Film Cooling Holes Performance on A Flat Plate

*Harbi A. Daud**
Middle Technical University, Institute of Technology Baghdad Iraq
**dr.harbidaud104@mtu.edu.iq*

Mohammed F.Mohammed
University of Technology, Baghdad Iraq

ABSTRACT

The present study aims to evaluate numerically the influence of film cooling holes configuration. Five models of holes configuration were studied to examine the effectiveness of a single film cooling hole (8 mm diameter). A cylindrical hole, square hole, rotated square, triangle hole and rotated triangle hole was employed to eject the coolant air with constant cross sectional area at 35^0 *angle of injection. A 3-D finite-volume method was utilized to describe the turbulent flow over flat plate surface (as a model of a turbine blade surface). The mainstream domain (40d x10d x4d) had been represented in ANSYS FLUENT 16 with a standard* $k-\varepsilon$ *turbulence model. So far, in term of effective cooling, the numerical results elucidate noticeable enhancement realized at BR=0.5 due to a reduction in plate surface temperature compared with BR =0.25 and 1 for cylindrical hole model. Moreover, the holes configuration performance demonstrate a better surface effectiveness about 1.6 % and 1.2 % for rotated triangle and square models respectively compared with the cylindrical holes model at BR=1. It was found that the numerical results had an excellent matching with related published results which show the solidity of ANSYS FLUENT code to predict the cooling efficiency with varying blowing ratio.*

Keywords: *Film Cooling; Effectiveness; Holes Configuration; Blowing Ratio; ANSYS FLUENT*

ISSN 1823-5514, eISSN 2550-164X

Received for review: 2021-12-19
Accepted for publication: 2022-07-17
Published: 2022-09-15

NOMENCLATURE

BR	Blowing ratio $=\rho_c V_c/\rho_h V_h$
D	Film hole diameter (8 mm)
K	Turbulence kinetic energy
T_i	Injection film temperature (K)
T_∞	Hot gas temperature (K)
T_w	Local wall temperature (K)
U_c	Coolant velocity (m/sec)
U_∞	Hot gas velocity (m/sec)
u_j	Velocity vector (m/sec)
x, y & z	Spatial coordinates

Greek Symbols

ρ_c, ρ_h	Coolant and mainstream density
$\partial P/\partial x$	Pressure gradient
$k-\varepsilon$	Turbulent model
μ	Laminar viscosity
μ_t	Turbulent viscosity
η	Film cooling effectiveness

Subscripts

B.C	Boundary conditions
B.L	Boundary Layer
CFD	Computational Fluid Dynamics
EWT	Enhancement Wall Treatment
RANS	Reynolds Averaged Navier-Stokes
RNG	Re-Normalization Group Model

Introduction

The Mixing process between the inflowing hot gas and the secondary flow (coolant) was studied experimentally using low speed wind tunnel and numerically using 2D and 3D finite volume method via commercial code. Film efficiency and heat transfer were considered as scales to measure the improvement in the shape of cooling holes. Series of researches were accomplished to study the effectiveness of one hole followed by a row of cylindrical film cooling holes [1]. Eriksen et al. [2] clarified the amount of heat transfer coefficient distribution and the results showed the peak of heat transfer had found at the edge of spreading jets with about 27% increasing of lateral heat transfer compared with no blowing. Moreover, the competition between the researchers were continued to study the effects of blowing ratio, air density and injection angle for single hole and one row of holes. The highest effectiveness was observed at BR = 0.5 [3][4][5]. Recently, gas turbine researchers have endeavored to improve holes configuration to increase the performance of aerodynamic flow and film cooling efficiency. Hence, a shaped hole was studied theoretically and experimentally. One of the researchers; Bell et al. [6] had measured the local and averaged adiabatic film efficiency for cylindrical hole and shaped hole. Specifically, large increase of spanwise-averaged adiabatic effectiveness was directly proportional with the mass ratio for shaped holes. Then, Brittingham and Leylek [7] completed numerical simulation on cooling hole with compound-angle shaped holes and determined that the compound-angle-shaped holes was designed to remove crossflow line-of-sight among adjacent holes. Thus, the shaped-holes were the most efficient with angle of injection 35 degree. While, the negative effects were found on film cooling performance for shaped cooling holes at increase free turbulence even with high BR [8]. Through testing 3 types of film holes geometry; laidback hole (expanded in lateral and streamwise direction) and cylindrical, fan shaped (expanded in lateral direction). Likewise, Peng and Jiang [9] had found that the lower film cooling efficiency was obtained for fan-shaped with 30 degree angle of injection compared with cylindrical holes. Jose et al. [10] investigated the cylindrical film cooling holes over flat plate numerically and the results showed the effectiveness cooling in lower value at high blowing ratio 2 to 4 due to attachment and detachment theory. The effectiveness value was strongly affected by the amount of DR and BR.

In the near past, Salimi et al. [11] applied the upstream jet hole to enhance the film effectiveness of main hole through keeping the cross sections constant of the main and upstream jets compared with ordinary cooling jet. A novel sister shaped (SSH) and novel combined hole configuration were examined for the cooling performance over flat plate surface [12]-[13] respectively. SSH obtained a higher film hole efficiency and less jet lift-off effect comparison with the cylindrical and shaped holes for BR = 0.5 and 1. Similarly, combined hole provided considerable improvement in effectiveness and more uniform surface protection than conical and fan shape hole. Ajlan et al. [14] examined the cylindrical film cooling holes model with different BR over flat plate surface and created an arc trench model to provide the greatest improvement in effectiveness at BR = 0.5 and 1 experimentally. The heat transfer of swirl jet flow studied numerally and experimentally by Zerrout et al. [15] using impingement technique and the turbulent flow was presented as standard K-ε. Furthermore, Zhang and Zhu [16] investigated the staggered holes arrangement as small auxiliary holes. In addition to, trench hole and cylindrical hole case. So, the numerical results display that large-scale vortices produced by the auxiliary hole injection obstruct the progress of vortices generated by the main hole in the streamwise direction. Meanwhile, Jiang et al. [17] studied numerically the vortex impact of cooling mechanism on plate surface effectiveness through changing coolant delivery configurations at different BR value.

