PROCEEDINGS OF THE INSTITUTE OF VEHICLES 3(99)/2014

Mateusz Wąsik[1], Mirosław Targosz[2], Wawrzyniec Panfil[3]

METHODOLOGY OF AERODYNAMIC ANALYSIS IN THE HYPERWORKS SOFTWARE OF THE CARS PARTICIPATING IN THE SHELL ECO-MARATHON RACE

1. Introduction

Numerical computational aerodynamic studies represent a significant part in the design and optimization of car bodies. Research carried out using numerical methods [5] [6] are relatively fast and inexpensive compared with tunnel research. Based on numerical methods of calculation it is possible to simulate any ambient conditions, referred to as boundary conditions, and visualize the distribution of parameter values, as desired, for example, the pressure distribution on the surface of the vehicle or the air speed distribution in the plane coinciding with the plane of symmetry of the car. Numerical investigations [5] [6] provide high accuracy data, depending on the developed numerical model. Their results can be compared with ongoing tunnel research on the physical model. This type of research using virtual models created in a CAD environment eliminates necessity to have physical model of the object. This approach makes it possible in the early stages of the body panel development to determine the right direction of its development and to optimize its shape for the least aerodynamic drag. Doing so significantly reduces the cost of developing the prototype. Optimisation of vehicle silhouette due to the aerodynamics significantly reduces the air resistance. It is extremely important, as it allows a significant reduction in the size of the required power units, which is associated with a reduction in vehicle weight and the energy absorbed by the power unit. Due to the continually rising fuel prices, as well as environmental considerations in the current trend of the development of the automotive industry, vehicle weight reduction and energy consumption are very important issues.

The aim of the study was to develop a methodology for aerodynamic analysis, prepare appropriate numerical models, determine drag forces and present the concept of development of a high-performance, eco-urban car Bytel designed to take part in the competition Shell Eco-marathon 2014 to 2015. The studies are part of a work carried-out by Smart Power Urban racing team from Silesian University of Technology. CAD models of the car and any modifications have been made in the software Catia V5. The numerical model was created in the software Altair HyperWorks using CFD module and the postprocessor AcuSolve. During the study there were developed methods for conducting aerodynamics research of this type of vehicle. The research ends with the numerical verification of the drag force of four models of the vehicle made for-Smart Power Urban.

[1] Eng. Mateusz Wąsik, Institute of Fundamentals of Machinery Design, Silesian University of Technology
[2] Eng. Mirosław Targosz, BSc, Institute of Fundamentals of Machinery Design, Silesian University of Technology
[3] Eng. Wawrzyniec Panfil, PhD, Institute of Fundamentals of Machinery Design, Silesian University of Technology

2. Theoretical description of the studied phenomena

In fluid mechanics, there are different flow distributions due to different criteria. Important for the aerodynamics is the division of flow due to the motion of fluid particles [1]. Consistent with this criterion the flow may be laminar or turbulent. Laminar flow [1] is a stratified flow, where the fluid flow creates a parallel, smooth layer. Depending on the shape of the walls of the object on which the fluid passes, the flow lines are straight or gently curved. The flow of this type is for small and medium speed for contractual Reynolds numbers less than 2300. Turbulent flow [1] is characterized by the presence of time-varying flow disturbances, so-called turbulences, manifesting with blenders fluid particles. The characteristics of the flow are variable in time and space. Turbulent flow occurs at high flow velocities Reynolds numbers above 2300.

On the car in motion in air act aerodynamic forces [2]. Aerodynamic force acting on the car has several components. The components of the aerodynamic force relevant to the issues to be analyzed are components acting along the longitudinal axis of the vehicle (so-called drag force) P_x (1) and the lift P_z (2).

$$P_x = \frac{1}{2} * \rho * S * v^2 * c_x \tag{1}$$

$$P_z = \frac{1}{2} * \rho * S * v^2 * c_z \tag{2}$$

where:

ρ is the air density, S the surface area of the object, v the speed, c_x coefficient of drag forces.

