DETERMINATION OF THE DRAG COEFFICIENT HIGH PERFORMANCE ELECTRIC VEHICLE

Wojciech Danek

Department of Theoretical and Applied Mechanics, Silesian University of Technology
wojciech.danek@polsl.pl

Summary

In the article there is presented the process of creating a numerical model aimed at analyzing the aerodynamic properties of a wheeled vehicle MuSHELLka involved in the cycle of the Shell Eco Marathon competition. This analysis was carried out in ANSYS CFX software and it was aimed at determining the drag coefficient of the vehicle and detecting places that produced large flow disturbances.

Keywords: aerodynamics, computational fluid dynamics

WYZNACZANIE WSPÓŁCZYNNIKA OPORU AERODYNAMICZNEGO POJAZDU ELEKTRYCZNEGO O DUŻEJ SPRAWNOŚCI

Streszczenie

W artykule przedstawiony został proces wykonywania modelu numerycznego przeznaczonego do wyznaczania własności aerodynamicznych pojazdu kołowego MuSHELLka biorącego udział w cyklu zawodów Shell Eco Maraton. Analizy numeryczne zostały przeprowadzone w oprogramowaniu ANSYS CFX i były zorientowane na wyznaczenie współczynnika oporu aerodynamicznego pojazdu oraz określenie miejsc, które generują największe zakłócenia przepływu.

Słowa kluczowe: Aerodynamika, komputerowa mechanika płynów

1. INTRODUCTION

Aerodynamic research has become increasingly important in the process of car designing. This is due to the desire for reduce the resistance force of the vehicle that will lead to lower fuel consumption, which price is constantly growing. In conjunction with this fact arose the idea of creating Shell Eco Marathon competition, which promoted highly efficient vehicles. One of these vehicles was created in the student scientific association Modeling of Mechanical Design which one in 2012 took 10th place passing 425km/kWh. In order to improve this results, it was decided to identify aerodynamic characteristics of the vehicle and to change the line of its shape.

Fig. 1. Vehicle MuSHELLka

For this purpose there was created a numerical model that accurately reflects the actual state. On the basis of the resulting there were defined places that can be improved in order to achieve better results for the next year competition.

2. AERODYNAMIC FORCES

The aerodynamics of the vehicle is mostly presented by two forces describing the dynamics of the model in the air. This is the lift force and the drag force [1,3].

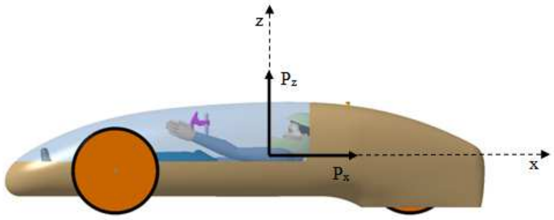

Fig. 2. Aerodynamic forces

Lift force is the force that acts in the perpendicular direction to the velocity vector. For negative values of the lift force there could be discussed the pressing force.

$$P_z = \frac{1}{2} * \rho * S * v^2 * c_z \quad (1)$$

Drag force is the force that is applied along the velocity vector. It is the vast majority of the resistance forces in the vehicle motion for speed above 16m/s.

$$P_x = \frac{1}{2} * \rho * S * v^2 * c_x \quad (2)$$

where:
ρ- air density,
S- the area of the object,
v- velocity,
c_z- dimensionless lift coefficient,
c_x- dimension drag coefficient.

3. POSITION OF THE VEHICLE RELATIVE TO THE ROAD

To obtain real conditions that prevail during the drive, it is necessary to determine the direction of the wind. Unfortunately, this is very difficult to determine. On the basis of the parameters of the wind in the period of one year there can be concluded that the research for sideslip angle in the research of aerodynamics of vehicles can be reduced to 14% [2,3].

