MULTIOBJECTIVE DUCT OPTIMIZATION WITH OPEN SOURCE CFD SOLVER

[1]Fabio Vicenza, [1]Daniele Obiso, [2]Stamatina Petropoulou
[1]Phitec Ingegneria Srl, Italy, [2]ICON Technology & Process Consulting Ltd, United Kingdom

KEYWORDS: adjoint optimization, multi-objective, open source, CFD, industrial design

ABSTRACT

During the development of duct systems, engineers often face various geometry and performance constraints forcing them towards non-optimal designs.
Efficiency in duct performance can be measured in various ways. Minimizing the pressure loss between the inlet and the outlet of the channel is the most significant requirement but improvements in other areas may be equally important. Further requirements could be the mass flow balance for multiple outlets or flow uniformity on the outlet.
The adjoint solver of iconCFD®* described in [2-4] has been used for the topology optimization of ducts. A new multi-objective approach has been developed integrating the latest features of the iconCFD Optimize software module and Beta CAE ANSA. Multiple objectives can be treated according to importance by the use of weighting factors. The automation of the optimized geometry extraction has been handled by a developed method based on Beta CAE ANSA. An industrial application example has been studied to prove the effectiveness of this solution.

1. INTRODUCTION

The aerodynamic design optimization of ducts systems (e.g. HVAC systems, cooling systems, etc) has acquired an important role in the automotive industry. Computational Fluid Dynamics (CFD) has proven to be a valuable tool for predicting duct performances (pressure losses and relevant flow features) both in early and advanced design phases. Standard duct optimization usually involves the use of black-box type optimization methods, coupled with CAD parameterization or geometry morphing techniques. A valuable alternative to the classical optimization tool is the adjoint optimization technique. The adjoint method has a number of advantages relative to other gradient-based methods. Apart from its rapid convergence, it provides the gradients of the cost function in a way that the computational effort required for this calculation is independent of the number of design variables [1].
The iconCFD Optimize software module developed by ICON Technology & Process Consulting, UK [2] contains adjoint optimization solvers addressing both shape optimization (external aerodynamics) and topology optimization (internal flow). The adjoint solver of iconCFD has been widely used for the topology optimization of duct systems [3-5] aiming at the optimization of a single cost function (pressure losses or flow uniformity). In the current study, a new multi-objective approach has been developed, integrating the latest features of iconCFD Optimize software module and Beta CAE Systems ANSA [6]. This approach is discussed in detail in the following paragraphs through its application on an industrial case of interest: automotive central cabin vent duct. The geometry was provided by Automobili Lamborghini S.p.A..

