Adjoint Shape Optimization of U-Bend Duct for Pressure Loss Reduction

G. Alessi[1,*], L. Koloszar[1], T. Verstraete[1,2], B. Blocken[3] and J. van Beeck[1]

[1] *von Karman Institute for Fluid Dynamics, Sint-Genesius-Rode, B-1640, Belgium*

[2] *Queen Mary University of London, London, E1 4NS, United Kingdom*

[3] *KU Leuven, Leuven, 3000, Belgium*

The pressure loss reduction inside a U-Bends is of crucial importance to increase the performance of cooling systems of gas turbines. The optimization technique proposed is a gradient based shape optimization method and is implemented in an open-source framework. The adjoint method has been used to efficiently compute the sensitivity. An optimum design that assure a minimum pressure loss is obtained via a steepest descent method, satisfying geometrical constraints to account for structural limits.

I. Introduction

Cooling systems of gas turbines are composed by internal channel connected by U-bend passages. Referring to these applications, in most cases the coolant is air bled from the high pressure compressor. A proper design should minimize the coolant mass flow rate, i.e. through a reduction of the pressure loss, maintaining the desired thermal exchange. The U-bends that connect consecutive passages are of paramount importance since they represent regions responsible of high pressure loss. As consequence, design improvements can be obtained through the pressure loss minimization of the U-Bends passage.

The adjoint shape optimization has been chosen in the present work to achieve the described goal. Among the different optimization method existing, the adjoint method is particularly interesting, for its computational cost independent from the number of design variables, opening up a vast design freedom.

II. Adjoint shape optimization in an open-source framework

The optimization will follow the schematic loop shown in Figure 1.

In the continuous adjoint formulation[1] two systems of equations have to be solved: the Navier-Stokes equations and the adjoint equations. The first step of the loop is to solve the Navier-Stokes equations (primal system) to obtain convergent flow variables; successively the convergent assessment is required for the adjoint variables. By solving both systems of equations, a surface sensitivity map can be extrapolated. A surface sensitivity map represents for each and every surface node how the objective function changes with respect to an infinitesimally small normal displacement of this surface node. Based on this information, the body contour has to be moved in the direction normal to the surface itself. In the present work, the geometry movement has been coupled with a mesh morphing solver, allowing automatic successive steps of the optimization process. In order to obtain a smooth surface, the sensitivity has been averaged and normalized. Geometrical constraints have been added to account for structural limits.

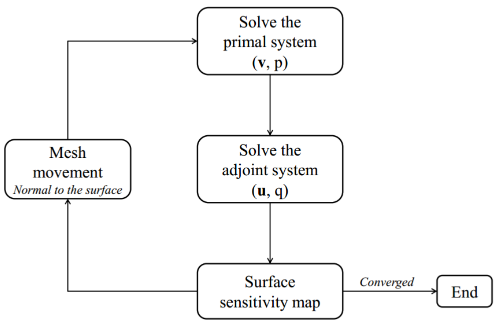

Figure 1. Optimization loop.

The different steps of the optimization loop (solution of the primal and adjoint field, evaluation of the surface sensitivity map, geometry movement and mesh morphing) have been performed by means of the open-source software OpenFOAM.

*PhD student, giacomo.alessi@vki.ac.be, Environmental and Applied Fluid Dynamics Department, Waterloosesteenweg 72 B-1640 Sint-Genesius-Rode Belgium.

III. Results

The first step of the optimization loop regards the evaluation of the flow field inside the U-Bend. The numerical domain is represented by a circular U-Bend of square section (hydraulic diameter D_h=0.075 m), whose geometrical details can be found in Verstraete et al.[2] The experimental tests, Coletti et al.,[3] were performed in atmospheric conditions, considering a bulk velocity of U_0 = 8.4 m/s and a turbulence intensity at the inlet of T.I.=5%.

In order to validate the numerical results, the numerical boundary conditions have to be representative of the experimental campaign. Figure 2 illustrates a comparison of the velocity profile imposed at the inlet of the numerical domain with the experimental one (across the center of the duct). The obtained profile is in good agreement with the experimental one. A zero pressure boundary condition is applied to the outlet.

