THE EFFECT OF ASPECT RATIO OF ELLIPTICAL COOLING HOLES ON THERMAL CHARACTERISTICS OF GAS TURBINE BLADE

Akram Luabi\textsuperscript{1,2}, Naseer Hamza\textsuperscript{1}
\textsuperscript{1}Mechanical Engineering Department, University of Al-Qadisiyah, Ad'Diwaniyah 58001, Iraq
\textsuperscript{2}Al-Diwaniyah Gas Turbine Power Plant, Ad'Diwaniyah 58001, Iraq
akramalkhanfar@gmail.com

Abstract
In advanced gas turbine, the improvement of the thermal efficiency and power output is required. Among these improvements, which still getting attention considerably, increasing turbine inlet temperature that may exceed the melting point of the blade material and to avoid that internal blade cooling technique is incorporated. The common traditional cross-sectional area of the cooling channels is circular. In this paper, a modification regarding the shape of cooling hole is suggested to enhance the heat transfer between the coolant in cooling serpentine and hot gases from the combustion chamber and thus minimizing the severe temperature usually occurs in the gas turbine blade especially at first row of the rotor. The proposed shape is elliptical and to discuss different scenarios of cooling schemes, the aspect ratio of ellipse axes and the distribution of whole holes are also analyzed to figure out the best results in terms of temperature distribution, heat transferred and cooling effectiveness of each model. The three new models consist of cooling holes of (5, 6 and 7) located at the center of the curvature line of blade. Results were compared with those obtained using the model of the GE company (model-1) which is already installed in (AL-DIWANIAH GAS TURBINE POWER PLANT). It were analyzed on the basis of real data, such as geometry profile, total cooling area, and boundary conditions. Results have been discussed and it is found that model-4 is the optimum solution, because of maximum total heat transfer rate and reduction in the blade trailing edge temperature about 24\% are attained in this model compared to model-1, and it is found increase number of cooling holes in the blade leads to lowers the blade trailing edge temperature, this depends on the design location of this holes on the blade surface. Meanwhile, the optimum ratio of the model-4 in the reduction of maximum temperature is (a / b = 2) by the dimensions of (a = 2 & b = 1). Steady state thermal analysis is carried out using CFD, Inconel 718 alloy selection as material and air is used as a coolant as the real condition of.

Keywords: gas turbine, heat transfer, temperature distribution, blade, CFD
1. Introduction

With the advent in Gas turbine technology, its usage as a prime mover has become prominent, since last few decades (1). One of the most important applications of gas turbines is in power generation, though it has been in use for aircraft propulsion since long time. The efficiency and power output of gas turbine plants is dependent on the maximum temperatures attained in the cycle. Advanced gas turbine engines operate at high temperatures ($1200 – 1500$ °C) to improve thermal efficiency and power output (2, 3). With the increase in temperatures of gases, the heat transferred to the blades will also increase appreciably resulting in their thermal failure. With the existing materials, it is impossible to go for higher temperatures. Taking into account the metallurgical constraints, it is necessary to provide cooling arrangement for turbine blades to keep their metal temperature within allowable limits. Therefore, developments in turbine cooling technology play a critical role in increasing the thermal efficiency and power output of advanced gas turbines.

Literature scanning mentions that conjugate heat transfer CHT examination performs an important function to guide the temperature distributions for the heat transfer representations. This shows the main characteristics effect on the internal cooling holes effectiveness, heat transfer coefficient and other parameters, in addition to holes configurations and Reynolds number for fluid flow (4-6).

Khari et. al (7) discussed four various models to analyze the heat transfer through gas turbine blade. Blade was analyzed with and without channels which have different hole numbers. In their results, the best cooling scenario was in terms of temperature distribution and transfer rate of heat which was in case of the blade with 13 holes.

Deepanraj et.al (8) considered in their study a blade manufactured from Titanium- Aluminum Alloy. They analyzed different models with different number of holes to find out the optimum number of holes for good performance. In Finite element analysis, first thermal analysis followed by structural analysis is carried out. Their results indicated a linear relationship between temperature distribution and number of holes.

