FAST ROBUST DESIGN OPTIMISATION OF AN UNDER-HOOD COOLING DUCT

Francisco CAMPOS MSc DIC[1]

Paolo GEREMIA MSc[2]

Simon WESTON MSc DIC[1]

[1] **ICON, Rofel House, Colet Gardens, London, W14 9DH, UK**
[2] **ESTECO UK, PO Box 54248, West Kensington, London, W14 9AF, UK**

SUMMARY

This paper presents the results of an automatic design optimisation study and fast robust analysis performed by Icon on a cooling duct located in the under-hood region of a vehicle. The optimisation procedure consisted of generating parametric CAD models, performing CFD analyses in batch mode, and applying the multi-objective robust design optimisation product modeFRONTIER to find the optimal design layouts. A fast robust design analysis, based on response surface methods, was also performed to screen the candidate solutions and select an optimum configuration in terms of stability. Both methods, traditional and robust design analysis, are compared and discussed.

KEYWORDS

Multi-objective, robust design optimisation, modeFRONTIER, OpenFOAM, STAR-CCM+, duct, under-hood, Pareto frontier, pressure loss, topology, automatic mesh generation, parametric, mesh morphing, response surface, statistical analysis.

1: INTRODUCTION

The traditional CFD design process is often based on a "trial-and-error" type approach. Starting from an initial geometry, CAD changes are introduced manually based on results from a limited number of design iterations and CFD analyses. The process is usually complex, time-consuming and relies heavily on engineering experience, thus making the overall design procedure inconsistent, i.e. different "best" solutions are obtained from different designers.

FAST ROBUST DESIGN OPTIMISATION OF AN UNDER-HOOD COOLING DUCT

The introduction of a mathematical frame to find an optimum design through the use of the latest optimisation techniques "standardises" the design procedure and eliminates most of the drawbacks found in the traditional approach. In this sense, designers are exploiting the benefits of parallel execution of CFD analyses by taking full advantage of the most sophisticated IT frameworks available. More precisely, distributed computing based on high-performing CPU technology allows a massive use of optimisation tools, especially for designs involving computational fluid domains whose size was considered prohibitive until a few years ago.

In order to reduce the product development time and satisfy the growing design requirements to stay competitive in the market, new design approaches have been introduced. Consequently, designers are giving more and more importance to the quality of their work complying with the principle of finding the best solution with the minimum effort.

Design parameterisation plays an important role in this context, since the definition of design parameters leads to a very efficient modification of the shape of the fluid domain geometry. Following this approach, the designer can investigate the effects of geometry parameterisation on model performance and find out which is the optimal shape that meets all the requirements needed. This task is normally accomplished by using optimisation tools, allowing the parametric model to be automatically driven by the optimisation software in order to achieve the optimal solution. For this purpose, different techniques are currently employed, namely: parametric CAD model, topological optimisation and mesh morphing approach.

Parameterisation of CAD models in the optimisation loop is based on the fact that a finite number of design parameters can define the geometrical shape of the model by using parametric curves and surfaces, following Bezier and NURBS theory [1]. This approach is widely used by engineers since most of the CAD tools available in the market feature parametric definition of the geometrical model. However, to integrate this process in the CFD optimisation loop a fully automatic mesh generation tool is required, leading to additional time and difficulty in the process. This is especially true in cases involving complex geometries which could result in poor quality meshes, thus affecting the reliability of the CFD analysis stage.

Topological optimisation is based on the idea that cells with back flow can be retrieved and discarded, thus isolating the "healthy" cells which feature no flow recirculation problems. The overall space available for the fluid can then be shaped by switching off recursively the "bad" cells. The main advantage is that just a single CFD model is required and, most importantly, no parameterisation is required. The main drawback is that topological optimisation does not apply to all CFD problems (e.g. maximise the swirl of an impeller, maximise the efficiency of an airfoil, etc.).

Finally, the mesh morphing approach allows for the model shape to be modified with no need for parameterisation of the CAD model or automatic mesh generation. An existing mesh is deformed based on predefined actions, such as stretching or contraction of nodes, thus saving computational time on each optimisation loop. However, a significant number of mesh changing parameters are needed to perform an efficient mesh modification while avoiding issues related to excessive mesh deformation.

In this work, the first approach was applied in the optimisation of a cooling duct placed on the under-hood region of a vehicle, as an extension of previous work presented in [2] using mesh morphing. The work was also extended to include the effects of design stability through the use of robust design optimisation using the open-source CFD code OpenFOAM.

