Aero-thermal optimization on multi-rows of film cooling over a flat plate has been performed to optimize the inclination angles. Hence three cylindrical holes with injection angles of $\alpha$, $\beta$, and $\gamma$ have been considered. The cooling hole has a 3 mm diameter and an inclined angle between 25-35°. Numerical simulations were performed at a fixed density ratio of 1.25 and blowing ratio of 0.5. The control-volume method with a SIMPLEC algorithm has been used to solve the steady-state RANS equations with SST $k$-$\omega$ turbulent model. The injection angles of the holes are selected as the design variables to perform the optimization of three rows of film cooling. In order to evaluate the performance of holes arrangement, two objective functions are defined based on aerodynamic losses and adiabatic film cooling effectiveness. The curve fitting method is used to find the optimal point of objective functions. The optimizations have been performed using the genetic algorithm method. Results of the present study show that the best performance of three rows of cooling holes was achieved in inclined angles 25.45, 32.85, and 33.1.

Key words: film cooling, genetic algorithm, inclined angle, numerical simulation

Introduction

In the modern gas turbines, substantial effort for increasing the thermal efficiency is directed to increase the inlet temperature of the turbine. Internal convection cooling and external film cooling should be applied to keep desirable life and operational demands under high temperature gases [1, 2]. The film cooling performance is highly affected by the hole shape and operating conditions [3-6].

Over the past decades, the numerical and experimental studies have been accomplished to understand the fundamental physics of film cooling flows [7]. Andrew et al. [8] studied normal (90°) and inclined holes (30° and 150°) and show that the inclined holes give better performance in comparison with normal holes. Abdullah and Funazaki [9] studied four rows of different inclined holes with two different angles of 20° and 35° and showed that more cooling effectiveness was achieved at a shallow hole angle of $\alpha = 20^\circ$.

Sarkar and Bose [10] showed that increasing the injection angle resulting in increasing the penetration of cooling air into the cross-flow which led to higher turbulence generation and reduction of cooling performance. Furthermore, the effects of cylindrical holes injection angle on the performance of cooling were simulated by Nasir et al. [11], Shine et al. [12], and

*Corresponding author, e-mail: rajabi@semnan.ac.ir
Hale et al. [13]. They highlighted that lower stream-wise injection angles perform better by producing a higher film cooling effectiveness. Yuen et al. [14] considered the effectiveness of film cooling by rows of cylindrical holes with different injection angles. Their experimental results show that the maximum efficiency corresponds to the injection angle of 30°. Guangchao et al. [15] investigated the effect of injection angle on cooling performance of dual fanned holes. Yuzhen et al. [16] results showed that the row spacing ratio affects the film cooling performance. They found that, better adiabatic cooling performance is achieved by using a smaller pitch, especially for the multi-hole patterns. Ai and Fletcher [17] concluded that for small hole spacing the cooling effectiveness at the area near to the exit of coolant is slightly higher than large hole spacing. Cun-Liang et al. [18] concluded that interaction between the adjacent jets at the hole spacing with small values lead to better effectiveness than large hole spacing at downstream locations.

Film cooling is a complex technique and a large number of geometrical parameters and flow can effect on this technique. Researchers have to considered multi-dimensional design space to find a global optimum solution. Lee and Kim [19] used coupling Kriging model and sequential quadratic programming to optimize a shaped hole. Johnson et al. [20] applied the genetic algorithm (GA) to optimize the film cooling array of a high pressure turbine on the vane pressure side. A multi-objective technique is performed by Lee et al. [21] to optimize a shape of one row of fan shape holes. Ayoubi et al. [22] optimized a round shape cooling hole by developing a non-dominated sorting GA. They improved the film cooling effectiveness over the surface while reducing the aerodynamic losses. Wang et al. [23] optimized of a fan-shaped hole by coupling the RBF neural network and GA. In general, previous studies are focused only to optimize the averaged film cooling effectiveness through the hole shape [21–24].