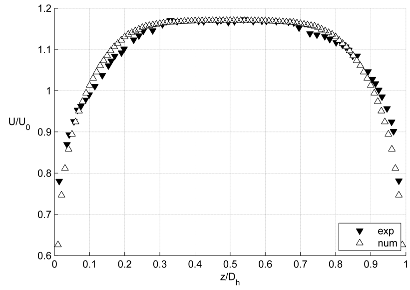

Figure 2. Comparison between the experimental and numerical velocity inlet profiles at the center of the duct.

The numerical simulation has been performed using a structured grid of 342x50x50 elements, assuring a maximum y+ value of 1.63. The Lauder-Sharma low-Reynolds k-ϵ turbulence model has been used. Figure 3 shows the velocity field at the middle plane of the U-Bend. The flow accelerates approaching the bend, it reaches the maximum velocity around the inner wall while it decelerates along the outer wall. The flow starts to separate before the end of the bend and form a separation bubble 1.7D_h long. The magnitude of the reverse flow reaches a maximum value of around 0.45U_0.
The main parameter to compare is represented by a normalized static pressure drop:

$$\Delta P = \frac{P_{s2} - P_{s1}}{\frac{1}{2}\rho U_0^2} \tag{1}$$

The obtained pressure drop is of ΔP=1.04, in good agreement with the experimental value of 1.03$\pm$0.03.

The successive step of the optimization loop regards the evaluation of the adjoint field and the attainment of the surface sensitivity map. The considered cost function is the minimization of the total pressure loss between the outlet and the inlet. Figure 4 illustrates the obtained surface sensitivity map at the middle plane. It indicates the modifications to perform for an improvement of the cost function, in particular: a movement away from the fluid of a positive area and a surface modification towards the fluid of a negative region. Modifications of a zero gradient region have little effect on the cost function.

Finally, the original geometry can be modified accordingly to the surface sensitivity map and the loop is restarted. The optimized shape is attained at the convergence of the loop.

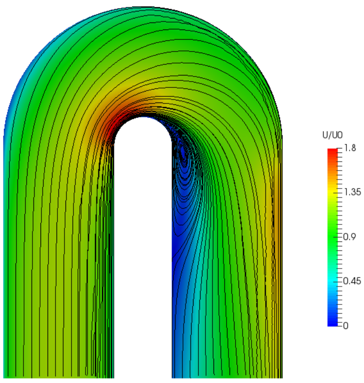

Figure 3. Velocity field at the middle plane.

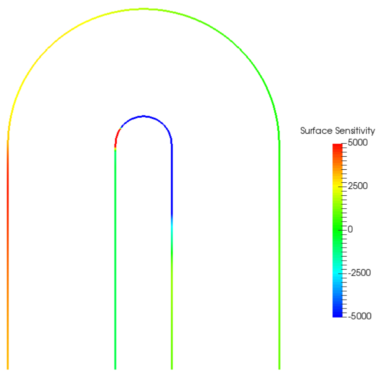

Figure 4. Surface sensitivity at the middle plane.

References

[1]C. Hinterberger and M. Olesen. Industrial Application of Continuous Adjoint Flow Solvers for the Optimization of Automotive Exhaust Systems. *ECCOMAS Thematic Conference - CFD & Optimization*, Antalya, 2011.

[2]T. Verstraete, F. Coletti, J. Bulle, T.V. der Wielen, N. van der Berge and T. Arts. Optimization of a U-Bend for Minimal Pressure Loss in Internal Cooling Channels - Part I: Numerical Method. *ASME Journal of engineering for Gas Turbines and Power*, 135:pp 051015, 2013.

[3]F. Coletti, T. Verstraete, J. Bulle, T.V. der Wielen, N. van der Berge and T. Arts. Optimization of a U-Bend for Minimal Pressure Loss in Internal Cooling Channels - Part II: Experimental validation. *ASME Journal of engineering for Gas Turbines and Power*, 135:pp 051016, 2013.