2: ROBUST DESIGN OPTIMISATION

Robust design optimisation allows the designer to find the best solution in term of performance stability by understanding the design responses to uncertainties on every design parameter. These uncertainties can be introduced in reality due to variability in the manufacturing process, weather conditions, etc.

Frequently, the optimal solution found by classical deterministic optimisation algorithms is not effectively the most stable solution, especially when working conditions change slightly and performance drops-off significantly. Consequently, the best robust solution cannot be always identified with the best global solution.

The deterministic approach used for standard optimisation is replaced by the stochastic approach, which applies a statistical distribution on the design variables around the nominal value of the design point considered. As a result, each variable is defined in terms of mean and standard deviation, and a single design is defined by a set of sampling points distributed according to the statistical distribution specified. The sampling points are created in order to define the fluctuations around the nominal value of the design point considered.

In terms of optimisation goals, the maximisation/minimisation problem has to take into account the mean and standard deviation of the objective function. Different approaches are applied. The classic maximisation problem is defined as:

$$\max \; f(x) = f(x_1, ..., x_n) \qquad (1)$$

where $x_1, ..., x_n \in \Re$.

The robust design maximisation problem as defined by Li and Padula [3] can be expressed as follows:

$$\max \ \bar{f}(x) = \frac{\sum_{i=1}^{N} f_i(x)}{N} \tag{2}$$

subjected to:

$$\min \ \sigma_{\mathrm{f}}(x) = \sqrt{\frac{\sum_{i=1}^{N} \left(f_i(x) - \bar{f}(x)\right)^2}{N-1}} \tag{3}$$

The $f_i(x)$ values of equations (2) and (3) are the performance functions calculated as a function of the x variables vector and for N sampling points.

As a result, the single-objective problem in (1) becomes more complex since the objective functions are re-defined as: (i) maximisation of the average value of $f(x)$ inside the variables distribution, and (ii) minimisation of the standard deviation of $f(x)$. This might result in prohibitive computational time required by the optimisation phase in order to get the optimal robust solution; therefore different techniques were developed and different approaches considered.

An innovative criterion is presented here based on a different approach which defines the maximisation problem as follows:

$$\max \ \left(\bar{f}(x) - 2\sigma_f(x) \ \right) \tag{4}$$

This could be considered a precautionary criterion because it preserves the optimiser from low values for f, and, at the same time, looks for low standard deviation values, by applying a confidence interval of 95% which corresponds to $2\,\sigma_f(x)$. Although the choice of confidence interval is somewhat arbitrary, in practice 90%, 95%, and 99% intervals are often used, with 95% being the most commonly used.

The criterion defined in (4) was successfully tested as a fast robust design optimisation methodology for use with CFD analysis, as stated in the example presented in Section 3. Different strategies were considered and then proposed as alternative solutions to the common robust design approach described by equations (2) and (3), as a way to shorten the computational time required while maintaining a good reliability of the results achieved. For this purpose, Response Surface Methodology (RSM) as well as state-of-the-art optimisation algorithms were employed.

FAST ROBUST DESIGN OPTIMISATION OF AN UNDER-HOOD COOLING DUCT

3: APPLICATION CASE: Optimisation of an Under-hood Cooling Duct

The duct geometry in this example is located in the under-hood of a vehicle and channels air from the side-grill (as shown in Figure 1) and directs it over the transmission for the purpose of cooling. Like most under-hood regions in modern vehicles there is a very compact and crowded compartment with complex paths for air movement. Providing convective cooling with a side-duct is considered beneficial for the reduction in the transmission surface and oil temperatures.
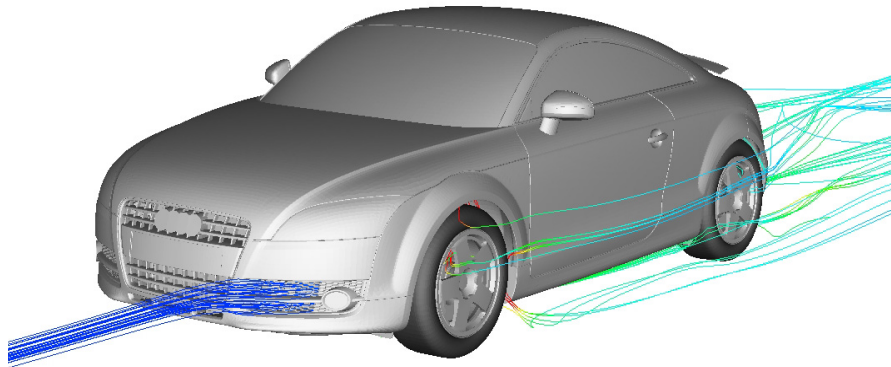
**Figure 1: Side grill inlet on a typical production car
(Presented with kind permission from Audi AG).**

