TOPOLOGY OPTIMIZATION IN CFD: USE OF INNOVATIVE TECHNIQUES FOR DESIGN SUPPORT IN THE INDUSTRY

BRUGIERE Olivier, CFD Numerics, olivier.brugiere@cfd-numerics.com
CORTEY Edgar, CFD Numerics
FRANCOIS Nicolas-Yoan, VALEO Thermal Systems BG

SUMMARY

For a few years, it is been possible to design a product using only computer aided design (CAD) tools and computer simulations. The reliability of numerical simulations and the increase in computing power allow users to model configurations increasingly complex. They are used for designing new products or modifying existing ones. Using digital technologies allows designers to avoid building prototypes and then decreasing development costs. With the ability to change the configuration faster and faster, engineers and designers can now perform better analysis of the configuration and make the process more robust.

To change the configuration rationally, it is possible to use optimization methods. The first optimization method used in industry is the shape optimization. It can be implemented through the method of Design Of Experiments (DOE) or based on the adjoint method. The shape optimization will modify the surface geometrically but not consider all possible designs because it’s based on the original duct. So to overcome this original duct, it's necessary to use methods which can change the topology of the flow, it's the topology optimization. It is possible to consider a large number of degrees of freedom, becoming one of the most interesting approaches for design assistance. The topology optimization can draw a piece from a working volume subject to targets. CFD-Numerics based its methodology on academic research and internal developments to implement topology optimization based on adjoint method applied to industrial configurations.

With the methodology implemented, it is possible to create an optimized geometry from the white sheet on the basis of the available volume and given objectives (reduction of pressure loss, a uniform supply through the heat exchanger, a better flow distribution ...). The proposed solution is based on the topological optimization that will allow digging in the useful volume and obtain an initial shape for design assistance. Then it will be possible to refine the performance of the solution using shape optimization after integrating design constraints. The surface will be locally modified to optimize performances of the configuration.

TOPOLOGY OPTIMIZATION IN CFD: USE OF INNOVATIVE TECHNIQUES FOR DESIGN SUPPORT IN THE INDUSTRY

1: Adjoint equations

An optimization problem is defined by objective functions which need to be minimized or maximized. The adjoint equations are presented in this section on general cost function which depend on flow variables $\mathbf{v}$ and p and the design variable α (porosity distribution). Equations that apply to this optimization problem are the incompressible Navier-Stokes equations.

$$\text{minimize } \mathrm{J} = \mathrm{J}\,(\mathbf{v}, \mathrm{p}, \alpha)$$

$$\text{such that } (\mathbf{v}.\nabla)\mathbf{v} + \nabla \mathrm{p} - \nabla.\left(2\nu \mathrm{D}(\mathbf{v})\right) + \alpha\mathbf{v} = 0;\ \nabla.\mathbf{v} = 0$$

with kinematic viscosity ν and the rate of strain tensor denoted $D(\boldsymbol{v}) = \frac{1}{2}(\nabla\boldsymbol{v} + (\nabla\boldsymbol{v})^T)$. We suppose that ν is defined as the sum of molecular and turbulent viscosities. The above equation introduces the design variable using Darcy's law. It allows for punishing counterproductive cells by increasing the porosity. This is the central component of topology optimization.

The optimization problem is constrained by the Navier-Stokes equations. This kind of problems has been addressed through introducing Lagrange multipliers reformulating the cost function as

$$\text{minimize } \mathrm{L} := \mathrm{J} + \int_{\Omega} (\mathbf{u}, \mathrm{q})\mathcal{R}\mathrm{d}\Omega$$

with Ω the flow domain, $\mathcal{R}$ the incompressible RANS and $(\mathbf{u}, \mathrm{q})$ are respectively the adjoint velocity and pressure introduced as Lagrange multipliers. Those velocity and pressure have not a physical meaning like primal variables. So they may not be interpreted as a velocity or a pressure in the physical sense. The names suggest that a similar solution procedure can be applied to resolve them. For example, the physical meaning of the adjoint velocity is a non-dimensional transfer function from forces applied in the fluid to the cost function. If in a cell, primal and adjoint velocities have not the same direction, this cell must be penalized to improve the solution of optimization.

The notation $(\mathbf{u}, \mathrm{q})\mathcal{R}$ shows that every state equations are multiplied with a Lagrange multiplier and their contribution are added and summed over all the domain. The total variation of L, $\delta \mathrm{L} = \delta_\alpha \mathrm{L} + \delta_{\mathbf{v}} \mathrm{L} + \delta_{\mathrm{p}} \mathrm{L}$, is studied to calculate the sensitivity according to the design variable. The adjoint velocity and pressure are chosen with respect to variations of the flow variables according to $\delta_{\mathbf{v}} \mathrm{L} + \delta_{\mathrm{p}} \mathrm{L} = 0$. From those equations, it’s possible to evaluate the sensitivity of the cost function with respect to the porosity in cell i as:

$$\frac{\partial \mathrm{L}}{\partial \alpha_{\mathrm{i}}} = \mathbf{u}_{\mathrm{i}}.\,\mathbf{v}_{\mathrm{i}} \mathrm{V}_{\mathrm{i}}$$

