Adjoint shape optimization of a duct for a wall jet film cooling setup

N N Kozyulin¹*, M S Bobrov¹, and M Y Hrebtov¹,
¹Institute of Thermophysics of SB RAS, 630090, Ac. Lavrentiev ave. 1, Novosibirsk, Russia
E-mail: nikozyulin@gmail.com

Abstract. The paper presents the results of optimization of the geometric parameters of the simplified wall jet cooling system using a modified Adjoint Shape optimization method for algebraic systems of equations (Discrete Adjoint Optimization). The modification consists in using a linearized discrete system of equations with the replacement of derivatives by their finite-volume approximations. The jet flowed through a duct and out from a nozzle. The duct was inclined at an angle of 35 degrees to the cooled wall. The mean velocity ratio between the jet and the main flow was set to 2. The total heat flux on the cooled wall was taken as a cost function. The problem was considered in a two-dimensional stationary turbulent formulation (RANS). As a result of optimization, the shape of the duct changed significantly, affecting the flow inside it. The optimization led to the disappearance of the recirculation zone and reattaching of the jet to the cooled wall. As a result of the optimization performed, the heat flux at the wall increased by 20%.

1. Introduction

At present, developing an effective algorithm for the optimization of geometry and other parameters for the turbulent flow of fluid is essential in many areas of engineering. One of the promising automatic optimization methods is the adjoint gradient method [1]. This method can be used for a large number of control parameters at the same time (complex geometry shapes, heat or mass sources distribution, etc.). In addition, this method allows choosing a cost function almost arbitrarily. Various kinds of integral characteristics of the optimized objects and their combinations can act as a cost function, such as, for example, the projection of the resulting force vector onto any direction, angular momentum, pressure drop, deviation from the reference data, etc. Another advantage of this method is relatively low computational costs. According to the adjoint gradient method, the computing time is practically independent of the number of parameters for each of which the gradient of the cost function is evaluated. The gradient computation time is approximately equal to the time for solving the main problem equations. The adjoint method can be applied to the third-party solver suitable for simulation of the problem of interest. The adjoint optimization method was successfully applied in the problems of optimizing the aerodynamic shape of an airfoil [1], multiphase flow through porous media [2], the shape applied to electromagnetic design [3], in the velocity reconstruction method from the experimental dataset, as well as optimizing interpolation weights and boundary conditions [4]. Also, this algorithm was used for optimizing the turbulent heat transfer problem in wall-bounded flows [5] and has proven itself. Thus, it is widely used in commercial CFD packages.

The film cooling devices design optimization has a long and rich history of studying [6, 7]. Many competing designs were introduced which helped a lot in optimizing certain parameters of the cooling
process (adiabatic cooling efficiency is the main one). However, it is always interesting to find the automatic way to optimize certain configurations. In the current study, we tested the adjoint shape optimization method with the simplified 2D film cooling device to see how it can perform in optimizing the shape of the internal surface of the jet nozzle (duct).

The current paper presents the results of optimization by using the adjoint gradient method of the geometric shape of the rectangular (2D) duct that directs the jet into a cross-flow for the simplified film cooling setup with a flat wall. The mean angle of the duct relative to the cooled surface was fixed, as well as the shape of the inlet and outlet sections of the duct. The adjoint gradient method was used to optimize the coordinates of points on the inclined part of the duct surface. The negative value of the total heat flux magnitude on the cooled wall was taken as a cost function.

2. Computational methodology

The simplified jet in a cross-flow setup was considered for simulation. The problem was considered in a two-dimensional stationary setting. The stationary RANS equations with a heat transfer (1-3) were solved:

\[
\frac{\partial \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \nu + \nu_t \right) \left( \frac{\partial \bar{u}_j}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_j} \right) - \frac{2}{3} k \delta_j,
\]

(1)

\[
\frac{\partial \bar{u}_j}{\partial x_j} = 0,
\]

(2)

\[
\frac{\partial \bar{T}}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\nu}{Pr} + \frac{\nu_t}{Pr_t} \right) \left( \frac{\partial \bar{T}}{\partial x_j} \right),
\]

(3)

where \( \bar{u} \) is the time-averaged velocity vector, \( p \) is the pressure, \( \nu \) and \( \nu_t \) are kinematic and turbulent viscosities, respectively, \( k \) is the turbulence kinetic energy, \( T \) is the temperature. The standard \( k-\varepsilon \) turbulence model was used for closing the RANS equations.

