CFD Analysis of the Central Engine Generic Sports Car Aerodynamics

Janusz Piechna¹, Leszek Rudniak², Adam Piechna³
¹Faculty of Power & Aeronautical Engineering
²Department of Chemical & Process Engineering
³Faculty of Mechatronics
Warsaw University of Technology, Poland

ABSTRACT

A lot of sport cars are using aerodynamic devices to generate aerodynamic downforce and reduce aerodynamic drag. The aim of the present work was analysis of influence of wings, potentially safe aerodynamic ads-on, on the generic sport car aerodynamic characteristics. Clean aerodynamically shape of the generic sport car was used as the reference model. Rather untypical front wing located in position guaranteeing aerodynamic shielding was applied. Rear wing located behind the body for maximum efficiency was used as an additional source of aerodynamic downforce. Different methods of meshing (tetrahedral and hybrid hexcore mesh with boundary layers, immersed boundary technique) had been used and the impact on calculated forces of the mesh type, density and quality compared.

INTRODUCTION

Fast cars take advantage of a high speed flow around the car body for generation of additional forces improving car performances [1,3,4]. Mid engine cars commonly have radiators located in the rear part of the fuselage. It is due to that front part of the car is relatively clean aerodynamically. In the presented work the modifications of the front car body have been mainly investigated. Typically front part of the car, which is responsible for the aerodynamic load of the front axis, is very difficult to design. Many teams have problems with front axis aerodynamic load [1,4]. It is not an easy task to aerodynamically balance a fast car [2,3]. Commonly used devices generating download aerodynamic force, for example in form of a horizontal plate shown in Fig.1, are sensitive to the clearance between the ground and car body. Such sensitivity is potentially danger due to lost of front axis download in unfavorable road conditions.

Fig. 1 Splitter plate as an aerodynamic down force generating device - streamlines

Because of that, an idea of use of the front wing located in significant distance over the ground, to be position insensitive, was investigated.
In the past, Ferrari was exploited such idea (see Fig.2).

![Ferrari Dino 2006 Competizione with front and rear wing](image)

Fig.2. Ferrari Dino 2006 Competizione with front and rear wing

![Prototype car with the front wing.](image)

Fig.3. Prototype car with the front wing.

No data are available describing efficiency of such solution. The main aim of the present work was collection of numerical data describing aerodynamic features of a car with front wing configuration.

**GENERIC RACE CAR MODEL**

For presented analysis not really existing fast car geometry was used. Only conditions of typical dimensions and relatively streamlined shape were used in geometry preparations. The analyzed car geometry schematically was presented in Fig.4.

![Schematically presented analyzed car geometry](image)

Fig.4. Schematically presented analyzed car geometry
For the flow analysis, the cooling air passages in the rear part of the central engine car body have been considered. Car body was designed in such way that residuary diffuser was located at the end of bottom part.

WING

A not typical airfoil has been used in recent investigation. Airfoil was located over the ground and in front of the car body. Due to car geometry constraints any typical airfoil could not be used. The shape of an airfoil was prepared after performing analysis of the path lines in front of the car body.

Fig.5. Path lines around the airfoil located in front of the car body.

Some tests of aerodynamic characteristics of the separate foil were performed but they were not taken into account due to not typical neighborhood during planed operation. Free airfoil characterized separation which had not be confirmed during two-dimensional analysis of the flow around the final configuration.

TWO- DIMENSIONAL ANALYSIS

For definition of final configuration of car body an initial two-dimensional tests have been performed. Flow around the car body is strongly three-dimensional and any two-dimensional simplifications can be misleading. Due to not very realistic flow conditions around the central part of the car (especially under the car body) in two-dimensional simplification, our activity was concentrated on the flow details near the front part of the car body. Results of flow visualizations are presented in Fig. 6 and Fig.7.

The flow separation over the airfoil have been searched in order to omit the configuration causing such phenomena.

Fig.6. Wing in high and low position – streamlines
Flow conditions around the airfoil depend on the wing position in relation to the ground and car body. In high wing position flow is attached and in low position a separation is clearly visible (Fig. 6).