The aim of the present numerical study is to investigate the effects of the inclined angles of three rows of cylindrical holes on area-averaged film cooling effectiveness and pressure difference, simultaneously. Optimum point of inclined angles is found with optimization of the objective functions by using the GA. A set of optimal designs were presented as Pareto-optimal solutions.

**Computational domain**

The computational domain of the present study has been shown in fig. 1. Injection angle of each cylindrical hole with 3 mm diameters is varied from 25°-35°. It is essential to model plenum, film hole and the cross-flow regions simultaneously for a detailed demonstration of interaction between jet and cross-flow [25]. The plenum is as a supplier of the cooling air and coolant enters the cross-flow area through the film hole. The computational model was extend-
ed 50 mm high from the test surface. The uniform air velocity is set for plenum inlet to apply the desired blowing ratio \((M = 0.5)\). The operating parameters used to compute the performance of the film cooling process was given in tab. 1.

Turbulence intensity is assumed to be 0.2% and 0.1% for free stream and coolant, respectively. Two lateral planes in the mainstream duct were assumed symmetry boundary condition. The top surface of mainstream duct is away from the test plate sufficiently and this is a reason that top surface defined as symmetry plane. Velocity inlet boundary condition was defined for the freestream and coolant inlet, and outflow boundary condition was applied at the outlet surface of mainstream duct, whereas the plate, coolant-pipe, and plenum were modeled as an adiabatic wall with the no-slip condition. The spacing of two rows of coolant holes in the streamwise direction is \(5D\) and the spacing between center-to-center of adjacent holes was set to \(4D\).

Table 1. operating parameters

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Freestream temperature</td>
<td>373.15 K</td>
</tr>
<tr>
<td>Blowing ratio</td>
<td>0.5</td>
</tr>
<tr>
<td>Density ratio</td>
<td>1.25</td>
</tr>
<tr>
<td>Coolant temperature</td>
<td>298.15 K</td>
</tr>
</tbody>
</table>

Numerical method

In the present study, the governing equations have been solved using ANSYS FLUENT 16.0.0 [26] to find the effects of the design variables on the objective functions. The steady-state Reynolds averaged Navier-Stokes equations, can be written:

\[
\frac{\partial U_i}{\partial x_j} = 0
\]

\[
\rho U_i \frac{\partial U_j}{\partial x_j} = - \frac{\partial p}{\partial x_j} + \frac{\partial \bar{\varepsilon}}{\partial x_j} - \frac{\partial}{\partial x_j} \left( \rho U_i u_j \right)
\]

\[
U_i \frac{\partial \Theta}{\partial x_j} = \frac{\partial^2 \Theta}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} \left( \bar{u}_i \bar{\theta} \right)
\]

where \(\bar{\mu}\) and \(\bar{\theta}\) are known as the Reynolds stress tensor and the turbulent heat flux vector, respectively. The SST \(k-\omega\) turbulence model is more accurate and reliable for a wide class of flows (e.g., adverse pressure gradient flows, airfoils and jet in cross-flow) in comparison with the standard \(k-\omega\) and the standard \(k-\varepsilon\) models. Details of the mathematical formulation can be found in [26, 27]. The pressure-velocity coupling was performed by the SIMPLC algorithm.

Structural meshes, fig. 2, were constructed by using ANSYS ICEM and grid nodes near the wall refined significantly where the first layer grid height is set to be small enough to make sure that \(Y^+\) value at the near-wall cell should be of the order of 1. The grid sensitivity test for lower bound is given in fig. 3. Three grids with different density and numbers of cells have been generated to achieve the optimal number of cells. Therefore, the grid with 1057095 cells has been selected for the computations. The adiabatic film cooling effectiveness on the centerline can be predicted directly by using:

\[
\eta = \frac{T_{\infty} - T_w}{T_c - T_{\infty}}
\]
Objective functions and design variables

