Numerical Study of the Effect Internal Holes Shapes in Blade Turbine Cooling

Hussein Talal Dhaiban
Refrigeration & Air-Conditioning Tech. - Dijlah University College. Qatar Al-Nada St, Baghdad, Iraq

Article Info

Article history:
Received 25 January 2016
Received in revised form
20 February 2016
Accepted 28 February 2016
Available online 15 June 2016

Keywords

Heat transfer, gas turbine and blade cooling

Abstract

As the temperature of combustion gases is higher than the melting temperature of the turbine materials, cooling of turbine parts in a gas turbine engine is necessary. The gap between rotating turbine blades and the stationary shroud provides an unintended flow path for hot gases. Gases that flow through the tip region cause pressure losses in the turbine section and high heat loads to the blade tip. This paper numerically studies the effect of change shapes of internal cooling holes and shows the effect of cooling at the tip of blade. Also measures the effect of pressure effectiveness and temperature distribution at the tip and alone holes channels.

In this study, results of change shapes of internal holes to circle, rectangle and triangle shows the circle shape is better in cooling by 45 ℃ and 125℃ when compared with rectangle and triangle shapes respectively. Streamlines results explain that they are very crowded at suction side and decreases at pressure sides and many of streamlines that passed through the gap and mixed with cooled flow. Results show also the temperature distribution alone the holes passages are reduced along them from shelf to tip and the maximum values at entrance region. Finally, the results show when the internal velocity of holes increases, the static temperature decreases in the blade.

1. Introduction

The gas turbine industry is always seeking to raise the thermal efficiency of the gas turbine engine by increasing the turbine inlet temperature. Increasing the operating temperature, however, leads to some major problems. Turbine blades, for example, are not able to withstand such high temperatures and thermal stresses. The design operating temperature in a gas turbine far surpasses the melting temperature of most materials. In modern gas turbines, sophisticated cooling schemes are implemented to help protect the blades and vanes from thermal failure.

Film cooling is an external cooling technique in which cool air is bled from the compressor stage, ducted to the internal chambers of the turbine blades, and discharged through small holes in the blade walls into the hot mainstream. This air provides a thin, cool, insulating blanket along the external surface of the turbine blade. As a result, the blade is able to sustain higher operating temperatures and achieve higher life cycles. Han et al. [1].

2. Review of Relevant Literature

For engine design it is important to have accurate information about the heat transfer and aerodynamics in turbine blade. Numbers of numerical and experimental studies were conducted to examine the effect of holes shapes in turbine blade. The configurations of heat transfer devices that were examined include holes geometries and angles on the film cooling effectiveness.

Je-Chin H. and Trent A. V. [2], studied the effect of holes geometries on the film cooling effectiveness using pressure sensitive paint. The cooling flow was used the Nitrogen so that the oxygen concentration levels can be obtained for the test surface. Five total hole geometries were tested: fan-shaped laidback with a compound angle, fan-shaped laidback with a simple angle, a conical configuration with a compound angle, a conical configuration with a simple angle,

Corresponding Author
E-mail address: husseinayo@gmail.com

and the reference geometry (cylindrical holes). The effect of blowing ratio on film cooling effectiveness was presented for holes geometry. The geometry of the holes has little effect on the effectiveness at low blowing ratios: 0.3, 0.6, 1.2 and 1.8%. The laterally expanded holes show improved effectiveness at higher blowing ratios. All experiments were performed at mainstream velocity of 34m/s.

Jr-Ming Miao and Chen-Yuan [3], studied numerically the effects of blowing ratio and holes shape on the distributions of flow field and adiabatic film cooling effectiveness over a flat plate collocated with two rows of injection holes in staggered-hole arrangement. The blowing ratio was varied from 0.3% to 1.5%. The geometrical shapes were cylindrical round simple angle CRSA, forward-diffused simple angle FDSA and laterally diffused simple angle LDSA. Diameter of different shape of cooling holes in entrance surface is 5.0 mm. This study reveals that (1) the geometrical shape of the cooling holes has great effect on the adiabatic film cooling efficiency especially in the area near to the cooling holes. (2) The thermal-flow field over the surface of the film-cooled tested plate dominated by strength of the counter-rotating vortex pairs (CRVP) that generated by the interaction of individual cooling jet and the mainstream. (3) The structure of LDSA can also increase the lateral spread of the cooling flow, thus improves the span wise-averaged film cooled efficiency.