Other forces and moments acting on the car in motion are the lateral force Py, the pitching moment My, heeling moment Mx, moment deflection Mz.

Aerodynamic resistance [3] is a movement counter force. Natural system of the test problems is the system of a vehicle. However, it is preferable to adopt analysis system associated with the axles of the vehicle, since the vehicle is at rest and moving air. A very important factor for consideration for car aerodynamic drag is a dimensionless drag coefficient c_x [7]. It is associated with the shape of the body. The greater is the drag coefficient, the greater is the aerodynamic drag of the vehicle. c_x coefficient is determined by measuring analytically drag forces. Boundary layer [8] is a layer of medium flowing in a short distance from the surface of the test object. In the boundary layer, there are significant changes in speed. On the surface of the object the speed of of medium flowing over the test object is equal to zero, and increases to the set speed with the distance from the surface. The phenomenon of the boundary layer is related to the fluid viscosity. It is very difficult to determine unambiguously the thickness of the boundary layer. It is assumed that the value of the boundary is the place where the speed is 99% of the set speed.

3. Methodology of numerical research

Methods of numerical research (Fig. 1) of the urban car Bytel have been developed on the example of the concept Bytel v1. The numerical model was created in the software HyperWorks HyperMesh using CFD module. Vehicle aerodynamic analysis process was performed in stages. In the first stage there has been created a CAD model in CATIA V5 software. Due to the symmetry of the model in a vertical longitudinal plane for further study was used only half of the model that was exported to STP format.