where V_i is the volume of cell i. From the condition $\delta_{\mathbf{v}}L + \delta_p L = 0$ and after derivation, it's possible to give the adjoint Navier-Stokes [1]:

$$-2D(\mathbf{u})\mathbf{v} = -\nabla q + \nabla.\left(2\nu D(\mathbf{u})\right) - \alpha\mathbf{u} + \frac{\partial J_\Omega}{\partial \mathbf{v}} \; ; \; \nabla.\mathbf{u} = \frac{\partial J_\Omega}{\partial p}$$

with J_Ω cost functions which contain contributions from the flow domain and $-2D(\mathbf{u})\mathbf{v} = -\nabla\mathbf{u}\cdot\mathbf{v} - (\mathbf{v}\cdot\nabla)\mathbf{u}$. The structures of the adjoint Navier–Stokes equations and the primal equations are similar. But the adjoint flow field has difficulty to converge. The main difference between those equations is the minus sign in front of the convective term of adjoint momentum equation. In fact, informations are convected upstream for adjoint solution rather than downstream for the primal flow.

As boundary conditions for adjoint velocity and pressure, we have :

- Adjoint boundary conditions for the wall and inlet:

$$\mathbf{u}_t = 0 \quad ; \quad u_n = -\frac{\partial J_\Gamma}{\partial p} \quad ; \quad \mathbf{n}.\nabla q = 0$$

- Adjoint boundary conditions for the outlet:

$$q = \mathbf{u}.\mathbf{v} + u_n v_n + \nu(\mathbf{n}.\nabla)u_n + \frac{\partial J_\Gamma}{\partial v_n} \quad ; \quad 0 = v_n u_t + \nu(\mathbf{n}.\nabla)u_t + \frac{\partial J_\Gamma}{\partial v_t}$$

The optimisation process is decomposed in three steps. In a first time, the physical field is solved with taking into account the turbulence equations. Then, the physical velocity and turbulent viscosity are used for the solution of the adjoint equations. Else, the calculation of the topological sensitivities with respect to the design variable is realised and the update of porosity field is performed using the conjugate gradients method.

2: Results

This method has been applied to improve and optimize the performance of a charge air cooler, in figure 1.

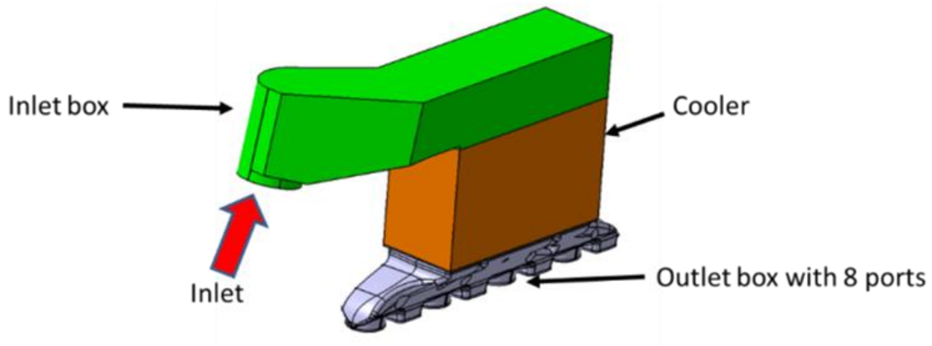

Figure 1: Charge Air cooler description

The inlet box (green part) represents the maximum volume (constraint due to the packaging of engine environment) to be optimized. The cooler (brown part) and outlet box (grey part) are frozen.

Figure 2: Final design of the geometry

The pressure drop of inlet box obtained by the topology optimization is around 10 mbar. This final design (Figure 2) has been compared to the original design computed by an iterative and classic simulation process. In this previous iterative study, the best value of pressure drop of inlet box performed was around 15 mbar. Thanks to the topology optimization, the pressure drop of inlet tank has been reduced by 32% and the global pressure drop of charge air cooler has been decreased by 12% compared to classic and iterative simulation process. In the meantime uniformity supply through the heat exchanger to maximize its thermal efficiency has been slightly improved with a reduction of velocity gradient by 2%.

The complete simulation process is performed in a few days including optimisation shape definition, validation and CAD format geometry reconstruction.

3: Conclusion

A fast and robust topology optimization has been implemented into a VALEO CFD simulation process from OpenFoam [3] based on continuous adjoint solver and a porosity field as term source. The result is very useful and relevant. This method offers a great advantage for improving and optimizing quickly a design in industrial context. It proposes significant improvement way of design in order to reduce the pressure drop and homogenize flow distribution through the cooler in a controlled development time.

REFERENCES

[1] C. Othmer, "*A continuous adjoint formulation for the computation of topological and surface sensitivities of ducted flows*", Int. J. Numer. Meth. Fluids (2008)

[2] R.P. Dwight and J. Brezillon, "*Effect of various approximations of the discrete adjoint on gradient-based optimization*", AIAA-2006-0690, 2006

[3] www.openfoam.org