This work aims at finding out the optimal shape design for the cooling duct. The optimisation problem was stated by means of two objective functions defined as: (i) minimisation of the pressure drop between the inlet and the outlet of the duct, and (ii) maximisation of the outlet flow velocity magnitude.

The overall optimisation procedure, performed using the multi-objective optimisation product modeFRONTIER, can be summarised as follows:

- Generate parametric CAD model in Catia v5
- Create a polyhedral mesh of the flow domain in STAR-CCM+
- Setup and perform CFD analysis in OpenFOAM
- Setup optimisation workflow within modeFRONTIER
- Perform automatic fast robust design optimisation
- Results assessment

3.1: Geometry and Meshing

The first stage in the work was to retrieve all the available CAD data for the full under-hood simulation, which also included the baseline duct geometry.

FAST ROBUST DESIGN OPTIMISATION OF AN UNDER-HOOD COOLING DUCT

Once all of the data had been retrieved, any un-necessarily detailed components such as the engine and transmission were re-built as de-featured or simplified CAD parts in Catia V5. The final CAD geometry is presented in Figure 2.
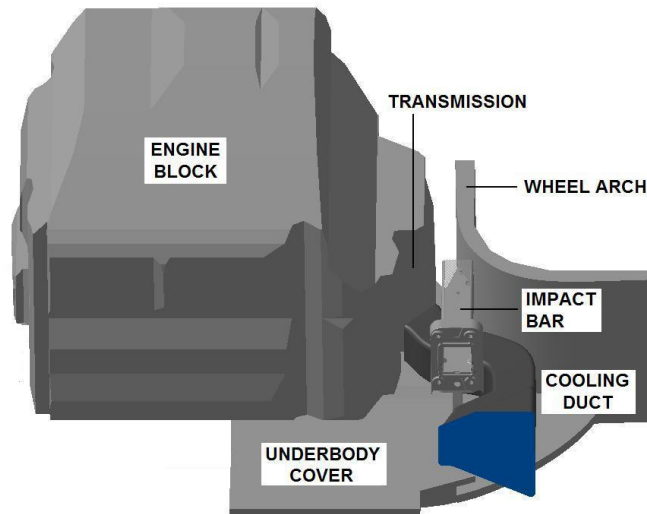

Figure 2: Under-hood components included in the duct optimisation model.

The next step consisted on defining the CAD parameters responsible for modifying the shape of the duct, and also their limits. Figure 4 illustrates two of a total fourteen (14) parameters employed for the case presented in this paper. Two example variations in the duct geometry are also presented for deliberately extreme parameter values.
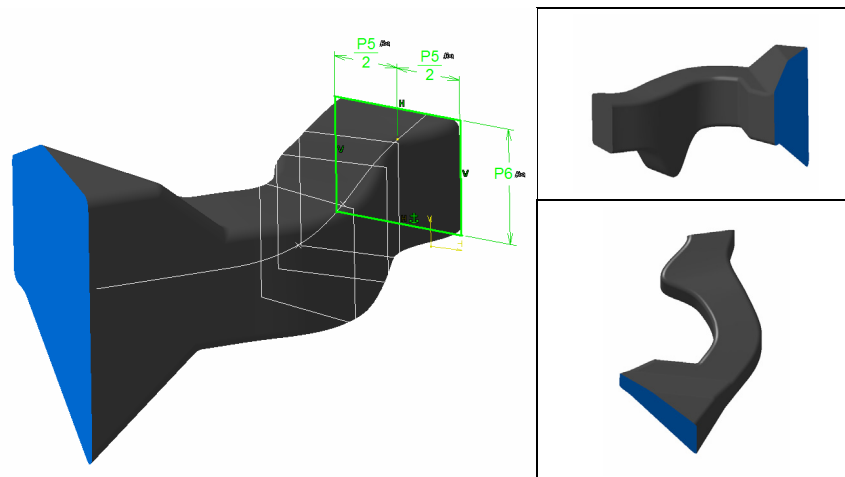

Figure 3: Examples of duct parameters and duct shapes.

