The Study of the Aerodynamic Behavior of an Electric Kart Using CAD and CFD Methods

Constantin Cioroianu[1], Ion Tabacu[2], and Adrian Iorga[3]

[1]DPR Draexlmaier Procese de Productie, Pitesti, Romania, [2]University of Pitesti, Romania,
[3]Engie Romania, Pitesti, Romania

`constatin.cioroianu@yahoo.com, ion.tabacu@upit.ro,`
`adyiorga@yahoo.com`

Abstract. This work presents the aspects of modelling, simulation and results using CAD (Computer Aided Design) and CFD (Computational Fluid Dynamics) methods. This simulation was applied to an electric powered kart developed within the research center "Automotive Engineering" of the University of Pitesti. Based on the existing kart, a geometrical model was developed using Catia V5 software. The respective model was used to simulate the aerodynamic behavior with the help of Ansys Fluent software.

Keywords: electric kart; aerodynamics.

1 Introduction

In this paper it is presented a model for studying the aerodynamic behavior of an electric kart. In comparison with a petrol powered kart, the electric kart has batteries placed between the driver's seat and the side bumpers. For the studied case, presented in Fig. 1 and Fig. 2, due to their flat surface, the batteries could have a negative impact on the karts' aerodynamic performance. An electric kart was chosen for this study due to the lack of air and sound pollution.

If a large flat plate is inserted into an airstream at right angles to the flow, then air particles moving in the column of air near the center would be decelerated to a stop at the plate and all of their kinetic energy would be converted to an increase in static pressure. This increase in static pressure is called the "stop" pressure, numerically equal to the dynamic pressure [1].

International Congress of Automotive and Transport Engineering, CONAT 2016
Transilvania University of Brasov, 2016
ISSN 2069-0401

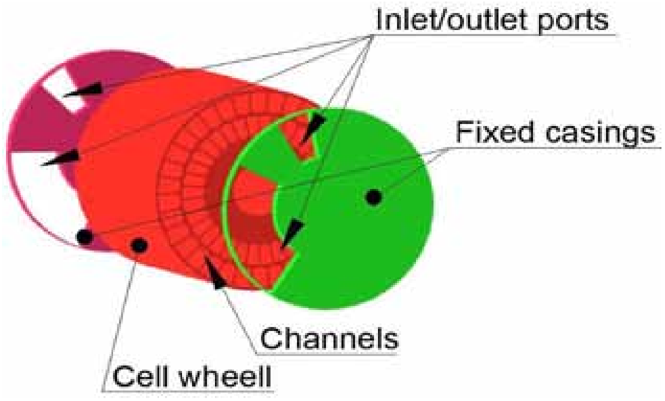

Fig. 1. Dynamic or "stop" pressure, "q" [1]

2 Methodology

In the first stage, a CAD model was created using CATIA V5 software. The numerical model is generated from the geometrical model. After generating the numerical model, the aerodynamic behavior is simulated by CFD, with the help of ANSYS FLUENT software.

2.1 Geometric Model

With the sets of commands and functions available through CATIA V5 a geometric model of the kart and driver was generated. The geometric model corresponds to a medium-sized individual with a height of 1750 mm.

The generated CAD model and components are shown in Fig. 2.

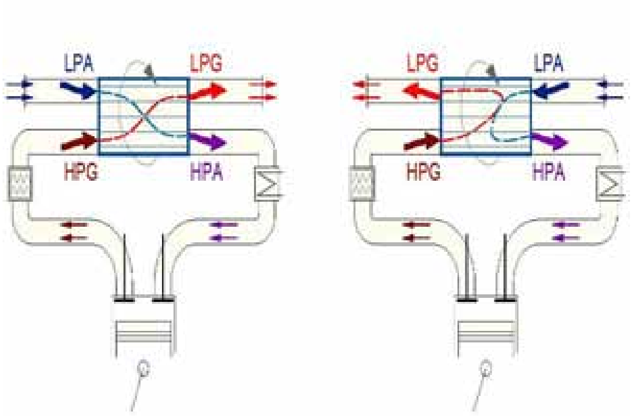

Fig. 2. Geometrical model and components

This model was used taking into account the dimensions of the kart and the anthropometric dimensions of the users of this kart. In CATIA V5, model type dummies are used for evaluation of ergonomics performance of various technological operations. A simplified model of the driver will reduce the necessary computational resources. Therefore, an ellipsoid model was developed and it corresponds to a medium-sized individual with a height of 1750 mm.

