Use of the CFD for the Aerodynamic Optimization of the Car Shape: Problems and Application

Giovanni Lombardi, Marco Maganzi
Department of Aerospace Engineering, University of Pisa, Italy

Ferdinando Cannizzo, Enrico Cardile
Ferrari S.p.a., Maranello (Modena), Italy

ABSTRACT
In the initial phase of a car project, difficulties arise from the high number of parameters involved. A systematic aerodynamic analysis taking into account the effects of all these parameters appears to be difficult. In the analysis through numerical optimization, an aerodynamic code is coupled in a loop with an optimization routine, to automatically manage the values of the design variables, with the aim of minimizing a given objective function. This approach is extremely flexible, and capable of meeting multi-disciplinary requirements. The developed project has demonstrated the applicability of optimization procedures in the context of automotive industry by using CFD for the aerodynamics solver. The integration of the aerodynamic optimization in the design phase allows engineers to interact with the other groups without excessive delays. It becomes possible to search for solutions that do not negatively affect car style or performance, while providing a high degree of efficiency and safety.

1 INTRODUCTION

The traditional design processes cannot longer be met competitive with the increasing performance requirements and the economical pressure to increase efficiency of ground transportation vehicles. Current practice is to move the design of complex equipments away from a process involving a sequence of specialist departments and to emphasize its multidisciplinary nature through the use of integrated product teams. These commercial trends, together with the immense volume of design, manufacturing and maintenance data inherent to complex modern equipments, demand for a heavily computerized environment. Multidisciplinary Design and Optimizations (MDO) envisions a parametric description format of input data, which will generate, for a specific set of values of the parameters, a new vehicle description that in turn is used to generate input for Computer Aided Engineering, including Computational Fluid Dynamics (CFD).

The aerodynamic design plays a crucial role in the development phase of new automotive configurations and, due to its intrinsic complexity, the designer needs as much aids as possible to strengthen his/her choices and discard unsuitable solutions. In this context, the possibility of evaluating performances of different configurations is of utmost importance;
however, difficulties arise due to the high number of geometrical parameters involved, which are necessary for defining each configuration. A systematic analysis taking into account the effects of all these parameters is very difficult, given the complexity related to both aerodynamic load evaluation and the assessment of mechanics, stylist, commercial and others requirements.

A direct numerical optimization technique may be satisfactorily employed to find one’s way through this complex survey. Indeed, with this kind of aid the designer has a great flexibility in the choice of the design variables and the problem may be addressed systematically. The activity on the study of optimization has significantly increased over the last years, driven by advances in computational methods and improvement in computer performances, and this aspect has been particularly significant in the aerodynamic design, especially for land and flying vehicles. Several examples of this trend can be found, for instance, in Ref. 1. From the analysis of the existing bibliography, two different aspects can be highlighted.

The first aspect is the need to improve the accuracy and the validity range of the results, to obtain a realistic representation of the aerodynamic flow; this implies using a sophisticated flow solver within the optimization procedures. The second aspect is the requirement to obtain the results in short time. These two aspects have the common targets to achieve an improvement in the configuration performances and to reduce the “time to market”. Clearly, they act in opposite directions, because high accuracy flow solutions imply long computing time. As a consequence, an important research activity on the optimization techniques is in progress, especially in the aeronautical field (see Ref. 2 for a review).

Different methodologies have been proposed to solve this classical accuracy-time dilemma. In general the results are procedures with high accuracy coupled to long processing time or, conversely, rapid time responses obtained with simplified aerodynamic solvers. An example of the first type is described in Ref. 3, while an example of the second type can be the procedure described in Refs. 4 and 5, where a simple potential flow solver was used.

An approach to increase the obtain a good compromise between accuracy and time is that to use “mixed” optimisation algorithms, as described, for instance, in Ref. 6, where a description of the different optimisations algorithms are also presented.