The final stage in the development of the parametric CAD model was to setup and perform a component intersection check using an external script for Catia v5.

The cooling duct geometry was then imported using step format into STAR-CCM+ v2 for surface and volume meshing. The computational grids were polyhedral with three wall extrusion layers. The number of cells in the meshes totalled approximately 35,000 and required less than 3 minutes to be generated in a single CPU. The meshing process was recorded in a batch script to be employed in the optimisation procedure. An example of a computational grid is shown in Figure 4.
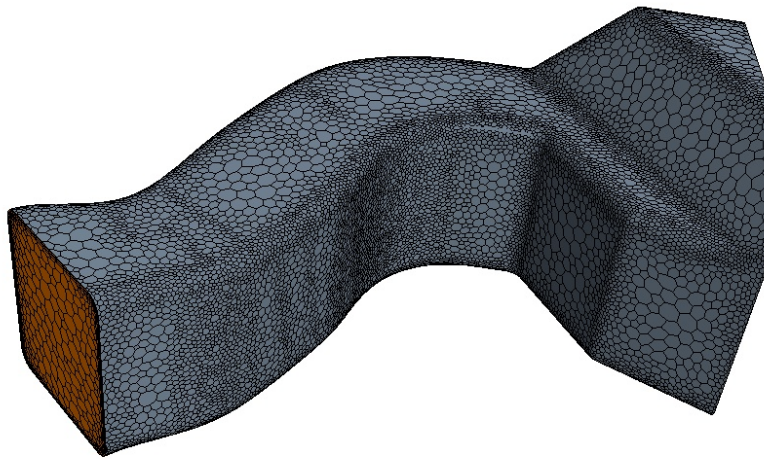
Figure 4: Polyhedral mesh example of the under-hood duct.

3.2: CFD Modelling in OpenFOAM

The CFD solver selected for this example was OpenFOAM, an open source CFD suite. Automatic design optimisation generally requires many CFD computations to converge towards the best design; consequently, an open source solution offers the freedom to exploit a large number of hardware resources without the cost limitations imposed by commercial licensing.

The polyhedral mesh created in the previous stage was first imported into OpenFOAM format using the `ccm26ToFoam` conversion tool. A series of input text files (or scripts), written in OpenFOAM scripting language, were then employed to: (i) set up the CFD problem, i.e.: apply boundary conditions, thermophysical properties, turbulence models, solver controls, etc.; (ii) solve the CFD problem to convergence; and (iii) post-process the CFD solutions to extract the values of the flow field variables corresponding to the objectives

functions of the optimisation problem. Each of these actions is automatically handled by modeFRONTIER.

The air flow was computed as an ideal-compressible-subsonic turbulent gas. The standard high Reynolds k-ε turbulence model with non-equilibrium wall function was applied. The y+ values were verified at the walls on the model for selected geometries. The SIMPLE solution procedure for pressure-velocity coupling was employed in the calculations with the conjugate gradient linear solver. The default differencing schemes were utilised for all the flow variables.

A mass flow boundary condition was applied at the duct inlet with a value of 0.506 kg/s. The prescribed flow direction was normal to the boundary with an incoming air temperature of 300 K. The turbulence intensity was set to 10%, while the length scale was set to 0.025 m. These conditions were approximated from aerodynamic simulations and wind-tunnel data for the full vehicle for a driving speed of 250 kph. At the duct outlet, the static pressure was fixed at 0 Pa (relative to the operating pressure of 101,325 Pa). The surfaces representing walls in the model were all defined to be adiabatic and no-slip stationary boundaries.

An upper limit of 500 iterations was specified in the event that a case would not converge. Post-processing of the simulation results was also automated so that modeFRONTIER could run the model analysis and extract/post-process the necessary information in batch mode. This information included the area average total pressure difference between the inlet and outlet as well as the area weighted average velocity on the outlet boundary.

3.3: modeFRONTIER General Setup

The optimisation software employed in this work was modeFRONTIER v4. This package allowed the process integration of STAR-CCM+ and OpenFOAM with the optimisation strategy employed. This was achieved by reading a specific set of CAD parameters as input variables for Catia v5, and retrieving velocity magnitude and pressure drop values as output variables from OpenFOAM. Both the data and the logic process flow were defined in the Workflow environment, as depicted in Figure 5.