Hyams and Leylek [4] have focused on the detailed analysis in physics of film cooling process for five various kinds of shaped, stream wise-injected, inclined jets. This study suggested that the crucial flow mechanisms downstream of discrete-cooling hole could be clarified from vortices point of view. It can be observed from their results that the shape of the cooling holes has great effect to the downstream film cooling performance.

Bell et al. [5] also study the effects of the shape and angle orientation of the film cooling holes, such as: CYSA, LDSA, LDCA, FDSA and FDCA on the film cooling effectiveness by using experimental method. They have discovered that LDCA has the best film cooling performance, followed by FDCA over the range of blowing ratios from 0.5% to 2.0%. This was caused by the film diffusion from expanded holes shape. This consideration in lateral spreading of injection can reduce the exit momentum of the cooling flow and increase the diffusion effect at the lateral direction. Therefore, less jet penetration effect and lower velocity gradient were expected.

Gritsch et al. [6] has used infrared thermal-photographic system to analysis the heat transfer coefficients in the near-hole region of various shapes of the cooling hole with a flat plate model. The results show that fan type design with laterally expanded hole will have a lower heat transfer coefficient distribution when the blowing ratio was increase.

Jun Y. et al [7] they studied the numerical effect of hole shape on blade adiabatic cooling effectiveness has been carried out on four geometry models comprising a standard cylindrical hole, a cylindrical hole with an upstream ramp, a shaped diffuser, and a double console slot. In all the cases, the holes centerline has an inclination angle of 35 degree against the mainstream gas flow. Results of the cylindrical holes model were in good agreement with available experimental and other numerical data. For the other three holes geometry variants considered, it was found that the cooling effectiveness has been considerably enhanced by a max factor of 2, compared to that from the base model. The physical mechanism for this was mainly due to the weakening of coolant flow penetration in the vicinity of the holes exit, thus reducing the level of mixing and the entrainment with the surrounding hot gas flow.

The objectives of the study presented in this paper are to present numerically the effect of change shape of holes in turbine blade cooling at two blowing ratio.

2. Blade Model Configuration

A Three-dimensional blade profile was created for low speeds. The scaling and design of blade profile is discussed in Hohlfeld [8].The important geometric features of the study and the blade profiles are shown in Fig. (1).

Some important geometric and measurements of the pitch, chord, axial chord, inlet flow angle, blade rotation angle, inlet Reynolds number, and the tip gap height are tabulated in Table 1. These values were used in conjunction with the blade profile to construct computational models. The cooling holes locations in a blade turbine are shown in Tables 2.

3. Governing Equations

The numerical model for describe the fluid flow and heat transfer by the basic equations of mass, momentum and energy equations is developed under the following assumptions:

- Steady – three dimensional – irrational flow.
- Fully developed – turbulent – incompressible flow.
- Constant fluid properties.
- Ignored body forces.

The general form of governing equations is Verestage and Malalasekera [9]:

$$\frac{\partial(\rho u \phi)}{\partial z}+\frac{1}{r}\frac{\partial(\rho r v \phi)}{\partial r}=\frac{\partial}{\partial z}\left(\boldsymbol{\Gamma}^{\phi}\frac{\partial \phi}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(r\boldsymbol{\Gamma}^{\phi}\frac{\partial \phi}{\partial r}\right)+S_{\phi} \quad \dots (1)$$

The transform equation (1) from physical domain to computational domain, and that lead to obtain the transformation of the general governing equations as follow:-

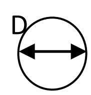
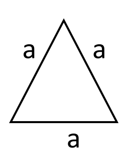
$$\frac{\partial(\rho G_1 \phi)}{\partial \zeta}+\frac{\partial(\rho G_2 \phi)}{\partial \eta}=\frac{\partial}{\partial \zeta}\left(\boldsymbol{\Gamma}^{\phi}\, J\, a\frac{\partial \phi}{\partial \zeta}\right)+\frac{\partial}{\partial \eta}\left(\boldsymbol{\Gamma}^{\phi}\, J\, c\frac{\partial \phi}{\partial \eta}\right)+S_{total} \dots (2)$$

where a,c is transformation coefficient

Table1: Geometry and flow condition of the blade.