2.2 Numerical Model

The computational grid (Fig. 3) is defined by taking the geometrical model as a carrier defined above. The numerical model defines the computing grid necessary for numerical solving of the set equations describing the fluid flow. At this stage, the boundary conditions are defined (wall, interaction area, entrance area, output area) and the conditions necessary to establish the set of equations describing fluid flow are also created.

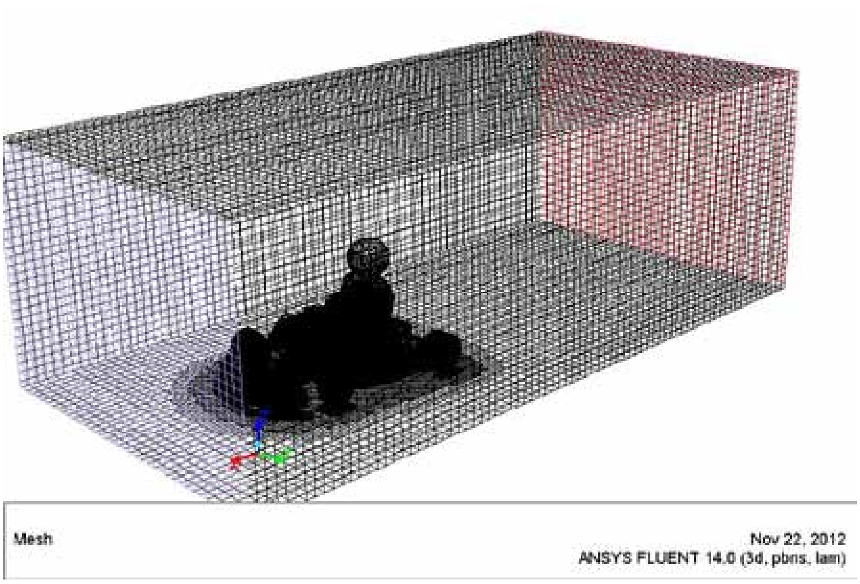

Fig. 3. Computational grid

The geometric model of the electric kart and driver represents the structure – defined as a wall – around which airflow is studied. The outer faces of the geometric body representing air are defined as the borders of the field of computing. These borders can be associated with different properties such as wall, input or exit. The numerical values that are initialized to these areas are the air flow rate that gives the speed of the vehicle and pressure. If the output value of the surface pressure is equal to atmospheric pressure thereby defining a free flow conditions. Solving numerical model provides the user with a series of results necessary for evaluating the aerodynamic performance.

2.3 Conditions of simulation and turbulence model

The influence of the ground on the main aerodynamic characteristics of the car, drag and lift, is studied in two ways, commonly used in wind tunnels, respectively without ground effect (fixed wheels and no relative motion between car and road), and with the moving wall approach [2]. This study is carried out considering fixed wheels and no relative motion between kart and road for three velocities of 15, 20 and 25 m/s. Analyses were performed in steady state, for a reference pressure of the air p=1 At. The SST (Shear Stress Transport) turbulence model is used to solve the simulation process.

3 Results

The pressure diagram plots the pressure values obtained by solving the numerical model. Information about total, static and dynamic pressure and pressure coefficient is available.