For every step of the optimisation loop, Catia v5, STAR-CCM+ and OpenFOAM were run in batch mode in order to evaluate each specific design configuration. Furthermore, modeFRONTIER parallel optimisation algorithm capabilities were exploited to allow concurrent OpenFOAM analyses to be performed in a multi-processor cluster machine.
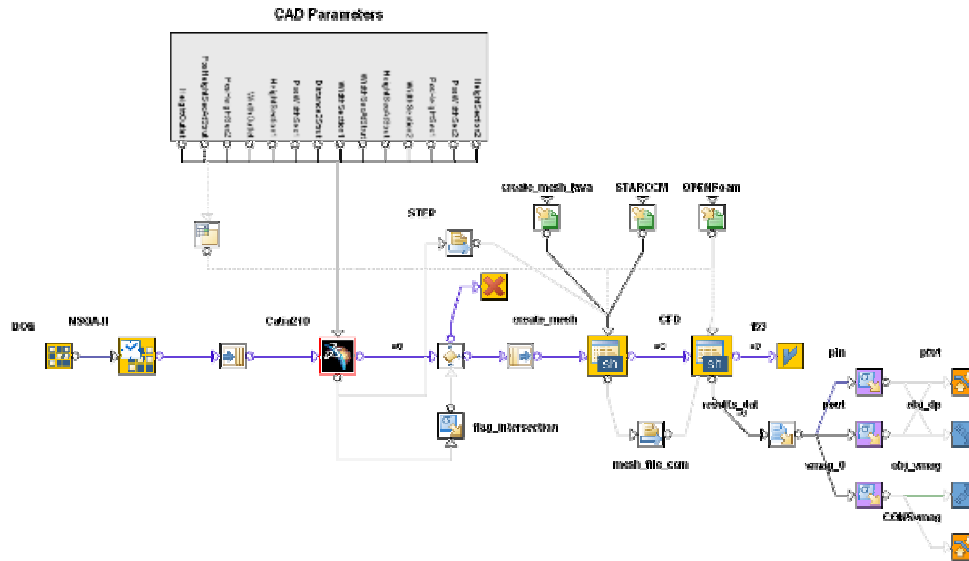
Figure 5: modeFRONTIER Workflow definition.

3.4: Robust Design Analysis

The optimisation goals for this problem were to increase the outlet duct flow velocity by minimising at the same time the pressure drop between inlet and outlet. In order to achieve the optimal solution in terms of operational stability, the robust design criterion introduced in equation (4) was applied.

The use of stochastic variables available in modeFRONTIER allowed the robustness analysis of the cooling duct performance. Accordingly, the design objectives were redefined by taking into account the mean value of the objective functions as well as the corresponding standard deviation. The modified objective for velocity can be written as:

$$\max \ \left(\overline{v}_{mag} - 2\sigma_{Vmag} \right) \qquad (5)$$

whereas the pressure drop is:

$$\min \ [\overline{p}_{in} - \overline{p}_{out} + 2(\sigma_{pin} + \sigma_{put})] \qquad (6)$$

The fast robust design optimisation strategy employed in this paper can be divided into two stages: Latin Square Design of Experiments (DOE) exploration [5] and multi objective robust design optimisation (MORDO) [4].

The Latin Square preliminary exploration sequence was set with 10 levels for each of the input variables, resulting in a total of 100 design experiments