Parameter	Value
Axial Chord, B_x	35 cm
True Chord, C	53 cm
Pitch, P	43 cm
Span, S	55.2 cm
Inlet Angle, θ	16.5°
Blade Angle, ϕ	50°
Tip gap, H	1 cm

Table2: Blade holes locations

Holes shape	Holes No.	X(cm)	Y(cm)	Dimension (cm)
Circle 	1	3.43	40.5	D=0.7
	2	8.3	37.5	D=0.7
	3	20.3	31.48	D=0.6
	4	26.02	21.857	D=0.6
	5	30.88	10.66	D=0.6
Triangle 	1	3.43	40.5	a=1
	2	8.3	37.5	a=1
	3	20.3	31.48	a=0.8
	4	26.02	21.857	a=0.8
	5	30.88	10.66	a=0.8

Rectangle a b	1	3.43	40.5	a=1,b=0.8
	2	8.3	37.5	a=1,b=0.8
	3	20.3	31.48	a=0.8,b=0.6
	4	26.02	21.857	a=0.8,b=0.6
	5	30.88	10.66	a=0.8,b=0.6

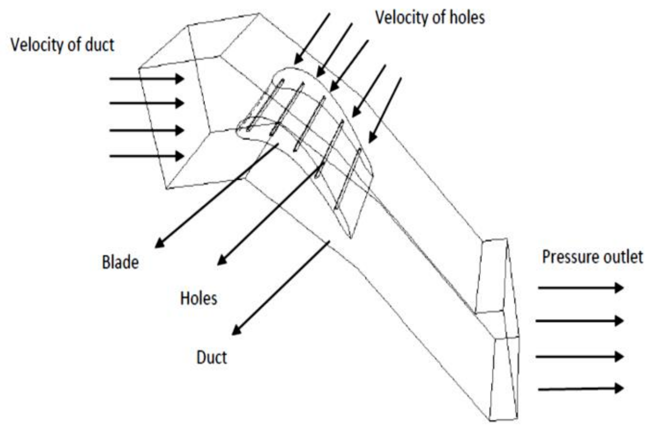
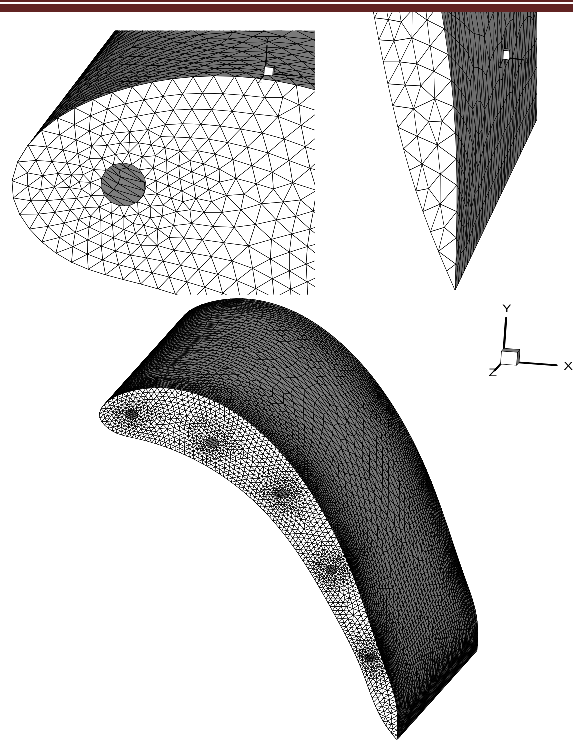