Another type of investigation presented in the study of (9) where heat transfer characteristics were experimentally investigated on a heated flat plate equipped with micro-channels in order to simulate the real situation of cooling channels of gas turbine blade. They found a sensible difference between macro and micro scale of the process of forced convection heat transfer in terms of its parameter especially Nusselt number.

Some researchers investigated the model of k- turbulence model and the wall function to predict the flow characteristics more accurate model as in the work of (10). They found perfect coincidence of their model with experimental results.

The type of alloy that gas turbine blade is made from also had a great attention of researchers. In the works of (11-13) a detailed discussion of temperature distribution and associated thermal stresses had been performed. In related level, Reddy (14) had investigated the thermal and structural analysis of stainless steel blade. Four variant cases containing of blade without and with different number of passages (7, 8, 9 and 10) were examined to discover the perfect number of cooling channels.

The coolant that flows in the serpentine of cooling channels in gas turbine blade also plays an important role. Moskalenko et al (15) tested two types of coolants: air and steam. A comparison of coolant air and steam was carried out, and the
computations were done using ANSYS Fluent. Results showed that coolant steam is more efficient than the air when examined using the same conditions.

Abdulla and Isam (16) fulfilled the numerical analysis of different cases of heat transfer at different orientation for a geometry of semi-circular. Their goal is to get high amount of transferred of convective heat transfer at minimum surface area.

A new method of calculation coolant air requirement is presented in the work of (17, 18). They used an in-house code and compare its results with other adopted methods. The advantage of their method over the others was its ability to calculate in more reliable way at different operating conditions.

Hasanpour et al (18) took into their consideration the effect of selection the turbulence model on the conjugate heat transfer. Many models were used based on the arrangement and rearrangement of cooling holes ant their positions. Their results in principle coincided with the literature regarding the highest temperature in the trailing edge zone.

Although there are many studies that have been performed in the field of gas turbine research as aforementioned earlier, little attention has been paid to the shape of cooling channel itself. In this paper different cooling channels have been considered and compared their characteristics with common circular one.

In this paper the duct configuration has been changed into three cases studies to get optimum temperature distribution and heat transfer and the analysis is performed on the GE gas turbine blade over the blade surfaces, leading edge and trailing edge, comparison is made based on average and local Nu number at different blade configurations. Additionally, numerical analysis was done for different geometric shapes such as ellipse and square model shape. The geometric of GE model and other models are simulated based on the experimental setup of AdDiwaniyah power plant. The CFD analysis are developed in the framework of Ansys CFX 18.1. The average heat transfer coefficient and the maximum surface temperature are used to examine the performance for comparison target.

2. Governing Equations

The fluid flow governing equations and heat transfer that represent the case study are introduced below:

- Mass Equations (continuity equations)

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0
\]  

(1)

- Momentum equations

Component of the momentum equation at x-axis

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{\mu}{\rho} \frac{\partial u^2}{\partial x^2} + \frac{\mu}{\rho} \frac{\partial u^2}{\partial y^2} + \frac{\mu}{\rho} \frac{\partial u^2}{\partial z^2}
\]  

(2)

Component of the momentum equation in the y-axis

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \frac{\mu}{\rho} \frac{\partial v^2}{\partial x^2} + \frac{\mu}{\rho} \frac{\partial v^2}{\partial y^2} + \frac{\mu}{\rho} \frac{\partial v^2}{\partial z^2}
\]  

(3)

And Component of the momentum equation in the z-axis

\[
\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \frac{\mu}{\rho} \frac{\partial w^2}{\partial x^2} + \frac{\mu}{\rho} \frac{\partial w^2}{\partial y^2} + \frac{\mu}{\rho} \frac{\partial w^2}{\partial z^2}
\]  

(4)

- Energy equation

The energy equation is given in form:

\[
\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} = \frac{K}{\rho C_p} \left[ \frac{\partial T^2}{\partial x^2} + \frac{\partial T^2}{\partial y^2} + \frac{\partial T^2}{\partial z^2} \right] + \frac{1}{\rho C_p} \frac{\partial P}{\partial t} + \Phi
\]  

(5)

- Turbulence model
Due to its importance and significance, the turbulence model has a notable attention in previous studies to simulate the turbulent heat transfer, among them, the Shear Stress Transport (SST) turbulence model (19) and the Renormalized Group (RNG) k-ε model (20, 21) which show a good convergence in their result. A low-Reynolds number (LRN) k-ω turbulence model, with improved heat transfer predictions, is proposed by Bredberg J. in his works (22-28) which shows better results and convergence, especially as it takes into account the rotational effect of the gas turbine and separation of flow at walls and thus it has been selected in this paper.