verified via CFD calculations. The distribution of points was such that all the 10 levels available for each design parameter were explored following the orthogonality criterion introduced by the Latin Square method. Consequently, the correlation index between input variables was drastically reduced to almost nil, as shown in green in Figure 6, thus confirming the fact that Latin Square distributes efficiently the points around the design space in this particular case.
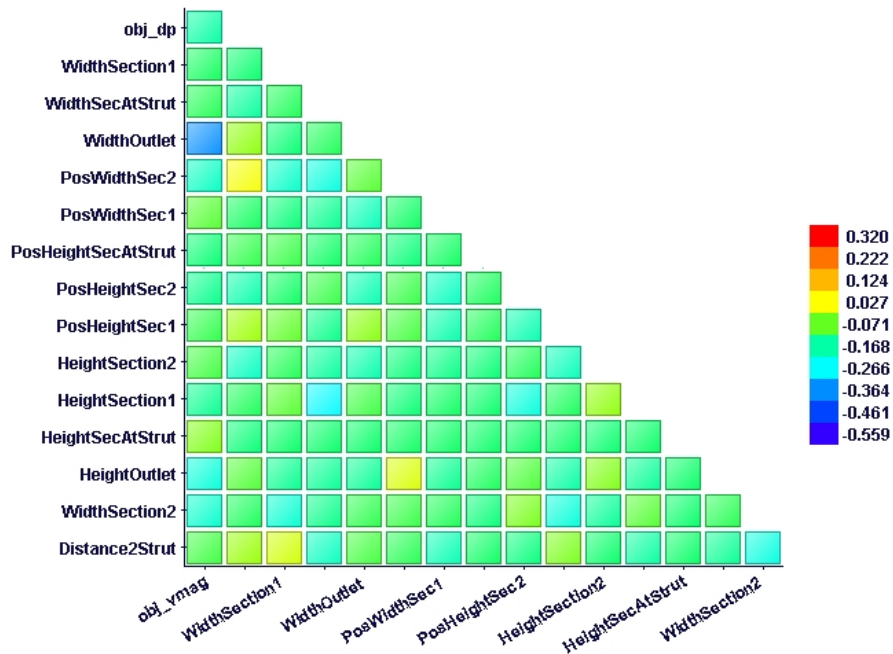

Figure 6: Correlation matrix for Latin Square design variables.

In the next stage, the robustness check was carried out by running the optimisation algorithm NSGA-II [6] using as initial population the Pareto frontier achieved by Latin Square sequence, plus some additional points randomly distributed. The MORDO tool was then employed in combination with the RSM tool to generate 100 samples for each nominal configuration. At the end of the robust design optimisation run, a total of 560 designs were evaluated (1 real, 99 virtual using dynamic adaptive Kriging response surface).

The results obtained using MORDO and RSM were finally validated to check the interpolation error on the evaluation of the mean and standard deviation of the velocity magnitude and pressure losses. For this, the mean values from the robustness check were compared against available solutions from a deterministic NSGA-II optimisation with 1,300 OpenFOAM CFD calculations. Figure 7 shows good agreement along the Pareto frontier between the predominantly virtual calculations (blue) and the high fidelity CFD calculations from the classical optimisation (grey).
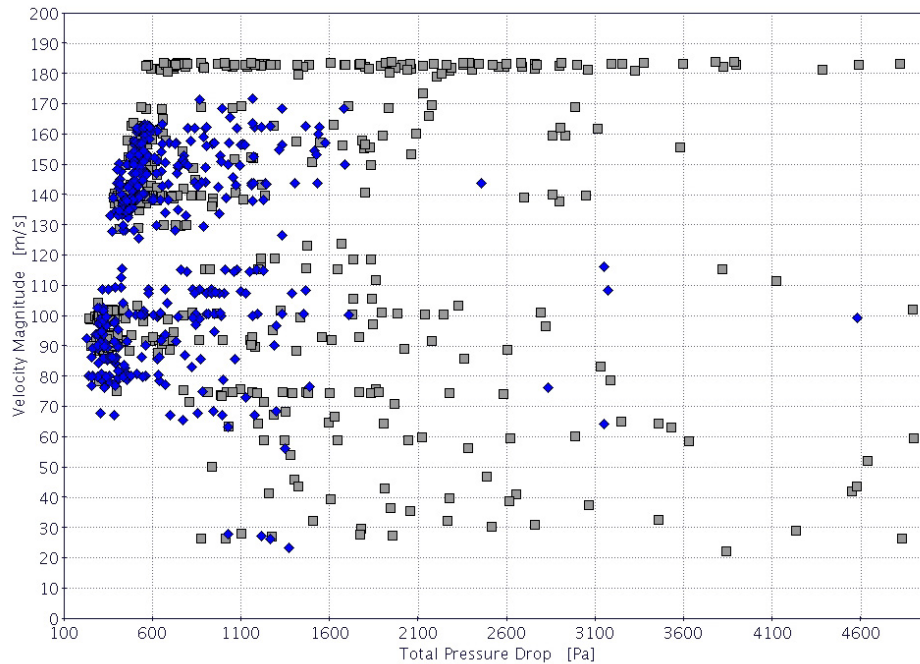
Figure 7: Pareto frontier – classical optimisation (grey) and MORDO (blue).