A more recent example that combines sophisticated flow solvers and efficient optimization techniques, applied to 2D sails, is described in Ref. 7: the technical strategy is to merge together CFD and numerical optimization, thereby facilitating a much broader utilization of these simulation technologies in vehicle design. This procedure gives very good results, but it is related to a specific problem and flow solver, and, therefore, not indicated for industrial applications.

The potential impact of optimisation procedures extends across many aspects of vehicle engineering. Fluid dynamic analysis, including heat transfer, is the basis of design not only for the external shape of the vehicle, but also for the prime mover and power train (cylinders, valves, intake and exhaust systems, transmission, and cooling), passenger comfort and climate control (noise reduction, heating, ventilation and air conditioning), and subsystems (such as windscreen de-icing). Automatic aerodynamic optimization can be used in the context of any of these design tasks by helping to achieve the best possible solution in each case, while simultaneously reducing the duration of the design cycle and time to market.

In the analysis through numerical optimization, an aerodynamic code is coupled in a loop with an optimization routine, to automatically manage the values of the design variables, with the aim of minimizing a given objective function. This approach is extremely flexible, and capable of meeting multi-disciplinary requirements. Therefore, in the past, in the
collaboration between the Department of Aerospace of Pisa and Ferrari, an advanced, integrated design and development environment for optimizing car aerodynamics, under certain geometrical and physical constraints, was developed (Ref. 8).

The results were encouraging, but there was the problem that a potential flow code was used in Ref. 8; obviously, the capabilities of this model are restricted. The important increases in both the CFD algorithms and computing capabilities, suggested to possibility to use RANS solver in the optimisation procedure, maintaining a simple “black box” scheme for the procedure. Therefore, an activity to insert in the procedure a RANS code was carried out. In a first phase the procedure was developed for aerodynamics bodies (the keel of an America’s Cup yacht), characterised by a flow with not significant separated zones, and therefore less complex to analyse (Ref. 9).

In this paper the general scheme of the procedure is described, and an example of the capability to improve the aerodynamic characteristics of a high performance car is presented. By using this approach, increases in the results are obtained both in the initial phase of the project, the definition of the car shape, and in the final phase, in order to optimize the details.

2 METHODOLOGY APPROACH

In the analysis through direct numerical optimization, an aerodynamic code is coupled in a loop with an optimization routine, in order to automatically manage the values of the design variables – typically concerning geometry modifications – with the aim of minimizing a given scalar quantity (objective function). This approach is extremely flexible, and capable of meeting multi-disciplinary requirements. The general flow chart of the procedure is shown in Fig. 1, in the simple “black box” version. In this approach, any block is independent, and can be easily substituted with a different tool (for instance, to use a different flow solver, or a different CAD representation). The advantage of this scheme is evident: a great flexibility, essential for industrial applications, is obtained.

![Flowchart of the optimisation procedure](image)

The optimisation procedure requires the geometry definition in a parametric form, in order to
manage the geometry modification in an automatic way. This represents one of the more critical points of the optimisation procedure with RANS methods. In fact, the number of parameters necessary for the car shape definition, independently by their form, is very high (some thousand of parameters). This fact leads to two main problems:

- It is impossible to directly manage the geometrical parameters as design variables.
- The management of the parameters defining the general shape (the “volume” of the car) is completely different from the management of the parameters defining the “details” of the car.

To solve these problems two different approaches are identified, and the optimisation procedure is different when the optimisation of the “volumes” of the car are required (typically in the first phase of the project) or if the optimisation of a detail (typically in the last phase of the project) are required.

For the “volume” problem a procedure for the parametric definition of the car geometry was developed: at the present, the external shape of the car, without details, can be satisfactory represented by means of some hundreds of parameters, and it is possible to make the optimisation procedure on the entire car, following the “standard” procedure as indicated in Ref. 9 for non lifting bodies.

When the “detail” optimisation is required the problem appear quite different. In fact, in this case, only a limited part of the geometry is involved in the modification. This implies two facts:

- Only a small part of the geometry can be varied during the optimisation process.
- The parameters defining the details under considerations can be usually used directly as design parameters.

