chap. 5, pp. 147.

[2] Toyota Prius 2008, "Specs, Dimensions." URL: http://www.toyota.com/prius-hybrid/specs.html [cited 24 February 2008].

[3]Honda Certified Used Cars – 2004 Insight, "Overview." URL: http://automobiles.honda.com/certified-used/2004/insight/specifications.aspx [cited 24 February 2008].