In Fig. 7, typical pressure distribution near the front part of the car is presented. Drag and lift force generated by the airfoil was recorded and was taken as a very rough estimation.

**THREE – DIMENSIONAL ANALYSIS**

Taking into account results of preliminary two-dimensional analysis of the airfoil geometry and wing position, a three-dimensional model of a car was prepared. Used computational domain is shown in Fig. 8 and dimensions of the bounding box were following:

<table>
<thead>
<tr>
<th>X min</th>
<th>X max</th>
<th>Y min</th>
<th>Y max</th>
<th>Z min</th>
<th>Z max</th>
</tr>
</thead>
<tbody>
<tr>
<td>-8 [m]</td>
<td>20 [m]</td>
<td>-0.24 [m]</td>
<td>5.74 [m]</td>
<td>0 [m]</td>
<td>6 [m]</td>
</tr>
</tbody>
</table>

Simulations were performed in rectangular section with following boundary conditions: “velocity inlet” at inlet cross-section, “pressure-outlet” at the outlet cross-section, “symmetry” at symmetry plane, top plane and side plane, “moving wall” at the bottom. Rotation of wheels was not simulated.
GRID GENERATION

Due to rather complicated shape of the body two different techniques of the grid generation have been tested.

Fig. 9. Considered car geometry: basic shape, shape with forward wing and shape with forward and backward wing and cooler

The first one was a typical grid generation with triangular surface mesh on the car body, addition of boundary layer grid and unstructured grid generated in surroundings.

Fig.10. Exemplary mesh with details of prismatic boundary layer mesh generated by Gambit and Tgrid.

The second one approach used the new technique developed by ANSYS and called immersed boundary method. Such technique is recently under tests and such purpose was used in recent work.

Fig.11 Computational domain and ranges of regions used in immersed boundary technique.

The immersed boundary method implemented in Fluent solver uses a special correction technique to obtain accurate results with non-body conforming meshes. Such approach significantly reduces the time required for geometry and mesh preparation. Accurate and robust reconstruction schemes are enforced, resulting in numerical boundary conditions that emulate body-fitted behavior. Full description of this method can be found in ANSYS Fluent 12 documentation [5].
Use of immersed boundary method was easy and effective way for mesh generation even in very complicated geometry. The geometry in STL format can be read by Fluent solver (TOMMIE module) and then TOMMIE module creates a hexahedral mesh with isotropic or anisotropic grid refinement based on the users input.

![Image](image1.png)

Fig.12. Details of cooling air inlet edge causing problems for Tgrid and successfully meshed by TOMMIE (Fluent immersed boundary module)

![Image](image2.png)

Fig.13. Details of the cooling duct outlet, rear wing and wing pylons causing problems for Tgrid and successfully meshed by TOMMIE (Fluent immersed boundary module)

Turbulent flow was simulated using mesh generated in TGrid and by immersed boundary method with the same boundary conditions. Results were analyzed and compared in order to investigate new Fluent functionality – immersed boundary method. Immersed boundary technique was found as an easy and effective way for mesh generation even in very complicated geometry. Comparison of simulation results obtained by both methods have shown 20% difference in drag prediction and 4% difference in lift and momentum calculation.

RESULTS AND DISCUSSIONS

Several modifications of initial geometry were analyzed. For reference purposes a clean race car configuration was prepared and drag, lift and momentum values were recorded. All simulations were modeled as turbulent incompressible flow with second order discretisation. S-A and k-e with standard wall function ($y^+ 40-120$) turbulence model were used in TGrid cases and k-e turbulence model with enhanced wall treatment was used in TOMMIE cases. In TOMMIE cases PRESTO! pressure discretisation method was used.

In Fig. 14 variation of the pressure at the symmetry plane of reference car shape and car with wing is shown. One can notice that great changes of the pressure at the front part of the body as well as some changes at the rear under the car exist.
Fig. 14. Comparison of pressure distribution at the symmetry plane of the reference shape (left) and car with the front wing (right). Color coding: red – positive pressure, blue – negative pressure.