The objective of this paper is to study the effect of film cooling holes configuration as cylindrical, square, rotated square, triangle, and rotated triangle with 35° an angle of jet flow over the flat plat surface with constant cross sectional area. In addition, the study aims to assess robust ANSYS FLUENT code to predict the cooling efficiency via simulation of the effects of changing blowing ratio compared with previous published paper.

Computational Flow Model

To predict aerodynamic flow and highly nonlinear, steady transient flow, jet cross flow numerically; a Computation Fluid Dynamics (CFD) has appeared as the best efficient tools to solve three dimensional Reynolds Averaged Navier- Stokes (RANS) flow equations and the energy equation.

To simplify solving the conservation equations, a number of assumptions have been taken as steady state, incompressible flow and turbulence model demonstrated via $(k-\varepsilon)$ turbulent equations which are employed to expect the heat transfer of jet cross flow over flat plate surface. These partially differential equations take the form: Equations (1), (2), and (3).

$$\frac{\partial u_j}{\partial x_j} = 0 \qquad (1)$$

$$u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\frac{\mu + \mu_t}{\rho}\frac{\partial u_i}{\partial x_j}\right] \qquad (2)$$

$$u_j \frac{\partial T}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\frac{1}{\rho}\left(\frac{\mu}{\mathrm{Pr}} + \frac{\mu_t}{\mathrm{Pr}_t}\right)\frac{\partial T}{\partial x_j}\right] \qquad (3)$$

The standard $(k-\varepsilon)$ equation was used in this study as a common turbulent model, with a chosen effect of enhanced wall treatment. SIMPLE algorithm was selected to control the solution by resolving velocity-pressure coupling with implicit technique. The 2nd order upwind discretization results were approved and all numerical calculations in this paper were boosted with $1\text{x}10^{-7}$ as a factor of convergence.

Geometry and mesh for film holes

Figure 1 illustrates the geometry of single film cooling hole. This film cooling model was planned and analyzed [12]-[13]. The description of the holes in correspondence with previous studies is denoted by a film cooling diameter and angles of injection is 8 mm, and 35° respectively. Figure 2 shows the film cooling holes configuration as cylindrical, square, rotated square, triangle, and rotated triangle to inject the coolant liquid as a secondary flow over the flat plate surface with constant cross sectional area.

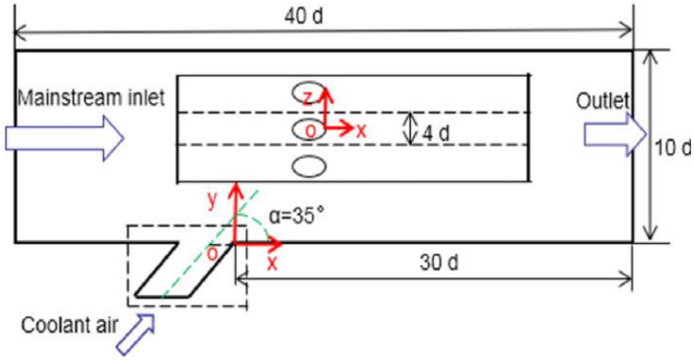

Figure 1: Geometry of film holes along streamwise direction [13]

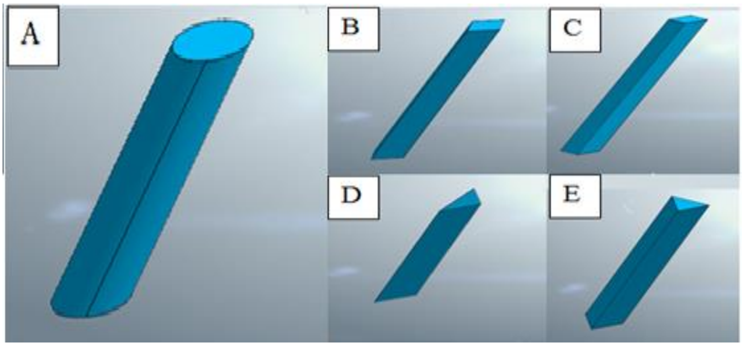

Figure 2: Film cooling holes configuration models: a) cylindrical hole, b) square hole, c) rotated square hole, d) triangle hole and e) rotated triangle hole

The computational domain in this paper was extracted from [12]-[13]. The mainstream domain is 40d long x 10d height x 4d width and the hole diameter is 8 mm. A structured mesh has been employed for the mainstream and film cooling holes domains with the FLUENT Wall Boundary Layer mesh option (WBL) (mesh system was utilized for all models included in this study). Numerical accuracy satisfactorily accomplished and amplified through refining the WBL mesh at the sub-layer area by present (y^+) value. The y^+ value is a specifically essential parameter to focus on the significance in simulations of CFD represents a great degree of accuracy y^+~12 as recommended by [18] when employing the RNG turbulence model with EWT. But, the value of $y^+<1$

for LES as recommanded by [19]. In addition to the simulation cases with SST turbulent model the y^+ value sufficientlly less than 1 despite of the simulation of y^+ surface indecated that 35.27, which falls under the layer of log-law area [20]. However, best quality of mesh for all domains (mainstream, cooling holes) keeps the value of y^+ in this study nearly y^+~3. So, enhancement wall treatment (EWT) has been selected to achieve numerical simulation. The final mesh generated for all models volume includes more than 2 million cells (elements).