In the present study, aero-thermal optimization on multi-rows film cooling holes has been performed. Hence two objective functions based on adiabatic cooling effectiveness and aerodynamic losses have been defined. The non-dimensional pressure ($\Delta P/P_{plenum}$), is chosen as the objective function for aerodynamic loss. The averaged film-cooling effectiveness over the test surface has been considered as the objective function for film cooling performance:

$$\bar{\eta}(x,z) = \frac{1}{4D \times 23D} \int_{-2D}^{2D} \int_{-3D}^{17D} \eta(x,z) \, dx \, dz$$

In the present study, three geometrical parameters have been considered as design variables. These parameters including the injection angle of the holes $\alpha$, $\beta$, and $\gamma$. These angles are shown in fig. 4. The results of previous numerical studies have been considered in selecting the ranges of the design variables. Also, primary simulations for each design variable have been accomplished in wide ranges. These design variables and ultimate ranges are given in tab. 2.

Table 2. Lower and upper limits of the design variables

<table>
<thead>
<tr>
<th>Design variables</th>
<th>Lower bound</th>
<th>Upper bound</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$ [°]</td>
<td>25</td>
<td>35</td>
</tr>
<tr>
<td>$\beta$ [°]</td>
<td>25</td>
<td>35</td>
</tr>
<tr>
<td>$\gamma$ [°]</td>
<td>25</td>
<td>35</td>
</tr>
</tbody>
</table>

Optimization procedure

The optimization procedure has been used in this study is shown in fig. 5. Three design variables are chosen, and the design space was determined at first. At the design points, Numerical simulation is used to compute the values of objective function. Multi-objective optimization needs many assessments of the objective function find the optimal solutions. In the present study, the objective function values are evaluated by the curve fitting method (CFM), which prevents numerical cost. The CFM is based on a methodology that fits the sum of exponential or polynomial functions to numerical simulation results.
Results and discussion

Twenty seven cooling holes configurations are modeled to considered the effect of design variables on the objective function. From these 27 configurations, 20 groups have been selected for initial population or training samples (TRS). The remaining (7 groups) are also considered for the testing sample (TES). Numerical simulations have been done for all 27 cooling configurations. In the present study, initial population is chosen accidentally from all configurations, and the randomly design points are spread through the design space evenly.

Figure 6 shows the non-dimensional temperature contours at \(x/d = 0, 5,\) and 9 for upper (three holes with inclined angles 35°) and lower bound (three holes with inclined angles 25°) of the configurations. The jets interaction with cross-flow is more when large streamwise angle are used and consequently due to jet lift-off, the mainstream penetrates more into the jets flow [11]. In the downstream, the coolant air is moved away from the wall due to the presence of the counter-rotating vortex structure. These vortex structures push the cooling air upward and pull the hot mainstream gases toward the wall. As expected, move away from the coolant jet/wake from the wall along the centerline is higher for upper bound rather than lower bound configuration.

Figure 7 shows a pair of symmetric vortices (kidney vortices), which are created due to the interaction between the cooling air and hot mainstream. These kidney vortices significantly
reduce the effectiveness of film cooling by pushing the coolant from the target surface. This figure also shows that center of the counter-rotating vortexes for the lower band is closer to the surface than those obtained for the upper band. This is the main reason for higher film cooling effectiveness in this case. As shown in fig. 6(a), the holes with three inclined angles 25°, show better adiabatic effectiveness on the test surface. The objective functions values for each design variable in the initial population are listed in tab. 3.

The adiabatic effectiveness results from the simulations showed significant improvements from the lower bound when compared to upper bound configuration. This table shows that reducing the inclined angles from 35-25°, will result in approximately 21% increasing in area-averaged film cooling effectiveness, as mentioned by previous studies [11-14]. But it is notable that, reducing the inclined angles will also leads to increasing in pressure difference. The difference between the averages of the three jets output pressure and the inlet pressure of the plenum is increased from 27.711-34.034 Pa, that’s means approximately 23% increasing in pressure difference.

