Aerospace Science and Technology 58 (2016) 18–25

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Optimization of a fan-shaped hole to improve film cooling performance by RBF neural network and genetic algorithm

Chunhua Wang [a,*], Jingzhou Zhang [a,b], Junhui Zhou [a]

[a] Nanjing University of Aeronautics and Astronautics, Jiangsu Province Key Laboratory of Aerospace Power System, Nanjing, 210016, China
[b] Collaborative Innovation Center of Advanced Aero-Engine, Beijing, 100191, China

ARTICLE INFO

Article history:
Received 31 May 2016
Received in revised form 1 August 2016
Accepted 1 August 2016
Available online 5 August 2016

Keywords:
Gas turbine
Fan-shaped film cooling hole
Optimization
Radial basis function neural network
Genetic algorithm

ABSTRACT

To improve the cooling performance, shape optimization of a fan-shaped film cooling hole was carried out. Three geometric parameters, including incline angle, lateral expansion angle and hole length, were selected as the design parameters. Numerical model of the film cooling system was established, validated, and used to generate training samples. Radial basis function neural network (RBF-NN) was applied for surrogate model, and the optimal design parameters were determined by a kind of genetic algorithms. At low blowing ratio ($M = 0.5$), the area-averaged film cooling effectiveness can reach its maximum value in the design space as incline angle, lateral expansion angle and hole length-to-diameter ratio are 40.5°, 23.97° and 7.43. At $M = 1.5$, the optimal values of length-to-diameter ratio, lateral expansion angle and incline angle are 20.1°, 23.92° and 7.65. RBF-NN coupled with genetic algorithm is an effective scheme for the optimization of fan-shaped film cooling holes.

1. Introduction

Demands for increased power output from gas turbine engines and decreased fuel consumption have caused engine designers to increase combustion exit gas temperature. One of the consequences of this is the potential failure of components in the turbine section due to large thermal load [1]. Film cooling scheme is one of the external cooling technologies being used to reduce this problem. One approach to improve film cooling performance is to alter the geometry of a film-cooling hole from a standard cylindrical shape to one with a diffused exit shape. The flow deceleration in the diffuser section of the hole allows the coolant jet attach more closely to the wall surface. Moreover, the diffuser promotes lateral spreading of the coolant, and thus provides a more uniform film on the surface [2–4].

Many experimental and numerical investigations of shaped film cooling holes have been carried out to analyze cooling efficiency and aerodynamic performance. In the experiments by Kohli and Bogard [5], the hole with the larger forward diffusion angle was expected to keep the coolant jet closer to the surface, however thermal performance of this hole in terms of film cooling effectiveness was lower than that of the hole with the smaller forward diffusion angle. Colban et al. [6] developed a semi-empirical correlation to predict film cooling effectiveness of laidback fan-shaped holes. They indicated that the film cooling effectiveness is strongly affected by pitch-to diameter ratio, blowing ratio, hole size, area ratio and width of hole at trailing edge of hole breakout. Saumweber et al. [7] reported that the effects of large free-stream turbulence intensity and periodic unsteady wakes on the film cooling performance of fan-shaped holes are always detrimental. Lutum et al. [8] reported that the hole with an additional forward diffusion performed slightly worse and attribute this behavior to local flow separations within or just downstream of the hole. Taking into account the known problems from the literature, it can be found that film cooling of shaped holes is a complex system affected by a large number of geometric parameters, and designers have to deal with a high-dimensional design space, where a global optimum solution needs to be found for a given set of requirements.

Developing an efficient design and optimization method for film cooling systems is attracting more and more attention from researchers. Lee and Kim [9] developed a method by coupling Kriging model and sequential quadratic programming for optimizing a shaped film cooling hole. Lee et al. [10] applied response surface approximation method and a hybrid evolutionary algorithm for multi-objective optimization of a row of film cooling holes. Ayoubi et al. [11] developed a non-dominated sorting genetic algorithm for optimizing round film cooling hole in the leading edge of blade. Nowak and Wróblewski [12] used conjugate heat transfer analysis and evolutionary optimization algorithm for optimization of blade cooling system. Evolutionary algorithm has also been ap-

* Corresponding author. Fax: +86 025 84892200.
E-mail address: chunhuawang@nuaa.edu.cn (C. Wang).