Boundary condition

The primary boundary condition (B.C) details introduced in the ANSYS FLUENT code as the inlet velocity by 10 m/s. Blowing ratio is the ratio of the coolant mass flux to the mainstream mass flux. the BR value for the plenum had been set to be as BR = 0.25, 0.5 and 1. In the present paper the jet cross flow through the cooling holes defined as coolant air. Thus, the ratio of the incoming temperature to the injection temperature was prescribed as $T_\infty / T_i = 1.333$ where; T_∞ was the inlet hot gas temperature ($T_\infty = 400K$). All these B.C were extracted from [12]-[13]. The outlet flow in this article was defined as the static pressure.

Results and Discussion

The main objective of this study is to achieve the influence of film holes configuration as cylindrical, square, rotated square, triangle, and rotated triangle with an angle of jet flow 35° over the flat plat surface with constant cross-sectional area. In addition, to assess the dependability of ANSYS FLUENT 16 code to predict the cooling effectiveness. Therefore, the first section of results represents the validation of cylindrical film cooling holes for BR = 0.5 with the active turbulence model against the published experimental data and numerical results. In this section, the centreline and laterally averaged effectiveness over the flat plat surface are presented for BR = 1.

The second section of results is associated with film holes configuration of cylindrical, square, rotated square, triangle, and rotated triangle by preserving the cooling holes area constant for all models. Hence, the influence of the exit holes shape and the BR on effectiveness cooling at the jet crossflow are investigated, as well as the film effectives contours on the flat plat surface at different places downstream of the hole injection. Local effectiveness (η) is calculated as a role of wall temperature (T_∞), (T_W) and (T_i) using Equation (4) [21]:

$$\eta = (T_w - T_\infty)/(T_i - T_\infty) \tag{4}$$

Validation

Figure 3 illustrated the local centerline effectiveness for the single cylindrical cooling hole at BR = 0.5 with different turbulent model: K-ε, RNG, RKE and SST K-ω validated against the experimental data [5]. In addition, results compared with previous numerical simulation [12][22].

In general, the prediction of single centerline effectiveness result at BR = 0.5 is acquired with the RNG, realizable K-ε turbulent model which were identical with [22] result curve especially at X/D<10. On the other hand, the effective cooling associated with SST K-ω turbulent model have over prediction at X/D >5 compared with [5][12] for previous numerical and experimental data respectively. Whereas, the film cooling efficiency result obtained with K-ε model is superior to Ely and Jubran [22] effectiveness result curve at X/D<17 and an excellent agreement with the experimental data of [5] at X/D>10. Consequently, the prediction of single cooling effectiveness in the present study depending on K-ε turbulent model is more recommended due to matching with experimental data [5] and mediate between previous numerical data [12][22] compared with other turbulent model results. So, all the numerical simulation had used K-ε turbulent model to predict the performance of film cooling configuration.

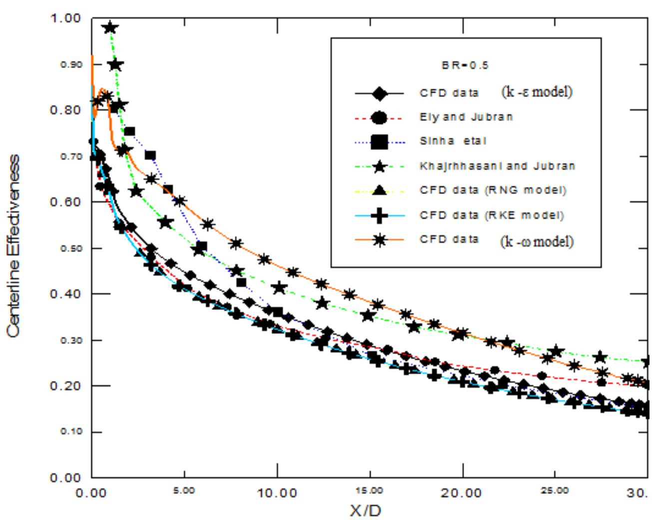

Figure 3: Centerline film cooling effectiveness at different turbulent model compared with previous published paper at BR = 0.5

Figure 4 demonstrated the second step of validation through accomplishing the prediction of lateral averaged effectiveness of single cylindrical hole at BR = 0.25 and 1 compared with previous numerical published results [13].

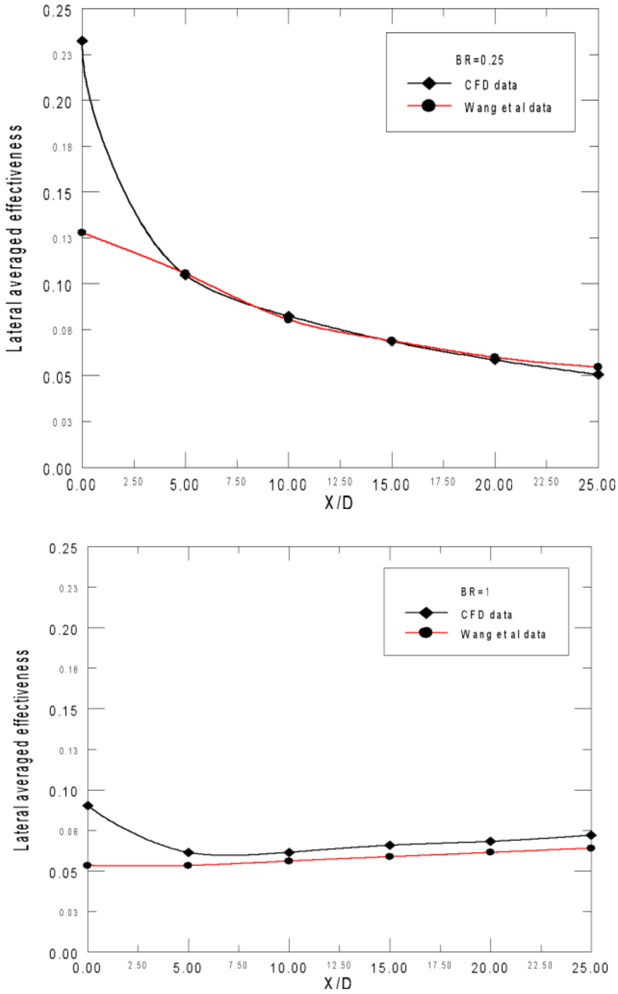

Figure 4: Lateral averaged effectiveness compared with previous published data [13] at BR = 0.25 and 1

