Mechanics and Mechanical Engineering
Vol. 20, No. 3 (2016) 295–308

Investigation on Aerodynamics of Super–Effective Car for Drag Reduction

Artur Cieśliński
Wojciech Prym
Mateusz Stajuda
Dariusz Witkowski
Lodz University of Technology
Stefanowskiego 1/15, Łódź, Poland

Received (15 August 2016)
Revised (16 August 2016)
Accepted (17 September 2016)

This paper focuses on shape optimization of a car body to be used in Shell Eco Marathon race. The work consists of the review of aerodynamic performance for currently used shapes, definition of the design constraints for the vehicle and recommendations for the final shape to be used in the oncoming Shell Eco Marathon editions. The designs are inspired by winning models, but adjusted to Iron Warriors technology and scaled accordingly to the driver's space requirements. A range of velocities from 20 km/h to 45 km/h with 5 km/h interval is tested, giving idea about the full model performance. Results are then compared and the best solutions, concerning the coefficient and parameter taking into account the frontal area influence are recommended.

Keywords: super efficient vehicle, drag reduction, aerodynamics, CFD analysis.

1. Introduction

Growing environmental awareness and high fuel prices have been an incentive for development of more and more efficient vehicles. Except for producers intensified work on this topic, it is also a subject of various student competitions. Shell Eco Marathon is the most popular student competition for a vehicle with lowest energy consumption. The aim of student teams from all over the world is to create a car consuming as little fuel as possible. Such task is very complex, as combining many aspects is needed for a working vehicle. The lowest fuel consumption can be attained by diminishing the resistance of motion, simultaneously increasing the vehicle efficiency. One of the main sources of vehicle motion resistance is aerodynamic drag, which strongly depends on a shape of the car body and its frontal area. The importance of aerodynamic drag reduction for Shell Eco Marathon vehicles is very high as drag can account for up to 50% of total energy consumption [1]. To reduce the drag, all cars participating in Shell Eco Marathon have very small frontal area, and are streamline to reduce the drag coefficient. As there is no obvious solution, the cars differ significantly in shape, even designers of the best cars tend to have different approaches, which raises the question of the best aerodynamic shape. Recent development in the field of CFD and PC computational power increase has given easier access to the sophisticated simulation tools, making it possible to test multiple shape designs with lower cost than ever before. Despite relative ease of aerodynamics simulations of super–efficient cars, not many articles were created in this field. It is probably due to the fact that in highly competitive environment, the students do not want to share their findings and give advantage to the opponents. One of the most comprehensive aerodynamics studies was presented by [1] in 2005. The whole process of car body shape development was presented there, however changes in the Shell Eco Marathon regulations make the design no longer valid. Other works, like [2, 3] were performed for different competitions or categories therefore different assumptions were made at the beginning. Car body optimisation was also a subject of numerous thesis, for example [4] was describing the car body development for DNV GL Fuel Fighter, the car of Trondheim University. All of the aforementioned works focused on introducing minor changes to defined shapes of the car body and tested their influence on the drag. None of the works gave general benchmark of different shapes which could be used for a Shell Eco Marathon. As many different designs of vehicles are winning Shell Eco Marathon, the need for testing and assessment of their performance emerges.

1.1. *Aim of the work*

The aim of this work is to analyze shapes currently used by different teams in the competition, compare it with the shape used by Iron Warriors team and design a new car body with better aerodynamic performance which could be used by the team in the oncoming events to improve their result and break current Polish record of 830 km per liter set by the team. The car body, even in Shell Eco Marathon vehicles, is not only the outcome of aerodynamic consideration, but also mechanical possibilities, space constraints and competition regulations. Having those considered as initial conditions, the most optimal shape should be found.

1.2. *Aerodynamic drag and its sources*

Aerodynamic drag has been a subject of research for over 100 years. It has been primarily the interest of aviation industry, focusing on shaping the wings of airplanes in the best manner. At the beginning, it was neglected by automotive industry, but as cars become faster, its importance drastically increased. The aim of numerous scientist was to fully understand the air flow around bodies and provide guidelines for the most optimal shapes. Aerodynamic drag is a force induced by the relative velocity of a car and a fluid, which acts along the direction of a fluid flow, therefore opposite to the vehicle driving direction.