Fig. 1 Boundary conditions placed on the blade model



1%

2 %

1%

2 %

Fig. 2 shows the mesh of the test rig at circular holes shaped

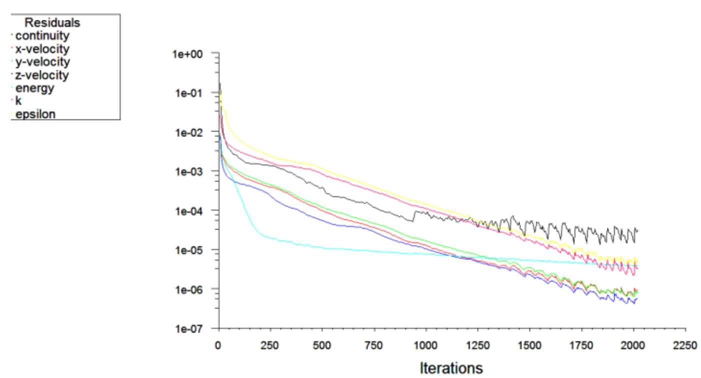


Fig. (3) Typical residual convergence for a large tip gap model

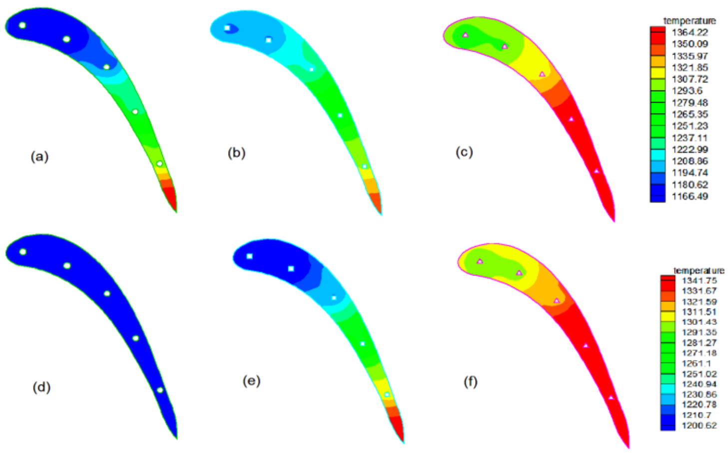


Fig. 3 the contour of temperature distribution at the tip of blade with (1% and 2%) blowing ratio at different shapes of holes

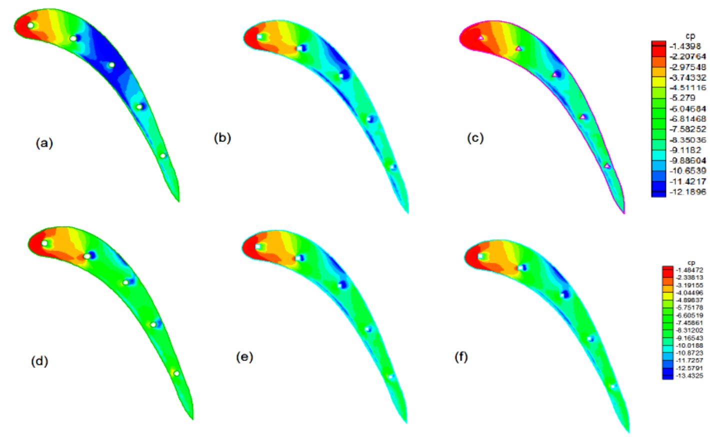


Fig. 4 The contour of pressure effectiveness (Cp) at the tip of blade with (1% and 2%) blowing ratio

Table 3: Summary of computational methodology

GAMBIT			
Operation			
Geometry	**Mesh**	**Zones**	**Tools**
Create real vertex	**• mesh edge**	Zone type: fluid	**Mesh face: using size function method**
Create edge from vertex :interpolate	mesh the holes with interval size = 20	Define faces as main and cooling velocity and pressure outlet with some walls	Type :meshed
Create face from wireframe	**• mesh face**		Growth rate:1.1
Stitch faces to great volume	element triangle, type pave		Max. size = 2
unite, subtract and intersect volumes	Mesh volume		
FLUENT			
Define			
General	**Models**	**Materials**	**Boundary Conditions**
• Solver	Energy: ON	**Fluid, air**	No slip between the blade surface and the air
Type: Pressure-Based	Viscous: Turbulent	Density: 1.225 kg/m^3	No relative velocity between blade and the shroud.
Time: Steady		Specific Heat: 1.007 J/kg•K	All walls of duct were adiabatic.
Velocity: Absolute		Thermal Conductivity: 0.0263W/m•K	The main inlet velocity through the duct is 11.3 m/s
3D Space		Viscosity: 1.846e-05 kg/m•s	The inlet temperature through the duct is 1523 K.
• Gravity: NO		**Wall, Stainless Nickel**	The temperature through the holes of blade is 523 K.
		Specific Heat: 460.6 J/kg•K	
		Density: 8900 kg/m^3	
		Thermal Conductivity: 91.74W/m•K	
Solve			
Solution Methods	**Solution Controls**	**Monitors**	**Solution Initialization**