A good matching was obtained and the averaged effectiveness results showed excellent agreement mainly at X/D>5 for BR = 0.25 and 1. While, at the region near hole injection, X/D<5 the variation in averaged effectiveness data is due to the complication of mixing procedure (incoming and coolant flow) after the cooling hole. So, the turbulent model plays major role to predict the flow at this area (Realizable K-ε turbulent model used by [13] compared with present results). The influences of BR on the local film cooling efficiency have also been predicted for the cylindrical hole. Hence, three values were selected (BR = 0.25, 0.5 and 1) to evaluate the performance of downstream effectiveness.

Figure 5 shows centerline cooling efficiency of the cylindrical hole in downstream direction. From the first view, it is clear that the downstream effectiveness cooling gradually decreases after the hole. Therefore, the best effectiveness pattern is observed for BR = 0.5 compared with effectiveness curve for BR = 0.25 and 1 particularly at X/D>4. Besides, increasing the value of blowing ratio (BR = 1) in Figure 5 reveals drawback in film cooling effectiveness at X/D<22 compared with effectiveness curve for BR = 0.5 due to the penetration of secondary flow (the coolant flow is a way from the plate surface). Therefore, there is no improvement in the film effectiveness for cylindrical hole model on the plate with increasing the value of BR.

Flat plate surface temperature contour also presented in Figure 5 with BR = 0.25, 0.5 and 1. The numerical average surface temperature reduced about 0.5% with increased BR from 0.25 to 0.5 and increased about ~0.3 with increased BR from 0.25 to 1.

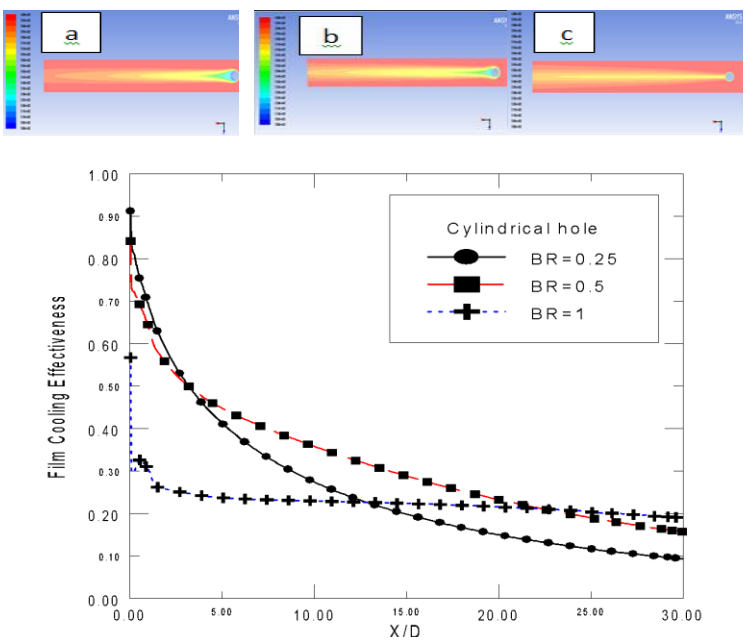

Figure 5: Effect of BR for cylindrical hole on film cooling effectiveness with temperature contours for a) BR = 0.25, b) BR = 0.5 and c) BR = 1

Holes configuration data

Figure 6 shows the lateral effectiveness cooling for five-hole configuration models with BR = 0.25 and 1 at X/D = 0. For all models with the BR = 0.25, the lateral distribution of the film cooling efficiency displays a similar value at Z/D = 0 but then it changes due to holes configuration shape. Subsequently, the coolant air was provided best protection from incoming hot gas in spanwise direction for triangle and square holes shape especially $|Z/D| < 0.7$ through the elevation of effectiveness cooling value in lateral direction compared with the other models. On the other hand, the model of cylindrical and rotated triangle holes model supplied higher effectiveness at $1.1 > |Z/D| > 0.7$ and the values of the film cooling efficiency were close to zero at $|Z/D| > 1.2$ for all models.

The effects of holes configuration on the performance of film cooling with increasing BR = 1 in lateral direction was also illustrated in Figure 6 at X/D = 0 through the film cooling pattern. At Z/D = 0 the film cooling efficiency value diverged due to the source of flow penetration with increased BR. In lateral direction, the triangle and square holes models provided perfect superiority in effectiveness value $|Z/D| < 0.7$ compared with other models. While, the models (rotated triangle, square and rotated square) got a small enhancement between $1 > |Z/D| > 0.7$ compared with triangle hole model. In general the effective cooling of cylindrical hole model declined in lateral direction mainly at$|Z/D| > 0.2$.

Figures 7 and 8 show the lateral effective cooling for five-hole configurations with BR = 0.25 and 1 at X/D = 5 and X/D = 10. Obviously, the effective cooling pattern at BR = 0.25 for all models were identical and the effect of holes configuration doesn't influence the numerical results. While, the influence of increasing BR from 0.25 to 1 had been enhanced for rotated triangle and square holes configuration compared with cylindrical holes model as $|Z/D| < 1.2$ and $|Z/D| < 1.4$ for X/D = 5 and X/D = 10 respectively.

At the same time, the lateral effective cooling for rotated square and triangle holes configuration revealed decrement in numerical results compared with cylindrical hole model data. Certainly, this was due to the film cooling holes configuration design which change the effects of coolant path in downstream and spanwise direction.