Aerodynamic drag is a force induced by the relative velocity of a car and a fluid, which acts along the direction of a fluid flow, therefore opposite to the vehicle driving direction. This force can be described by the formula [1]:

$$F_D = \frac{1}{2}\rho v^2 A C_D \quad (1)$$

where:

F_D – drag force [N],

ρ – density of the fluid [kg/m^3],

v – velocity of the object relative to the fluid [m/s],

A – reference area [m^2],

C_D – coefficient of drag [-].

Velocity of flowing fluid, area of reference, density of fluid, and non-dimensional coefficient influence the aerodynamic drag force. Reference area in automobiles and many other applications is equal to the frontal projection area of the body. The coefficient of drag represent the body aerodynamic potential, therefore it is important to understand where its source is.

The vehicle drag is caused by several factors. It can be generally divided into two main components:

- Friction drag
- Pressure drag

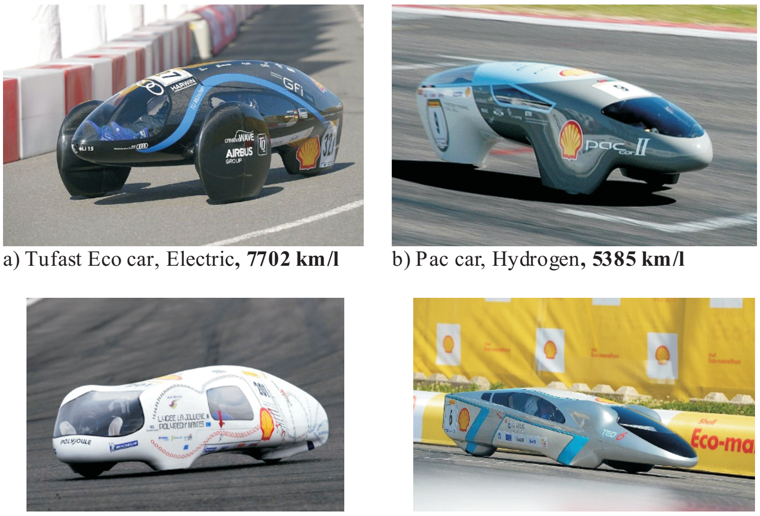
Frictional drag comes from friction between the fluid and the surfaces over which it is flowing. This friction is associated with the development of boundary layers, and it scales with Reynolds number.

Pressure drag comes from the eddying motions that are set up in the fluid by the passage of the body. This drag is associated with the formation of a wake, which can be readily seen behind a passing boat, and it is usually less sensitive to Reynolds number than the frictional drag. Formally, both types of drag are due to viscosity (if the body was moving through an in viscid fluid there would be no drag at all), but the distinction is useful because the two types of drag are due to different flow phenomena. Frictional drag is important for attached flows (that is, there is no separation), and it is related to the surface area exposed to the flow. Pressure drag is important for separated flows, and it is related to the cross-sectional area of the body [5, 6].

The aerodynamic drag force is a combination of several parameters. The vehicle designer attempting to decrease the vehicle drag may decrease frontal area, drag coefficient or both. Therefore the aim of the designer is to design a car body with the smallest possible frontal area with shape providing lowest possible coefficient. The aerodynamic drag coefficient for vehicles participating in Shell eco marathon is much lower, reaching extreme values like 0.08 for the record holder, PacCar II [1]. The same formula as (1) holds for lift force, with only difference being drag coefficient replacement with lift coefficient.

1.3. *Designs used in Shell Eco Marathon*

Since the emergence of Shell Eco Marathon many different shapes were used in the competition. Fig. 1 from a) to d) shows wining designs from all categories. As it can be noticed, different approaches are used. Most of the teams decides to have an unibody construction with 3 wheels, two in front and one in the back, all placed in drop-shaped car body. A different and less popular approach is to create a narrower body and place front wheels outside it. The second approach also has different aspects, as wheels may be covered or exposed to the air flow. The teams do not share information about vehicle aerodynamic performance, therefore its assessment is required to choose the best design as the fact of winning may not be due to optimized aerodynamics, but other mechanical aspects.



a) Tufast Eco car, Electric, **7702 km/l**
b) Pac car, Hydrogen, **5385 km/l**
c) Microjoule car, CNG, **2551 km/l**
d) TedAsso car, Gasoline, **2308 km/l**

Figure 1 Winning car body designs, their categories and fuel consumption [12-15]

1.4. *Iron Warriors Eco Arrow 2.1 & body optimization*

The design currently used by Iron Warriors team is an unibody construction, being a compromise between aerodynamic performance and mechanical construction of the vehicle components. The shape was created in 2015, being second design of the team since establishment of Iron Warriors. It was not subjected to any CFD analysis before. The body used in races from 2015 to 2016 can be seen in Fig. 2.