2. BASE GEOMETRY AND DESIGN SPACE

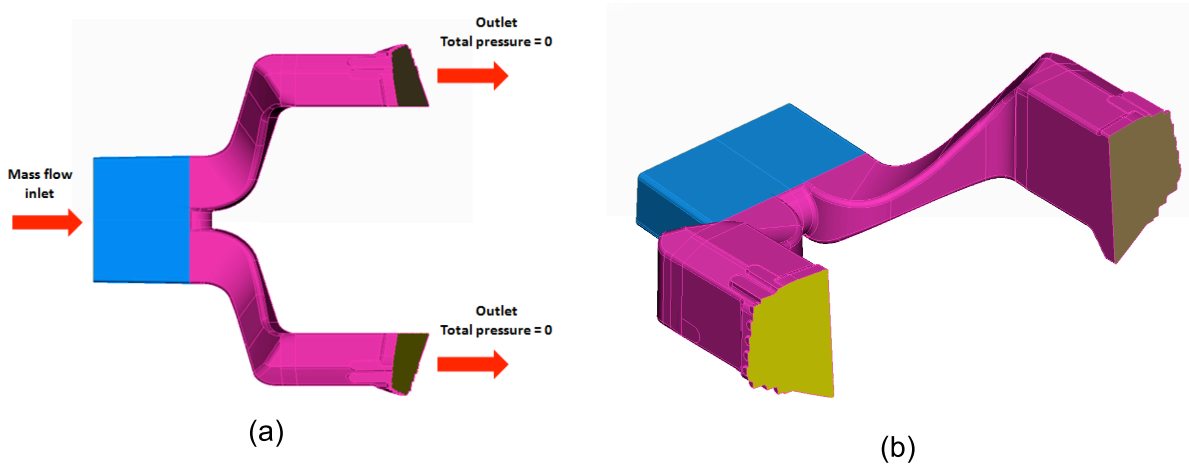


Figure 1 – Base geometry: (a) top view; (b) isometric view

The base geometry of the automotive cabin vent duct to be optimized, shown in figure 1, was provided by Automobili Lamborghini S.p.A. (LB). The original LB design performance has been evaluated with a steady state RANS solver on an unstructured Hex-Dominant mesh with prism layer. The k-ω SST turbulence model was used.
A mesh convergence study has been performed in order to assess the independency of the grid on the accuracy of the solution.
Figure 2a shows the relevant flow features appearing in a cross section of the duct and the velocity contour on the outlet area. A strong separation occurs after the second bend of the duct, leading to a strong non-uniformity of the velocity on the exit surface of the duct (Figure 2b).

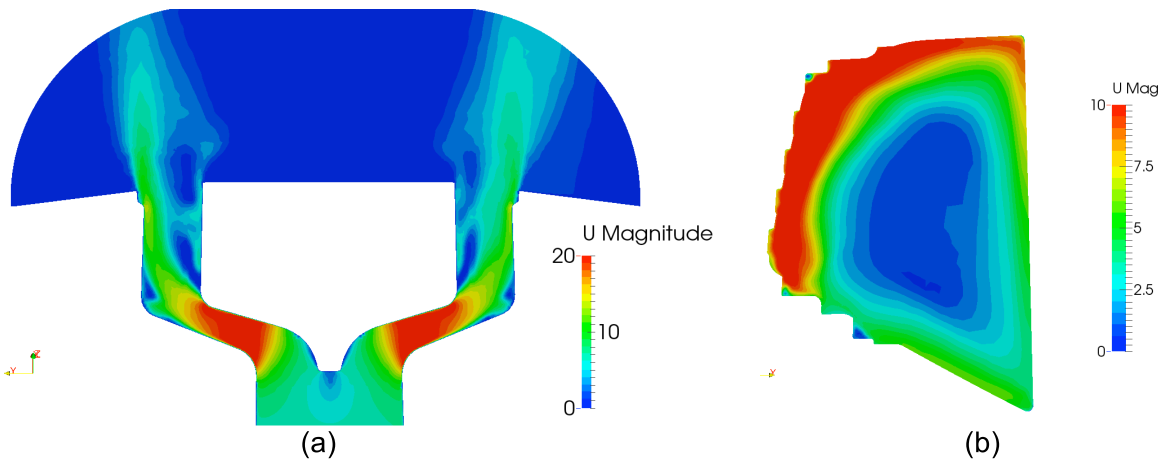


Figure 2 – Velocity magnitude contours [m/s] on: (a) transversal section of the duct; (b) outlet section.

The performance of the duct was evaluated using the following two parameters:

1. Net inward flux of energy (Eqn. 1)

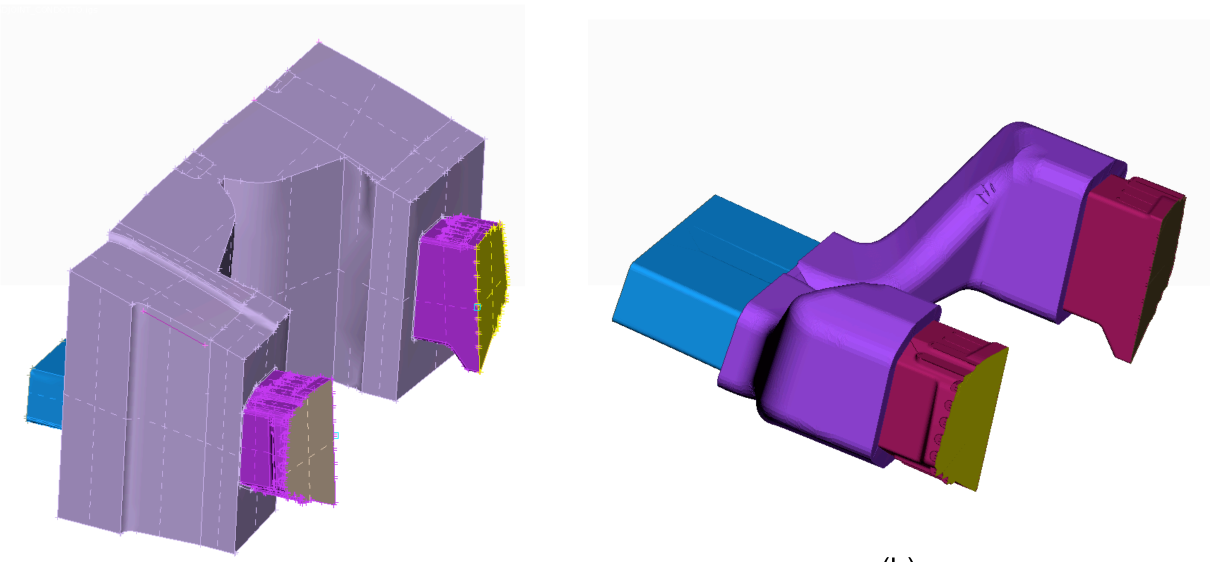
$$J_1 := -\int_{inlet} dS\left(p + \frac{1}{2}v^2\right)\boldsymbol{v}\cdot\boldsymbol{n}$$

2. Flow uniformity index at each outlet (Eqn. 2)

$$J_2 := \int_{outlet} dS\frac{1}{2}\left(\boldsymbol{v} - \boldsymbol{v}^{\boldsymbol{d}}\right)^2$$

where $\boldsymbol{v}^{\boldsymbol{d}}$ is the desired velocity on the surface.

The original LB design shows a net inward flux of energy of 4.40 $W/m^3/kg$ and a flow uniformity index at each outlet of 0.051 m^4/s^2.
In order to take advantage of the benefits of the topology optimization it is necessary to start from a larger design space and allow the method to predict the final flow path.
Being at an early design stage, there is a lot of freedom in the definition of the available design space. Moreover some manufacturing constraints apply: on the z-axis direction the design is restricted by the upper part of the dashboard, in y-axis direction the central space between the two branches of the duct should not be modified. Due to further manufacturing constraints, the duct close to the outlet is excluded from the optimization. This is in order to avoid interfering with the dashboard style and vent flap mechanics. An additional manufacturing constraint is that the flow path should be divided into no more than two channels (one for each outlet). Multiple small channels would be more difficult to manufacture.
The design space that is obtained by fully using the allowed manufacturing freedom can be seen in Figure 3a. For CFD purposes the starting geometry that was used for the topology optimization is an offset of 10mm of the original geometry as it can be seen in Figure 3b.
The procedure described below was tested with several design spaces, and every design led to consistent improvements in the performances of the duct. The design described below gave the best performance improvements.