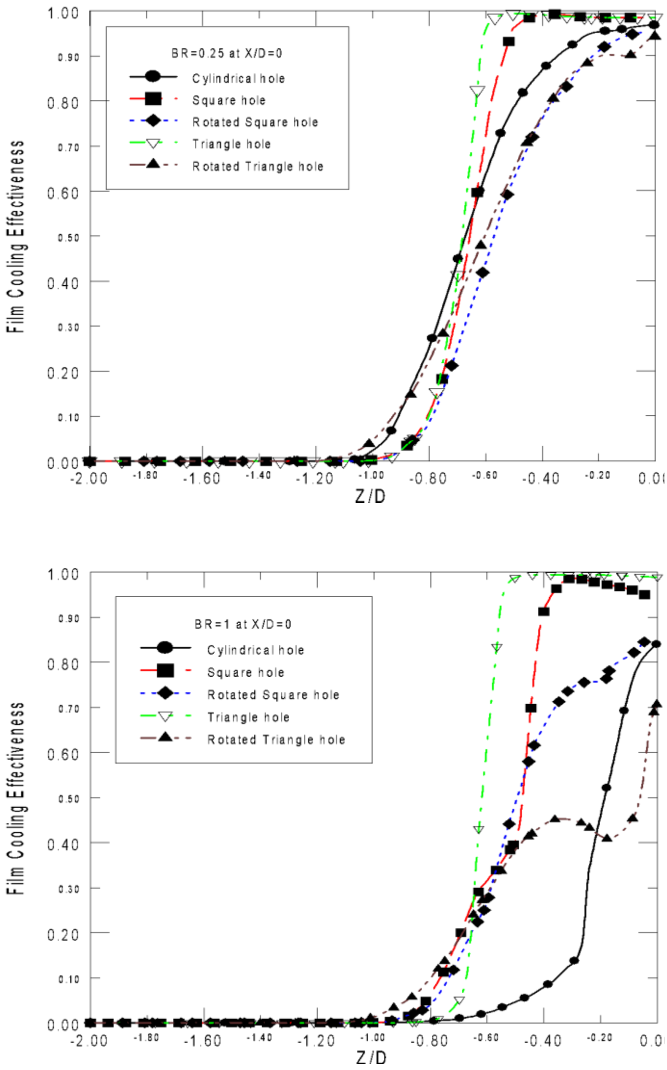

Figure 6: Lateral effective cooling holes configuration at X/D = 0 for BR = 0.25 and 1

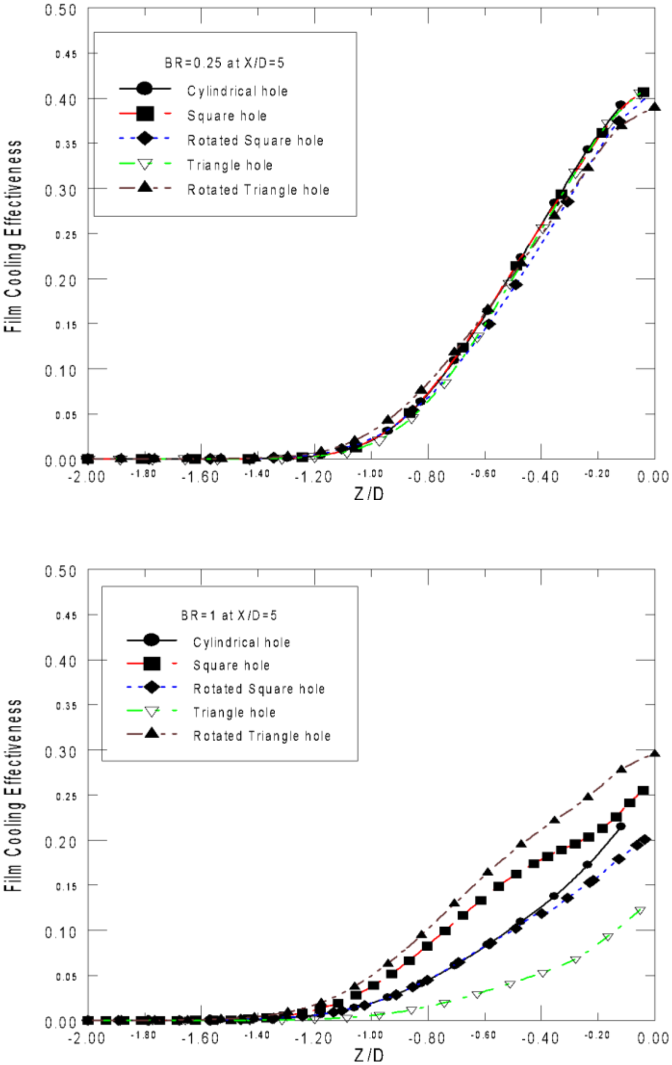

Figure 7: Lateral effective cooling holes configuration at X/D = 5 for BR = 0.25 and 1