- **Heat Transfer**

  In the gas turbine blades the most dominant transfer of heat is convection heat transfer. Convection heat transfer actually occurs by two mechanisms that are the combined effect of conduction, and heat transfer by flow of hot gases from coming from the combustor. Therefore many researchers adopt the concept of conjugate heat transfer to model the transfer of heat in the domain of gas turbine blade.

  \[
  \text{Conduction Heat Transfer} \quad q_{\text{cond}} = -k \frac{dT}{dx} \quad (6)
  \]

  \[
  \text{Convection Heat Transfer} \quad q_{\text{conv}} = h(T_f - T_b) \quad (7)
  \]

  The flow can be approximated to the flow over a flat plate which is justified by many researchers (29-32),

  For hot gasses \( Nu = 0.664 \ Pr^{1/3} \ Re^{1/2} \) \( (8) \)

  For coolant air \( Nu = 0.023 \ Pr^{0.3} \ Re^{4/5} \) \( (9) \)

  \*

  \* Radiation Heat Transfer (RHT)

  In this study the radiative heat transfer calculation was neglected since there is a very small effect of RHT compared with convective and conductive heat transfer.

  \*

  \* Cooling effectiveness (\( \phi \))

  \[
  \phi = \frac{T_g - T_b}{T_g - T_c} \quad (10)
  \]

  Where \( T_w, T_g \) and \( T_c \) denote to the temperature of the metal at wall, temperature of the gas and the temperature of the coolant, respectively

  - **Boundary conditions**

    Theoretical estimation has been based on the real geometrical and operating conditions at AL-DIWANIAH GAS TURBINE POWER PLANT. Some of these data are listed in Table 1.

  - **Design methodology**

    The goal of the research paper is to suggest another cooling scheme that reduces the cooling hole number and / or improver the thermal performance of the cooling system incorporated inside the gas turbine blade. Thus another geometry, number and distribution are suggested to be investigated and assessed whether they are more efficient than the existing cooling scheme already installed in gas turbine blade or not. The whole data and details of the proposed cooling channels are listed in Table 2.
### Table 1. The boundary conditions of case study

#### Hot gases boundary conditions

| Properties                              | value | Units   |
|-----------------------------------------|-------|---------|
| Inlet pressure                          | 9.8   | bar     |
| Mass flow rate                          | 6.4   | kg/sec  |
| Inlet flow velocity                     | 154   | m/sec   |
| Inlet Mach number of peak mode          | 0.17  | ------- |
| Exit Mach number of peak mode           | 0.132 | ------- |
| Turbulent intensity of hot gas for all cases | 5%    | ------- |

#### Coolant air boundary conditions

| Properties                              | value | Units |
|-----------------------------------------|-------|-------|
| Inlet temperature                       | 366   | K     |
| Inlet pressure                          | 9.8   | bar   |
| Model no.          | Configurati on type of the model | Number of cooling holes (N) | Dimensions and type |
|-------------------|----------------------------------|-----------------------------|---------------------|
|                   |                                  |                             | Ordinal hole no.    |
| Model-1 (Original design) | circular                         | N = 11                      | Diameter (mm)       |
| 1st               | 2nd                              | 3rd                         | 1th                 |
| 2nd               | 3rd                              | 4th                         | 2th                 |
| 3rd               | 4th                              | 5th                         | 3th                 |
| 4th               | 5th                              | 6th                         | 4th                 |
| 5th               | 6th                              | 7th                         | 5th                 |
| 6th               | 7th                              | 8th                         | 6th                 |
| 7th               | 8th                              | 9th                         | 7th                 |
| 8th               | 9th                              | 10th                        | 8th                 |
| 9th               | 10th                             | 11th                        | 9th                 |
| 10th              | 11th                             |                             | 10th                |

