Aerodynamic shape optimization of racing car front wing

M Kalinowski¹², M Szczepanik²

¹ Alstom Konstal S.A., Metalowców Street No.9, 41-500 Chorzów, Poland
² Silesian University of Technology in Gliwice, Faculty of Mechanical Engineering, Department of Computational Mechanics and Engineering, Konarskiego Street No.18a, 44-100 Gliwice, Poland

*Author’s e-mails: marcin.kalinowski@alstomgroup.com; miroslaw.szczepanik@polsl.pl

Abstract. Nowadays, the automotive industry is one of the largest areas of CFD (Computational Fluid Dynamics) simulation applications. The big challenge that engineers dealing with CFD simulations in this industry have to face is shape optimization, which is often very time-consuming and requires a lot of iterations. To reduce the time needed to achieve an optimal shape and make the design process more efficient, more and more CFD analysts are turning to Adjoint Solver (AS) formulas, a new methodology used for optimization in the automotive as well as railway industry. The AS method calculates gradients (directions, quantities) directly by solving conjugate equations, which makes them independent of design variables. The article concerns optimization the shape of the front wing of the racing car in order to obtain the highest possible downforce-to-drag ratio. At the beginning, 2D shape optimization of the front wing was performed. The next stage of work was the preparation of a CAD (3D) model that took into account the change in the shape of the wing, obtained during the optimization process, and then performing a flow analysis for it.

1. Introduction

Nowadays, the automotive industry is one of the largest areas of CFD (Computational Fluid Dynamics) simulation applications. The big challenge that engineers dealing with CFD simulations in this industry have to face is shape optimization, which is often very time-consuming and requires a lot of iterations. To reduce the time needed to achieve an optimal shape and make the design process more efficient, more and more CFD analysts are turning to Adjoint Solver formulas, a new methodology used for optimization in the automotive as well as railway industry. The AS method calculates gradients (directions, quantities) directly by solving conjugate equations, which makes them independent of design variables. ANSYS Fluent software offers shape optimization capabilities that can automatically adjust the geometrical parameters of a specific design to specific optimization goals. The tool responsible for shape optimization in ANSYS Fluent is the Adjoint Solver module. It provides information on how to modify the geometry to achieve the goals of the design (e.g., minimalization of the drag or lift force). This information is communicated by modifying the mesh to see the effect of the recommended changes. The Adjoint Solver can be used to optimize downforce (F1, sports cars), reduce drag (for passenger cars) and also to reduce overall pressure drop (for ducts and pipes, Figure 1). [1,2]

So far, few articles dealing with the problem of aerodynamic optimization using Adjoint Solver have been written. In article [3] authors describe variant optimization of the open-wheel race car, focusing
mainly on the change in the drag coefficient due to the change in radiator cooling channel angle. Optimization method presented in this article consist on choosing the best of the "n" variants under consideration and does not concern the determination of the extremum (local or global) of the objective function. On the other hand authors of the article [4] present shape optimization of racing cars using Computational Fluid Dynamics optimization process, which is based on using the mesh morphing techniques to create new designs for analysis by morphing the CFD mesh of the original design. Article [4] is the most similar to the subject under consideration in this article, but focus only on the optimization of the downforce (not downforce-to-drag ratio as presented in this article) for rear wing of racing car (not front wing as presented in this article).

Figure 1. Optimizing the shape of the L-shaped conduit using the Adjoint Solver module [1]

The article is divided into two main parts. In the first part of the work, optimization of the shape of the front wing was carried out. In order to simplify the problem, the analyzed model was a 2D model and contained only a cross-section through main plane and upper flap. The second part of the work was the preparation of a CAD model, which took into account the change in the shape of the wing, obtained during the optimization process, and then performing a flow analysis for it. Aerodynamic analysis was carried out in the ANSYS Fluent software. The shape optimization, which is the main goal of this article, was made in the "Adjoint Solver" add-on, built into Fluent.

2. The principle of operation of the Adjoint Solver module

During the standard (conventional) aerodynamic calculation process in ANSYS Fluent, the input values are geometric properties, finite element mesh, fluid properties (e.g. air, water). After the aerodynamic analysis, we get maps (e.g. velocity, pressure), graphs, scalar values (forces: resistance, lift) on the output. Based on them, engineer decides to change the shape. The conventional process of carrying out aerodynamic analyses is presented in the Figure 2 [5,6].