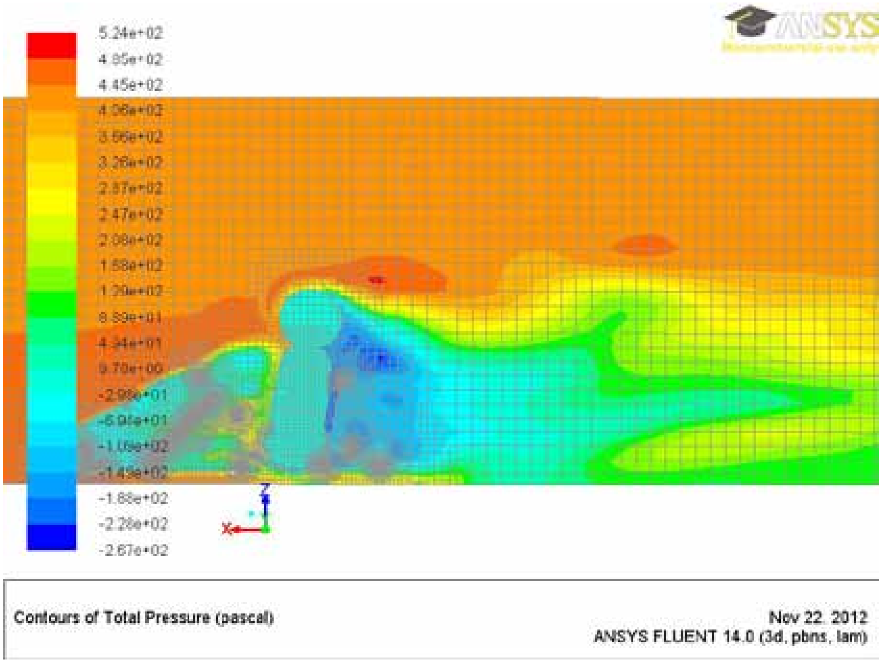

Fig. 4. Numerical results of total pressure (velocity 25 m/s)

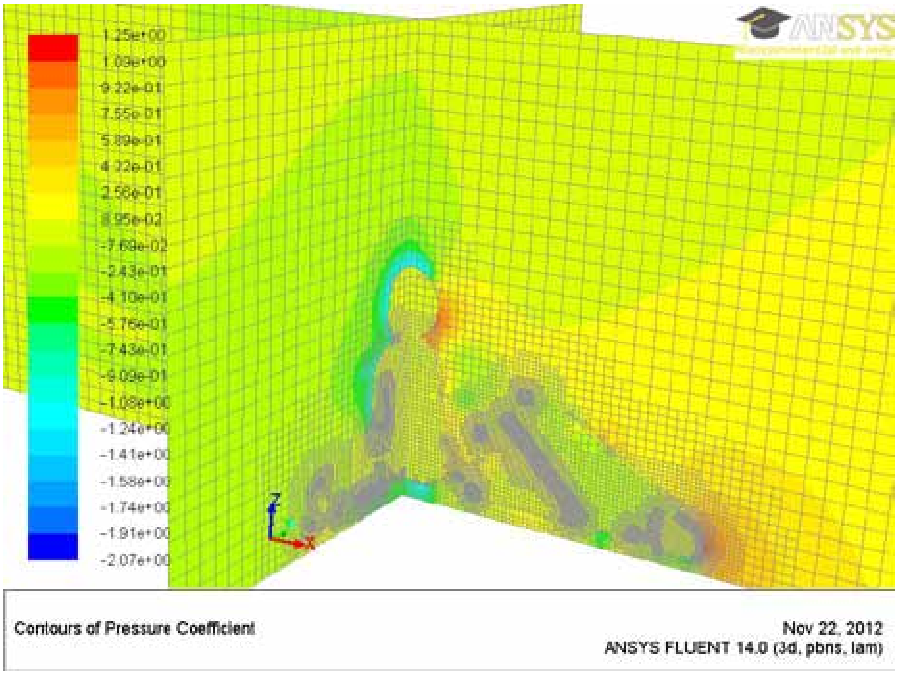

Fig. 5. Numeric results of the pressure coefficient

Figure 4 shows the variation of *total pressure* for the studied kart. Figure 5 shows the calculated value of the pressure coefficient.

4 Contribution of the Kart's Components in Generating Drag

In order to evaluate the contribution of the kart's components to the aerodynamic force, a set of 3 numerical tests were made at speeds of v¥=15 m/s, 20 m/s and 25 m/s. Table 1, Table 2 and Table 3 show the results of calculated aerodynamic drag forces that each component generates.

Table 1. Calculated aerodynamic drag forces for fluid speed v=15 m/s

Zone	Pressure	Viscous	Total
Front bumper	-8.564	-0.047	-8.611
Steering column protection	-4.549	-0.011	-4.560
Chassis	-3.627	-0.050	-3.677
Frame (front protection)	-0.006	0.000	-0.006
Right front wheel	-0.811	-0.020	-0.831
Left front wheel	-0.810	-0.021	-0.830

Steering column	0.080	0.001	0.081
Steering wheel	-0.812	-0.004	-0.816
Floor panel	-0.100	-0.012	-0.111
Driver	-22.050	-0.051	-22.101
Seat	-1.274	-0.005	-1.279
Right batteries	-7.910	-0.012	-7.922
Left batteries	-9.096	-0.013	-9.109
Side left protection	-0.866	-0.041	-0.907
Side right protection	-2.027	-0.046	-2.073
Engine	-3.947	-0.003	-3.950
Rear axle (assembled)	-8.321	-0.055	-8.376
Net	-74.689	-0.391	-75.079