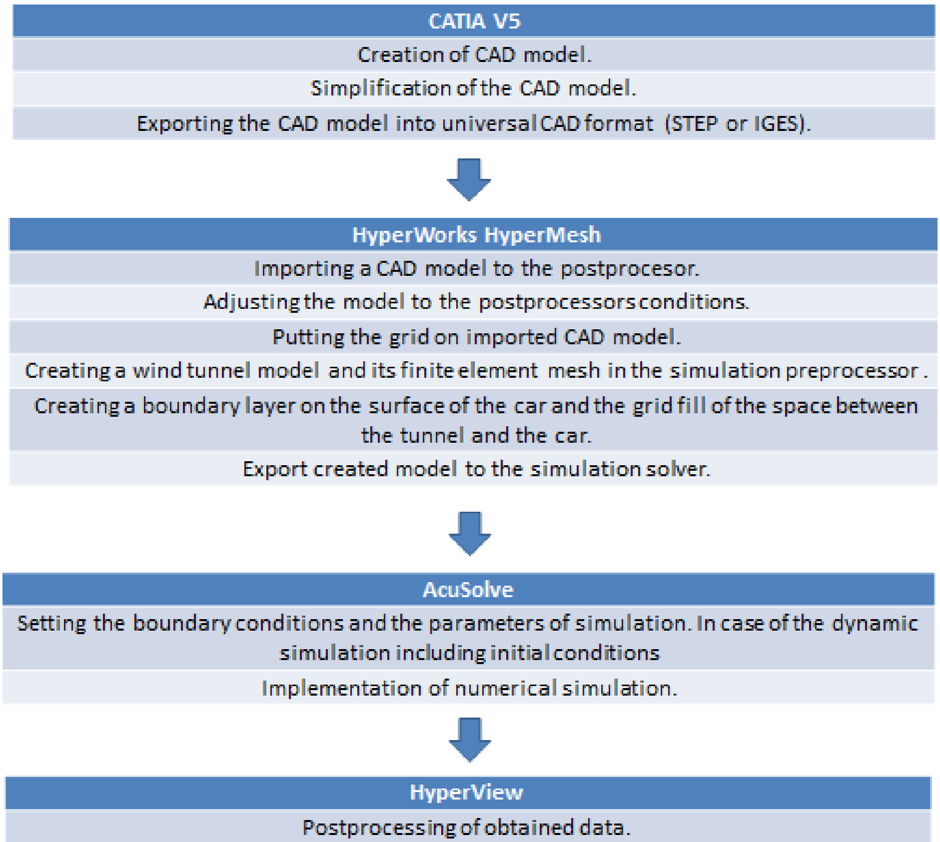

Fig. 1. Diagram showing the developed methodology of the research

The next step was to import the model into the software HyperWorks HyperMesh, which adapted the model to the simulation. Then the finite element mesh was applied, the wind tunnel was generated, the boundary layer for the car and the boundary conditions were determined. The mesh on the surface of the car is composed of triangular elements with a maximum size of 8 mm, the growth factor of 1.2, the maximum angle of 15 degrees of adhesion and coefficient of deviation equal to 50%. For a tunnel with a length of 15.5 m, a height of 2.5 m and a width of 1.2 m square elements are used with a maximum edge length of 40mm. So prepared the numerical model was exported to the solver AcuSolve, where the assignment of appropriate conditions for the simulation was made. The analysis of results was performed in the postprocessor HyperView. Simulations were performed for a medium flowing in the form of air, having a temperature of 25 ° C, a pressure of 1 atm and an initial speed of 10 m/s. As a model for the simulation of turbulence was used Spalart-Almaras model with turbulence coefficient equal to 5%. Selection of parameters has been carried out experimentally. As a baseline for the analysis of air flow in the wind tunnel default software parameters were adopted. Any deviation from the baseline parameters used in the analysis are the values that give adequate accuracy at the same time the demand for computing power. Simulation conditions were chosen according to the conditions during the Shell Eco-marathon race in Rotterdam and in accordance with generally accepted principles of research in the tunnel of the Institute of Aviation in Warsaw [2] [4]. In this study the resulting values were the distribution of the drag forces and pressure distribution on the surface of the vehicle. In addition, using HyperView software air distribution lines abounding vehicle have been obtained. On the basis of the obtained values of the aerodynamic drag force on the surface of the vehicle aerodynamic coefficient c_x was analytically calculated.

4. Analyzed models

There have been analyzed four models of the Bytel vehicle, called successively as v1, v2, v3 and v4. Version v1 (Fig. 2a) is the basic version subjected to testing. On the basis of the oncoming results further development versions of the car have been elaborated. First model was used for analysis of the open wheel wells. In the first modification of the underlying body marked as version v2 (Fig.2b) several significant changes have been made in relation to the conceptual version v1. The v2 version of the car has been extended, i.e. its rear part has been completely redesigned, including covering the rear wheel arches. The front part also has been remodeled including transition between the hood and windshield of the car and the shape of the pillars. These modifications have also contributed to the lateral outline of the car. The simulation was performed with covered front wheel arches.

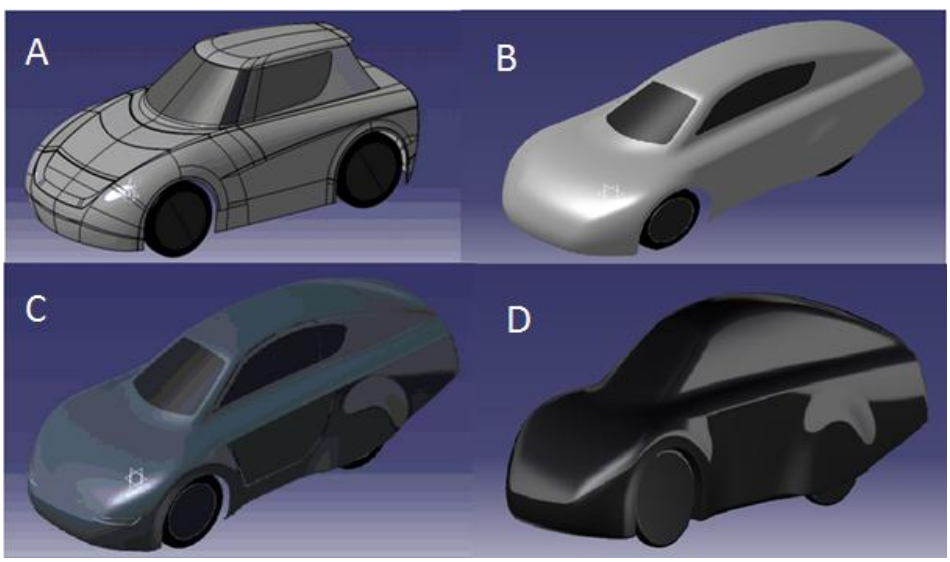

Fig. 2. CAD models of the urban car Bytel in the development versions a) v1 b) v2 c) v3 d) v4[1]

