Effect of Lid Height and Blowing Ratio on Film Cooling Effectiveness of a Novel Lidded Hole configuration

[1] Space Mechanical Research Department, Centre of Satellite Development (CDS), Algerian Space Agency (ASAL), BP 4065 Ibn Rochd USTO Oran, Algeria

[2] Laboratory of Applied Mechanics, Faculty of Mechanical Engineering, University of Sciences and Technology Mohamed Boudiaf, Oran, Algeria

[3] Laboratory of Materials and Reactive Systems, Department of Mechanical Engineering, University Djilali Liabes, Sidi Bel Abbes, Algeria

[4] Department of Engineering Design and Mathematics, University of the West of England, Bristol, BS16 1QY, UK.

Corresponding Author Email: guelailia@yahoo.fr

Abstract

Film cooling is one of the promising technologies used for protecting rocket nozzles and turbine blades from combustion chamber hot gases. This paper proposes a novel shape of film cooling injection hole, called lidded hole, that can offer significant enhancement of cooling performance. ANSYS CFX is used to perform 3D numerical simulations of a flat plate with a single row of lidded holes, in which the k–ε model approximates turbulence effects. Four cases are investigated to highlight the influence of the hole's lid height (H/d = 0, 0.25, 0.5, 0.75). The effect of blowing ratios (M=0.5, 1, 1.5) is also analyzed for each configuration. The numerical results of this study are compared with available experimental data, and, generally, a good agreement is achieved. The results obtained show that the lidded hole configuration reduces the coolant flow separation which improves significantly the film cooling effectiveness. In addition, increasing the blowing ratio leads to an increase in lateral and centerline cooling effectiveness. Comparing all studied cases, the optimum coolant coverage was obtained for the lidded hole configuration with H/d=0.25 at M=1.5.

Keywords: Computational fluid dynamics, Film cooling effectiveness, Blowing ratio, Heat and mass transfer, Lidded hole.

1. Introduction

Improving the performance of thermal engines such as turbojets, rocket engines and gas turbines is one of the most challenging tasks for researchers and engineers. The performance of such engines can be enhanced by raising the temperature in combustion chambers. However, the hot gases released from combustion can exceed the maximum temperature of a component's materials. Efficient cooling technology is thus required to ensure an adequate thermal environment for all components exposed to high temperature gases. The method of film cooling has been widely used to protect material surfaces from high thermal loads. In this technique, a cold jet is injected through holes on the surface to be cooled. The interaction between the mainstream flow and the secondary cooling flow creates a thin thermal insulation layer on the cooled wall to protect it from being overheated by the hot flow.

For several years, great effort has been devoted to enhancing the film cooling effectiveness which is influenced by several geometrical and physical factors. Previous studies indicate that turbulence intensity, blowing ratio and density ratio are among the physical flow parameters that mostly affect film cooling performance. Ammari et al.[1] conducted an experimental work to investigate the effect of density ratio on heat transfer coefficient from a film cooled flat plate. The results indicated that the heat transfer coefficient was highly affected by the variation of density ratio, especially for the injection angle 35°. In the paper of Sinha et al.[2], the effect of blowing ratio and density ratio on film cooling effectiveness was experimentally investigated. Laterally averaged effectiveness was found to be dependent on density ratio and blowing ratio. Drost et al.[3] studied the film cooling and heat transfer on a gas turbine airfoil. Tests were conducted at several exit Reynolds numbers and two mainstream turbulence intensities. Based on their results, variations in Mach and Reynolds number changed the film cooling on the section (you mean "suction"?) side. Heat transfer ratios on the pressure side were observed to be lower at high turbulence intensity.