Figure 2. Conventional process for conducting aerodynamic analyses

Thanks to the use of the Adjoint Solver module, we can, based on the results, change the shape and finite elements mesh in the examined flow problem in order to maximize / minimize the observed output value of the aerodynamic analysis (e.g. drag force). In other words, in a conjugate solver, the output values...
have an effect on the input variables. The process of conducting aerodynamic analyses with the Adjoint Solver is shown in Figure 3 [2].

A more detailed comparison of the standard process for calculating flow properties and calculations performed with a coupled solver is presented below.

The standard process of calculating flow properties begins with the designer creating a 3D model. The next step is to create a finite element mesh, which may be external (external flows, e.g. a car while moving) or internal (internal flows e.g. cooling fluid flow through a pipe). The production of a high-quality mesh is very important to obtain correct and accurate results. On the other hand, too fine mesh results in a very long computational time.

The next step in the standard flow analysis process is flow analysis from which the output is derived. Only on their basis, the CFD engineer makes decisions about the changes necessary to obtain the optimal flow. These changes are successively presented to the designer and, after obtaining his approval, are introduced into the CAD model. Detailed scheme of the fluid properties calculation process is presented in Figure 4 [5,6].

As mentioned before, this approach is very ineffective as it requires a lot of iterations. Moreover, with the complex geometry of the analysed model and with non-standard flow conditions (e.g. turbulent flow), it is very difficult to indicate the appropriate "direction" of the changes made.

These inconveniences can be solved by using the coupled solver, which is an extension of the conventional CFD solver. The process of performing a flow optimization can be described in the following few steps:

1. **Step I:**
   Standard flow analysis. This step covers the first 3 steps of the standard flow property calculation process (i.e. CAD model building, model discretization, flow analysis).
2. **Step II:**
   Activation of the Adjoint Solver module and determination of the observed values (e.g. drag force, lift force, lift to drag ratio, pressure etc.), which will be arguments of the optimization objective function. In this paper, the observed value is the ratio of the downforce to the drag force, which is to achieve the highest possible value.
3. **Step III:**
Selection of Adjoint Solver parameters and conditions of convergence.

4. **Step IV:**
Perform a flow analysis consisting of initialization and iteration until convergence.

5. **Step V:**
Post processing stage. During it, velocity/pressure maps can be generated, convergence graph can be checked and the change in the observed value can be evaluated.

6. **Step VI:**
Conversion of the mesh using a dedicated tool. This tool defines the dimensions of a quad element (in the case of 2D analyses) or a hexagonal element (in the case of 3D analyses) inside which the mesh morphing is to take place. The actual amount of change within a quad/hexagonal depends on the number of control points and the percentage of modification allowed. Before modifying the shape, the morphing tool reports the predictable effect on the change in the observed value, taking into account the sensitivity data and morphing settings.

Optimization in the Adjoint Solver is gradient-based topology optimization which is combined with a mesh morphing tool. Parameters in optimization task considered in this article are the coordinates of each and every point on the boundary of the region (contour of the main plane and upper flap) thereby allowing efficient shape improvements to be made.

Due to the fact that the optimization process in Adjoint Solver is iterative, after completing the sixth step, solver go back to the fourth step (re-analysis of the flow is carried out in order to evaluate the change made). Figure 5 shows a diagram showing the process of calculating flow properties using the Adjoint Solver module [1,2].

![Diagram of the flow properties calculation using the Adjoint Solver module](image)

**Figure 5.** Diagram of the flow properties calculation using the Adjoint Solver module [2]

3. **Optimization**

3.1. **The input model to the optimization process**
As mentioned in previous chapters, the input model for the optimization process is based on the front wing CAD model of the racing car.

Due to the fact that the optimization process is usually a multi-iteration process, a very important aspect is the maximum limitation of the duration of one iteration. As mentioned in previous chapters, the finite element mesh (dense mesh - long computation time) has the greatest impact on the calculation time. Therefore, the 3D model has been replaced with a 2D model. As a result, the number of mesh nodes was reduced to 171,000 (previously 750,000), and the number of elements to 60,000 (previously 3,200,000).