The development version v3 (Fig.2c) has increased with tears and rounded back of the body. In front of the car the width of pillars has been changed and the bottom of the bumper part rounded. The simulation was performed with a curtained front wheel arches. The development version marked as v4 (Fig. 2d) has rounded sidelines, undercutted the front part to increase its angle of inclination and modified pillars. The analysis was performed on the exposed front wheels.

5. The results of the verification

For the analyzes examined values are drag force P_x determined on the basis of computer simulation and drag coefficient c_x determined analytically for each of the models on the basis of the resulting drag forces. The following table (Tab. 1.) shows a comparison of the drag force and drag coefficient for each of the development version.

Table. 1 Comparison of drag force and drag coefficient c_x for each body style

The development version	Drag force [N]	Drag force coefficient c_x [x]
Bytel v1	21,66	0.349897
Bytel v2	19,8	0.31985
Bytel v3	18,57	0.299981
Bytel v4	19,78	0.319527

Basic v1 showed the greatest drag coefficient. The smallest drag coefficient has development version v3. As can be seen, the drag coefficient c_x decreases with subsequent development concepts. The exception is the development version v4, at which you will see an increase in the value of the drag coefficient from the previous development version, but you can see a slight drop in the resistance to the development version v2. Versions v3 and v4 have the best results. They were subjected to a detailed comparison. Versions v2 and v4 have similar drag coefficient c_x, however the analysis was carried out for v4 without simplification via a covered front wheel.

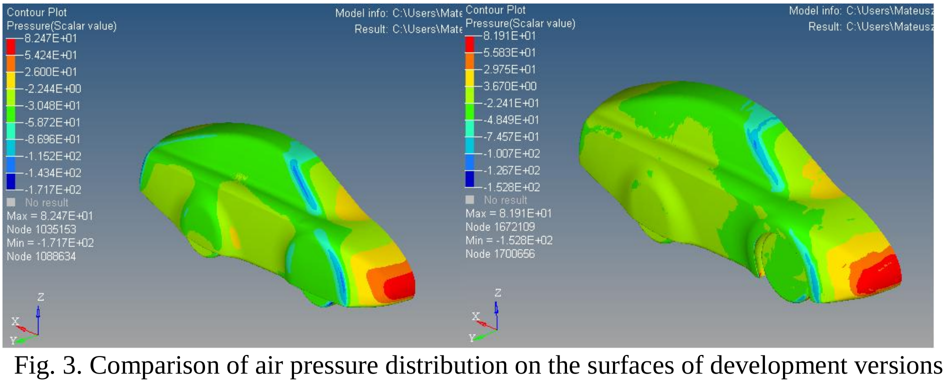

Fig. 3. Comparison of air pressure distribution on the surfaces of development versions v3 (pictured left) and v4 (pictured right)

Detailed comparison of development versions v3 and v4 shows that the second one may be considered as better, even with greater drag coefficient c_x. Version v4 has a favorable pressure distribution (Fig. 3) on the surface and has less air turbulence behind the car (Fig. 4) (Fig. 5). It is important to analyze the v4 with outdoor front wheel arch and v3 with covered wheel arches. Comparing these two studies it can be concluded that the model v4 with concealed front wheel arches obtained the value of c_x lower compared to v3. Number of the analyzes was limited with amount of time to complete the optimization model. Due to the time of calculations performed by the computer (approx. 12-48 hours depending on the model) and the time needed to create more CAD models only the concepts that dictate the right direction of development of the initial solid have been analyzed.