• Pressure-Velocity Coupling	• Non-Iterative Solver Relaxation Factors	• Residuals	• Reference Frame: Relative to Cell Zone
Scheme: SIMPLE	Pressure: 0.2	Options- Print, Plot: ON	• Initial Values
• Spatial Discretization	Momentum: 0.5	Monitors convergence criteria	Gauge Pressure: 0 Pa
Gradient: Least Squares Cell Based	Energy: 0.9	Continuity:0.0001	x Velocity: 11.3 m/s
Pressure: Standard	Turbulent kinetic energy :0.5	Momentum:0.0001	y Velocity: 0 m/s
Momentum: Second Order Upwind	Turbulent dissipation rate: 0.5	k and ε:0.0001	Temperature: 1523 K
Energy: Second Order Upwind	Viscosity: 0.7	Energy:1E-7	
Turbulent: Second Order Upwind			
Display			
grid - contours - vectors - path lines - xy plot			

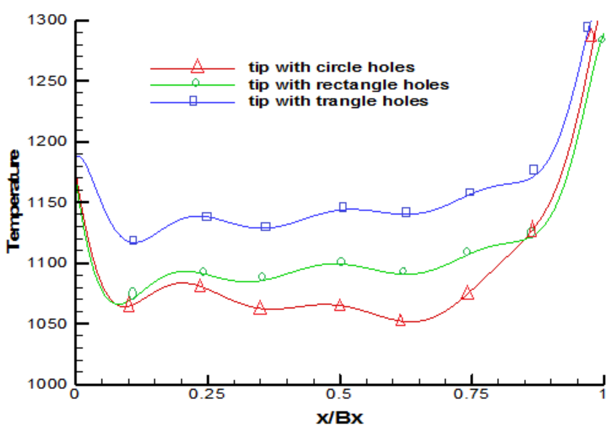
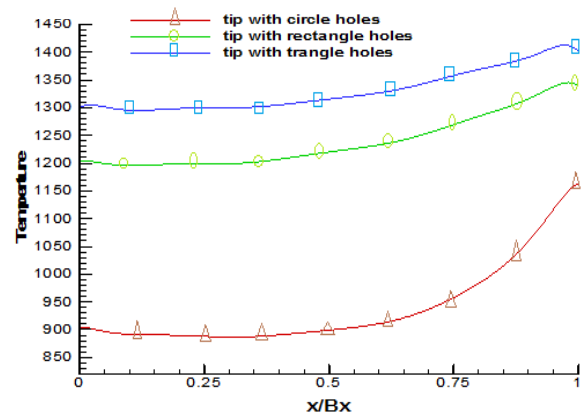
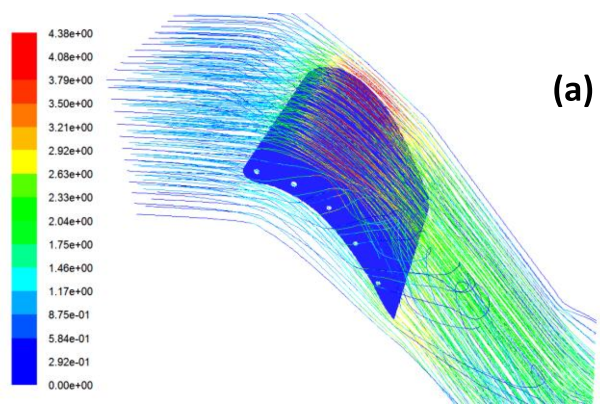


Fig. 5 The relation between the temperature ($°K$) and dimensionless distance (x/Bx) at the tip with blowing ratio 1%

u/umax





(a)

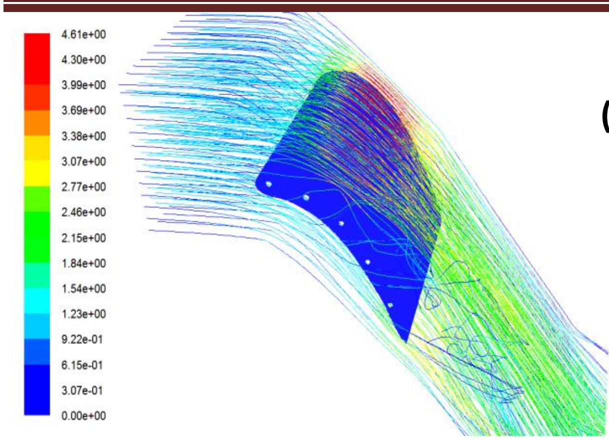


Fig. (6) The relation between the temperature ($°K$) and dimensionless distance (x/Bx) at the tip with blowing ratio 2%