<table>
<thead>
<tr>
<th>Initial configurations</th>
<th>Design variables</th>
<th>Objective functions</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$\alpha$ [°]</td>
<td>$\beta$ [°]</td>
</tr>
<tr>
<td>TRS 1</td>
<td>25</td>
<td>25</td>
</tr>
<tr>
<td>TRS 2</td>
<td>25</td>
<td>25</td>
</tr>
<tr>
<td>TRS 3</td>
<td>25</td>
<td>30</td>
</tr>
<tr>
<td>TRS 4</td>
<td>25</td>
<td>30</td>
</tr>
<tr>
<td>TRS 5</td>
<td>25</td>
<td>35</td>
</tr>
<tr>
<td>TRS 6</td>
<td>25</td>
<td>35</td>
</tr>
<tr>
<td>TRS 7</td>
<td>30</td>
<td>25</td>
</tr>
<tr>
<td>TRS 8</td>
<td>30</td>
<td>25</td>
</tr>
<tr>
<td>TRS 9</td>
<td>30</td>
<td>30</td>
</tr>
<tr>
<td>TRS 10</td>
<td>30</td>
<td>30</td>
</tr>
<tr>
<td>TRS 11</td>
<td>30</td>
<td>30</td>
</tr>
<tr>
<td>TRS 12</td>
<td>30</td>
<td>35</td>
</tr>
<tr>
<td>TRS 13</td>
<td>30</td>
<td>35</td>
</tr>
<tr>
<td>TRS 14</td>
<td>35</td>
<td>25</td>
</tr>
<tr>
<td>TRS 15</td>
<td>35</td>
<td>25</td>
</tr>
<tr>
<td>TRS 16</td>
<td>35</td>
<td>30</td>
</tr>
<tr>
<td>TRS 17</td>
<td>35</td>
<td>30</td>
</tr>
<tr>
<td>TRS 18</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>TRS 19</td>
<td>35</td>
<td>35</td>
</tr>
<tr>
<td>TRS 20</td>
<td>35</td>
<td>35</td>
</tr>
</tbody>
</table>

The relationship between the objective functions and design variables are obtained by using curve fitting. This method (CFM) is usually applied for analyzing the results of numerical and experimental studies in film cooling technique [28-31]. The optimum design point can be detected by the following curve fittings equations:
\[
\eta = 0.2354 \left( \tan \alpha \right)^{0.2812} \left( \tan \beta \right)^{-0.116} \left( \tan \gamma \right)^{-0.0772}
\]  \hspace{1cm} (6)

\[
\Delta \frac{P}{P_{\text{plenum}}} = 0.494 \left( \tan \alpha \right)^{0.0701} \left( \tan \beta \right)^{-0.1106} \left( \tan \gamma \right)^{-0.124}
\]  \hspace{1cm} (7)

Seven TES were used to evaluate the accuracy of CFM method vs. the numerical simulation. These TES are listed in tab. 4. With these curve fittings, the maximum error in predicting the objective functions are approximately 1.5% and 0.8% for area-averaged film cooling effectiveness, \( \eta \), and aerodynamic loss, respectively.

<table>
<thead>
<tr>
<th>Testing sample</th>
<th>Design variables</th>
<th>Objective function 1 (( \eta ))</th>
<th>Objective function 2 (( \Delta \frac{P}{P_{\text{plenum}}} ))</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( \alpha [^\circ] )</td>
<td>( \beta [^\circ] )</td>
<td>( \gamma [^\circ] )</td>
</tr>
<tr>
<td>TES 1</td>
<td>25</td>
<td>25</td>
<td>35</td>
</tr>
<tr>
<td>TES 2</td>
<td>25</td>
<td>30</td>
<td>30</td>
</tr>
<tr>
<td>TES 3</td>
<td>25</td>
<td>35</td>
<td>25</td>
</tr>
<tr>
<td>TES 4</td>
<td>30</td>
<td>25</td>
<td>30</td>
</tr>
<tr>
<td>TES 5</td>
<td>30</td>
<td>35</td>
<td>25</td>
</tr>
<tr>
<td>TES 6</td>
<td>35</td>
<td>25</td>
<td>25</td>
</tr>
<tr>
<td>TES 7</td>
<td>35</td>
<td>30</td>
<td>35</td>
</tr>
</tbody>
</table>

In the present study, the cooling optimization has been done based on the GA. For film cooling applications, satisfactory solutions can be given quickly by GA [20, 24]. Darwin’s theory of evolution is the motivation of the GA. Finding the best available global design is the purpose of the GA process. In this study, the GA is written in C programming language.