Although the team has achieved quite an impressive result (830 km per litre) considering its second start in the competition, there are still several issues which may be improved. One of them is the car body construction. The main problem of Eco–Arrow is its excessive size. This results in two negative aspects which strongly affect the fuel consumption. One is the frontal area of the vehicle influencing the aerodynamic properties. Second is simply the amount of additional material which increases the total mass of the car.

Figure 2 Eco–Arrow [16]

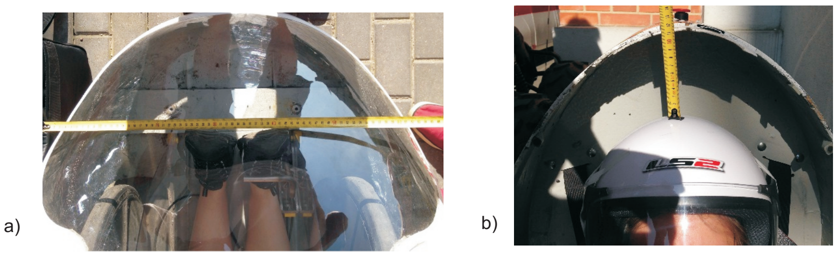


Figure 3 Measurement of the current car body loses: a) width near the feet (on the left); b) height above the helmet (on the right)

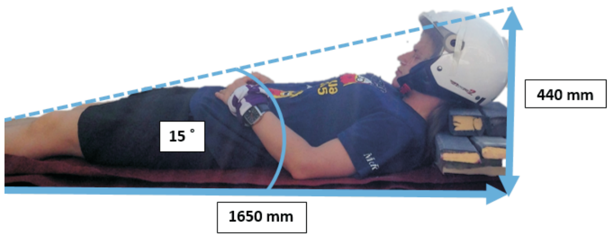


Figure 4 New angular position of the driver's seat

In certain areas there is too much space left without any purpose and those are the spots were optimization may take place. Fig. 3 presents some of the places which were subjected to change.

The idea was to minimize the amount of space near the pedals and adjust the height of the vehicle near the helmet area. What is more, after a discussion with the team driver an agreement has been reached that the whole seat can be adjusted at a smaller angle and so the driver would take more laying position. This type of change has given an opportunity to lower the car body much further than expected. Finally, it appeared that the whole car body may be lowered about 10 cm. To define the constraints measurements of the driver in an expected driving position were taken. Fig. 4 shows space required in this configuration.

Despite the new position of the driver, the upcoming regulations for the Shell–Eco Marathon 2017 had to be taken into consideration [7]. They impose the vehicle turning radius of 6 m instead of 8 m. As a result, the front wheels require more space for turning. That aspect was also taken into consideration within the new car body designs.

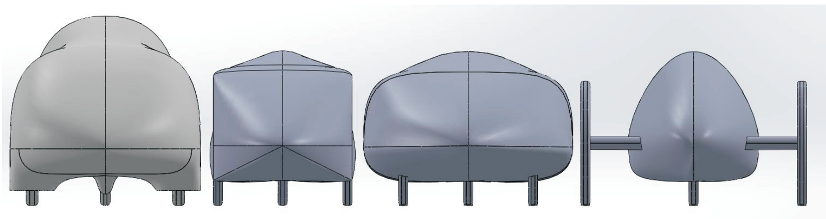


Figure 5 Car body shapes: a) Eco-Arrow; b) Prototype 1; c) Prototype 2; d) Prototype 3

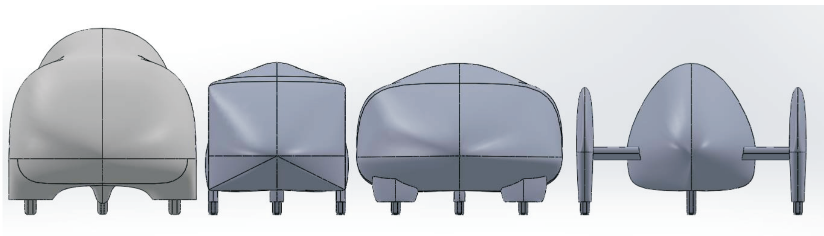


Figure 6 Car body shapes with fairings: a) Eco-Arrow; b) Prototype 1; c) Prototype 2; d) Prototype 3