Therefore, it was decided to approach the problem in a different way, following the procedure that will be described in the following, for the example of the optimisation of the rear diffuser.

3. THE NUMERICAL APPROACH

From the aerodynamic point of view the problem is related to the evaluation of lift and drag in incompressible flow, with a complex geometry characterized by extended zones of separated flow. In particular, the more important aspect is the capability to evaluate the small differences that occur in the aerodynamics forces for configurations characterized by small differences. Therefore, an accurate method is necessary, and a RANS approach seems to be suitable.

Obviously, if a RANS model is used, the procedure will require very high computational capabilities in order to obtain the optimised configuration in a reasonable time. Therefore, this aspect assumes a crucial role in the activity.

3.1. COMPUTING CAPABILITIES

A Linux cluster comprised of 16 SUN Fire AMD Opteron Servers was used for the simulations. Each server is a SUN Fire X4100 equipped with 2 AMD Opteron 285 (Dual Core) processors and 8GB RAM each. All nodes are connected with a standard IP connection based on Gb Ethernet to the backbone network. In addition, they are connected through a high-performance Myrinet 2000 switch. Myrinet provides a low-latency, high-bandwidth and low-cost solution to connect the system.

A SUN Fire X4200, equipped with 2 AMD Opteron 252 (single core) processors and 16Gb
RAM acts as a front-end to the cluster.

For the software, a full installation of Linux SUSE SLES9 SP3 for AMD64 processors is provided on all the nodes. Additional drivers for Myrinet (GM 2.0.26) and interface control programs are installed on the systems to provide Myrinet connection.

3.2 THE SOFTWARE
Due to the flow complexity, the aerodynamic solutions require a RANS solver. For this, FLUENT® was used in the present phase of the activity. Clearly, the use of a different RANS solver does not affect the layout of the procedure.

The parametric CAD model used for the optimization loop was built with CATIA® V5R16, while an automatic procedure for the grid generation was developed with GAMBIT®.

The optimisation procedure was driven by the multidisciplinary optimization code modeFRONTIER®.

4 EXAMPLE OF DETAIL OPTIMISATION PROCEDURE
In the following an example of optimisation of a detail is described. The detail considered is the rear diffuser of the Ferrari F430. The present configuration is shown in Fig. 2, while in Fig. 3 the complete CAD representation is shown.

Because of the interest is focused on the rear diffuser, only a part of the geometry will be varied during the optimisation process. Therefore, the CAD geometry is divided in two parts: the “fixed” part (red in Fig. 4) and the “detail” part (yellow in Fig. 4). Only the “detail” part is parametric, and its geometry will change during the optimisation process.
Therefore, the grid volume is subdivided in two parts (Fig. 2): the fixed part, representing the geometry of the car not changing in the optimisation, and the parametric part (yellow), defining the rear diffuser, and changing during the optimisation following its parameters variations. At any step of the optimisation, the grid volume is obtained merging the fixed part with the parametric one.

The general flow chart of the optimisation procedure, shown in Fig. 1, is then modified as reported in Fig. 6.

**Fig. 4 – Scheme of “fixed” and “detail” parts in the CAD geometry**

**Fig. 5 – The grid volume merging procedure**

**Fig. 6 - Flowchart of the optimisation procedure for the “detail” approach**
The advantages of this approach is that the pre-processing phase (modification of the CAD geometry and generation of the corresponding grid), necessary at each step of the optimisation procedure, is made only on the parametric volume (the rear diffuser). As a consequence, important reductions in the computational time and memory requirements are obtained. Furthermore, the grid of the main part of the car is not modified: therefore, there are no problems of undesired shape modifications.

4.1 The parametric representation
The corrected representation of all the details involved in geometry would require a large number of grid cells (more than 20 millions of cells). This, clearly, would lead to a very high time to carry out the optimisation procedure. In order to reduce the time, it was decided to simplify the geometrical representation of small details. Anyway, it is necessary to take into account that, in an optimisation procedure, we are not interested in the “absolute” values, but only in the differences in the aerodynamics performances related to the variation of the parameters. Furthermore, an excessive refinement in the detail representation could give an undesirable numerical sensitivity to the parameters representing small geometry modifications.