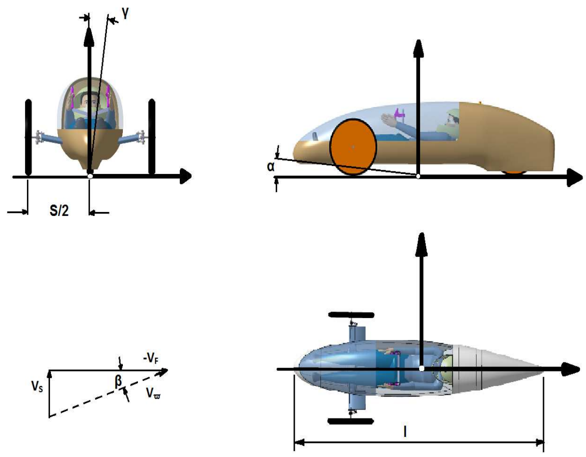

Fig. 3. Position of the vehicle relative to the road

Analyses were carried out for sideslip angles in the range of 0 to 20 degrees, so in order to determine the value of the drag coefficient it was necessary to establish the surface where the force acted. For this purpose, the model was created with the software Inventor where you could determine the surface area of the vehicle for different values of sideslip angle β.

Table 1. The values of surface area depending on sideslip angle β

Value of sideslip angle β	Surface area [m^2]	graphical illustration
0 ˚	0,297	
5 ˚	0,337	
10 ˚	0,405	
15 ˚	0,439	
20 ˚	0,539	

4. THE PHYSICAL MODEL

CAD model that was used to carry out the numerical analysis was developed in cooperation with student scientific association “Modeling of Mechanical Design”. It was made based on surface modeling technique in CATIA software. This technique was used due to complicated geometry of the car.

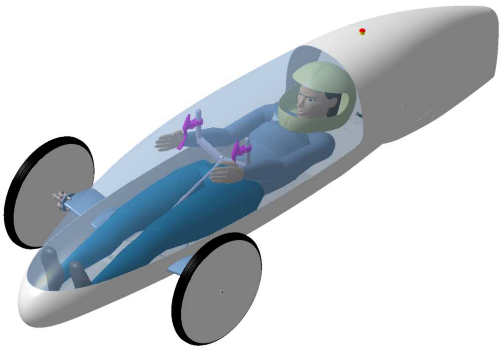

Fig. 4. Geometry of the vehicle made in CATIA software

For such a geometry some simplification were adopted. They involved removal of the elements inside the car due to the fact that they did not affect the results of the analyses, as well as they did not omit certain details that had a negligible impact on the results of numerical simulations such as screws or wires.

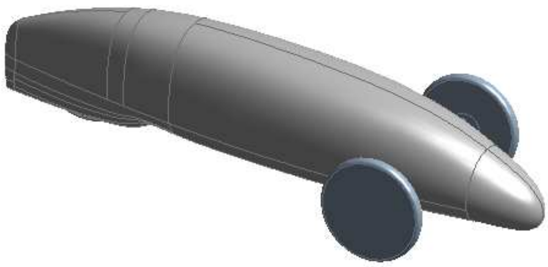

FIG. 5. Simplified vehicle geometry

For such a simplified model which uses boolean operations there was generated a model of the wind tunnel where the vehicle MUSHELLka was located [2,3].

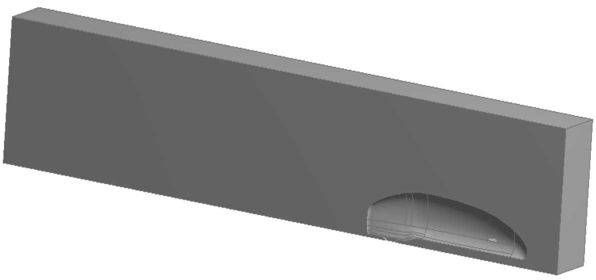

Fig. 6. form of the wind tunnel with the marked surface of the vehicle

5. NUMERICAL MODEL

5.1 FINITE ELEMENT MESH

For thus created geometry it was applied finite element mesh that was compatible with the CFX module. For this purpose, the geometry was divided into sections that were selected to facilitate the work of the original model of mesh compaction and the inflation layer addition.