(a) (b)

Figure 3 – (a) Available design space; (b) design space used for CFD purposes

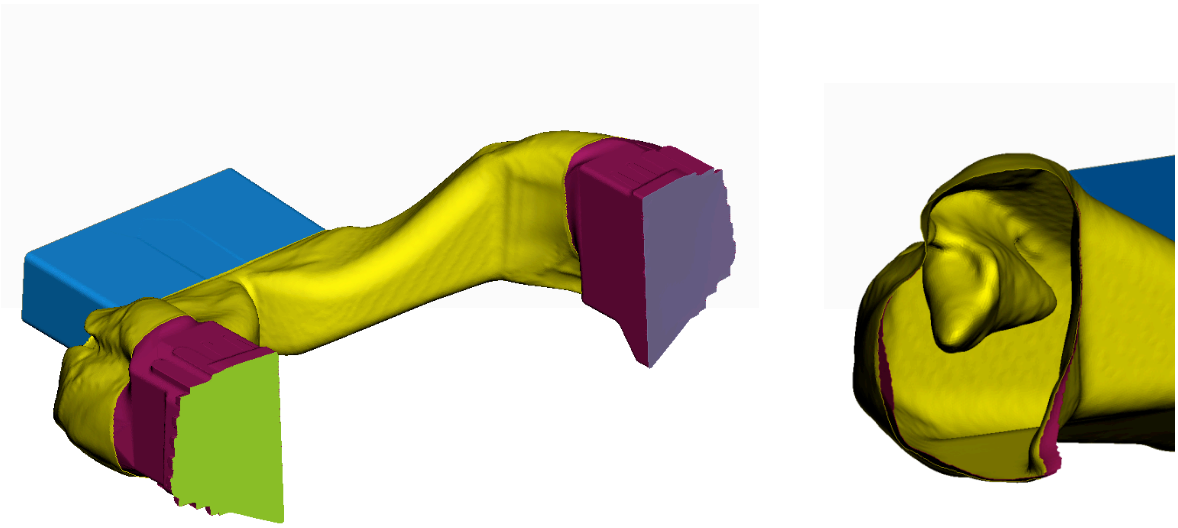
3. MULTIOBJECTIVE ADJOINT OPTIMIZATION AND OPTIMIZED GEOMETRY EXTRACTION

The topology adjoint method solves the constrained optimization problem by blocking “non-productive” regions of the domain with regards to the objective (cell porosity update). In the current analysis a multi-objective approach has been developed, optimizing for both the net inward flux of energy (Eqn. 1) and the flow uniformity index at the outlet (Eqn. 2). Multiple objectives can be treated according to importance by the use of weighting factors.

As the topology optimization progresses, several regions of the domain are blocked with porosity allowing an optimal flow path to form the inlet to the outlet. Analysing the flow field resulting from the optimization provides an indication of the final improvement that can be obtained.

After the optimization is complete an iso-velocity surface is automatically extracted from the optimized flow field and loaded in ANSA. At this stage several automatic ANSA routines are used to delete “bubbles” (small porosity regions detached from the boundaries), remove double or collapsed elements and to connect the iso-contour to the initial inlet and outlet sections of the domain. A surface smoothing is then performed in order to improve the surface quality of the resulting STL description of the new geometry (Figure 4).

The resulting STL description of the new geometry is then tested with CFD (Figure 5) to assess the performance improvement. In the current example the energy flux at the inlet reduced from 4.40 $W/m^3/kg$ of the original design to 1.16 $W/m^3/kg$ and the flow uniformity index at each outlet is reduced from 0.051 m^4/s^2 of the original design to 0.019 m^4/s^2.



(b)

(a)

Figure 4 – STL description of the optimized geometry: (a) general view; (b) detail of the fin shape inside the second bend of the duct.

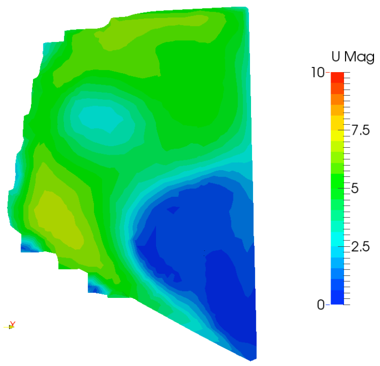


Figure 5 – Velocity magnitude contours [m/s] on the outlet section of the optimized CFD geometry.