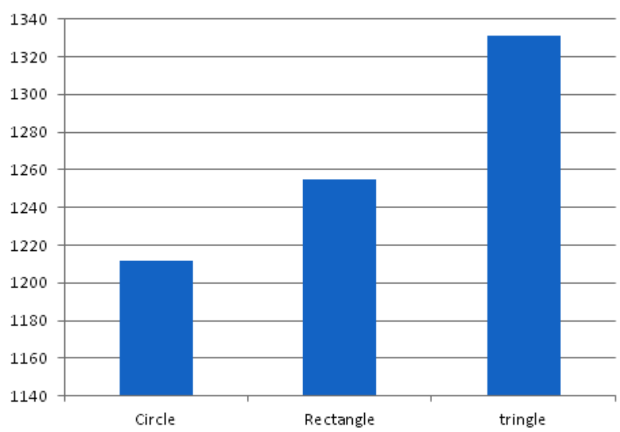


Fig. 7 the bar of area averaged temperature plotted at blowing ratio 1% for circle, rectangle and triangle holes

Fig. 8 Non-dimensional velocity of the streamlines for the internal circle holes at (a) 1% and (b) 2% blowing ratio

4. Computational Methodology

Computational fluid dynamics (CFD) simulation is performed to analysis numerically the effect of internal passages shapes in blade turbine cooling. The computational process can be divided into three main components consisting of a pre-processing stage, solver stage (processing), and finally a post-processing stage. The preprocessing by GAMBIT involves constructing the computational domain when the geometric assembly of the model then meshing, and finally the application of boundary conditions. Figs. 1,2 show the boundary conditions and mesh at tetrahedral cells of the test model.

The solver stage involves sending the model to a Fluent 6.3.26 (2009) [10] and solving the governing equations to provide results. The results from the solver are then analyzed in a post-processing stage in the same program. Fig. 3 shows scaled residuals of continuity, momentum, energy, k and ε as the solution progresses to convergence around 2000 iterations to reach the specified convergence levels. Table 3 shows the summary of computational methodology in Gambit and Fluent programs.

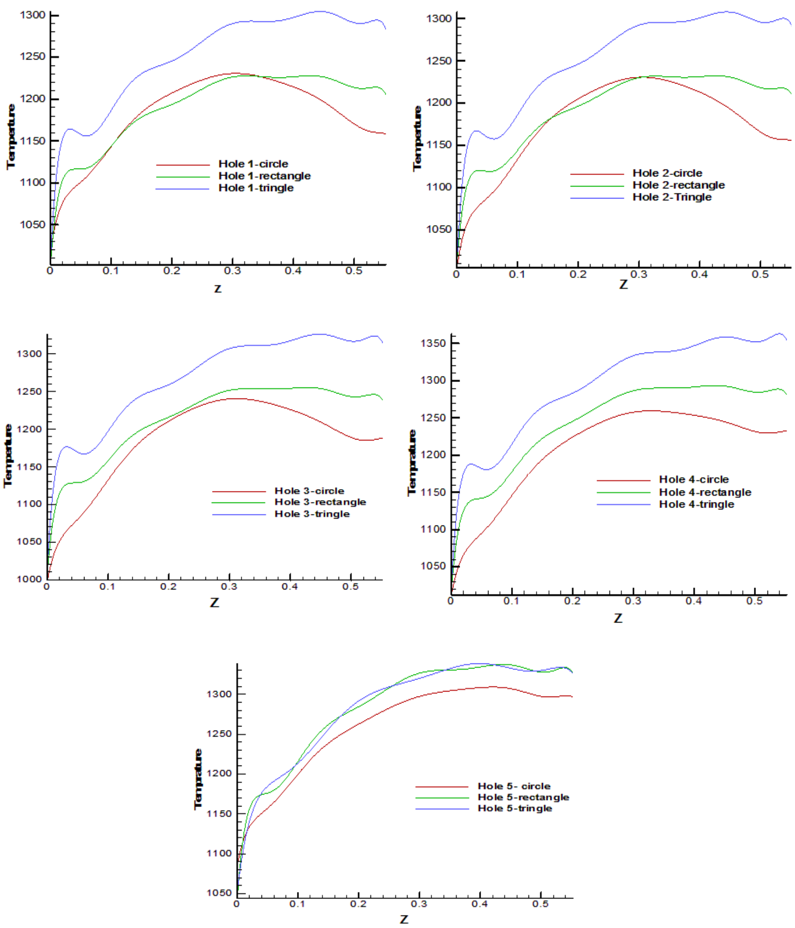


Fig. 9 wall temperature distribution through the length of holes (1, 2, 3, 4 and 5) in circle, rectangle and triangle shapes