For the section corresponding to the geometry of the car it was made the mesh refinement. The size of finite elements on the surface of the car was 0.006 [m]. However, for the geometry of a simulated wind tunnel, the size of finite elements was equal to 0.04 [m]. For thus, to the mesh that was created it was added the inflation layer consisting of 20 layers with the grown rate that was equal 1.2 [4,5].

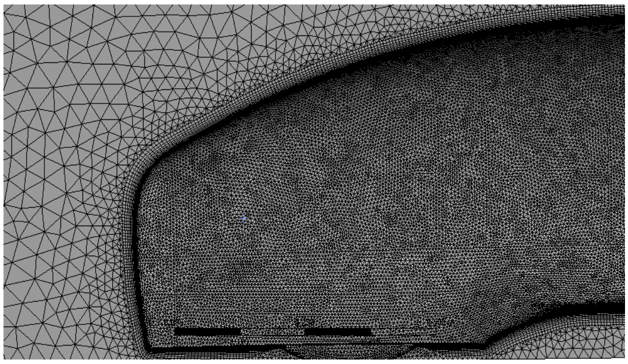

Fig. 7. Finite element mesh on the surface of the vehicle with the inflation layer

5.2. BOUNDARY CONDITIONS

The next step was to impose boundary conditions on the specified geometry. This was the most important stage of preparing the model for analysis. In case of incorrect imposition of the boundary conditions the results would not be reliable.

For this purpose the fluid model was chosen for the computation:

- air at 25 ˚ C and a pressure of one atmosphere,
- isothermal heat transfer,
- turbulent flow model according to the k-Epsilon.

and the area in which they were identified:

- air inlet,
- air outlet,
- symmetry,
- vehicle surface.

First, boundary conditions were imposed air inlet. It was determined velocity the air inlet, which was used in the analyses. In this case, two analyses were performed two different speeds. One equal to 25 km / h, which was the minimum average speed of a vehicle that had achieved during the race and the other equal to 35 km / h, which was the maximum speed of which vehicles move the track. Another parameter specified in that condition was the turbulence factor, which was equal to 5%.

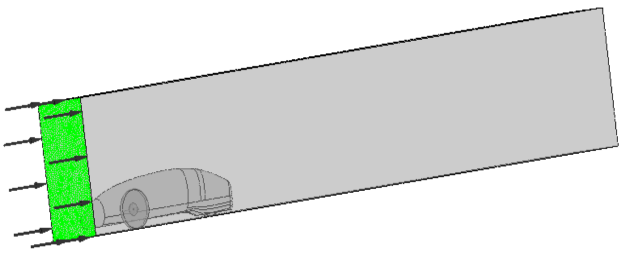

Fig. 8. Surface of the air inlet

Air outlet was another boundary condition that was imposed. In this case, it was necessary to determine the relative pressure prevailing in the outlet. For this analysis there was relative pressure equal to 0 adopted.

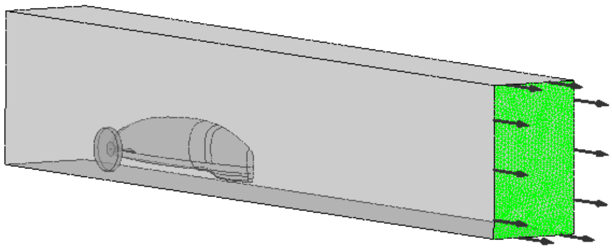

Fig. 9. Surface of the air outlet

In order to determine the value of the drag force on the surface of the car, it has been imposed the condition on the surface of the wall of the analyzed model. This was possible thanks to earlier forms of the division of wind tunnel at the appropriate parts.