| Proposed cooling configuration scheme |
|---------------------------------------|
| Model-2 (suggested design)            |
| elliptical                            |
| N = 5                                 |
| Ordinal hole no.                      |
| major length (a, mm)                  |
| minor length (b, mm)                  |
| Aspect ratio                          |
| 1st                                   |
| 2.5                                   |
| 20                                      |
| 2th                                   |
| 2.4                                   |
| 20                                      |
| 3rd                                   |
| 2.4                                   |
| 20                                      |
| 4th                                   |
| 2.4                                   |
| 20                                      |
| 5th                                   |
| 3.1                                   |
| 20                                      |

| Model-3 (suggested design)            |
| elliptical                            |
| N = 6                                 |
| Ordinal hole no.                      |
| major length (a)                      |
| minor length (b)                      |
| Aspect ratio                          |
| 1st                                   |
| 2.5                                   |
| 2.4                                   |
| 2th                                   |
| 2.5                                   |
| 2.4                                   |
| 3rd                                   |
| 2.5                                   |
| 2.4                                   |
| 4th                                   |
| 2.5                                   |
| 2.4                                   |
| 5th                                   |
| 2.5                                   |
| 2.4                                   |
| 6th                                   |
| 2.5                                   |
| 2.4                                   |
3. **Numerical analysis in ANSYS-CFX:**

The three dimensional domains involve one blade inside the flow path of hot gases domain, with boundary conditions utilized to simulate the blade model. The domain extended 11.7 cm from the hub to the tip which represent the height of blade. The computational domain of model-1 (GE blade) is generated and the same way of computational domain is used for three suggested models that are listed their details in **Table 2**. A section of single blade was taken and represented by the ANSYS CFX definitions to simulate the actual flow domain of all blades, which represent the first stage of gas turbine.

The subsequences figures illustrate the reference model and the proposed models (elliptical shape) of gas turbine blade with cooling channels that are suggested by current study. **Figure 1** shows the model-1 (GE model) with 11 cooling holes distributed at the centerline of tip section from the leading edge to the trailing edge, while **Figure 2** shows the model-2 with 5 cooling holes configuration and **Figure 3** displays model-3 with 6 cooling holes configuration and the **Figure 4** shows the model - 4 with 7 cooling holes, which were distributed as the same as model-1, extending from the shroud to the hub surface of blade perpendicularly.

| Model-4 (suggested design) | elliptical | N = 7 | Ordinal hole no. | major length (a) | minor length (b) | Aspect ratio |
|----------------------------|------------|-------|------------------|------------------|------------------|-------------|
|                            |            |       | 1<sup>st</sup>   | 4                | 2.5              | 1.6          |
|                            |            |       | 2<sup>nd</sup>   | 2.5              | 1.5              | 1.67         |
|                            |            |       | 3<sup>rd</sup>   | 2.5              | 1.5              | 1.67         |
|                            |            |       | 4<sup>th</sup>   | 2.5              | 1.5              | 1.67         |
|                            |            |       | 5<sup>th</sup>   | 2.5              | 1.5              | 1.67         |
|                            |            |       | 6<sup>th</sup>   | 2.5              | 1.5              | 1.67         |
|                            |            |       | 7<sup>th</sup>   | 2                | 1                | 2            |
Figure 1 shows the model-1 (GE model).

Figure 2 shows model-2 with 5 elliptical cooling holes.

Figure 3 shows model-3 with 6 elliptical cooling holes.
4. Mesh generation

The model system is meshed by ICEM CFX. Unstructured tetrahedral mesh is used in the current work which minimize the time spent for generating meshes and it is of acceptable accuracy. For GE model blade with 11 cooling holes, the option of global size function is curvature to possess the features of the curves. In the present model, the patch conforming mesh with tetrahedral type has been used to generate mesh for the flow domain and solid blade. This method applies a path that starts from the bottom up to the crest, that meaning it will arrived the edges first and then followed by faces extending to the full size. The overall elements number are 4,126,444. Growth rate of 10% has been maintained and 24 degree curvature normal angle to ensure that the mesh count does not go high while maintaining the accuracy and features of the model. The flow domain has been inflated to pick up the boundary layers effective. Five layers have been prepared for boundary layer zone with a first aspect ratio of 5 and growth rate of 10%. The minimal element size is 0.1 mm and size of max face is 5 mm and the volume size of tetrahedral is about 10 mm. Similarly for all other models, same growth rate, minimum and maximum values have been adopted respectively. Figure 5 shows mesh of all models at the same method and setup.
5. Results and discussions

Calculations were fulfilled in ANSYS CFX software for the four models under real boundary conditions to determine the heat transfer analysis and temperature distribution between three domains hot gasses, blade surfaces and coolant air.