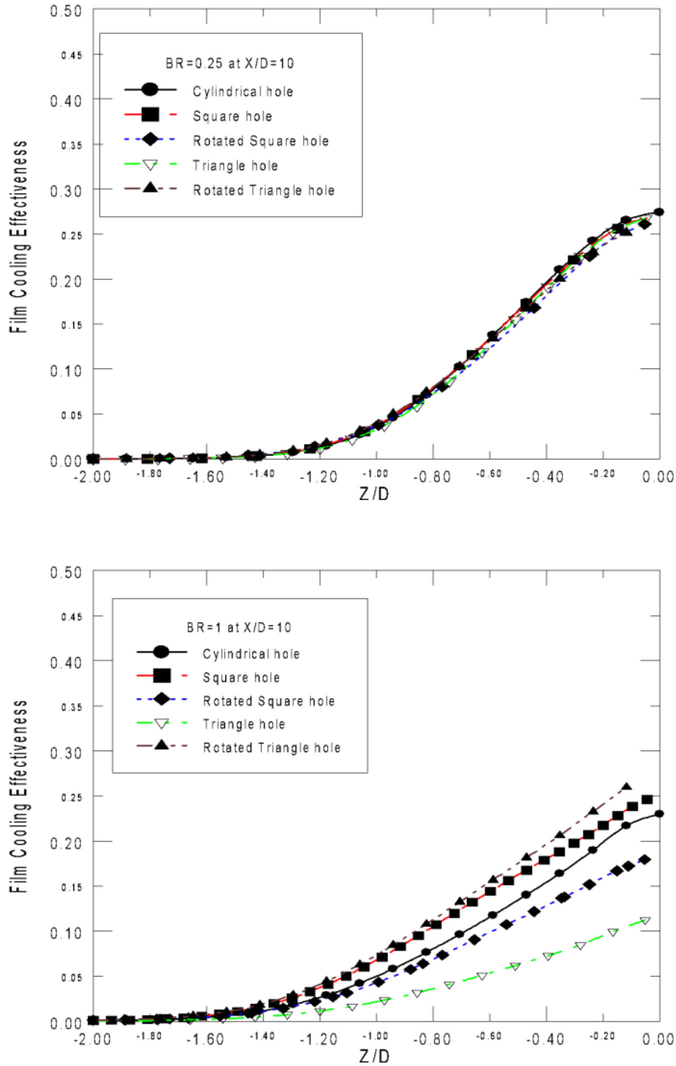

Figure 8: Lateral effective cooling holes configuration at X/D = 10 for BR = 0.25 and 1

Numerical simulation temperature

The CFD numerical results represent the performance of film cooling holes configuration geometry and evaluate the temperature distribution over the flat plate surface by assuming the holes area remain constant. Figure 9 to Figure 13 illustrate the effects of the five study models on temperature distributions of flat plate surface in downstream with six sections located at X/D = 0, 5, 10, 15, 20 and 25. Figure 9 to Figure 13 illustrate the temperature distribution of all models over flat plate at BR = 1 and the effect of coolant air reveals downstream direction as a lower temperature (at mid line of plate) and gradually expanded in lateral direction, but the effects of cooling was insufficient owing to mixing process . Moreover, the temperature distribution at downstream section at X/D = 0, 5, 10, 15, 20 and 25 at BR = 1 indicates that the coolant air was away from the surface due to penetration process. So, objective of this study will appear in understanding the film cooling holes configurations which plays an influential role to enhance the performance cooling hole during reducing the surface temperature with constant cross section area of holes.

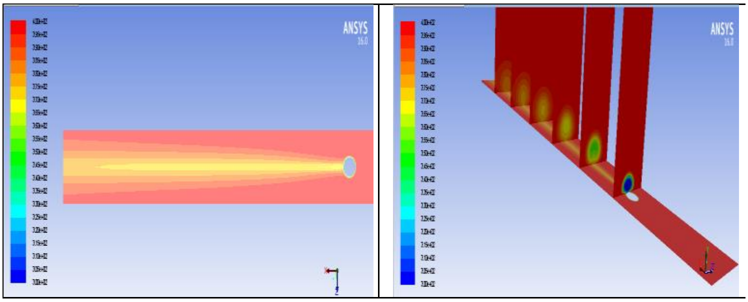

Figure 9: Temperature distribution of cylindrical hole model over flat plate surface and downstream section X/D = 0, 5, 10, 15, 20 and 25 at BR = 1

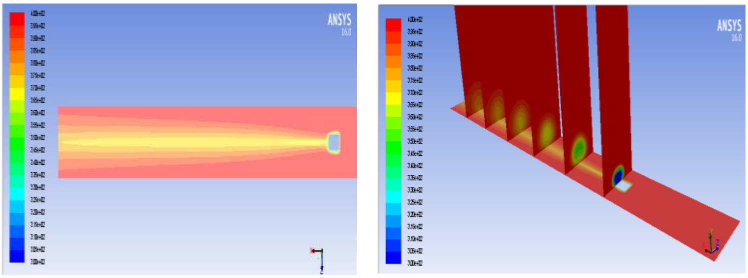

Figure 10: Temperature distribution of square hole model over flat plate surface and downstream section X/D = 0, 5, 10, 15, 20 and 25 at BR = 1

Accordingly, the temperature contours were formed on the surface due to film cooling holes configuration and the best protection provided through good prevalence of coolant air in lateral direction in Figure 10 to Figure 13 which represent square and rotated triangle models respectively. The average surface temperature dropped about 1.6% and 1.2% compared with cylindrical hole model shown in Figure 9. Hence, the effect of the coolant air penetration due to high BR = 1 was reduced.

Rotated square hole and triangle hole model shown in Figures 11 and 12 provided negative protection through increasing the average temperature about 0.15% and 0.6% respectively compared with cylindrical hole model. The coolant air becomes away from flat plat surface especially in downstream direction which is unexpected prediction due to the high penetration occurred.