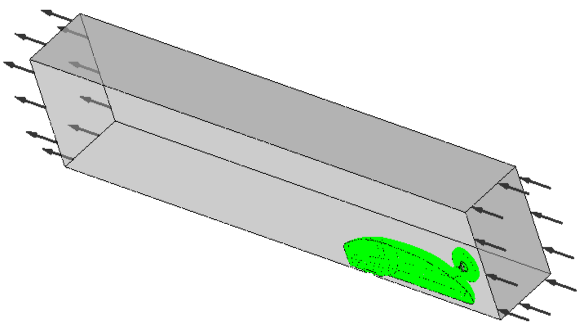

Fig. 10. Surface of the vehicle

Thanks to the fact that the tested geometry was symmetrical, there could be applied to the additional condition of symmetry. Therefore, for further calculations it was tested only on half of the model. This made it possible to reduce the simulation time and better thicken the finite element mesh around the test object.

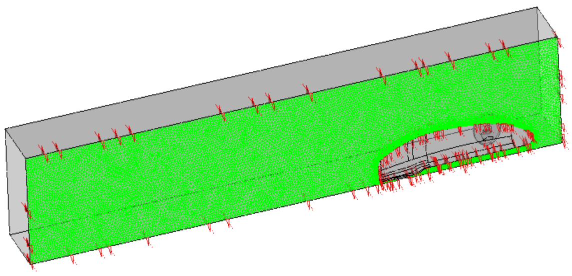

Fig. 11. Surface symmetry

6. RESULTS OF NUMERICAL ANALYSIS

For so created a numerical model has been designated the drag forces for different sideslip angles. On the basis the surface areas shown in section 3 designated the values of the drag coefficient in the range of sideslip angle between 0 and 20 degrees. The results obtained from drag coefficient were used in a further step to create a strategy for driving the vehicle during the competition Shell Eco Marathon. The main objective of this analysis was to determine the places that brought the greatest disturbance of flow and caused increase the Cx coefficient, allowing improvement of the results obtained in previous competitions [2,4,5].

6.1 RESULTS FOR VELOCITY EQUAL 25 KM/H

The values of the drag force and the drag coefficient are shown in the table below.

Table 2. C_x-values for different sideslip angle β

Sideslip angle β	Force [N]	Drag coefficient
0 ˚	2,1546	0,255
5 ˚	2,3836	0,2497
10 ˚	2,9116	0,253
15 ˚	3,5096	0,281
20 ˚	4,2034	0,274

Graphic illustration of this data has been shown on the graph.

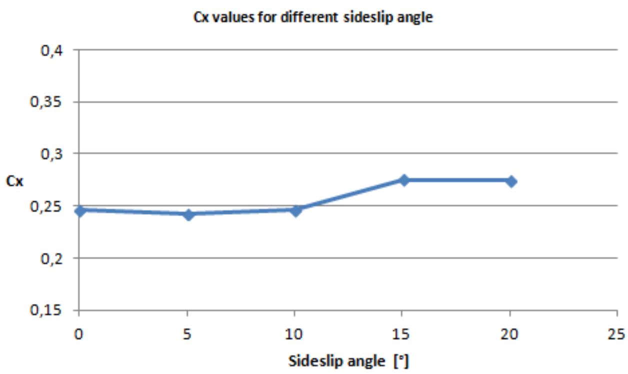

Fig. 12. C_x-values for different sideslip angle β

In addition to the drag force it was obtained the pressure distribution on the surface of the vehicle, the lines of air flow around and the plane of velocity. This enabled the determination of the most loaded places and the places where there was significant of the flow.