**Verification and validation of current study**

*Verification numerical results with other research papers*

At first the numerical results are tested by applying geometry and boundary condition of other research papers and tested in the current software to check the similarity and agreement between their results and results that will be obtained by using this software. HARSHAD A. (33) designed and modelled the blade in CATIA v5 and ICEM CFD software with 12 cooling holes. Their case study is applied in current model and the results show a good agreement and approximately the same as shown in Figure 6.
Results of current numerical code | Results of [33]
--- | ---

**Figure 6. Verification with the study of HARSHAD A. [33].**

Hasanpour, A (18) also the numerical analysis of gas turbine blade equipped with 10 cooling holes and compared the results. Another verification study blade with six channels at varying configuration was compared with the [COMSOL Multiphysics] reference model.

Again their geometry and boundary conditions for both cases of study are applied and the results were very close. It is observed that for both works the temperature is maximum at the trailing edge and less severe at the leading edge of the blade as shown in **Figure 7.**
Validation of the results

A blade with circular cooling holes is already installed in AL-DIWANIAH GAS TURBINE POWER PLANT and there are several thermocouples that measure the temperatures at different zones. The numerical results of the current study of first model are compared directly with these values to check its accuracy. According to comparison of results, the software and its setup for suggested models are judged to be accurate enough and its results are reliable and acceptable.

By comparison the values of real data obtained from CCR and the numerical value of points that are at the same positions and orientation on the surface of gas turbine blade; for instance taking top and bottom points at the trailing edge; the deviation in calculation of temperatures is found to be:

- \[ top \text{ point deviation} = \frac{791 - 738.6}{791} = 6.6\% \]
- \[ bottom \text{ point deviation} = \frac{775 - 730}{775} = 5.8\% \]

As illustrated in Figures 8 - 9.

Figure 8. Photo of central computer in CCR in AL-DIWANIAH GAS TURBINE POWER PLANT
Figure 9 shows capture of (CCR) with real data.

**Behavior of temperature distribution**

Figure 10 shows the temperatures distribution of all studied models. The maximum temperature of all models occurs at the trailing edge, where the thickness of blade is the minimum. Therefore, the existence of cooling holes with appropriate amount of cooling air is crucial matter to reduce the maximum temperature.

In the model-1 (GE blade), the cooling channels located at the rear of the trailing edge are the minimum ones in size compared to other cooling holes which have bigger diameters, this greatly affects the rate of reduction of the maximum temperature in this region and may be expose it to critical hot spots. As proposed solution in this study the size of fourth and fifth cooling channels in the models 2, 3 and 4 are bigger than model-1, therefore the drop of maximum temperature is higher. Taking into consideration the total cooling area of all models is the same.

In addition to the size of the cooling holes, the location and configuration of channels are playing a significant role. As shown in the top view and contour of each model.

Cooling holes in the first and second suggested models are big dimensions and few numbers, thus the coolant air flows at high streamline and absorbs the largest possible amount of temperature from the blade surfaces to reduce the maximum temperature.
The results show that the maximum temperature of model-1 is about (900 K), while in model-2 is (792 K), model-3 is (783 K) and model-4 is (774). These differences of reduction in the maximum temperature for the four models are due to variety of the configuration of holes as shown in figure 12 which affect the heat transfer parameters like Nusselt number, total local exposed area, flow patterns, etc.