http://dx.doi.org/10.1016/j.ast.2016.08.004
1270-9638/

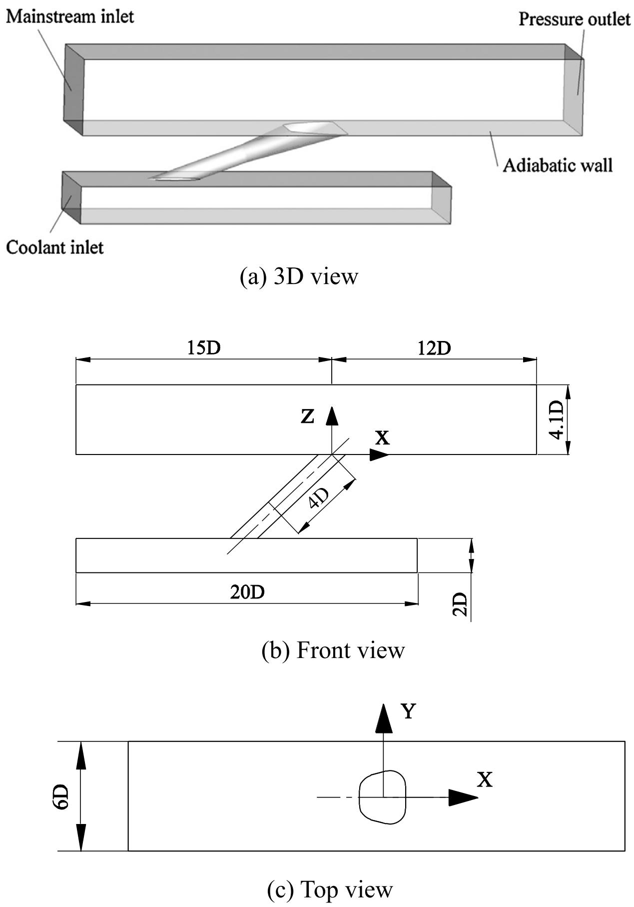

Fig. 1. Geometry of computational domain.

plied for optimization of a high-pressure turbine vane pressure side film cooling array by Johnson et al. [13]. Based on these previous studies, authors propose a data mining method for shape optimization of a fan-shaped hole. In the present optimization method, radial basis function neural network (RBF-NN) is used for surrogate model. RBF-NN, which is originated from the well-studied subject of multivariate regularization theory, provides powerful method for hyper-surface reconstruction coupled with efficient noise elimination [14,15]. Furthermore, a kind of genetic algorithms was introduced to determine the optimal design parameters. Genetic algorithm belongs to the larger class of evolutionary algorithms (EA), which generates solutions to optimization problems using techniques inspired by natural evolution [16,17].

In this paper, the optimization problem of a fan-shaped hole is introduced firstly; and then the CFD model is established; finally, the optimization by RBF-NN and genetic algorithm is performed, and the optimization results are analyzed in detail.

2. Design variables and objective function

The geometry of the computation domain in current research is shown in Fig. 1. The computation domain consists of mainstream duct, coolant plenum, and a fan-shaped hole. The hole diameter (D) is 10 mm. The mainstream duct length, width and height are $27D$, $6D$ and $4.1D$ respectively. The coolant plenum length, width and height are $10D$, $6D$, and $5D$ respectively. To facilitate validating the mathematical model, the boundary condition is the same to that in the experiment by Saumweber and Schulz [18]. The mainstream velocity is 139 m/s, and the mainstream total temperature is 540 K. The coolant temperature is 290 K. The turbulence intensity of mainstream is 5.2%, and integral length scale is $1.1D$.

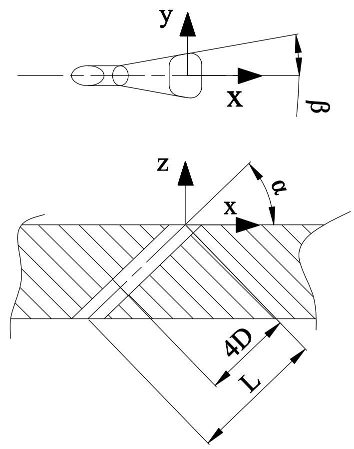

Fig. 2. Definition of design variables.

Table 1
Lower and upper limits of the design parameters.

Design variable	Symbol	Unit	Lower bound	Upper bound
Hole incline angle	α	°	20	50
Lateral expansion angle	β	°	12	24
Hole length	L	mm	50	80

Area-averaged adiabatic film cooling effectiveness ($2D < x < 12D$, $-3D < y < 3D$) is used to evaluate the film cooling performance quantitatively. Area-averaged adiabatic film cooling effectiveness, η_{avg}, is defined as:

$$\eta_{avg} = \int_{2}^{12} \bar{\eta}(x/D)d(x/D) \tag{1}$$

$$\bar{\eta}(x/D) = \frac{1}{\Delta y}\int_{-3}^{3} \eta(x/D, y/D)d(y/D) \tag{2}$$

$$\eta(x/D, y/D) = \frac{T_{ad}(x/D, y/D) - T_{\infty}}{T_c - T_{\infty}} \tag{3}$$

where T_{ad} is the adiabatic wall temperature, T_{∞} is the mainstream temperature, T_c is the coolant temperature, and $\triangle y$ is equal to $6D$ in the present model.

Three geometrical parameters, including hole incline angle (α), lateral expansion angle (β) and hole length (L) are selected as design variables. The definitions of these three parameters are shown in Fig. 2. The lower and upper limits of these three parameters are listed in Table 1. During the optimization, the length of the diffuser section of hole is fixed.

3. Methods for solving optimization problem

3.1. Numerical analysis

To investigate the influences of the design variables on the film cooling effectiveness, numerical experiments by Ansys-Fluent 14.0 are performed. The working fluid is ideal gas. At the gas inlet, constant mass flow rate and temperature is specified. At the mainstream outlet, the gauge pressure is set to 0 Pa. At the wall, adiabatic and non-slip condition is specified. Because of high prediction performance for overall level of film cooling effectiveness, renormalized group (RNG) k–ε model is applied for solving turbulence [19].

Mesh grids are generated using GAMBIT with structured topology grids (shown in Fig. 3). Near the wall, the first grid points are