Moreover, it is known that the presence of the exposed rotating wheels has a great impact on drag generation. According to [8], [9], the influence may be even up to 40% of the vehicle total drag for the Formula One racing cars. Although, the mentioned vehicles have much wider wheels, turning with much higher velocities, an extra drag generation is most probably present within the thin wheels used in Eco–Arrow. Taking that into consideration, the analysis of aerodynamic properties for proposed designs was split into two cases. The first one considered the wheels to be exposed (Fig. 5). Second one assumed that the wheels were partially covered by fairings (Fig. 6), in order to define the improvement in drag reduction due to such feature.

For the following study three different car body prototypes have been drawn. Figs 5 and 6 present comparison of the frontal area of the current Eco–Arrow vehicle (first on the left) and proposed prototypes designs. It is easily visible that all the prototypes are much lower vehicles thanks to a new angular position of the driver. Prototype 1 (b) is a new model with wheels aligned with the car body, but not covered from the outside. This design was created partly due to the Iron Warriors team request to increase the wheels accessibility for the maintenance purposes. Prototype 2 (c) being similar to the Eco–Arrow but much lowered. Prototype 3 (d) - having an extraordinary shape - protruding wheels and sharply pointed front.

Some details concerning the frontal area are listed in the Tab. 1. It appears that the biggest drop in frontal area is in case of Prototype 3, however it has to be taken into account that the wheels in this case are protruding and that they will generate some additional drag. Moreover, area for the prototypes with fairings has been included to visualize the change in the frontal area. It appeared that in some cases (Prototype 2 and 3) the additional coverings significantly enlarged the area.

Table 1 Car body shapes: frontal area comparison

	Eco–Arrow	Prototype 1	Prototype 1 + fairings	Prototype 2	Prototype 2 + fairings	Prototype 3	Prototype 3 + fairings
Area [m^2]	0.413	0.262	0.263	0.356	0.368	0.250	0.270
Diff.	-	-37%	-36%	-14%	- 11%	- 40%	-35%

The size of the frontal area give some idea of an aerodynamic improvement although that has to be verified by means of CFD analysis.

2. CFD Analysis

2.1. *Domain and mesh preparation*

The 3D car models were originally designed in one of the CAD programs and then imported to Ansys software. Due to the symmetry of the car, it was assumed that the flow on its both sides was identical and only half of the model was used for simulation. That assumption had a great impact on the mesh size, simulation time and computational domain shape.

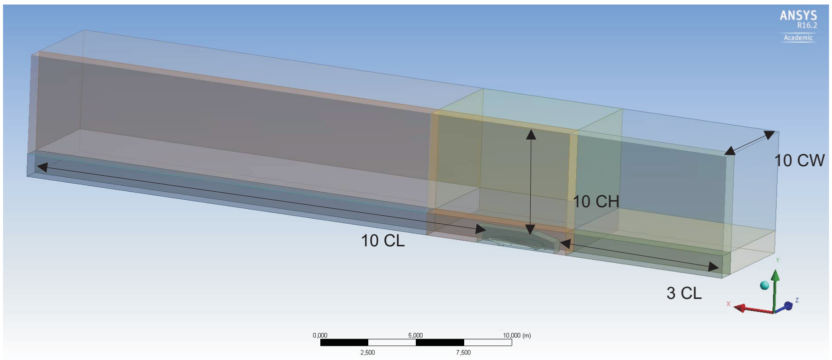


Figure 7 Computational domain; CL - car length; CH - Car height; CW - car width

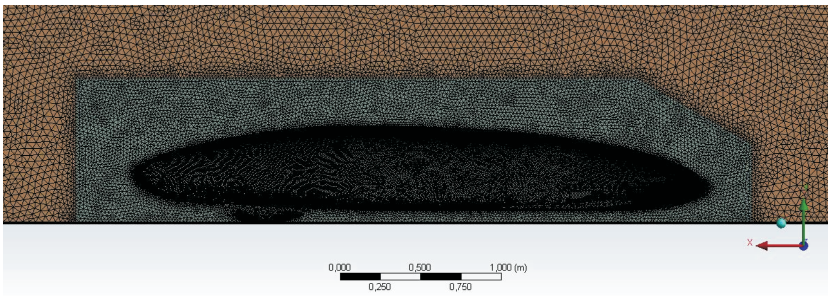


Figure 8 Mesh in the nearest neighborhood to the vehicle body

The main dimensions of the computational domain are presented in Fig. 7. The flow inlet plane was located three car lengths away from the front of the car body, whereas the outlet plane ten car lengths from the rear. The side plane was located sufficiently far from the car body side surface (about 10 car widths), providing vast simulation domain. The height of the domain was set to ten times the height of the car body. The ground was located at the distance 0.06 m from the car body bottom surface.

