ABSTRACT

Because the aerodynamic loads, which are acting on the high-speed vehicles, play a significant part concerning the dynamic behaviour of the latter, the aerodynamics is one of the most important design considerations for cars such as Indy or Formula 1. In this study, the main goal is to investigate the influence of the boundary conditions at the level of the ground on the main aerodynamic characteristic of an open-wheel race car using the facilities offered by the ANSYS CFX, CFD code. 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. The conclusions are demonstrated by the results, using the relative increment of drag, lift and pitching moment, and by the computer-graphics visualizations. There are also presented some considerations concerning the importance of the rotating wheels in aerodynamics of the road vehicle and the opportunity to simulate it in a virtual environment.

1. INTRODUCTION

Mathematically, the aerodynamic load that an air stream exerts on a car is the integral, over all surfaces exposed to the stream, of the local streamwise component of the normal (pressure) and tangential (skin friction) forces, \( F = F_p + F_f \), where \( F_p \) is the normal component of force, due to the pressure distribution and \( F_f \) is the tangential component, due to the skin friction. In this way, direct evaluation of the contribution on total drag of an element of the vehicle, as mirrors, wheels, driving axels and others elements, requires knowledge of the detailed stress distributions over all element surfaces, also for entire vehicle. To obtain this experimentally is very difficult, except for simple shapes, even in a research sense. However, direct evaluations of components of \( F_p \) have been made in selected regions of a vehicle body. This is feasible operation in the regions where the pressure distribution is reasonably uniform: engine bonnet, windscreen and other continuously uniform surfaces. In this way, direct evaluation of the previously mentioned component of the aerodynamic loads, is not practical for bodies with the complex geometry e.g. road vehicle. On the other hand, detailed stress (pressure and friction) distributions are the specific output of CFD (Computational Fluid Dynamics) analysis [1].

Until recently, studies of fluids in motion were performed in the laboratory, but with the rapid growth in processing power of the computers, software applications now bring numerical analysis and solutions of flow problems to the desktop. In addition, the use of common interfaces and workflow processes make fluid dynamics accessible to designers as well as analysts. In this context, adequately validated computational codes can contribute greatly to a better understanding of road vehicle aerodynamics. The main reason for using numerical methods in the study of cars is that they can generate information before a testable model even exists. In addition, CFD analyses are not necessarily burdened with the limitation of size and geometry of the test section of the wind tunnels [2]. In this sense, computational
space can be made large enough to eliminate blockage effects (according to the available hardware resources). Simulation of the relative motion between vehicle and road is also comparatively easy to accommodate, even including rotating wheels [3]. On the other hand, once the equations of the mathematical model have been solved, there is much more information available than from a routine experiment, as shown in Figures 5 – 7.

In this study, the main goal is to investigate the influence of the boundary conditions at the level of the ground on the main aerodynamic characteristic of an open-wheel race car using the facilities offered by the ANSYS CFX, CFD code. The influence of the ground on the 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) as shown in Figure 1, and with the moving wall approach, see Figure 2, in a range of speed between 50 - 300 km/h.

![Figure 1: Principle of simulation with no relative motion between car and road and rotating wheels](image1)

![Figure 2: Principle of simulation with ground effect using moving belt approach](image2)

The conclusions are demonstrated by the results, using the relative increment of drag, lift and pitching moment, and by the computer-graphics visualizations. There are also presented some considerations concerning the importance of the rotating wheels in aerodynamics of the road vehicle and the opportunity to simulate it in a virtual environment.

ANSYS CFX (Computational Fluid dynamiX) is a powerful fully integrated fluid analysis software tool, which combines CAD (modeling and input), complex meshing solutions, fast solution algorithm and post-processing facilities. It can be used successfully for fluid dynamics simulations of all levels of complexity [4].

2. CFD METHODOLOGY

The flow field around a vehicle is physically very complex. In consequence, the efficiency of an aerodynamic CFD simulation depends on many factors. Creation of the model geometry and its integration in a physical domain, grid generation and choice of a suitable numerical computing scheme are significant factors that can determine the level of success of the simulation process. The main steps of the simulation processes are described in the following paragraphs.

2.1 CAD Model

The body surfaces of the vehicle were drawn as CAD data (see Figure 3), with the aid of a professional software-package, Pro-ENGINEER 2000i2 and imported in ANSYS Workbench environment as an IGES file. The exterior of vehicle was very carefully reproduced, including...
the exterior surfaces of the wheels in order to obtain realistic results, as much as possible. The cockpit of the driver, which is closed for this study, represents the major exception.

The geometry of computational domain, a rectangular enclosure, was such of dimensions that the adverse pressure effect between the vehicle and the walls are negligible, as shown in Figures 5 and 6. The dimensions of the physical domain as functions of \( L \) (the length of car) are the following:

- \( 2.0 \, L \) in front the car and in the upper side, from the level of ground;
- \( 2.5 \, L \) in lateral side, from the symmetry plane;
- \( 5.0 \, L \) behind the car.