Since it is of principal matter to operate blade before its metallurgical limit in reliable and smooth operation, it is clear that – based on the aforementioned findings - the elliptical shape with 7 cooling channels in the model-4 is the optimum solution scheme of the present study as shown in Figure 11.
For the model-4 the dimensions of seventh channel close to the trailing edge have been changed to get the optimum \((a / b)\) ratio, which gives the best reduction in the maximum temperature. This should affect on the temperature distribution at this region (trailing edge).

Five different dimensions were simulated by changing the value of \((a, b)\) to achieve the optimum ratio in the reduction of maximum temperature. The ratio is \((a / b = 2)\) by the dimensions \((a = 2 \& b = 1)\) were adopted in the model-5 with maintaining the same surface area of cooling as shown in Table 3 and Figure 12.

Table 3 shows the dimensions of seventh hole.

| a    | b    | a / b ratio |
|------|------|-------------|
| 1.65 | 1.5  | 1.1         |
| 1.88 | 1.2  | 1.566667    |
| 2    | 1    | 2           |
| 2.12 | 0.7  | 3.028571    |
| 2.21 | 0.3  | 7.366667    |

Figure 12 relation between max temperature and a / b ratio

Therefore it should be careful in the dimensional selection of elliptical shape. It notes that the elliptical with 7 cooling channels with ratio of \((a / b = 2)\) is the optimum solution of the present study due to the lowest temperature at trailing edge as compared with the other models as shown in Figures 11-12.

As for the heat transfer, the rate of heat flux is performed by analyzing the conjugate heat transfer between the solid-fluid domains at steady-state conditions. The results show that the rate of flux in the model-1 is \((445 \text{ kw/m}^2)\), model-2 \((525 \text{ kw/m}^2)\), model- 3 \((530 \text{ kw/m}^2)\) and \((558 \text{ kw/m}^2)\) in the model- 4, which is the maximum and dominant as compared to the others because of the behavior and amount of cooling rate in the model-4.

This gives a pure explanation for the temperature drop in the elliptical model because of there is a direct relationship between the rate of heat transfer and temperatures distribution through domains. The maximum increase in the total heat transfer is about \((558 \text{ kW/m}^2)\) in the model-4 as shown in Figure 13 - d.
Figure 13 shows the heat flux of different models.

On the other hand, with it has been noted that the hot gases enter at a steady rate velocity, which represents the inlet value of boundary condition of the hot gasses, then its velocity increases to the maximum value at the suction side and reduces at the pressure side, this change happens mainly due to the geometrical profile of the blade which fluctuates pressure and kinetic energies and switches between them as shown in Figures 14-15.

Figure 14. Velocity profile along hot gasses domain model-4.
As it is well known, there is an inverse relation between the pressure and velocity of flow, when the velocity of a hot gasses speeds up some of the energy from the random motion of fluid is used to move faster in the fluid’s direction of motion according to the law of conservation of energy and this results in a lower pressure. Figures 16-19 display the pressure variation of the blade on both sides and indicate that the pressure is maximum at pressure side and minimum at suction side because of the high velocity of hot gasses at the suction side.
Figure 17. Pressure distribution of Model-2.

Figure 18. Pressure distribution of Model-3.

Figure 19. Pressure distribution of Model-4.
The main objective of the suggested models in the present study is to improve the rate of heat transfer and reduce the maximum temperature created in the blade surfaces selecting the optimum serpentine configuration of cooling holes. But, at the same time, this should not affect the other performance parameters of gas turbine engine such as its power output. Therefore, it is necessary to keep an adequate amount of cooling flow from compressor at sufficient amount to cool down the blade but not so much to the limit that losses the gas turbine its pressure ratio as shown in Figures 20-21.

Figure 20. Velocity streamline of all models.