In order to conduct accurate analysis, a precise mesh had to be used. The computational domain was divided in a few regions, in order to introduce different mesh definitions. The most important region is the one in the nearest neighbourhood to the vehicle body (Fig. 8). Due to irregular shape of the car surface the tetrahedral mesh was applied. The mesh was refined in this zone and inflation layers of prismatic elements were used to properly solve the boundary layer flow, which is crucial for this type of analysis.

To ensure the good quality of the results, a few mesh densities were proposed, on the basis of which, the most satisfactory was chosen for further simulations. In order to check quality of the generated mesh, parameters such as aspect ratio, skewness were verified. Aspect Ratio is a ratio of the longest to the shortest length of the mesh element and should not exceed 100. Skewness describes angular deformation of the mesh elements. It is defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume and its max value should be kept below 0.95, with an average value that is significantly lower [10].

The max. aspect ratio on average was about 500, yet only in places far away from car body - in the region of small mesh density and negligible velocity gradients. The region in the closest neighborhood of the car body had aspect ratio below 50. The skewness appeared to be of max value equal to 0.98. Due to [10], such a high value of skewness may lead to some convergence difficulties, however, this problem did not take place.

The mesh on average was composed of 6 million nodes and 10 million elements in total.

2.2. *Pre–processing*

For the purpose of CFD simulation some assumptions were made regarding the domain properties. The flowing air was treated as an ideal gas at the temperature of 25 $^\circ$C. The Shear Stress Transport (SST) turbulence model was used, which provides good results in a wide range of flow types. The solver settings were defined as listed in Tab. 2.

Table 2 Boundary conditions for 3D car body analysis

Turbulence model:	Shear Stress Transport (SST)
Fluid type:	Air: Ideal gas
Temperature of the fluid:	25 $^\circ$C
Fluid velocity range:	20, 25, 30, 35, 40, 45 km/h

To enhance the credibility of the solution two important aspects were comprised in the simulation. First, an imitation of a relative velocity of the car with respect to the ground. A non–slip, smooth wall condition with velocity was implemented at the bottom wall of the domain with the velocity of the moving ground depending on the velocity of the moving air. Second, a rotation of wheels to account for the resultant flow turbulences.

2.3. *Solution quality*

In order to provide reliable numerical solution of flow structure in the boundary layer region, an appropriate refinement of the computational mesh should be applied. The quality of solution in that layer can be evaluated by the Yplus (y^+) parameter, which describes the non–dimensional wall distance for a wall–bounded flow:

$$y^+ = \frac{u_* y}{\nu} \tag{2}$$

where u_* is the friction velocity at the nearest wall, y is the distance to the nearest wall and ν is the local kinematic viscosity of the fluid [11]. Whether y_+ the parameter of the first mesh node near the wall is of a proper value, it is dependent on the needs of solution accuracy and flow representation. Once the y_+ value is below 1 (2 in ANSYS CFX), the software may fully solve the flow in the boundary layer. In case the value of the parameter is greater than 1, the flow structure is reproduced with the use of the wall function. The most important conclusion is that the smaller the y_+ value, the more precise the solution near the wall, however the mesh size and simulation time is longer. In the case of attached flow simulations usually significant difference in the solution is observed neither for full solution of the boundary layer nor for wall function. However, in cases of separated flows differences can be significant, therefore, if it is possible, full solution of the boundary layer is recommended.

In order to determine the reliability of the solution, a few parameters were analyzed and verified. Domain imbalance (should be lower than 0.1%) and a couple of the last iterations should be controlled to ensure that the results were obtained without any failure. For all the mass and momentum equations the imbalances appeared to be below 0.1%, indicating stable solution. Moreover for all the solutions, the mass and momentum residuals reached a level lower than 2e-5. This level of convergence was obtained after around 200 iterations. The residuals did not attain lower values due to unsteady flow eddies behind the vehicle.