4. GENERATION OF THE NEW CAD DESIGN AND PERFORMANCE ASSESMENT

Following the optimization procedure described above, a consistent performance improvement is obtained. However, the resulting shape cannot be produced with traditional manufacturing techniques. At this stage, an engineering approach should be selected to either shift to a modern manufacturing technique such as 3D printing or alternatively to add a further step to the optimization process in order to obtain a more traditional shape from the optimized duct.
This further step is performed in ANSA. The optimized STL description is cut using several parallel planes normal to the outlet section of the duct (Figure 6). The information about each cross section is extracted and every section is substituted by an equivalent rectangle, which has the same area and centre of gravity. The resulting cross sections are then connected with surfaces and finally a fillet is added on the 4 corners of the resulting geometry. Figure 7 shows the CAD geometry resulting from this process.

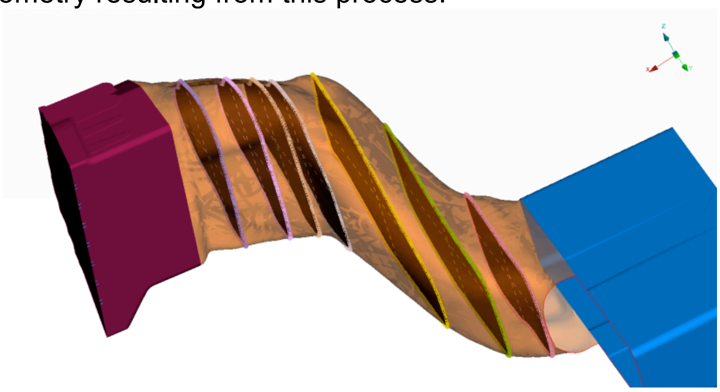


Figure 6 – Extraction of the cross sections information from the STL description

The performance improvements on the final CAD design are assessed with CFD (Figure 8c and 9). As summarized in Table 1, the CFD simulation on the resulting duct shows a further improvement on the energy flux at the inlet (1.14 $W/m^3/kg$) than the previous design but a slight increase in the flow uniformity index at the outlet (0.024 m^4/s^2).

Design	Inlet Energy Flux [$W/m^3/kg$]	Flow uniformity index [m^4/s^2]
Original LB Design	4.40	0.051
Optimized CFD	1.16	0.019
Optimized CAD	1.14	0.024

Table 1 – Comparison of the three design performances.

The overall improvement with respect to the original design is still consistent and a new CAD of the internal surface is ready to be passed to the design department in STEP or IGS format.