For these dimensions the assigned blockage ratio is:

\[
\frac{\text{Model frontal area}}{\text{Cross section area of the physical domain}} \times 100 = 2.5 \, \%
\]

Note: According with SAE Standards [5], in order to have results unaffected by the size of the test section, the blockage ratio for experiments, must be smaller than 5 \%.

2.2 Computational Grid and Boundary Condition

The computational grid was generated using a multi-block scheme with tetrahedral and wedge elements nearest to surfaces of the vehicle, in order to solve accurately the flow in the proximity of the later. In this sense, the side length of the elements on the surface of the vehicle was in range of \((0.002 - 0.02) \, \text{m}\). Also, the maximum distance from surface of the vehicle to the first layer of grid points was \(0.00045\, \text{m}\), in order to fulfil the condition concerning the \( y^+ \) for computations where is used the “wall function” approaching. A \( y_{\text{max}}^+ < 200 \) is acceptable in the case of using automatic wall treatment (the specific wall function of the used turbulence model) [4].

For a half model with longitudinal symmetry plane, the dimensions of this are:
- 1.666.975 grid points for entire computational domain;
- 188.683 grid points for body surfaces of the vehicle.
- 53.208 grid points for wheels.

According with previously mentioned the boundary conditions of the processes were the following:

- no slip conditions for the velocity on surfaces of the vehicle $v_X = v_Y = v_Z = 0$, in a reference frame according with SAE Standards [5], including the wheels for the analyses without ground effect;
- an uniform and constant velocity $v_Y = v_Z = 0$ were imposed at the inlet boundary; same conditions were considered for the surface which represent the ground, and in addition, the option solid moving wall was activated for the analyses with relative motion between vehicle and road;
- at the outlet boundaries, a zero (gauge) pressure condition was imposed: $p = 0$;
- for the grid points on surfaces of the wheels in motion, a rotating was considered with the angular speed $\omega = \omega_c / R$ ($R$: radius of the wheels), in some reference frames having the origin in the centers of the wheels and orientation as general frame of the motion;
- for the rest of the surfaces, fluid walls and free slip conditions were assigned.

2.3 Conditions of Simulations and Turbulence model

For computing of the variables of flow, the code solves the full RANS (Reynolds-Averaged Navier-Stokes), mass (continuity) equation and energy equation, if necessary.

The analyses were performed in steady state, adiabatic, fully turbulent conditions, for a reference pressure and temperature of the air $p_a = 1$ At, $T_a = 288$ K. The study was performed for six velocities, from medium to higher, as following: $v_1 = 50$ km/h , $v_2 = 100$ km/h , $v_3 = 150$ km/h , $v_4 = 200$ km/h , $v_5 = 250$ km/h and $v_6 = 300$ km/h.

ANSYS CFX contains a wide variety of turbulence models to provide accurate and computationally efficient results for almost every application. The widely proven SST (Shear Stress Transport) $k-\omega$ turbulence was used to solve the simulation processes in combination with the automatic near-wall treatment developed for the $k-\omega$ based model [4]. This is a two-equation eddy-viscosity model which offers significant advantages for nonequilibrium turbulent boundary layer flows and provides excellent results on a wide range of flows and near-wall mesh conditions [6]. Also, according with previous experience [7] this turbulence model has a good behaviour in adverse pressure gradients and separating flows. The tendency to produce large turbulence levels in regions with large normal strain, like stagnation regions and regions with strong acceleration, is much less pronounced than other turbulence models used in aerodynamics applications, as $k-\varepsilon$ ones.

3. RESULTS

The solutions were considerate finished, when the variations of normalized rate of change for the variables of processes were insignificant for the final steps of iterations. These variables include the components of velocity, pressure, and turbulence quantities. The main convergence criteria checked very carefully were the followings:

- decreasing of the residuals below $1e - 003$, as shown in Figure 4;
- variations of the aerodynamic loads which are acting on the vehicle lower than 0.5% for the final steps ($i$) of the iterations;

![Momentum and Mass](image1)

$\text{Figure 4: Variations of the residuals for the pressure and components of the velocity}$

- an adequate value of $y^+$, to the first grid points above the surfaces of the vehicles, smaller than 200 [4].
- a continuous and physically realistic distribution of the variables of the process, as shown in Figures 5 - 7, for $v_\infty = 200 \text{ km/h}$.