3. Results

The results obtained from simulations provide number of parameters which may be used for assessment of body shape performance. The graphical representation of the flow is not useful in case of model performance assessment, therefore the authors limit themselves to presentation of standard aerodynamic coefficients allowing for easy comparison. The most important one is drag coefficient C_D, but the choice of the final shape should also consider the frontal area of the car as their product influences the total force acting on a body. Due to this, AC_D coefficient is introduced, being the product of frontal area of the car A and drag coefficient C_D. Another important factor is lift coefficient C_L, which should be as close to zero as possible, not to introduce additional induced drag or down force. The calculated drag force may be additionally compared with other resistances, so that the team can decide whether further optimization in field of aerodynamics is important, or maybe they should shift their priorities to other areas where energy losses are much higher. The results of simulations for 25 km/h, being an average velocity of the vehicle during the race are presented in Tab. 3. For the purpose of result presentation and analysis, the prototypes were denoted as following: Prototype 1 without fairings – v1_1, Prototype 1 without fairings – v1_2, etc.

Table 3 Boundary conditions for 3D car body analysis

Model	Eco arrow	V1_1	V1_2	V2_1	V2_2	V3_1	V3_2
C_D [-]	0.096	0.093	0.104	0.105	0.092	0.197	0.182
AC_D [m^2]	0.0396	0.0244	0.0274	0.0372	0.0339	0.0492	0.0491
C_L[-]	0.0825	-0.0908	-0.0096	-0.0693	-0.0909	0.0082	0.0087
F_D [N]	1.12	0.69	0.77	1.05	0.96	1.39	1.39

Drag coefficient C_D for all the velocities are presented in Fig. 9. As it can be noticed, they decrease with an increase of the car velocity. The lowest drag coefficients are obtained for model v2_2, which indicates that its shape is the most streamlined. The values are slightly lower than those for EcoArrow or model v1_1. Concerning the latter, its good result is surprising, as rolling wheels tend to induce higher drag [8]. The root of this result may be idealization of the geometry, which assumes that wheels are exactly in–plane with the car body and there is no gap between the wheel and a car body. Results of v1_2 are noticeably worse than those of the best models. The C_D values obtained for v3_1 and v3_2 models are very high, mostly due to detachments of the flow at the wheels or wheel fairings.

The values of C_D coefficients are lower for models with fairings for two cases (v2_2 and v3_2), but opposite result is visible for model v1_2 having much higher C_D than model without fairings. This may be due to smaller fairings having different task than those in models v2_2 and v3_2.

When the product of area and drag coefficient AC_D, being indicator of car performance is analyzed (Fig. 10), the best results are obtained for model v1_1. This is mostly due to low frontal area, as wheel covers increasing the width of a car body are not present. The obtained characteristics is however valid only for driving straight, with wheels unturned. In case of any turn, the area would increase and wheel would be exposed to the flow, which would definitely increase the drag coefficient. This is also a threat for model v1_2, having second best result. The third best result concerning the $AC_D$is obtained for v2_2 model, despite its higher frontal area than that of v2_1. The results obtained for models v3_1 and v3_2 are much above others, indicating that the concept connected with shape of versions v3_1 or v3_2 is not promising and should not be furtherly developed.

Considering lift coefficient $C_L$for most of the designs it is negative, resulting in creation of a downforce acting on a vehicle. The influence of this force is however small, reaching values below 2.5 N which is negligible in comparison with the total vehicle weight. Nevertheless, the non-zero value of lift indicates that further improvement can be made to the geometry, so that the induced drag component would be lowered.