The input model consists of a cross-section through the main plane and the upper flap. The input model for optimization in the Adjoint Solver module is shown in Figure 6.
3.2. FEM model

The FEM (Finite Element Method) model consists of a rectangle simulating the environment (air) in which the shape of the front wing has been cut out (Figure 7). The FEM model was created in the CAD software through a Boolean operation, consisting in subtracting from the rectangle (environment) the cross-section of the front wing (main plate and upper flap).

The dimensions of the rectangle should be selected depending on the length of the model. Due to the fact that this analysis is two-dimensional and only covers the cross-section of main plate and upper flap, it is not physically possible to determine the total length of the model. Therefore, it was assumed that the front wing model would not be longer than 1000 mm.

For the assumed model length, the rectangle dimensions are as follows:

- length (-X): 3 * L = 3000 mm
- length (+ X): 5 * L = 5000 mm
- height (+ Y): 2 * L = 2000 mm

The above dimensions of the rectangle simulating the environment were selected on the basis of the document [7] and own observations resulting from CFD analyzes. The distance from the ground was set to 70 mm.

The FEM model is a non-deformed (rigid) model. This means that the front wing of the car under analysis does not deform under the influence of the flowing air. The following air properties were assumed:

- density = 1.225 kg/m$^3$
- viscosity = 1.7894e-5 kg/ms
A computational pressure-based solver was used in the analysis. This solver is well suited for vehicle aerodynamic calculations, as it gives precise solutions in the speed range up to 367 km/h, when the air flow can be treated as incompressible. Temperature is not taken into account. During optimization, only one case was considered, characterized by the air stream speed equal to 100 km/h.

### 3.3. Mesh

As mentioned earlier, the model was meshed approximately with 171,000 nodes and 60,000 elements (triangle and quadrilateral). The sizes of the elements differ depending on the location in the FEM model. Elements at a considerable distance from the front wing model are large, but while approaching to the front wing model are getting smaller and smaller. The refinement of the mesh near the analysed model allows for more accurate results of the flow analysis (mainly in terms of pressure and velocity values). Further away from the model, the mesh can be coarse as the airflow is almost intact there. The "Sphere of influence" function was used to condensate the mesh (Figure 8).

![Figure 8. Mesh](image)

### 3.4. Results

This section presents the results of flow optimization in the Adjoint Solver module. The most important result is a change in the shape of the front wing and thus a change in the downforce-to-drag ratio. The drag and the lift coefficients, respectively, have a decisive influence on the drag force and the downforce. These coefficients directly depend on the shape of the analysed body, so their change will be directly proportional to the change of the corresponding forces. Forces (drag and downforce) also depend on the frontal area, which will also change, but compared to the change in the coefficients (drag and lift), this change is negligible.

Figure 9 shows the changes in the coefficients of the drag force and the downforce as a function of the number of iterations. Drag force coefficient decreases, while the downforce coefficient increases (in absolute value). It follows that the ratio of the downforce to the drag force tends to infinity. It is worth mentioning here that the negative sign at the value of the downforce coefficient only informs about the direction.
Optimization was interrupted when the change in the value of both coefficients became negligible. In order to check the correctness of the calculations, the model convergence should be checked by assessing the residual values of continuity, velocity in the horizontal and vertical directions (x-velocity and y-velocity), kinetic energy of turbulence (k) and the coefficient of kinetic energy dissipation of turbulence ($\varepsilon$). In a correctly conducted CFD analysis, the values of these values should decrease until reaching a steady state. For multi-stage and multi-iterative analysis, the convergence plot should take the shape of a comb. With each successive step of the analysis, forced by the change of the mesh, the graphs should in the first step of the new step increase very rapidly to the level that was reached at the beginning of the previous step, and then gradually decline until they reach a steady state. The result of such behaviour of the function course is the "comb" shape of the graph [7].