The present study aims are to find an optimum point which in this point, the area-averaged film cooling effectiveness has a maximum value and the ratio of the pressure difference to the inlet pressure of the plenum is also minimized. A set of optimal designs have been presented as Pareto-optimal solutions. With this process, the optimum point of design variables \( \alpha \), \( \beta \), and \( \gamma \) are 25.45, 32.85, and 33.1, respectively.

Numerical simulation has also been used for the optimal point which concluded from CFM-GA (\( \alpha = 25.45[^\circ], \beta = 32.85[^\circ], \text{and} \gamma = 33.1[^\circ] \)). The film cooling effectiveness and the ratio of the pressure difference to the inlet pressure of the plenum at optimal points are compared in tab. 5. The optimal film cooling effectiveness gained from CFM and CFD simulation are 0.315 and 0.316, respectively. Also, the optimal ratio of the pressure difference to the inlet pressure of the plenum calculated from CFM and RANS solution is 0.576 and 0.577, respectively, and the difference between these values is less than 0.2%.

<table>
<thead>
<tr>
<th>Design variables</th>
<th>Objective function 1 (( \eta ))</th>
<th>Objective function 2 (( \Delta \frac{P}{P_{\text{plenum}}} ))</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha [^\circ] )</td>
<td>( \beta [^\circ] )</td>
<td>( \gamma [^\circ] )</td>
</tr>
<tr>
<td>Optimal design by GA</td>
<td>25.45</td>
<td>32.85</td>
</tr>
</tbody>
</table>
Figure 8 compares the centerline adiabatic effectiveness at $M = 0.5$. As mentioned by Walters and Leylek [32] the pressure gradient in the injection region causes the increase of the momentum of the fluid in the downstream region of the jet exit. At the streamwise inclination of 35°, the effectiveness is lower than that for 25° streamwise injections, as the higher streamwise angle lead to greater jet penetration and result in lower coolant coverage. Under a specified blowing ratio ($M = 0.5$), the three holes with inclined angles 35° shows a lower film cooling effectiveness in comparison with other cases. When the multi-hole injection with inclined angles 25° is used, the decrease in strength of kidney vortices leads to a significant increase of the film cooling effectiveness. The most notable improvement of the film cooling effectiveness is considered in the between of the holes region. The optimum design that is calculated by the GA improves the distributions of the film-cooling effectiveness along the centerline in comparison with the upper bound case.

Figure 9 shows the film-cooling effectiveness contours on the test surface for optimum and original geometries (upper and lower bound cases) at different $x/d$. According to these figures and previous studies [11-14], the distribution of cooling air for the three holes with inclined angles 35° is lower than other cases in both directions (streamwise and spanwise). This can be the consequence of the jet lift-off and the dilution of the coolant air downstream of the film hole, as shown in fig. 6, which results in low cooling effectiveness. These figures also show that the penetration of the holes with inclined angles 25° is weaker than that for the other cases. As shown in fig. 9, the inclined angles of multi-hole arrangement, can significantly affect the distribution of the adiabatic film cooling effectiveness.

The comparison of the two objective functions at optimal design point and reference geometries is shown in fig. 10. This chart shows that the optimized shape improved area-averaged film-cooling effectiveness over the test surface by approximately 13% in comparison with the upper bound reference geometry. Also, this shape improved second objective function ($\Delta P/P_{plenum}$) by approximately 8% in comparison with the lower bound reference geometry.
Effect of the different blowing ratio on the second objective function ($\Delta P/P_{\text{plenum}}$) is shown in fig. 11. Eight blowing ratios, included $M = 0.3, 0.5, 0.8, 1, 1.5, 2, 2.5, \text{and } 3$, were investigated for this plot. Since the optimum and reference geometries show a similar trend, therefore, the optimum and lower geometries are selected for display. The pressure ratio was inversely proportional to the area-averaged film cooling effectiveness. The high momentum of the cooling air forms a strong CRVP with increasing of blowing ratio, which lifts the cooling air from the target surface and consequently the cooling air replaces with the hot mainstream in the near-wall region [4]. This results in decreasing film cooling effectiveness [11]. As shown in fig. 11, the pressure ratio increase with increasing the blowing ratios. For $M < 1.0$, the increasing trend of the second objective function is more significant, especially between the blowing ratios 0.3-0.8.