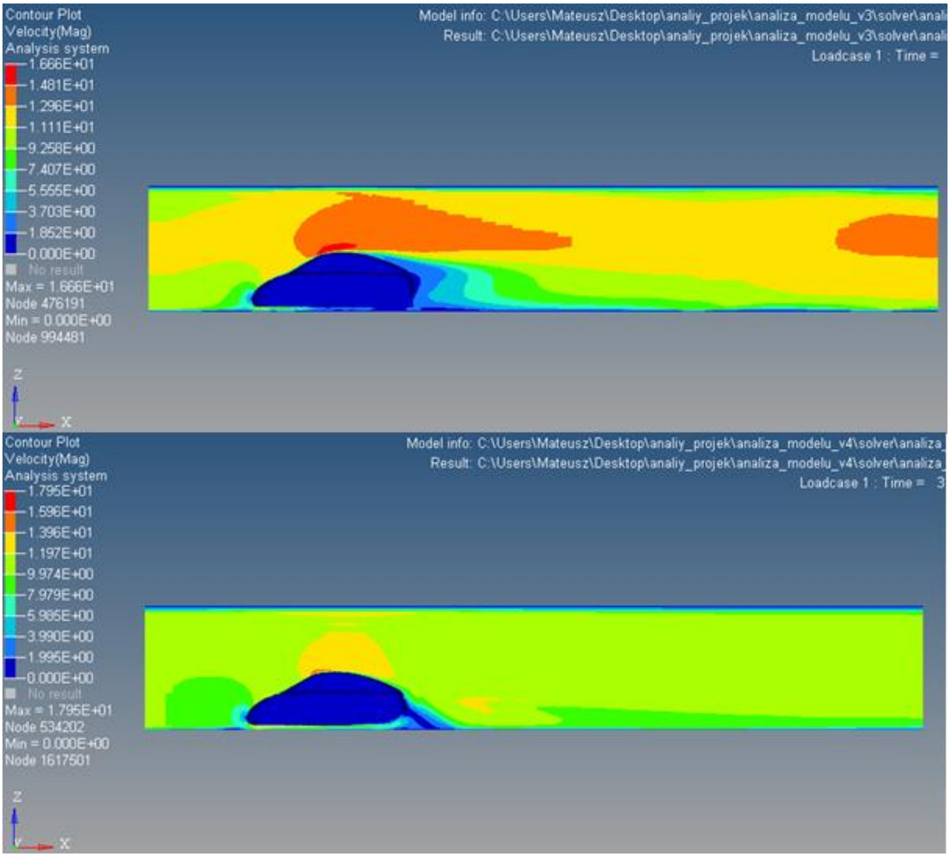

Fig. 4. Comparison of air velocity distribution in the plane coinciding with the plane of symmetry of the car. Development version v3 (pictured top) and v4 (pictured below)

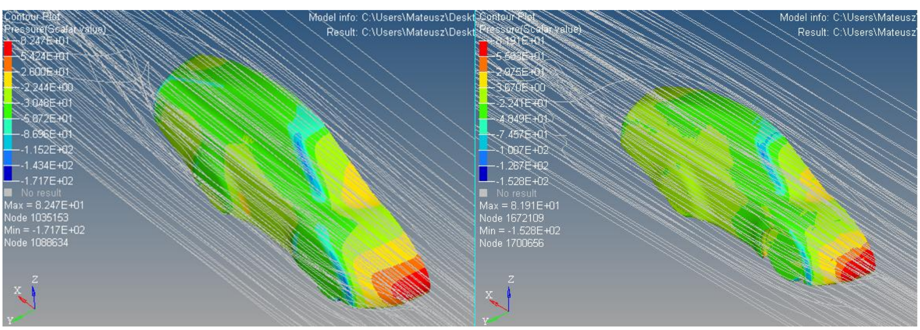

Fig. 5. A comparison of air streams amounting car. Development version v3 (pictured left) and v4 (pictured right)

6. Summary

The analyzes suggest that version v4 is considered as the most advantageous from the point of view of minimizing drag. On the basis of the model will be made a prototype of the car. Analyzes outlined a direction for developing the solid of version v4 for the next seasons of the Shell Eco-marathon. To reduce the pressure on the rake face of the car it will be provided with a grating decreasing the pressure that will be placed at the site of the highest hypertension. Drawings of the vehicle were transferred to contractors at the beginning of January 2014. Car races start at the Shell Eco-marathon 2014 in Rotterdam in May 2014. Upon completion of construction of the car it is planned for the verification of numerical investigations by performing aerodynamic testing in the wind tunnel.