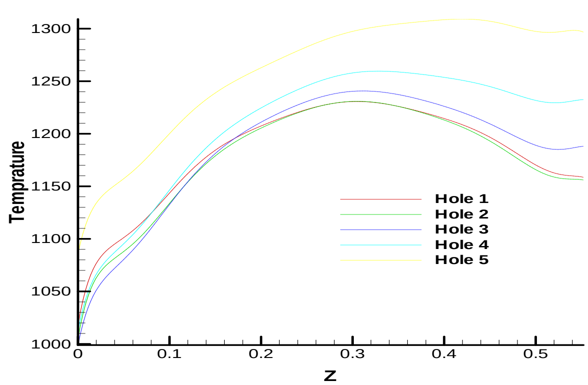


Fig. 10 the wall temperature distribution alone the blade with circle shape at 1% blowing ratio

5. Results and Discussion

Results are shown for cases where coolant is injected through internal holes with different shapes (circle, rectangle and triangle). Cooling levels of 1% and 2% blowing ratios, they are studied to explore thermal and flow effects within the internal cooling passage of the blade.

Fig. (3) Shows the contour of temperature distribution at the tip with two blowing ratios(1% and 2%) at different shapes of holes. This figure shows good tip cooling when coolant levels are at circle holes and shows better cooling near the leading edge. As expected additional coolant provides better cooling near the leading edge for rectangle holes and alone the blade at circle holes and remain the triangle shapes with small change in temperature around the holes. The temperature contours at the lower blowing ratios (1%) show the low momentum cooling jets being swept across the tip along with the hot tip leakage flow. Only as the cooling increases to the previously mentioned 2%, are enough cooling mass and momentum to penetrate upstream into the tip. For all cases, the trailing edge is thermally unaffected by the internal holes coolant.

Fig. (4) Shows pressure contours along the tip for different shapes of holes at 1% and 2% blowing ratios. When looking at the tip pressure contours one can see that there are many variations in these two blowing. As expected the leading edge shows a continual evolution with the addition of coolant. The pressure near the blade stagnation location increases with additional coolant. In addition, there is trend showing lower pressures within the film holes as more coolant is added. This figure also shows around the cavity that is increasingly present with additional blowing. These low pressures indicate high velocities, and these higher velocities result from the additional coolant added to this region. Pressure contours along both the suction and pressure sides of the blade that are within the main passage remain constant as the blowing level increases from 1% to 2%. This is the result of a more effective flow blockage from the film holes coolant that results in additional flow stagnation and thus higher pressures.

Fig. (5) Shows relation between the temperature and dimensionless distance (x/Bx) for different shapes of holes at the tip for 1% blowing ratio. Temperature has been averaged along the tip for tip with (x/Bx = 0) being at the blade stagnation point and (x/Bx = 1) at the trailing edge. The peak temperature occurs at the tip with triangle holes when compared with other shapes, but low temperature at the circle holes. Also from the figure the high temperature occur at the entrance region of dimensionless distance at leading edge and highest temperature at the end of dimensionless at trailing edge, because the coolant that floods the tip gap from the holes is eventually mix with the mainstream flow and does not remain in the tip gap. Fig. (6) shows the temperature at the tip with different shapes of holes degreases as the amount of coolant flow increases.

Fig. (7) Shows the bar of area averaged temperature plotted at blowing ratio 1% for different shape (circle, rectangle and triangle) holes at the tip. These results indicate the circle shape is better in cooling from triangle and rectangle shapes and these results compatible with result in Fig. (5).

Fig. (8) Depicts streamlines released upstream of the blade for a blowing ratios at 1% and 2%. In figure (8a) can clearly see that streamlines being large number of hot gas flow into the gap. This figure though should not be interpreted to mean that there is no tip leakage around the holes. We are only looking at streamlines released from a single plane upstream that obtaining a general understanding of the flow. Fig. (8b) there is minimal effect on the mainstream flow patterns in the gap. Along the suction-side, just above the leading edge of the blade for 2% blowing one can notice the large number of streamlines that are not passing over the blade when compared with 1% blowing. The tip leakage vortex does not seem to change substantially with the holes blowing, but along the suction side of the blade just above the holes there is a noticeable decrease in the number of streamlines that are diverted into the gap with the addition of cooling blowing.