Table 2. Calculated aerodynamic drag forces for fluid speed v=20 m/s

Zone	Pressure	Viscous	Total
Front bumper	-15.396	-0.063	-15.458
Steering column protection	-8.328	-0.015	-8.343
Chassis	-7.441	-0.068	-7.509
Frame (front protection)	-0.072	-0.001	-0.074
Right front wheel	-0.736	-0.027	-0.763
Left front wheel	-1.401	-0.025	-1.425
Steering column	0.049	0.000	0.049
Steering wheel	-2.144	-0.005	-2.148
Floor panel	-0.170	-0.017	-0.186
Driver	-38.546	-0.067	-38.613
Seat	-3.408	-0.006	-3.414
Right batteries	-14.208	-0.017	-14.225
Left batteries	-15.720	-0.018	-15.739
Side left protection	-3.926	-0.061	-3.987
Side right protection	-2.637	-0.044	-2.682
Engine	-4.793	-0.005	-4.798
Rear axle (assembled)	-14.728	-0.071	-14.799
Net	-133.605	-0.510	-134.115

Table 3. Calculated aerodynamic drag forces for fluid speed v=25 m/s

Zone	Pressure	Viscous	Total
Front bumper	-24.04	-0.08	-24.11
Steering column protection	-11.86	-0.02	-11.88
Chassis	-10.12	-0.07	-10.19
Frame (front protection)	-0.09	0.00	-0.09
Right front wheel	-2.33	-0.03	-2.36
Left front wheel	-3.39	-0.03	-3.42
Steering column	0.34	0.00	0.34
Steering wheel	-1.44	-0.01	-1.45
Floor panel	-0.26	-0.01	-0.28
Driver	-70.30	-0.08	-70.38
Seat	3.28	-0.01	3.27
Right batteries	-22.65	-0.02	-22.67
Left batteries	-23.87	-0.02	-23.89
Side left protection	-4.17	-0.08	-4.25
Side right protection	-4.64	-0.06	-4.69
Engine	-6.37	-0.01	-6.37
Rear axle (assembled)	-21.03	-0.09	-21.12
Net	-202.94	-0.61	-203.54

In order to highlight the results, the figure 6 shows, in percentages, the geometric elements with significant contribution in generating the aerodynamic drag force.

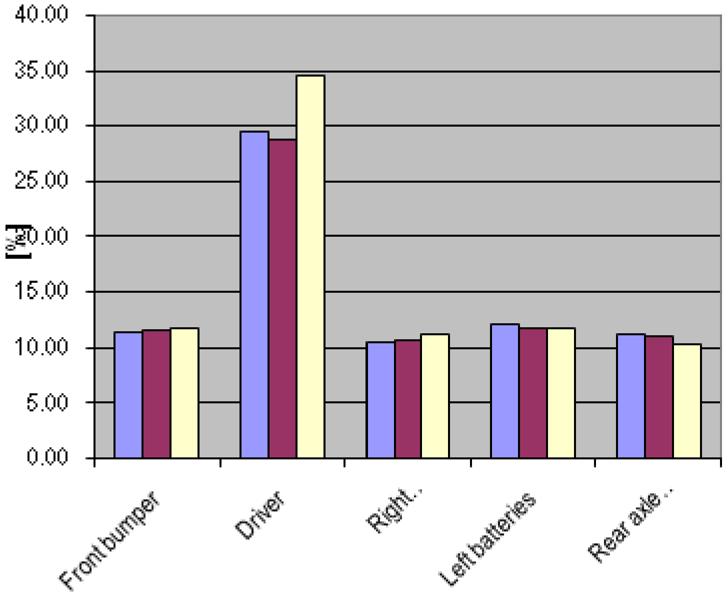

Fig. 6. Geometric elements with significant contribution in generating aerodynamic drag force

5 Conclusions

Analyzing the results presented in Figure 6 one can see that within reasonable limits of + / -2% the batteries have about 25% contribution in generating drag at various speeds. It is quite significant and it is a negative impact on aerodynamic behavior. This happens due to the front vertical flat surface of the batteries. The problem may solved by placing wind deflectors in front batteries or use protective covers with a shape that would have less contribution in generating drag. Another possibility would be to place the batteries in the middle of the kart, between the steering column and the seat.

This computational model allows further investigation of the influence of driver size and can be used to optimize vehicle outer shape for improved aerodynamics.

References

1. Milliken W. F., Milliken, D. L. (1995) *Racecar vehicle dynamics.* Society of Automotive Engineers Inc., Warrendale 3:92
2. Huminic A., Huminic G., *CFD Investigations of an Open – Wheel Race Car.* EASC 2009.