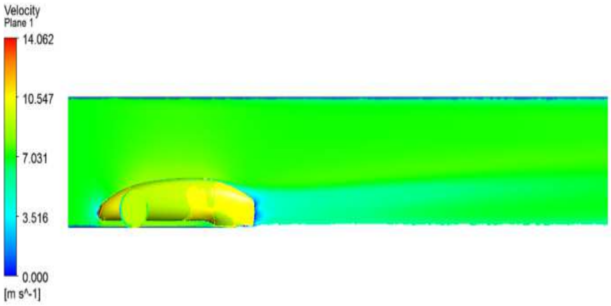

Fig. 13. Velocity plane

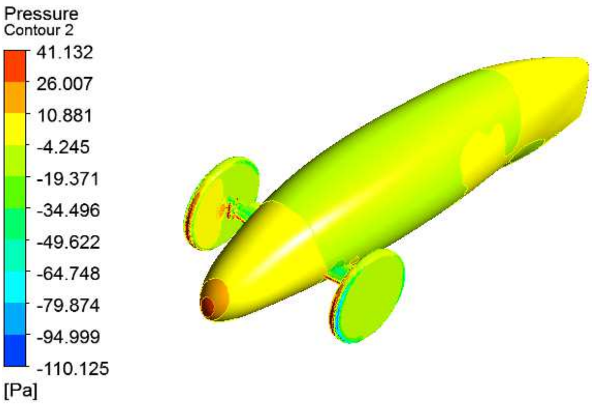

Fig. 14. Pressure distribution on the surface of the vehicle

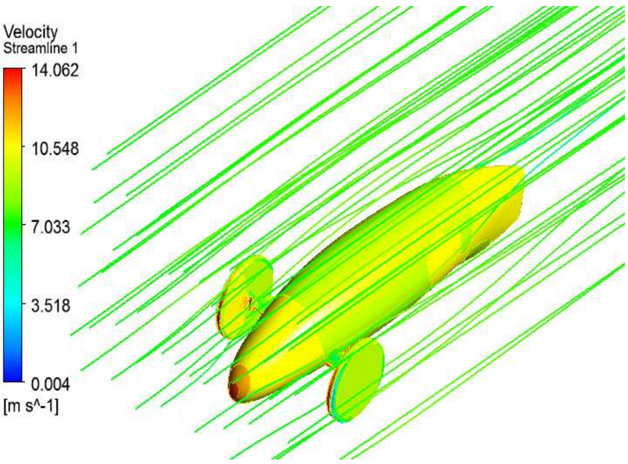

Fig. 15. Lines of the air flow around the vehicle

Analysing the data obtained on the basis of a numerical analysis there can be determined the places where there are the greatest flow disturbances and the location of the highest pressure on the surface of the vehicle. In places such as the front wheels and front beam, and therefore further consideration of these elements should be taken into account

6.2 RESULTS FOR VELOCITY EQUAL 35 KM/H

The values of the drag force and the drag coefficient are shown in the table below.

Table 3. C_x-values for different sideslip angle β

Sideslip angle β	Force [N]	Drag coefficient
0 °	4,2346	0,246
5 °	4,75	0,243
10 °	5,803	0,2474
15 °	7,0128	0,275
20 °	8,5784	0,274

Graphic illustration of these data has been shown on the graph.

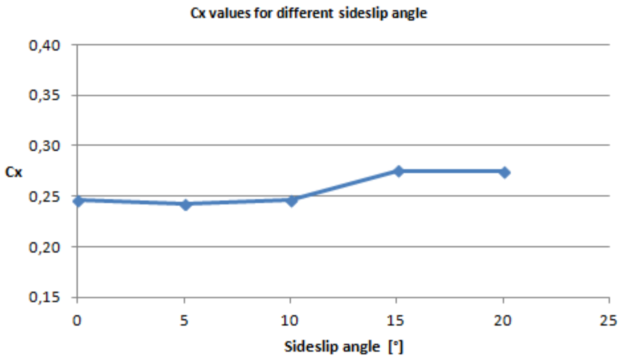

Fig. 16. C_x-values for different sideslip angle β

In addition to the drag force there was obtained the pressure distribution on the surface of the vehicle, the lines of air flow around and the plane of velocity. This enabled the determination of the most loaded places and the places where was significant of the flow.