Bibliography:

[1] White Frank M. „Fluid Mechanics 4th Edition", Mcgraw- Hill College, 1998.

[2] Piechna Janusz. „Podstawy aerodynamiki pojazdów", Warszawa: Wydawnictwa Komunikacji i Łączności, 2000.

[3] Wolf Heinrich Hucho, (red.), „Aerodynamika samochodu : od mechaniki przepływu do budowy pojazdu", tł. z niem. Krzysztof Wituszyński, Warszawa: Wydawnictwa Komunikacji i Łączności, 1988.

[4] Raport Instytutu Lotnictwa w Warszawie pt. "Badania bolidu MuSHELLka. Badania wagowe współczynników aerodynamicznych i wizualizacja opływu.", Warszawa: 2013.

[5] Anderson John D. „Computational Fluid Dynamics: The basics with applications", McGraw-Hill, Inc., New York, 1995.

[6] Ferziger J. H., Perić M., „Computational Methods for Fluid Dynamics", Springer, New York, 2002.

[7] Łanecka-Makaruk Wiesława, Łucjanek Wiesław, „Mechanika lotu", Wydawnictwa Komunikacji i Łączności, Warszawa , 1966.

[8] Abłamowicz A., Nowakowski W. „Podstawy aerodynamiki i mechaniki lotu", Wydawnictwa Komunikacji i Łączności, Warszawa, 1980.

Abstract

The subject of study was the methodology for aerodynamic analysis of the cars built for the Shell Eco-marathon race. The research was carried out as the part of the racing team Smart Power Urban Silesian University of Technology work. The study helped to develop methods for conducting aerodynamic research for such types of vehicles. The main idea of the research was the numerical calculation of the aerodynamic drag force and its coefficient c_x and visualisation of the pressure distribution on the car surface and the air flow lines along the vehicle. As the base for the research there has been used a first conceptual CAD model of the Bytel (Fig.2a) - a vehicle built by Smart Power Silesian University of Technology racing team for the Shell Eco-marathon race 2014 in the urban category. The developed methodology was used in designing and optimization of the final body of the Bytel car.

Keywords: CFD, Aerodynamics, Numerical methods, Fluid mechanics

METODYKA PROWADZENIA BADAŃ AERODYNAMICZNYCH SAMOCHODÓW PRZYGOTOWANYCH DO STARTU W WYŚCIGU SHELL ECO- MARATHON BAZUJĄCA NA OPROGRAMOWANIU HYPERWORKS

Streszczenie

W ramach badań opracowano metodykę prowadzenia numerycznych badań aerodynamicznych pojazdu mającego wziąć udział w wyścigu Shell Eco- marathon. Badania były prowadzone w ramach prac zespołu wyścigowego Smart Power Politechniki Śląskiej. Na podstawie opracowanej metodyki utworzono modele numeryczne oraz wykonano symulacje numeryczne czterech koncepcyjnych wersji rozwojowych pojazdu. Wartościami wynikowymi badań były rozkład wartości siły oporu aerodynamicznego oraz rozkład ciśnienia na powierzchni pojazdu. Bazowym modelem do utworzenia metodyki prowadzenia badań dla pojadów tego typu był model CAD pojazdu Bytel (Rys. 2a). Utworzona metodologia przyczyniła się do zaprojektowania oraz zoptymalizowania kolejnych wersji rozwojowych samochodu Bytel.

Słowa kluczowe: CFD, Aerodynamika, Metody numeryczne, Mechanika Płynów