Figure 21. Pressure behavior of all models.
6. Conclusions

A GT gas turbine blade has been modeled in ANSYS CFX to simulate the temperature, pressure and heat transfer throughout its surfaces and inside the cooling channels serpentines. In order to minimize the severe temperatures that appear at hot spots on the blade surface, the conjugate heat transfer between hot incoming gases from the gas turbine combustor and coolant air from the compressor should be maximized. In the current study, the focus is directed mainly at the geometry of these serpentine channels and the effect of changing them into elliptical shape with various hole numbers rather than the common circular ones. The results show that the elliptical channels achieve lower temperatures than circular and the model-4 is the best. The temperature reduction in case of model-2 is 12\% in comparison with the original geometrical design of circular shape, while it is 13\% in case of model-3 and it is 14\% in the model-4 as summarized in Table 4. According to the aforementioned results, it may be concluded that the proposed geometries are better than existed geometry if the manufacturer can avoid the difficulties of casting them. The elliptical geometry seems to be the best compromise as compared with circular and it has promising results. The temperature distribution in model 4 is less severe rather than other models and consequently less expected thermal stresses.

| Table 4. Summary of results of the models |
|------------------------------------------|
| Type of model | Boundary conditions | Results |
| Mass flow rate (kg/s) | Velocity (m/s) | Pressure (bar) | Coolant air temperature (K) | Total cooling area (cm²) | NO. of channels | Max temperature LE (K) | Max temperature TE (K) | Maximum heat flux (kW/m²) |
| Model-1 Circular | 6.4 | 154 | 9.8 | 366 | 113 | 1 | 1 | 790 | 900 | 444 |
| Model-2 Elliptical 5 holes | 6.4 | 154 | 9.8 | 366 | 113 | 1 | 5 | 750 | 792 | 525 |
| Model-3 Elliptical 6 holes | 6.4 | 154 | 9.8 | 366 | 113 | 5 | 6 | 745 | 783 | 530 |
| Model-4 Elliptical 7 holes | 6.4 | 154 | 9.8 | 366 | 113 | 7 | 7 | 735 | 774 | 558 |
References

1. Han J-C. Recent studies in turbine blade cooling. International Journal of Rotating Machinery. 2004;10.
2. Chiesa P, Consonni S, Lozza G, Macchi E, editors. Predicting the ultimate performance of advanced power cycles based on very high temperature gas turbine engines. Turbo Expo: Power for Land, Sea, and Air; 1993: American Society of Mechanical Engineers.
3. Kurt H, Recebli Z, Gedik E. Performance analysis of open cycle gas turbines. International Journal of Energy Research. 2009;33(3):285-94.
4. Kassab A, Divo E, Heidmann J, Steinthorsson E, Rodriguez F. BEM/FVM conjugate heat transfer analysis of a three-dimensional film cooled turbine blade. International journal of numerical methods for heat & fluid flow. 2003.
5. Ho K-S, Urwiller C, Konan SM, Liu JS, Aguilar B, editors. Conjugate heat transfer analysis for gas turbine cooled blade. Turbo Expo: Power for Land, Sea, and Air; 2014: American Society of Mechanical Engineers.
6. Yoshiara T, Sasaki D, Nakahashi K, editors. Conjugate heat transfer simulation of cooled turbine blades using unstructured-mesh CFD solver. 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition; 2011.
7. Brahmaiah KH, Kumar ML. Heat transfer analysis of gas turbine blade through cooling holes. International Journal of Computational Engineering Research, ISSN. 2014:2250-3005.
8. Deepanraj B, Lawrence P, Sankaranarayanan G. Theoretical analysis of gas turbine blade by finite element method. Scientific world. 2011;9(9):29-33.
9. Choo KS, Youn YJ, Kim SJ, Lee DH. Heat transfer characteristics of a micro-scale impinging slot jet. International Journal of Heat and Mass Transfer. 2009;52(13-14):3169-75.
10. Öztekin E, Aydin O, Avci M. Heat transfer in a turbulent slot jet flow impinging on concave surfaces. International Communications in Heat and Mass Transfer. 2013;44:77-82.
11. Vaferi K, Nekahi S, Vajdi M, Moghanlou FS, Shokouhimehr M, Motallebzadeh A, et al. Heat transfer, thermal stress and failure analyses in a TiB2 gas turbine stator blade. Ceramics International. 2019;45(15):19331-9.
12. Moghanlou FS, Vajdi M, Motallebzadeh A, Sha J, Shokouhimehr M, Asl MS. Numerical analyses of heat transfer and thermal stress in a ZrB2 gas turbine stator blade. Ceramics International. 2019;45(14):177.50-42
13. Nekahi S, Vaferi K, Vajdi M, Moghanlou FS, Asl MS, Shokouhimehr M. A numerical approach to the heat transfer and thermal stress in a gas turbine stator blade made of HfB2. Ceramics International. 2019;45(18):24060-9.
14. Reddy R. THERMAL STRUCTURAL ANALYSIS OF GAS TURBINE BLADE USING SOLIDWORKS.
15. Moskalenko A, Kozhevnikov A. Estimation of gas turbine blades cooling efficiency. Procedia Engineering. 2016;150(1):61-7.
16. Al Ali AR, Janajreh I. Numerical simulation of turbine blade cooling via jet impingement. Energy Procedia. 2015;75:3220-9.
17. Torbidoni L, Horlock J. A new method to calculate the coolant requirements of a high-temperature gas turbine blade. J Turbomach. 2005;127(1):191-9.
18. Hasanpour A, Farhadi M, Ashorynejad H. Hole configuration effect on turbine blade cooling. World Academy of Science, Engineering and Technology. 2011;49(1):240-4.
19. Hwang S, Son C, Seo D, Rhee D-H, Cha B. Comparative study on steady and unsteady conjugate heat transfer analysis of a high pressure turbine blade. Applied Thermal Engineering. 2016;99:765-75.