The jet flowed out from a nozzle located at an angle of 35 degrees to the cooled wall (Fig. 1, left). The blowing ratio was set equal to 2. The temperature of the coolant in the jet was set 10 degrees below the temperature of the main flow (280 K and 290 K, respectively). The isothermal condition was set on the cooled wall (with a temperature of 290 K). Reynolds number based on for the jet width was set equal to 10,000, and the molecular Prandtl number was equal to 0.71 (Pr = 0.9). The computational grid consisted of \( 10^7 \) nodes in a two-dimensional setting (Fig 1, right). This resolution was chosen after a grid convergence study.

The optimization algorithm by the adjoint gradient method [8] consists in determining the gradient of the cost function at the control points and then changing the parameters at the control points along these gradient vectors. After several iterations, the cost function reaches its local minimum. This method allows for one calculation to obtain the gradients of the cost function at all points of interest, which is much less computationally expensive compared to approaches without taking into account the form of equations of a continuous medium.

Let \( R(w, s) \) denotes a constraint function (for which at any point \( R(w, s) = 0 \) should hold) and \( J(w, s) \) denotes a cost function. The expression for the variation of the cost function \( \delta J \), expanded in a Taylor series with discarding small second-order quantities, is written as follows:

\[
\delta J \approx \left[ \frac{\partial J}{\partial w} \right]^T \delta w + \left[ \frac{\partial J}{\partial s} \right]^T \delta s,
\]

(4)

where \( w(s) \) are flow field variables, \( s \) are optimized variables. Analogically, an expression for the variations of the functions of restrictions \( \delta R \) is written as follows:
\[
\delta R \approx \left[ \frac{\partial R}{\partial w} \right] \delta w + \left[ \frac{\partial R}{\partial s} \right] \delta s = 0
\]  

(5)

Subtracting from expression (4) the expression (5) multiplied by the vector of Lagrange multipliers \( \lambda \), the variation of the cost function can be written as follows:

\[
\delta J \approx \left[ \frac{\partial J}{\partial w} \right] - \lambda^T \left[ \frac{\partial R}{\partial w} \right] \delta w + \left[ \frac{\partial J}{\partial s} \right] - \lambda^T \left[ \frac{\partial R}{\partial s} \right] \delta s
\]

(6)

To avoid calculating the variation \( \delta w \), we select the vector of Lagrange multipliers \( \lambda \) in such a way that the following equation holds:

\[
\left[ \frac{\partial R}{\partial w} \right] \lambda = \left[ \frac{\partial J}{\partial w} \right].
\]

(7)

After the vector of Lagrange multipliers is found from the adjoint equations (7), the variation of the cost function can be written as follows:

\[
\delta J = G^T \delta s \quad G^T = \left[ \frac{\partial J}{\partial s} \right] - \lambda^T \left[ \frac{\partial R}{\partial s} \right],
\]

(8)

where \( G \) is the gradient vector by which the direction of change of the variables can be determined to minimize the cost function.

**Fig. 1.** Computation domain and flow schematics (left), computational grid fragment (right).

After finding the gradients at the control points located on the modified area of the surface, the surface is displaced towards the gradients by a small amount, after which the calculation process is repeated. Such an iterative process leads to the convergence of the configuration to a local minimum of the objective function.

For the calculations, we used the open package daFoam [9], based on the finite-volume calculation code OpenFoam (www.openfoam.com). In this work, we used a modification of this method for algebraic systems of equations (Discrete Adjoint Optimization) [10]. The modification consists in using a linearized discrete system of equations with the replacement of derivatives by their finite difference approximations. Usually, this method is used to optimize the hydrodynamic resistance of airfoils flown around [1]. In this work, we have applied this method to optimize the efficiency of heat transfer in the problem of a coolant jet in a cross-flow.