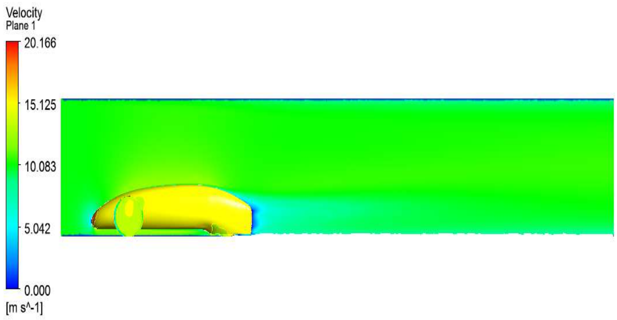

Fig. 17. Velocity plane

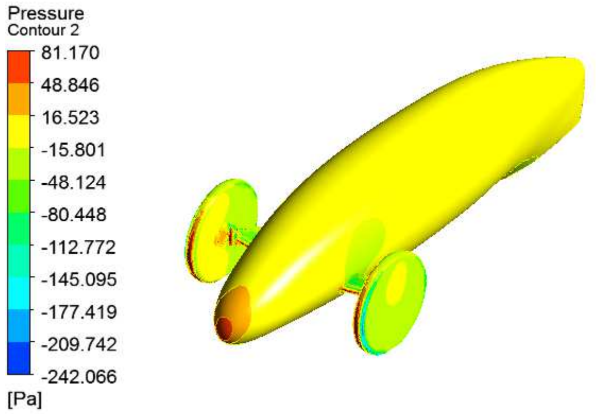

Fig. 17. Pressure distribution on the surface of the vehicle

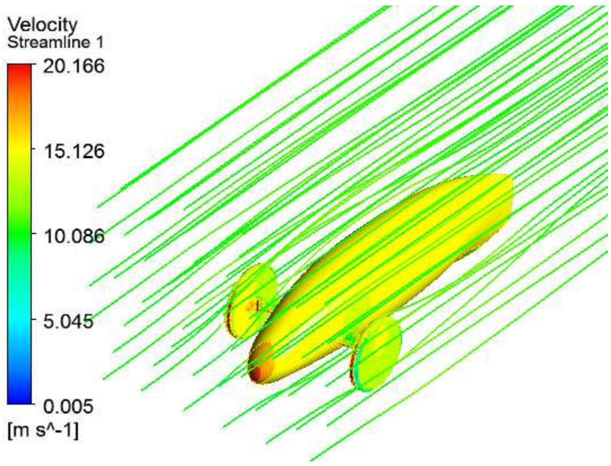

Fig. 18. Lines of the air flow around the vehicle

Analyzing the data obtained on the basis of numerical analysis you can be able to confirm the results obtained for a velocity equal 25 km / h, so the next step will be an attempt to eliminate the places where there are the greatest flow disturbances and places where there is the highest pressure.

7. CONCLUSIONS

With the numerical analysis it was possible to determine the aerodynamic properties of a wheeled vehicle MuSHELLka without the expensive tests in the wind tunnel. So there are pre-determined places that need improvement in order to achieve better results in the next year competition Shell Eco Marathon. These places were the front wheels and the front beam, which introduced the largest flow disturbances based on analyses made, it was found that the value of the drag coefficient C_x does not depend on the velocity of the air inlet, but only on the geometry of the tested vehicle. It was also concluded stability of drag coefficient for the entire range of sideslip angles.

References

1. Łanecka-Makaruk W., Łucjanek W.: Mechanika lotu. Warszawa: WKŁ, 1966.
2. Wolf-Heinrich Hucho: Aerodynamika samochodu: od mechaniki przepływu do budowy pojazdu. Warszawa: WKŁ, 1988.
3. Piechna J.: Podstawy aerodynamiki pojazdów. Warszawa: WKŁ, 2000.
4. Anderson J. D.: Computational fluid dynamics: the basics with applications. New York: McGraw-Hill, Inc, 1995.
5. Ferziger J. H., Perić M.: Computational methods for fluid dynamics. New York: Springer, 2002.