Therefore, it was decided to simplify the geometry of the rear diffuser, as indicated in Fig. 7. However, after the optimisation procedure, a RANS evaluation with a very refined grid on the starting and modified geometries was carried out, in order to verify, with high accuracy, the results of the optimisation procedure. This activity will be described in the following section.

In particular, as can be seen by comparing parts a) and b) of Fig. 7, in the simplified geometry some surfaces with very small curvature radius are eliminated, and the external sector is not considered lower than the car backside.

The geometry shown in Fig. 7b can be represented by the 13 parameters indicated in Fig. 8; a complete description of these parameters can be found in Ref. 10.

Fig. 7 – Original and simplified CAD geometry of the rear diffuser
With this number of parameters it is reasonable to approach the optimisation procedure considering the parameter as design variables.

4.1 The RANS evaluations
For the RANS evaluations, carried out at a car speed of 35 m/s, the code FLUENT was used. After a sensitivity analysis, a grid domain of 22 car lengths, 20 car widths and 15 car heights was chosen, and a $k-\varepsilon$ realizable turbulence model was used.

Because of the symmetry of the problem, only one half of the car was represented. About 2.5 millions of cells were used in the computational domain.

4.2 The control of the optimisation procedure
The optimisation procedure was driven by the code ModeFrontier, following the scheme reported in Fig. 6. By means of this code, starting from the initial geometry, at any step were automatically managed the following codes:

- CATIA for the CAD modification of the diffuser geometry
- GAMBIT to generates the new grid for the parametric volume and to merges it with the fixed grid
- FLUENT for the RANS evaluation

The used optimisation algorithm was the Moga-II (Multiple – Objective Genetic Algorithm II), a genetic algorithm implemented in ModeFrontier.

The object function was related to both the total drag and the vertical download of the car. Constrains were imposed on the maximum variations of the parameters, and, in particular, on the minimum volume of the gearbox and of the lateral side of the diffuser. A further constrain was required on the vertical download, that cannot be reduced ($C_Z \leq C_{Zref}$).

By using the previously described computer, about 26 minutes were necessary for each step, with 13 minutes for the RANS solution.
4.3 Results
A number of 42 initial base data (DOE, Design Of Experiments) were used, with 16 new populations. A total number of 570 different geometries were evaluated. Results are summarized in Fig. 9, reporting the scatter chart in terms of variation, on respect to the original configuration, of the drag and vertical load coefficients.

![Scatter Chart](image)

**Fig. 9 – The scatter chart (points). Negative values of c_z = download force**

The configurations satisfying the vertical download constrain are 202, and in Fig. 9 are in green. By moving along the Pareto frontier, two interesting configuration can be identified: the "Low Drag" and "High Load" configurations. The first is characterised by a drag that is as lower as possible (without increase in vertical down-load), the latter by a high value of the vertical download, with a low increase in drag.

The results for these configurations are summarised in Tab. 1, where, for clarity, the difference on respect the starting configuration are multiplied for 100 ("points").

<table>
<thead>
<tr>
<th></th>
<th>Δ c_x</th>
<th>- Δ c_z</th>
</tr>
</thead>
<tbody>
<tr>
<td>Low Drag</td>
<td>-1.52</td>
<td>0.30</td>
</tr>
<tr>
<td>High Load</td>
<td>0.26</td>
<td>1.94</td>
</tr>
</tbody>
</table>

**Tab. 1 – Results of the optimisation procedure**

The “Low Drag” configuration is shown in Fig. 10 (in red) compared with the starting one (yellow).
It is characterised by a more complex shape of the inner sectors, and opposite curvature of the gearbox lateral side. In this way, as shown in Tab. 1, a reduction of drag coefficient of about 1.5 points is obtained, with also a small increase in the vertical download, 0.3 points. The “High Lift” configuration is shown in Fig. 11 (in blue) compared with the starting one (yellow).