Conclusions

The optimization of film cooling performance was carried out numerically by GA. The numerical investigation is performed on three rows of the cylindrical holes which every hole has a 3 mm diameter and the inclined angle between 25-35°. Numerical simulations are done at a fixed density ratio and blowing of 1.25 and 0.5, respectively. The spacing of two rows of coolant holes in the streamwise direction is 5D and the spacing between center-to-center of adjacent holes was set to 4D. The control-volume method with a SIMPLEC algorithm has been used to solve the steady-state RANS equations. The turbulent flow and heat transfer are modeled with the $k$-$\omega$ SST turbulence model. Reducing the inclined angles from 35-25° will result in approximately 21% increase in area-averaged film cooling effectiveness. It is notable that, reducing the inclined angles results in an increase in the pressure difference between the averages of the jets output pressure and inlet pressure of the plenum.

For the three design variables, namely, the inclined angles, 20 groups of TRS and seven groups of TES are generated. Optimizations have been performed for two objective functions, which are defined as area-averaged film cooling effectiveness, $\bar{\eta}$, and the ratio of the pressure difference to the inlet pressure of the plenum ($\Delta P/P_{\text{plenum}}$), simultaneously, and contribution effect of each objective functions on final optimization are the same. As the results, the difference between the value of two objective functions calculated by CFM and CFD was less than 1.5%. The optimum point of the design variables $\alpha$, $\beta$, and $\gamma$ are found in $25.45^\circ$, $32.85^\circ$, and $33.1^\circ$, respectively. The optimal film cooling effectiveness computed from CFM and RANS solution are 0.315 and 0.316, respectively. The average film cooling effectiveness obtained by CFM has less than 0.4% difference with its value calculated by the numerical simulation. Also, the optimal ratios of the pressure difference to the inlet pressure of the plenum obtained from CFM and CFD are 0.576 and 0.577, respectively, and the difference between these values is less than 0.2%. The optimized shape improved the first objective function by approximately 13% in comparison with the upper bound and the second objective function by approximately 8% in comparison with the lower bound reference geometry.
Nomenclature

\( D \) – diameter of the hole, [mm]
\( k \) – turbulent kinetic energy, \([\text{m}^2\text{s}^{-2}]\)
\( M \) – blowing ratio \(= \frac{(\rho U)}{\rho U_\infty} \)
\( T \) – temperature, [K]
\( x \) – x-direction, streamwise distance [mm]
\( y \) – y-direction, vertical distance [mm]
\( z \) – z-direction, spanwise distance [mm]

\( e \) – dissipation rate of turbulent kinetic energy, \([\text{m}^2\text{s}^{-3}]\)
\( \eta \) – adiabatic film cooling effectiveness,
\( \Theta \) – internal energy, [J]
\( \theta \) – heat flux, [Wm\(^{-2}\)]
\( \mu \) – dynamic viscosity, [kgm\(^{-1}\)s\(^{-1}\)]
\( \rho \) – density of the fluid, [kgm\(^{-3}\)]

Greek symbol

\( \alpha \) – injection angle of the 1st hole, [°]
\( \beta \) – injection angle of the 2nd hole, [°]
\( \gamma \) – injection angle of the 3rd hole, [°]

Subscript

\( \omega \) – adiabatic wall
\( c \) – coolant
\( \infty \) – free stream

References


[26] ***, ANSYS Inc. ANSYS FLUENT user’s guide, ANSYS FLUENT 16.0.0, Conon-sburg, PA, USA, 2014