Fig. (9) Show the relation between the temperature distributions alone the holes (1,2,3,4 and 5) at different shapes (circle, rectangle and triangle) with distance of span. From all results that show the triangle shape of holes is the higher temperature from the two shapes that below. Also the circle shape is given the lower temperature from other shapes.

From all results that prove the circle shape is better from other shapes, therefore many design engineers that made the internal holes from the circle shape to cool the blade material.

Fig. (10) Indicates the wall temperature distribution of the air flow through the circle holes passages that relation with the distance of span (z) at blowing ratio 1%. This figure shows hole 5 is at high temperature because the small distance between the suction and pressure sides and high temperature at trailing edge region. Additionally, this figure shows the low temperature at hole 2 when a large distance between suction and pressure and the high temperature at the surface does not affect largely on these hole. Finally, the wall temperature has minimum values of temperature at zero dimensional horizontal z (at entrance region), and maximum values at the end of dimensional distance.

6. Conclusions

In this study, the flow and heat transfer characteristics of flow the coolant in internal passages by different shapes (circle, rectangle and triangle) in turbine blade with two blowing ratios 1% and 2% are studied numerically.

- Results show the circle shape of internal holes is better for cooling the turbine tip by 45℃ and 125℃ when compared with rectangle and triangle shapes.
- Pressure effectiveness (Cp) results in the tip show that they are increased by increasing the blowing ratios.
- The results of flow calculations are used successfully to reshape the blade holes.
- Also, the temperature at the tip is not affected by change the shape at the triangle holes.
- Streamlines results explain that they are very crowded to suction side when increasing the blowing ratios.
- Results show that baseline temperature on the holes is reduced along the holes from shelf (base) to the tip.
- Finally, results show when the internal velocity is increased, the static temperature decreases in the blade.

Nomenclature

Symbol	Description	Dimension
ρ	density	Kg/m^3
Γ	diffusion coefficient	$N.s/m^2$
$\emptyset$	particles volume fraction	---
BR	blowing ratio (u_c/u_∞)	---
B_x	axial chord of the blade	M
C_p	pressure coefficient [$(P-P_{in})/0.5\rho U_{in}^2$]	---
G_1, G_2	contra variant velocity in ξ,η, respectively	m/s
J	Jacobean of coordinates transformation	---
P	static pressure	N/m^2
r, z	cylindrical coordinates	---
S_ϕ	source term of ϕ	---
S_{total}	total source term	---
T	temperature	˚C
u ,v	velocity component in r, z coordinates	m/s
Abbreviation		
SIMPLE	semi-implicit method for pressure linked equation	

References

[1] Han, J.C., Dutta, S., and Ekkad, S. Gas Turbine Heat Transfer and Cooling Technology, Taylor and Francis, New York, 2000, 540-559.

[2] Je-Chin H. and Trent A. V. Shaped Hole Effects on Film Cooling Effectiveness and a Comparison of Multiple Effectiveness Measurement Techniques, Texas A&M University, 2004.

[3] Jr-Ming M., Chen-Yuan, Numerical Approach to Hole Shape Effect on Film Cooling Effectiveness over Flat Plate Including Internal Impingement Cooling Chamber", National Defense University, 2005.

[4] Hyams D.G., Leylek J.A. A Detailed Analysis of Film Cooling Physics.Part III: Streamwise Injection with Shaped Holes, ASME J. Turbomach, 2000, 122-132.

[5] Bell C.M., Hamakawa H., Ligrani P.M., 2000, "Film Cooling From Shaped Holes, ASME J. Heat Transfer, 122, 224–232.

[6] Gritsch M., Schulz A., Wittig S., 2000,Film-Cooling Holes with Expanded Exits: Near-Hole Heat Transfer Coefficients, Int. J. of Heat and Fluid Flow, 146-155.

[7] Jun Y., Sohan S., Yufeng Y. and Tony W. D., Numerical Study of Hole Shape Effect on Blade Cooling, Modern Physics Letters, 24, 13. 2010.

[8] Hohlfeld, E. M. Film Cooling Predictions along the Tip and Platform of a Turbine Blade , Master's Thesis, Virginia Polytechnic Institute and State University, Blacksburg 2003.

[9] Verestage, H. K., and Malalasekera, W. An Introduction to Computational Fluid Dynamic-The Finite volume Method, Longman Group Ltd 1995.

[10] Fluent Inc., Fluent User's Guide, Version 6.3.26. 2009.