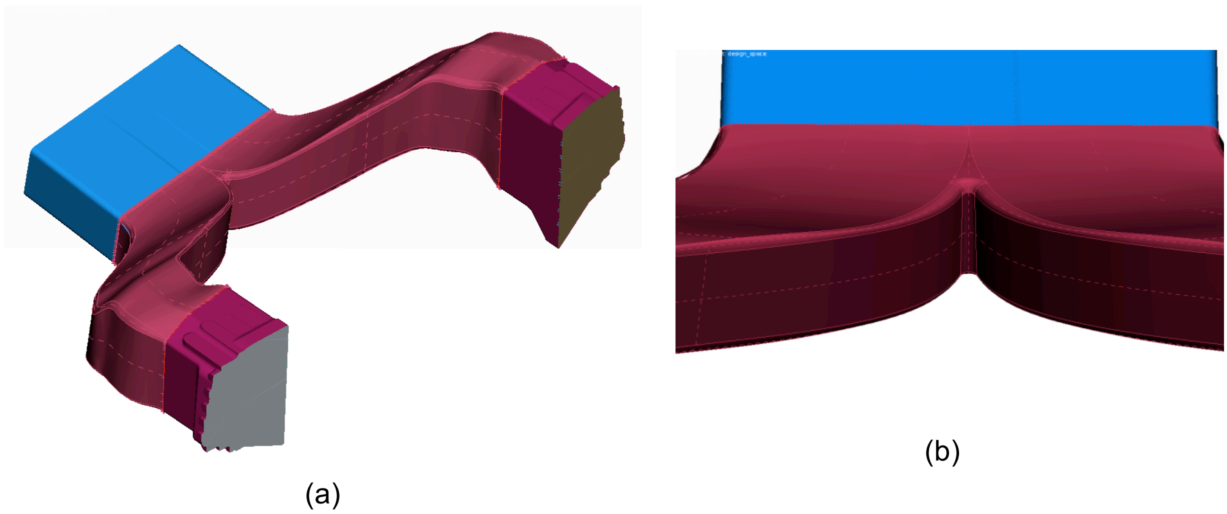


Figure 7 – CAD of the optimized geometry: (a) general view; (b) detail of the central area.

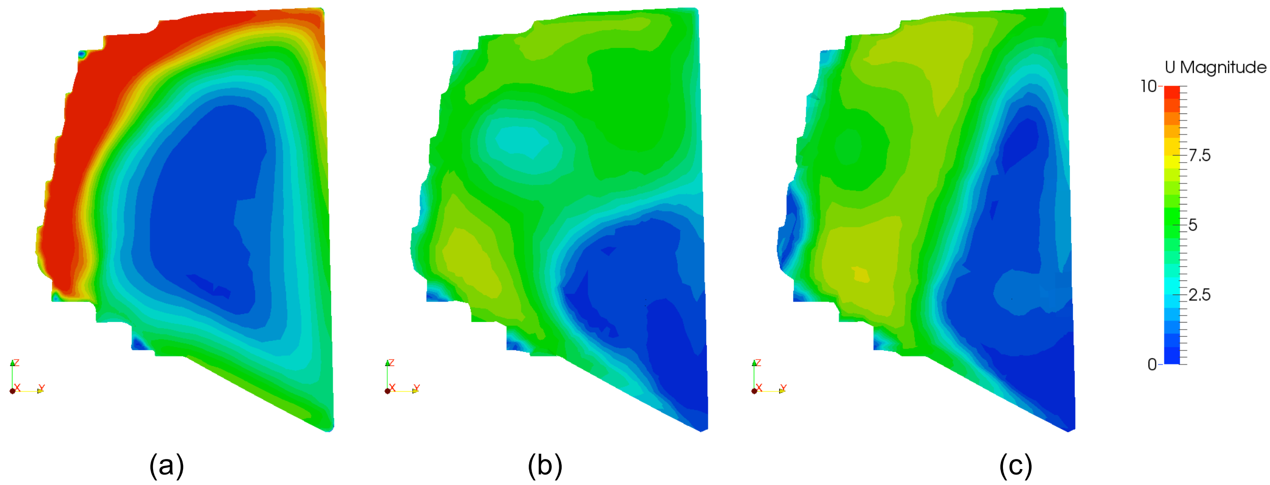


Figure 8 – Velocity magnitude contours [m/s] on the left outlet surface of: (a) original LB design; (b) optimized CFD; (c) optimized CAD.