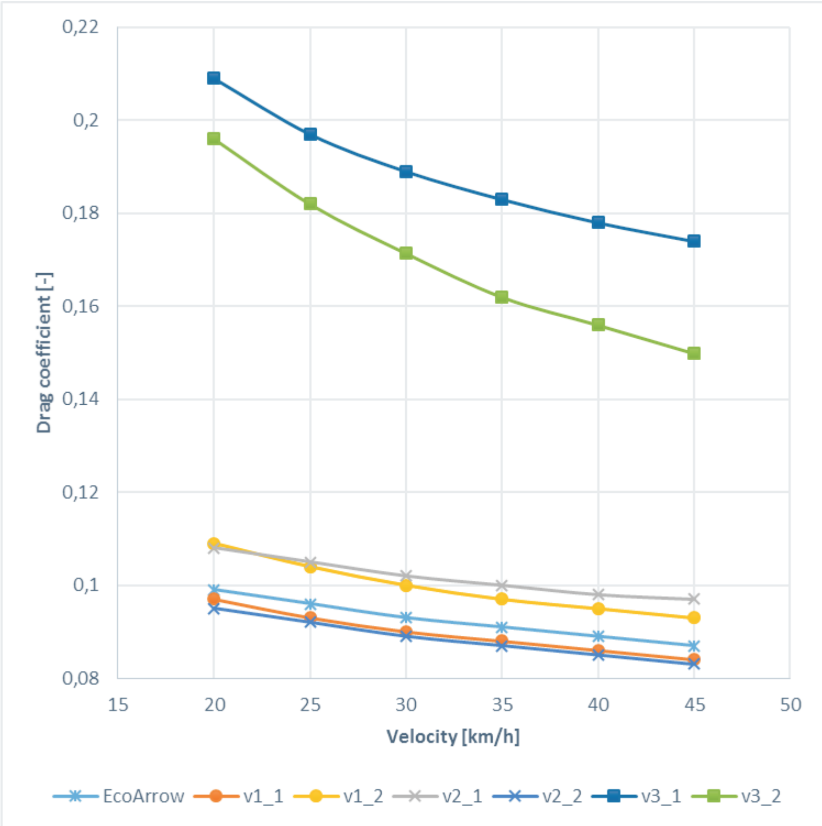


Figure 9 Drag coefficient - C_D values for different car velocities

The drag force decrease in comparison with Eco Arrow is significant, especially for v1_1, v1_2 and v2_2 models. Considering that aerodynamic drag constitutes up to 50% of the total resistance, the decrease in drag force of 14.6% should ideally result in 7.3% decrease in total resistance.

4. Conclusions

This article presents a numerical aerodynamics study of several vehicle shapes used in Shell Eco Marathon and their comparison with the Eco Arrow model, currently used by Iron Warriors Team. The results obtained thanks to this study are to be used for design of new model for the oncoming races.

The set of simulations was prepared for 6 models inspired by winning designs, with proper care taken for their sizing with respect to new regulations and driver's space requirements. Benchmark level was established form current car body simulations. A full range of velocities from 20 to 45 km/h was simulated, with step of 5 km/h. Obtained, and values indicate the general trend for the designs and possible areas of improvement.

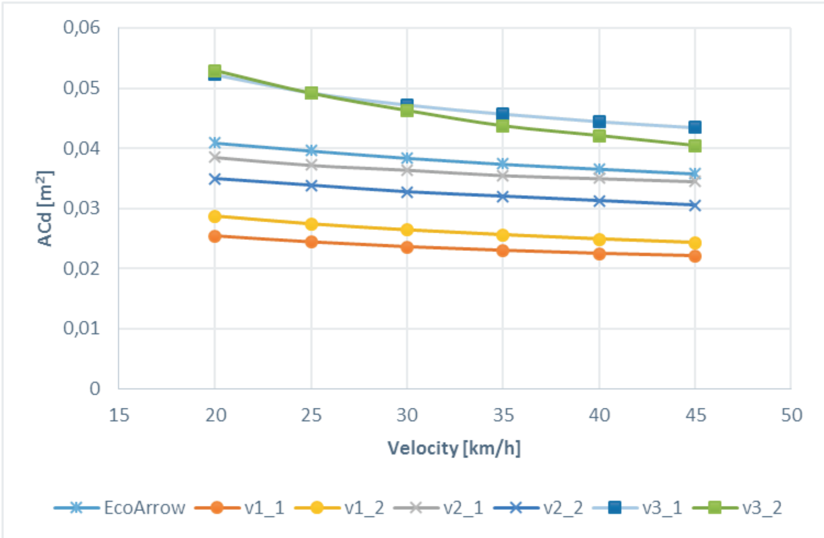


Figure 10 AC_D values for different velocities

Thevalues for a shape with wheels separated from the main body are the highest and make this type of design unworthy further consideration. The values for a design, having wheels partially exposed to the flow are surprisingly low, which may be attributed either to very low drag of this configuration or geometry simplification and ideally frontal air inflow, which results in better performance than in reality. The lowest values of 0.083 were obtained for an unibody construction with wheels covered and contained in fairings (model v2_2). The obtained value is however still higher than those of the best competition designs, which reach values as low as 0.075 [1].