In terms of velocity magnitude, the mean error from the RSM compared to CFD was -3.07%, while for pressure drop the mean error turned out to be +3.74%. The results also confirmed the support of the Kriging algorithm was very effective, since the grade of resolution of the response surface achieved nearby the high fidelity points evaluated from CFD analysis turned out to be acceptable.

4: CONCLUSIONS

Catia v5, STAR-CCM+ and OpenFOAM were successfully coupled with modeFRONTIER in order to manage fully automatically the whole design process as well as the robust design optimisation strategy applied.

A preliminary sampling using Latin Square sequence, as well as the fast multi-objective optimisation algorithm NSGA-II, combined with an efficient use of RSM tool based on Kriging method and an innovative definition of the objective goals, allowed a fast robust design optimisation of the shape of a cooling duct.

The use of RSM led to a drastic reduction in analysis time. A full robust design analysis that would have required more than 2000 CFD computations (maintaining the same robust design settings) was replaced with little more

than 500 design evaluations using the fast robust strategy described in this work.

Both design objective goals were successfully achieved in addition to good performance in terms of stability. The CFD results for velocity magnitude are compared in Figure 9 for both the baseline under-hood duct and the optimised configuration. Overall, the pressure drop was reduced from 355.6 Pa to 321.5 Pa, while the discharge velocity was increased from 40.7 m/s to 90.3 m/s.
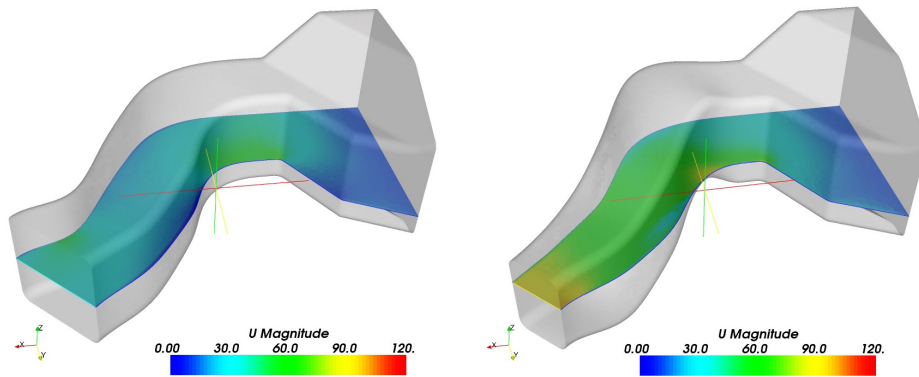

Figure 8: Velocity magnitude at mid-span (left: baseline design; right: optimised design)

REFERENCES

[1] Haslinger, J., and Mäkinen, R. A. E. **Introduction to Shape Optimization, and Computation**. SIAM, ISBN: 0898715369

[2] Campos, F., Geremia P., Weston, S. **Fast Robust Design Optimisation of a Cooling Duct**. Proceedings Fluent Forum 2006. Germany, 13 and 14 November 2006

[3] Wu Li, Padula S. **Performance Trades Study for Robust Airfoil Shape Optimization**. 21th Applied Aerodynamics Conference, AIAA-2003-3790

[4] ESTECO. **modeFRONTIER v3 User manual**. 2003

[5] Jacobson, M. and Matthews, P. **Generating Uniformly Distributed Latin Squares, Journal of Combinatorial Designs**. 1996. 4(6), 405-437

[6] Deb, K., Pratap, A., Agarwal, S., and Meyarivan, T. **A Fast and Elitist Multi-Objective Genetic Algorithm-NSGA-II**. 2000, KanGAL Report Number 2000001

FAST ROBUST DESIGN OPTIMISATION OF AN UNDER-HOOD COOLING DUCT

CONTACT

For information regarding this paper, please contact s.weston@icon-cg.co.uk

www.iconCFD.com

ICON is an independent provider of confidential CAE simulation services and has had a long association with industry by delivering CFD related projects at both OEM and supplier level since 1992. Close links with academic institutions, and continuous involvement with clients in various industries, has allowed ICON to establish a wide base of knowledge and offer technology transfer to its clients.

www.esteco-uk.com

modeFRONTIER is a multi-objective design optimisation software package developed by ESTECO. It is widely used in automotive and aerospace engineering to find, screen and select the optimal design in an organised fashion. modeFRONTIER can drive CAE tools to explore the design space and iterate towards these improved and/or optimal designs.

ACKNOWLEDGMENTS