Fig. 15. Pressure distribution on the airfoil surfaces

Pressure distributions presented in Fig. 15 and 16 at the wing surfaces, changes insignificantly along the wing (z direction). It means that side pylons covering front wing tips work correctly as the sideplates.

Fig. 16. Variation of the pressure along the front wing and pylons at different wing cross-sections.
In Fig. 17 pressure distribution on analyzed car surfaces and in car vicinity is depicted. It was expected that the wing will act as windshield for the car body and due to lower dimensions it reduces the total car drag.

The front part geometry and used coding names of considered wing-car configurations are presented in Fig. 18. The corresponding velocity and pressure distributions are shown in Fig. 19-24 distinguished by coding scheme from Fig. 18.
Fig. 19. Geometry comparison: configuration A in front, configuration B on the back (pressure distribution and velocity magnitude on the cross-section plane)

Fig. 20. Velocity distribution at symmetry plane – configuration B

Fig. 21. Velocity distribution at symmetry plane – configuration E
In Fig. 19, the geometry and pressure distribution differences in reference car shape and car with the wing have been shown explicitly. Additionally a velocity magnitude distribution in chosen cross-section is depicted.

Velocity magnitude contours are shown in Fig. 20 for configuration B, and in Fig. 21 for configuration E. Configuration B provide flow without separation in contradiction to configuration E.

Numerical results obtained from the simulations were collected in Table 1 and Table 2.

Table 1

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>Configuration</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cx</td>
<td>0.2351</td>
<td>0.2444</td>
<td>0.2432</td>
<td>0.2314</td>
<td>0.2374</td>
<td></td>
</tr>
<tr>
<td>Cz</td>
<td>0.1056</td>
<td>0.0660</td>
<td>0.0565</td>
<td>0.0508</td>
<td>0.1049</td>
<td></td>
</tr>
<tr>
<td>Cm</td>
<td>0.0070</td>
<td>-0.0790</td>
<td>-0.0930</td>
<td>0.0754</td>
<td>0.0114</td>
<td></td>
</tr>
</tbody>
</table>
Table 2

<table>
<thead>
<tr>
<th>Ratio</th>
<th>Configuration</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cx/Cxref</td>
<td>1</td>
<td>1.039766</td>
<td>1.034566</td>
<td>0.984459</td>
<td>1.009985</td>
<td></td>
</tr>
<tr>
<td>Cz/Czref</td>
<td>1</td>
<td>0.624922</td>
<td>0.534904</td>
<td>0.481</td>
<td>0.993247</td>
<td></td>
</tr>
</tbody>
</table>

The analyzed final car geometry with central engine has cooling air inlets located in a rear part of the body and is additionally equipped with the rear wing. Pressure distribution in such configuration is presented in Fig. 24.

![Fig. 24 Pressure distribution on the car body at full configuration.](image1)

Pathlines near the symmetry plane are shown in Fig. 25 and the pathlines at the cooling channel are visualized in Fig. 26. Pressure distributions at the car centerline for the basic and winged model are compared in Fig. 27. Increase of the pressure on the top car surface caused by the rear wing presence is clearly seen. The aerodynamic down force was strongly increased (Cz=-1.217) but with the aerodynamic drag build up (Cx=0.695).

![Fig. 25 Path lines near the symmetry plane](image2)
CONCLUSIONS

Taking into account results obtained from the simulations, it can be concluded that any action at the front part of the car strongly influence the flow conditions around the whole car body. Flow around the front part and rear part of the car is in strong conjunction. Due to that, it can not be possible to analyze separately any aerodynamic device located in front of the car body.

Analyzed front wings configuration was not so effective as expected. Application of front wing in almost any case, increased the aerodynamic drag about 3-4%. However the lift force were in many cases relatively reduced by half, but the absolute value of the lift force was generally low. It seems that front wing influences the position of the stagnation point on the car body and changes the proportion between the amount of air passing the car body under the car and over the car. Even small changes of the pressure values distributed over the relatively big car body area, generate aerodynamic force changes comparable to the value of the front wing aerodynamic forces.

REFERENCES