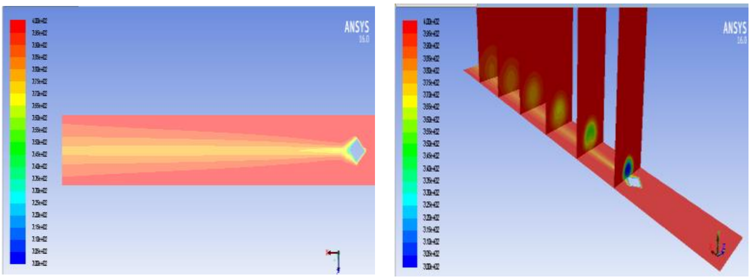

Figure 11: Temperature distribution of rotated square hole model over flat plate surface and downstream section X/D = 0, 5, 10, 15, 20 and 25 at BR = 1

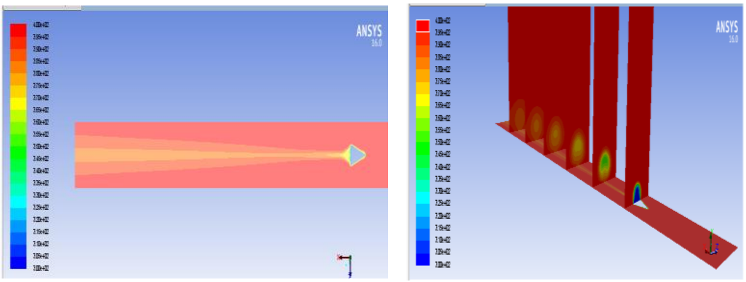

Figure 12: Temperature distribution of triangle hole model over flat plate surface and downstream section X/D = 0, 5, 10, 15, 20 and 25 at BR = 1

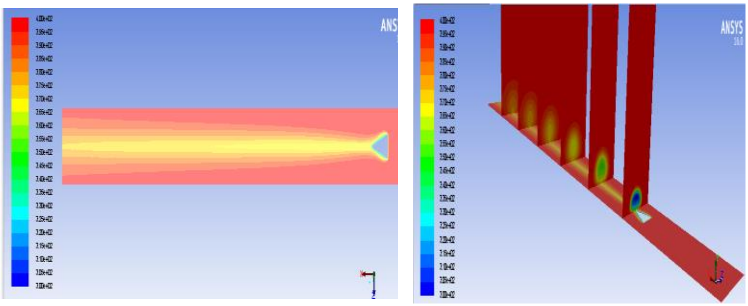

Figure 13: Temperature distribution of rotated triangle hole model over flat plate surface and downstream section X/D = 0, 5, 10, 15, 20 and 25 at BR = 1

The influence of film cooling holes configuration was elucidated in Figure 14 as centerline temperature for the five models (cylindrical hole, square hole, rotated square hole, triangle hole and rotated triangle hole). The lower centerline temperature usually has higher effective cooling. Consequently, the rotated triangle hole and square hole models provided lowest temperature compared with cylindrical hole model and the average centerline temperature reduction about ~ 2% and 1.5% respectively.

Undesirable performance was observed for rotated square and triangle hole models compared with cylindrical hole model through increasing the temperature as shown in Figure 14 due to effects of coolant penetration.

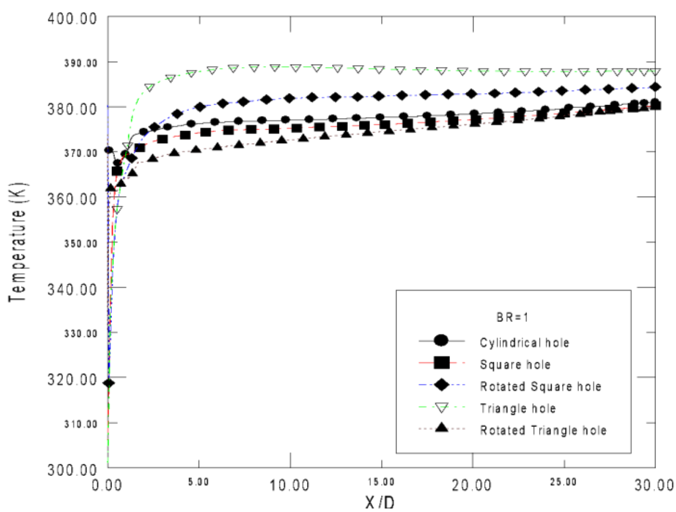

Figure 14: Temperature distribution of triangle hole model over flat plate surface and downstream section X/D = 0, 5, 10, 15, 20 and 25 at BR = 1

Conclusion

The film cooling performance for single film hole configurations with the effect of BR has been investigated numerically using ANSYS FLUENT 16. The main findings of this study are as follows:

1. Film cooling technique is an effective way to protect any surface from the hot mainstream flow due to cooling technique. Therefore, no improvement were found in film effective cooling for cylindrical hole on the flat plate surface with increasing BR from 0.25, 0.5 to 1 as the jet across flow penetrates the incoming hot gas which interrupt the film cooling layer (the coolant air is ejecting away from the surface). The best protection is obtained at BR = 0.5.
2. ANSYS FLUENT software reveals high accuracy and excellent agreement with previous numerical studies of flat surface film turbulent cooling.
3. This study has examined the effects the single film cooling holes configuration (holes cross section area remains constant). The performance of the holes configuration on the lateral effective cooling with BR = 0.25 for all models are identical. Whereas, with increasing BR = 1 the lateral effectiveness are improved and the average centreline effectiveness for the rotated triangle hole, square hole models are enhanced about $\sim 2\%$ and 1.5% respectively compared with cylindrical hole model.
4. The prediction of effective cooling is greatly dependant on film cooling holes configuration and the highest effective cooling corresponds to the lowest surface temperature. Thus, the average surface temperature is dropped about 1.6 % and 1.2 % for the Rotated triangle and square hole models respectively compared with cylindrical hole model. Hence, the effect of the coolant air penetration due to high BR = 1 was reduced.

This study recommends the use of rotated triangle and square hole models with high blowing ratio to provide more surface protection from incoming hot gas in lateral and downstream direction in addition to the solidity of ANSYS FLUENT code to predict the cooling efficiency via simulation of the effects of changing BR compared with previous published paper.

Acknowledgement

The first author would like to thank Middle Technical University, Institute of Technology Baghdad. Iraq for sporting this reserch.

References

[1] R. J. Goldstein, E. R. G. Eckert and J. W. Ramsey, "Film Cooling With Injection through Holes: Adiabatic Wall Temperatures Downstream of a Circular Hole", *ASME. Journal of Engineering for Gas Turbines and Power*, vol. 90, no. 4, pp. 384-393, 1968.