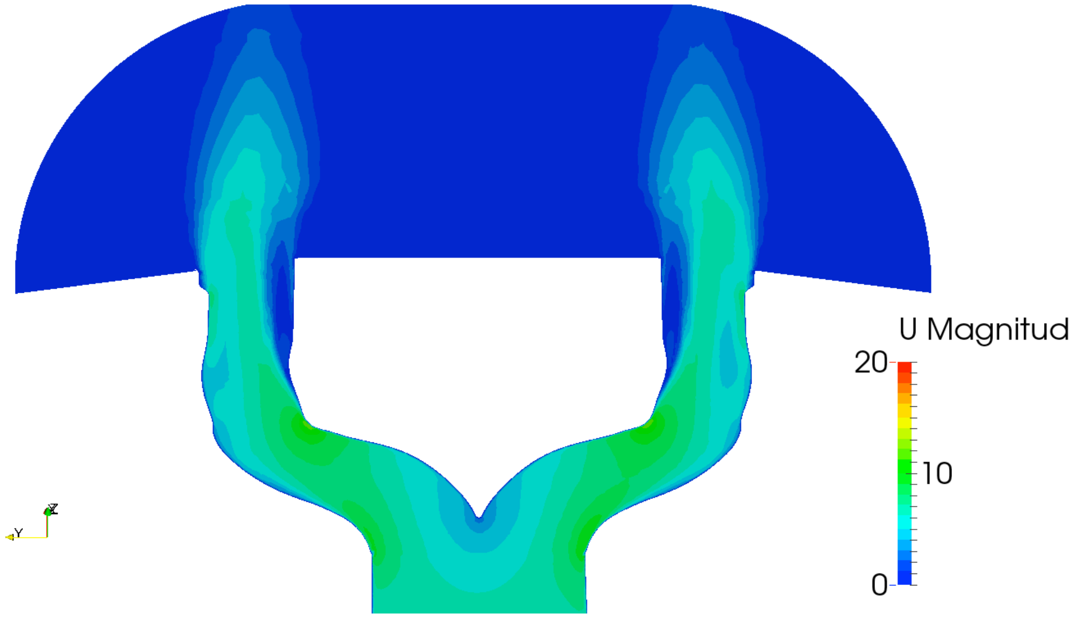


Figure 9 – Velocity magnitude contours [m/s] on a transversal section of the final optimized duct.

7. CONCLUSIONS

The multi-objective adjoint optimization capabilities of the iconCFD software suite, together with the ANSA geometry and STL surface handling capabilities, have proven to be an effective combination of tools for duct system optimization. In the current study, a complete duct optimization process has been developed. The efficiency of this process has been proven on a case of industrial interest: an automotive central cabin vent duct geometry provided by Automobili Lamborghini S.p.A.. After the optimization loop, a new CAD design of the interior surface of the duct is ready to be passed to the design department. The new design, when compared to the original, provides a 70% reduction in the pressure drop and a 50% increase in the flow uniformity at the outlet.
The optimization process is still not completely fine tuned but a high level of automation has been achieved. Two main research areas have been identified to further improve this procedure:

1. The extraction of the new STL geometry from the adjoint optimization results: the usage of the velocity magnitude iso-surface as a starting point for the generation of the new geometry usually does not delete all the unnecessary parts of the domain and requires some manual input from the user to remove them. In order to overcome these issues a new derived flow variable must be developed. This new variable must combine the velocity field and the adjoint optimization variables information, together with some geometric constraint information, in order to permit the direct extraction of the new STL geometry.
2. Extra automation of the complete process can be achieved by further developing the ANSA python scripting routines.

ACKNOWLEDGMENTS

The authors would like to thank Antonio Torluccio and Emiliano Dini of Automobili Lamborghini S.p.A. for sharing the base geometry of the duct, its operating conditions and information about the design space available.

REFERENCES

[1] S. Petropoulou, Industrial optimization solutions based on OPENFOAM®* Technologies, ECCOMAS CFD, Lisbon, Portugal, June 2010

[2] iconCFD® version 3.1.9 Technical User Guide, ICON Technology & Process Consulting Ltd.

[3] S. Petropoulou, Localised Adjoint Optimization – Better and Faster CFD-Driven Design, NAFEMS Optimization for Industrial Applications Seminar, Gaydon, UK, March 2012

[4] S. Petropoulou, Adjoint Solver Advances, Tailored to Automotive Applications, Flowhead Conference, Munich, Germany, 28th March 2012

[5] J. Palluch, S. Weickgenannt, M. Saroch (Friendship Systems, GER), B. Leroy, A. Zimmer (ICON, UK) Design Studies and Optimization of Functional Surfaces utilizing Open Source CFD, NAFEMS World Congress, Salzburg, Austria, 10th of June 2013

[6] ANSA version 15.2.3 User's Guide, BETA CAE Systems S.A., January 2015