Considering parameter and resistance power for average car speed, they are the lowest for model v1_1, having wheels aligned with car body, mainly due to low frontal area and very small influence of the exposed wheels on the value. This indicates that this model could be worth considering, however the results should be validated for situation with wheels turned, as then the drag could increase significantly due to higher wheels exposure and increase in frontal area. Certain study concerning the level of simplification on the results should also be performed. Additional element are design difficulties concerning mechanical solution for this type of model.

The overall influence of standard wheel fairings is positive, which is visible in comparison of models v2_1 and v2_2. Despite higher frontal area, the gain in drag coefficient is big enough to obtain lower parameter and therefore smaller resistance power. Results for fairings covering only front of the wheel (model v1_2) are negative, implying that in this type of design, it is better to leave the wheels uncovered.

The obtained result give plenty of data and allow to decide about the best design approach. Further work is required, which may be done either in the field of shape optimization for v2_2 model, or in further investigation of v1_1 model, as the preliminary results are promising, but should be validated for different situations.

Obtained lift coefficients show that further work may be done in a field of decreasing the induced drag. With current data and knowledge, the authors' recommendation for the team would be to use a design based on model v2_2. This choice would be a cautious one, as the aerodynamic characteristics of this model is predicted to be relatively stable for different angles of air inflow. With direct implementation of v2_2 design and no mechanical changes in the vehicle itself, the team should be able to increase its result from current 830 km per liter and set a new Polish record.

References

[1] **Satin J.J., Onder C.H., Bernard J., Isler D., Kobler P., Kolb F., Weidmann N., Guzzella L.**: The World's Most Fuel Efficient Vehicle: Design and Development of PAC-CAR II, *Zurich/Singen*, **2007**.

[2] **Oliveria Santos, R., Lyra, P. R. M., Araujo Souza, M. R., Souza Junior, M. A.**: Aerodynamic Design of Super Efficient Vehicle, *SAE International*, **2012**.

[3] **Wąsik, M., Targosz, M. and Panfil, W.**: Methodology of aerodynamic analysis in the Hyperworks software of the cars participating in the shell eco–marathon race, Proceedings of the institute of vehicles, 3(99), Warsaw, Poland, **2014**.

[4] **Nguyen, M.**: Aerodynamic Development of DNV GL Fuel Fighter, *Master Thesis Norwegian University of Science and Technology*, **2015**.

[5] **Kazimierski, Z., Orzechowski, Z.**: Mechanika Płynów, *Wydawnictwo Naukowe Politechniki Łódzkiej*, **1979**.

[6] **Katz, J.**: Race Car Aerodynamics: Designing for speed, *Robert Bentley Publishers*, USA, **1995**.

[7] Formula Student 2016-2017 Rules. http://www.shell.com/promos/download-the-shell-eco-marathon-global-rules/_jcr_content.stream/ 1446710609481/d8693d7d3c36e6a4836cb57274c0fdf09 a489b60da03734e2f7311cc153f094a/sem-2016-global-rules-chapter1-010715.pdf (accessed **2016**, June)

[8] **Diasinos, S.**: The Aerodynamic Interaction of a Rotating Wheel and a Downforce Producing Wing in Ground Effect, *School of Mechanical and Manufacturing Engineering*, **2009**.

[9] **Agathangelou, B. and Gascoyne, M.**: Aerodynamic Considerations of a Formula OneRacing Car, SAE 980399, **1998**.

[10] https://www.sharcnet.ca/Software/Ansys/16.0/enus/help/flu_ug/flu_ ug_mesh_quality.html, (accessed **2016**).

[11] http://www.cfd-online.com/Wiki/Dimensionless_wall_distance_%28y_plus%29, (accessed **2016**).

[12] Tufast Eco car, https://www.delo-adhesives.com/fileadmin/ _processed_/csm_2015_12_tufast_2_ b173a1d2ae.jpg

[13] Pac car. http://www.paccar.ethz.ch/pac_car_nogaro5830.jpg

[14] Microjoule car, http://tylkonauka.pl/sites/default/files/microjoule2.jpg

[15] Ted Asso car, https://scontent-waw1-1.xx.fbcdn.net/hphotos-xlp1/t31.0-8/12419074_1696843093885845_1514783238139333200_o.jpg

[16] Eco-Arrow car, https://scontent-waw1-1.xx.fbcdn.net/hphotos-xaf1/t31.0-8/11411808_935806526476768_7569419952882660466_o.jpg