As a result of the optimization carried out, the shape of the front wing was obtained, as shown in Figure 10. For the resulting wing shape, velocity and pressure maps were generated and compared with the aerodynamic analysis performed for the input geometry (Figure 11). Pressure and stream velocity maps for the optimized shape are much better aerodynamically than for the input geometry. By analysing only the area of the medium under the main plane for the optimized front wing, it can be seen that the pressure there is much lower (there is even a negative pressure) compared to the same area for the input shape. Moreover (according to Bernoulli’s law), under the modified main plane, one can observe a much higher air stream velocity than it was for the input shape.
The downforce / drag ratio for the optimized shape of the front wing is 12.65. It is worth remembering that this coefficient is calculated for a two-dimensional model with a large part of the unaccounted for components such as the nose of the car or end plates. In addition, elements omitted from optimization in the three-dimensional analysis (presented in the next chapter) may cause vortices adversely affecting the drag and pressure force (increasing and decreasing respectively). In fact, it should be expected that the value of this coefficient will be lower.

4. 3D flow analysis of the optimized front wing

The 3D model was made in the SolidWorks program based on the shape obtained in the optimization process. Due to the fact that the front wing model is symmetrical in the YZ plane, only half of the model was analyzed. Thanks to this, the computation time was significantly shortened and the model could be discretized with greater accuracy (finer mesh). The FEM model consists of a cuboid simulating the environment (air) in which the shape of one half of the front wing has been cut. Model was meshed approximately with 1,100,000 nodes and 5,000,000 second order tetras elements. The sizes of the elements differ depending on the location in the FEM model. Elements at a considerable distance from the front wing model are large, but while approaching to the front wing model are getting smaller and smaller. The refinement of the mesh near the analysed model allows for more accurate results of the flow analysis (mainly in terms of pressure and velocity values).

During 3D flow analysis, only one case was considered, characterized by the air stream speed equal to 100 km/h.

CAD model, FEM model and detailed mesh view were presented in the Figure 12 and results from 3D flow analysis were presented in the Figure 13.

The values of the aerodynamic forces (determined in the 3D analysis) acting on the front wing are presented in the Table 1.

| Drag force [N] | Downforce [N] | Downforce-to-drag ratio | Drag coefficient | Downforce coefficient |
|---------------|---------------|--------------------------|------------------|-----------------------|
| 34.16         | -152.78       | 4.48                     | 0.388            | -1.748                |
Figure 12. CAD model, FEM model and detailed mesh view

Figure 13. Pressure (1) and velocity (2) maps

5. Summary
Within the framework of this article, the shape of the front wing of the racing car was optimized in terms of maximizing the downforce / drag ratio, thereby achieving the goal of the study. The front wing was designed without relying on existing solutions (due to the lack of precise design data), it fulfils its primary function, i.e. generating the downforce. The downforce-to-drag ratio is equal 4.48, what is twice the size of the non-optimized front wing.

It is true that the resulting value of the downforce-to-drag ratio is more than twice lower than the value determined in the two-dimensional analysis (12.65), but it should be remembered that the two-dimensional analysis did not take into account, among others the prow of the car and endplates. The direction of further research may be a comparative analysis of the gradient optimization method with evolutionary algorithms or with swarm algorithms based on the aerodynamic optimization of the front wing of a racing car. The gradient optimization method has a solid mathematical basis, gives a strict solution and is characterized by high speed of operation, but its biggest disadvantage is the low resistance of the algorithm (there is a high risk of the algorithm converging to the local optimum). In the case of evolution and swarming algorithms, despite the longer waiting for a solution, the risk of the algorithm converging in the local optimum is much lower compared to the gradient method.
6. Acknowledgement
The research is co-financed under the program of the Ministry of Science and Higher Education "Implementation Doctorate" in accordance with the contract RJO15/SDW/002-04 concluded on 29.06.2020

REFERENCES
[1] http://www.ansys.com/
[2] Ansys fluent adjoint solver user’s guide
[3] Al-Obaidi A Sh, Sun L C, Calculation and optimization of the aerodynamic drag of an open-wheel race car. Journal of Engineering Science and Technology, EURECA 2013 Special Issue August, 1-15, 2014.
[4] Singh R, Golsch, K, A Downforce Optimization Study for a Racing Car Shape, SAE Technical Paper 2005-01-0545,
[5] Anderson J, Computational fluid dynamics. McGraw-Hill, New York, 1995.
[6] Malasekera W, Versteeg H K, An introduction to computational fluid dynamics. Pearson Education Limited, New Jersey, 2007.
[7] Lanfrit M, Best practice guidelines for handling automotive external aerodynamics with fluent. Fluent Deutschland GmbH, Darmstadt, 2005.