It is characterised by a more complex shape of the inner sectors (but less evident than in the “Low Drag” configuration), and a curvature of the gearbox lateral side that remains the same. As shown in Tab. 1, an important increase in the vertical download, about 2 points, is obtained, but with an increase in drag coefficient of 0.26 points. Anyway, the increase in drag is obtained with a high efficiency: in fact, the ratio between download and drag increases is about 6.5.

5 VERIFICATION OF THE OPTIMISATION RESULTS WITH REFINED GRIDS

As previously highlighted, to make possible the optimisation procedure in a reasonable time, the geometry of the rear diffuser was simplified. In order to verify, with high accuracy, the results of the optimisation procedure, a RANS evaluation with a very refined grid on the starting and modified geometries was carried out.

The CAD geometries, with the rear diffuser accurately defined, are shown in Fig. 12. As can be seen, the geometrical representation is completely detailed.
Fig. 12 – CAD geometry representation for the refined grids

The RANS evaluation were carried out with the same settling, with a grid with about 15 millions of cells (on the half car). The results are shown in Tab. 2.

<table>
<thead>
<tr>
<th></th>
<th>Optimisation</th>
<th>Refined grid</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$\Delta c_x$</td>
<td>$-\Delta c_z$</td>
</tr>
<tr>
<td>Low Drag</td>
<td>-1.52</td>
<td>0.30</td>
</tr>
<tr>
<td>High Load</td>
<td>0.26</td>
<td>1.94</td>
</tr>
</tbody>
</table>

Tab. 2 – Comparison with refined grids

From Tab. 2 it is evident a not negligible difference, related to the sensitive to the geometrical details representation, but the tendency is clearly maintained.

Probably, for practical application, a more refined grid could be used in the optimisation process. This is possible in industrial application: in fact, the entire procedure described in Sec. 4 required about 10 days, and it is appropriate to take into account that the increases in computational capabilities are so important that this time is expected rapidly reduced.

6 CONCLUSIONS

The developed project has demonstrated the applicability of optimization procedures in the context of automotive industry by using CFD for the aerodynamics solver. The integration of the aerodynamic optimization in the design phase allows engineers to interact with the other design groups without excessive delays. It becomes possible to search for solutions that do not negatively affect car style or performance, while providing a high degree of efficiency and safety.

The reduction of industrial costs is significant: in principle it is no longer needed to build many different models, to be subject to wind tunnel measurements: it is sufficient to test the final optimized ones.

The system satisfies the main requirements given by the end user. In particular, it is capable of yielding shapes having more favourable aerodynamic characteristics with very small geometry modifications, thus keeping the style substantially unchanged. Obviously, by this means it is not expected that very high decreases of the vertical load may be obtained. The objective is to find rapidly the best configuration within a certain small range of shapes that are considered to be acceptable from the industrial constraints, without resorting to time-consuming and expensive wind tunnel tests. In other words, while it is relatively easy, for an
experienced operator, to devise significant changes to a shape producing substantial increases in the aerodynamics performances, but also considerably altering the car layout, it is almost impossible to predict the effects of very small modifications of the geometry. In any case, the cost and the time for a wind tunnel campaign for this purpose would be prohibitive, and also the classical use of CFD, without the optimisation procedure, appears extremely time consuming and less accurate.

Finally, it should be emphasized that a numerical procedure of this kind can never be regarded as a completely automated tool; user’s experience and external control on the process are always needed to obtain the best results.

ACKNOWLEDGEMENTS

Thanks are due to FLUENT for the important support in the use of CFD and to AMD for the availability of an extremely powerful computing system. Thanks are also due to A. Ciampa and E. Mazzoni (INFN of Pisa) to render the computing system very efficient and easy to use, resulting from their research activities on computing networks, applied to our cluster.

REFERENCES