20. Al-Shamani AN, Sopian K, Mohammed H, Mat S, Ruslan MH, Abed AM. Enhancement heat transfer characteristics in the channel with Trapezoidal rib–groove using nanofluids. Case Studies in Thermal Engineering. 2015;5:48-58.

21. Immarigeon A, Hassan I. An advanced impingement/film cooling scheme for gas turbines–numerical study. International Journal of Numerical Methods for Heat & Fluid Flow. 2006.

22. Bredberg J. Prediction of flow and heat transfer inside turbine blades using EARSM, $k$-$\varepsilon$ and $k$-$\omega$ turbulence models. Report, Department of Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenburg Also available at www.tfd.chalmers.se/bredberg. 1999.

23. Bredberg J. On the wall boundary condition for turbulence models. Chalmers University of Technology, Department of Thermo and Fluid Dynamics Internal Report 00/4 Goteborg. 2000.

24. Bredberg J. On two-equation eddy-viscosity models. Department of Thermo and Fluid Dynamics, Chalmers University of Technology, Gothenburg, Sweden, Report. 2001.(8/01)

25. Bredberg J. Turbulence modelling for internal cooling of gas-turbine blades: Chalmers University of Technology Göteborg,, Sweden; 2002.

26. Bredberg J, Davidson L. Low-Reynolds number turbulence models: an approach for reducing mesh sensitivity. J Fluids Eng. 2004;126(1):14-21.

27. Bredberg J, Davidson L, Iacovides H, editors. Comparison of Near-wall Behavior and its Effect on Heat Transfer for $k$-$omega$ and $k$-$eps$ Turbulence Models in Rib-roughened 2D Channels. Turbulence Heat and Mass Transfer 3/Nagano, Y, Hanjalic, K, Tsuji, T; 2000.

28. Bredberg J, Peng S-H, Davidson L. An improved $k$–$\omega$ turbulence model applied to recirculating flows. International journal of heat and fluid flow. 2002;23(6):7.43-31

29. Burberi E, Massini D, Cocchi L, Mazzei L, Andreini A, Facchini B. Effect of rotation on a gas turbine blade internal cooling system: numerical investigation. Journal of Turbomachinery. 2017;139.(3)

30. Camci C. An experimental and numerical investigation of near cooling hole heat fluxes on a film-cooled turbine blade. 1989.

31. Cho S-Y, Yoon E-S, Choi B-S. A Study on an axial-type 2-D turbine blade shape for reducing the blade profile loss. KSME International Journal. 2002;16(8):1154-64.

32. Dey M, Jindal P, Roy AK, Kumar K. MAXIMIZATION OF COOLING EFFECTIVENESS OF TURBINE BLADE SURFACES USING DIFFERENT ARRANGEMENT OF COOLING HOLES AND VARIOUS BLOWING RATIOS. Journal of Engineering Science and Technology. 2018;13(4):909-24.

33. HARSHA D. CFD ANALYSIS ON RADIALCOOLING OF GAS TURBINE BLADE.