The negative value of the total heat flux on the cooled wall was chosen as a cost function. The heat flux was calculated from the wall’s normal temperature gradient at the horizontal wall in front of the nozzle. The functions of the constraints are the RANS equations for a two-dimensional stationary incompressible flow. The coordinates of points on the inner walls of the channel through which the cooler is supplied are used as optimization variables.
3. Results and discussion
According to the calculated results, it was found that before optimization, the jet was detached from the wall at the nozzle outlet point, forming a large recirculation zone (Fig. 2 a). This effect is often observed in film cooling [5,6,7] and it is the main negative factor that reduces the cooling efficiency for operational regimes with high coolant flow rates. It should also be noted that in this configuration, the temperature on the wall is slightly higher than in the coolant jet (Fig. 2 c).

As described above, the optimization points were located at the surface of the coolant jet duct, excluding the inlet and outlet sections (shown in Fig. 1, left). In the course of optimization (Fig. 2 c-f), the shape of the channel with the cooler changed significantly, with the formation of separated flows inside the channel. It should also be noted that this configuration led to a narrowing of the jet at the nozzle outlet from the channel. This restructuring led to a complete disappearance of the recirculation zone in the main flow, and the attachment of the jet to the cooled wall. It is also noticeable that the effective cross-section of the jet near the wall has significantly decreased (Fig. 1 e). As a result of the optimization, the heat flux on the wall increased by 20%. Figure 3 shows how the total wall heat flux changed during the optimization process. It can be seen that the increase in the heat flux is monotonical with decreasing rate. Only 7 iterations of the optimization algorithm were needed to get to the local minimum of the cost function.

![Image](image_url)

**Fig. 2.** Distributions of the velocity (a, c, e) and temperature (b, d, f) in the jet at 0-th (i.e. without optimization a,b), 3-D (c, d), and 7-th (e, f) optimization iterations.
Fig. 3. The heat flux evolution of cooled surface during the optimization process

The mentioned narrowing of the jet at the nozzle outlet that appeared during the optimization process led to an increase of the drag in the duct which was reflected in the pressure distribution (Fig. 4). The most prominent drag increase was observed for the later stage of optimization. At the beginning of the optimization, the change of the channel shape did not lead to a significant narrowing of the jet and hence the increase of the drag (Fig 4b) but still significantly reduced the size of the recirculation zone (Fig 2 cd) which is plausible. The additional constraints on the total drag in the channel might lead to a more balanced result.

Fig. 4. Pressure distribution in the duct at 0-th (a), 3-rd (b) and 7-th (c) optimization iterations.

4. Conclusion
As a result of the present work, the geometric shape of the duct for the coolant jet in a co-flow was optimized using the adjoint field method. The automatic adjoint optimization method has given a significant improvement in the total heat flux at the cooled wall. As a result of optimization, the recirculation zone in the main flow zone was eliminated, and the jet was attached to the cooled surface. Thus, this optimization made it possible to increase the heat flux on the cooled wall by 20%. This optimization is more of a research nature rather than a practical one since the initial setup is overly simplified. However, the method has shown its potential toward the heat transfer problems in a fluid flow and can be further applied to a more complex realistic configuration. The considered optimization method seems to be promising for improving the characteristics of devices with film cooling. The development of the method is necessary to take into account non-stationary effects in the flow. Some global search methods should be combined with the presented method to build an effective tool capable to provide complex optimizations.
Acknowledgments
This work has been supported by RSF (project No. 19-79-30075), numerical simulations were conducted under state contract with IT SB RAS.

References
[1] Nadarajah S, Jameson A 2000, 38th Aerospace Sciences Meeting and Exhibit, 667.
[2] Jansen J 2011, Computers & Fluids 46(1), 40-51.
[3] Lalau-Keraly C M 2013 Optics express 21(18), 21693-21701.
[4] Schneiders J and Scarano F 2016, Experiments in fluids 57(9), 1-22.
[5] Madrane, A., An, H., Leng, J., Schaanzer, M., Pham, M. Q., Bourgeois, G. International Journal of Thermal Sciences, 158, 106542.
[6] Bunker R S 2005. J. Heat Transfer, 127(4), 441-453.
[7] Zhang J, Zhang S, Chunhua W A N G, & Xiaoming T A N 2020 Chinese J of Aeronautics, 33(4), 1119-1136.
[8] Reuther J. et al. 34th Aerospace Sciences Meeting and Exhibit. 1996, 94.
[9] He P, Mader C A, Martins J R and Maki K J 2020, AIAA Journal, 58(3), 1304-1319.
[10] He P, Mader C A, Martins J R and Maki K J 2018, Computers & Fluids, 168, 285-303.