![Velocity](image2)

$\text{Figure 5: Velocity contours on the computational domain, } v_\infty = 200 \text{ km/h} \text{ (visualization in the symmetry plane)}$
Figure 6: Pressure contours on computational domain, $v_{\infty} = 200 \text{ km/h}$ (visualization in the symmetry plane and ground)

Figure 7: Pressure contours on body of car and ground, $v_{\infty} = 200 \text{ km/h}$

The used CFD code allows calculating the aerodynamic forces that are acting over surfaces of vehicle, as presented in the introduction. In this sense were evaluated main aerodynamic loads, drag and lift of the following components of the vehicles:

- front and rear wings;
- front and rear wheels;
- underbody geometry;
- aerodynamics elements such as fins and deflectors;
- blunt bodies, such as suspension and fastening elements of the fins, elements of the axles.
We focused mainly on these components taking into consideration that the most changes, from aerodynamic point of view, arise on structures which are running close to the ground, such as wheels, front wing, or underbody geometry, as shown in Figure 8. Concerning the Drag, the results are presented graphically in Figure 9 for each of previously mentioned components as percentage from resultant aerodynamic loads $D$. 

![Fig. 8 – Evaluated components of the car](image)

![Figure 9: Percentage variation of components on drag.](image)
Figure 10: The orientation of the aerodynamic loads for each of studied component. According with SAE Standards [5], the reference frame for vehicle aerodynamics has the orientation with the positive $z$ axis in down direction. In this way, the lift is negative and down force is positive. For the studied car, there are components which are generating positive vertical load, oriented in down direction, as the front and rear wings, and the others which are giving negative vertical load (lift), oriented in up direction, as the wheels (see Figure 10). The ratio of lift with moving belt approach and the lift with fixed ground approach $L_{MB} / L_{FG}$ is depicted by Figure 11 for each studied component and also for resultant lift.

Figure 11: The ratio of lift for studied cases
4. CONCLUSION

As expected, the method to simulate the ground has as results significant variations for the aerodynamic loads on the components of the car which are running in the proximity of the ground, mainly concerning vertical loads.

Thus, for both type of analyses, concerning the resultant drag, the most significant contribution is given by the wheels, more than 40%, due to their large size and the fact that they are wholly exposed to the air stream, as shown in Figure 9. Also, for the analyses with the wheels in motion, there are variations on drag with the air speed, mainly for the rear wheels, due to the supplementary vortices generated by the rotating wheels. The drag of the front wheels has a slightly decreasing with the air velocity, in opposite with the drag of the rear wheels which is increasing due to the flow around these is strongly affected by the structures in front of them.

The other components, such as front and rear wings, fins, blunt bodies has an increasing variation on drag with the air speed, except the lower geometry which has a slightly decreasing of the drag, due to the Venturi effect which is developing between underbody and ground. Both front and rear wings, together with the other aerodynamic elements (fins and deflectors) have a contribution on the resultant drag up to 30 %.

Concerning the lift, as shown in Figure 11, the simulations using moving belt approach has as result the variation of the vertical loads on the components of the car which are running in ground effect, the most significant being for the underbody geometry. For these surfaces, the ratio of the lift with moving belt approach and the lift with fixed ground approach are decreasing with the air speed, as similarly for the car. There are, also, components for which the vertical loads are not affected by the method for the simulation of the ground, as blunt bodies ad rear wing.

For the analyses using moving belt approach, due to the combined Magnus effect and ground effect, the influence of the wheels on lift is increasing with the velocity. From qualitative point of view, taking into consideration the motion of the wheels is important for a better evaluation of the aerodynamic characteristics of the car because this is the only way in which Magnus effect surrounding the wheels can be reproduced. Due to this effect the impact point on a rotating wheel is moving from the front of wheel to where the tire makes contact with the ground [8]. Also, due to Magnus effect, the wheels, which are wholly exposed to the air stream, generate a considerable aerodynamic lift, more than 40 % from the resultant lift [7]. The negative resultant lift (down force) of the car is mainly due to the two wings, the front one having the greatest contribution, more than 70% [7]. The lower side of the car generates a significant down force, too, due to the fact that the underbody is specially designed to meet the Venturi effect [9].

Consequently, in CFD computations, for this type of cars it is very important to simulate the ground effect, respectively the rotation of the wheels and the motion of the ground relative to the car. Not lastly, thanks to increases in computational power, CFD has become a valuable tool for fine-tuning the external shape of the road vehicles by providing much more complex information than from a routine experiment, before a testable model even exists.
REFERENCES


NOMENCLATURE. ACRONYMS

\[ D \] Drag  \\[ F \] Aerodynamic load  \\[ L \] Lift  \\[ v_\infty \] Velocity of air stream at infinity  \\[ y^+ \] Dimensionless distance from the walls  \\[ \omega \] Angular speed  \\[ AE \] Aerodynamic elements  \\[ BB \] Blunt bodies  \\[ FWg \] Front wing  \\[ FWs \] Front wheels  \\[ RWg \] Rear wing  \\[ RWs \] Rear wheels  \\[ UG \] Underbody geometry

CONTACT

Dr. Ing. Angel HUMINIC, Dr. Ing. Gabriela HUMINIC

Transilvania University of Brasov
Thermodynamics and Fluid Mechanics Department
29, Bulevardul Eroilor Street, Brasov 500036, Romania

angel.h@unitbv.ro  gabi.p@unitbv.ro

Copyright ANSYS, Inc.