[2] V. L. Eriksen and R. J. Goldstein, "Heat Transfer and Film Cooling Following Injection Through Inclined Circular Tubes", *ASME Journal of Heat Transfer*, vol. 96, pp. 239-245, 1974.

[3] G. Bergeles, A. D. Gosman and B. E. Launder, "Near Field Character of Jet Discharged Through a Wall at 30^0 to a Mainstream", *AIAA J. Air Transp*., vol. 15, no.4, pp. 499-504, 1977.

[4] A. J. H. Teekaram, C. J. P. Forth and T. V. Jones, "The Use of Foreign Gas to Simulate the Effects of Density Ratios in Film Cooling", *ASME Journal of Turbomach*inery, vol. 111, no. 1, pp. 57-62, 1989.

[5] A. K. Sinha, D.G. Bogard and M.E. Crawford, "Film cooling effectiveness downstream of a single row of holes with variable density ratio", *ASME Journal of Turbomachinery*, vol. 113, pp. 442-449, 1991.

[6] C. M. Bell, H. Hamakawa and P. M. Ligrani, "Film Cooling From Shaped Holes", *ASME Journal of Heat Transfer*, vol. 122, no. 2, pp. 224-232, 2000.

[7] R.A. Brittingham and J. H. Leylek," A Detailed Analysis of Film Cooling Physics: Part IV— Compound-Angle Injection with Shaped Holes", *ASME Journal of Turbomach*inery, vol. 122, no. 1, pp. 133-145, 2002.

[8] C. Saumweber, A. Schulz and S. Wittig, "Free-Stream Turbulence Effects on Film Cooling With Shaped Holes", *ASME Journal of Turbomachinery*, vol. 125, no. 1, pp. 65-73, 2003.

[9] W. Peng, and P.X. Jiang, "Eperimantal and Numerical Study of Film Cooling with Internal Coolant Cross Flow Effects", *Experimental Heat Transfer*, vol. 25, no.4, pp. 282-300, 2013.

[10] N. Jose, , J. S. Jayakumar, and G. B. Yepuri, "Numerical investigation of adiabatic film cooling effectiveness over a flat plate model with cylindrical holes" *Procedia Engineering*, vol. 127, pp. 398-404,2015 .

[11] M. R. Salimi, M. Ramezanizadeh, M. Taeibi-Rahni and R. Farhadi-Azar, "Film Cooling Effectiveness Enhancement Applying another Jet in the Upstream Neighbor of the Main Jet-Using LES Approach", *ASME Journal of Applied Fluid Mechanic,* vol. 9, no. 1, pp. 33-42, 2016.

[12] S. Khajehhasani and B. A. Jubran, "Film Cooling from Sister-Shaped Single-Holes. Proc", *ASME Turbo Expo* 2014, Du¨ sseldorf, Germany, GT2014–25971, *Journal American Society of Mechanical Eng*ineers, vol. 45721, pp. 414-435, 2014.

[13] J. Wang, K. Tian, J. Luo and B. Sundén, "Effect of hole configurations on film cooling Performance", *Numerical Heat Transfer*, vol. 75, no. 11, pp. 725–738, 2019.

[14] W.A. Ajlan, H. A Daud and N.J. Yasin, "Evaluate the Effectiveness of Cylindrical Cooling holes Embedded in arc Trench Over Flat Plate Surface using Infra-red Technology", *IOP Conference Series: Materials Science and Engineering*, vol. 881, no.1, pp. 012153, 2020.

[15] A Zerrout, A. Khelil, L. Loukarfi " Heat Transfer of Swirling Multi Jets Impinging System" *Journal of Mechanical Engineering*, vol. 17, no. 2, pp. 93-108, 2020.

[16] W. Zhang and H. R. Zhu,"Film Cooling Performance of the Staggered Arrangement of Auxiliary Holesand Main Holeson a Flat Plate", *ASME Journal of Applied Fluid Mechanics*, vol. 14, no. 3, pp. 741-752, 2021.

[17] Y. T. Jiang, H. F. Deng, X. L. You, H. J. Zhao and G. Q. Yue, "Numerical Investigation on Film Cooling Mechanism with Different Coolant Delivery Configurations", *ASME Journal of Applied Fluid Mechanics*, vol. 14, no. 1, pp. 175-185, 2021.

[18] H. A. Daud, Q. Li, O. A. Bég and SAA. AbdulGhani, "Numerical Investigations of Wall-Bounded Turbulence", *Journal of Mechanical Engineering Science, Part C:*, vol. 225, no. 5, pp. 1163-1174, 2011.

[19] F. Mohamad and T. Kajishima " Large-Eddy Simulation of Unsteady Pitching Aerofoil using a One-equation Subgrid Scale (SGS) Model based on Dynamic Procedure" *Journal of Mechanical Engineering*, vol. 18, no.1, 157-173, 2021

[20] S.N. Ashwindran, A.A. Azizuddin, A.N. Oumer , Azli Abd Razak "An Introductory CFD Analysis Study of Novel Cavity Vane Driven Wind Turbine Blade Design" *Journal of Mechanical Engineering*, vol. 17, no. 3, pp. 55-68, 2020.

[21] H. A. Daud, Q. Li, O. A. Bég and SAA. Abdul Ghani, "Numerical simulation of blowing ratio effects on film cooling on a gas turbine blade" *Computational Methods and Experimental Measurements XV*, vol. 51, pp. 279-292.2011 .

[22] M. J. Ely, and B. A. Jubran, "Film Cooling From Short Holes with Sister Hole Influence", Proc. *ASME Turbo Expo* Copenhagen, Denmark, GT2012–68081. *American Society of Mechanical Engineers*, vol. 44700, pp. 1185-1196 2012.