Also, film cooling effectiveness can be significantly enhanced by optimizing the shape of the film cooling holes, their distribution, and location. Film cooling on a turbine blade leading edge was experimentally studied by Liu et al.[4], where two injection hole configurations were investigated: cylindrical holes and converged slot holes. Guelailia et al.[5] numerically investigated the effects of mass flow rate on film cooling performance and heat transfer over a gas turbine rotor blade. Their results showed that converging slot holes provided better film cooling protection than simple cylindrical holes. Wang et al.[6] investigated the cooling effectiveness of several film cooling holes configurations including conical hole, fan-shaped hole and combined hole. Their results indicated that the combined hole provided a more uniform distribution of the cooling film than the other hole configurations. In a recent paper by Cao et al.[7], the effect of hole geometry and blowing ratio on film cooling effectiveness was numerically and experimentally studied. Tian et al.[8] investigated the influence of injection hole blockage configurations on film cooling characteristics. In the research works of Goldstein et al.[9], Schmidt and Bogard[10,11], the effects of surface roughness on adiabatic effectiveness were investigated. Guelailia et al.[12] conducted a numerical study to analyze the effect of deposit height and position on film cooling. Yuen and Martinez[13,14] presented in their studies detailed experimental data of heat transfer coefficient and film cooling effectiveness for single and multiple rows of holes on a flat plate. The heat transfer performance of showerhead film cooling as applied to the vane leading edge, and considering representative lean burn combustor swirling outflow was numerically investigated by Zhuang et al.[15]. Kouchih et al.[16] numerically investigated several film cooling configurations including Barshan-dune-shaped shells, forward injection hole and backward injection hole. The influence of wall radial curvature on film cooling efficiency was numerically analyzed by Guelailia et al.[17]. A numerical study of swirling flow in coolant jets was conducted by ZHU et al.[18] whose results demonstrated that film cooling effectiveness could be improved with jet swirl at high blowing ratios, and that swirl strength has significant influence on film cooling performance. The effects of mainstream turbulence intensity and blowing ratio on the endwall film cooling and heat transfer were numerically investigated by Yang et al.[19]. Results show that by increasing turbulence intensity, the variation of film cooling effectiveness was very limited.

Placing an obstacle downstream the injection hole is one of the new ways to enhance film cooling flow distribution on the cooled surface. For example, crescent-shaped block configuration promises significant improvement in film cooling as reported in the research work of Khorsi et al.[20]. Recently, Zhang et al.[21] investigated the effect of crescent-shaped block streamwise position on flat plate film cooling characteristics. Hefang et al.[22] numerically studied the effects of four types of winglet vortex generators upstream of the film hole on film cooling performance. In the work of Wilfert et al.[23], experimental tests were carried out in a high-speed cascade wind tunnel in order to understand the mixing processes between the cooling jets and mainstream flow.

Following literature, increasing the blowing ratio leads to the separation of coolant jet from the wall being cooled, which seems to impact film cooling effectiveness negatively. The results obtained by Thole et al.[24] indicated that the surface is not protected by the film cooling layer at high blowing ratios and momentum flux ratios due to the detachment of coolant jet. In the research work of Coulthard et al.[25], it was shown that the coolant's pulsation can improve film cooling effectiveness in cases where obvious jet-detachment occurs. However, in cases where the coolant jet fully attaches to the surface, the pulsation decreases the film cooling effectiveness. In addition, due to its highly-unsteady feature, the pulsed film cooling is affected by many operational pulsation parameters, making it complex for practical implementation.

To our knowledge, the previous film cooling studies have not considered the separation problem in sufficient detail. Saumweber et al.[26] studied the effects of free-stream turbulence on film cooling with shaped holes. They concluded that at low turbulence levels, the shaped holes do not show any detachment from the surface even at high blowing ratios. The momentum of the coolant decreases because of the expanded exit and hence the penetration of secondary fluid into the mainstream decreases. Keeping the coolant film attached to the wall at higher velocity will considerably enhance film cooling performance. With this goal, this study investigates the effect of a novel configuration of film cooling injection hole on the film cooling separation problem as well as the cooling effectiveness. More specifically, lidded injection hole is proposed to reduce the separation of the coolant fluid in order to enhance film cooling performance. Different blowing ratios will be investigated. The results of the present study are compared with the experimental results of Walters et al.[27] and Schmidth et al.[28] for validation purposes. In this paper, the following research questions are addressed:

a. Does the novel proposed lidded hole configuration reduce the separation of the coolant fluid from the surface to be cooled?
b. How can the lidded injection hole enhance film cooling performance and what is the influence of holes lid height (H/d) on the flow structure and coolant distribution?
c. How does increasing blowing ratio (M) affect the centerline and lateral film cooling effectiveness of lidded injection holes?

To address such questions, the problem is investigated numerically via 3D simulations using the commercial software code ANSYS CFX.

2. Research methodology

The software used in this three-dimensional investigation is ANSYS CFX. The ANSYS CFX solver uses an element-based finite volume method, which involves discretizing the spatial domain using a mesh (more details of finite volume method can be found in the work of Versteeg et al.[29]). The governing equations are the continuity, the momentum and energy equations as shown in equations (1), (2) and (3), respectively.

$$\frac{\partial \rho}{\partial t} + \nabla \bullet \left(\rho U \right) = 0 \quad (1)$$

$$\frac{\partial(\rho U)}{\partial t}+\nabla\bullet(\rho U\otimes U)=\nabla\bullet\left(-p\delta+\tau-\rho\overline{\mathrm{u}\otimes\mathrm{u}}\right)+\mathrm{S_M} \quad (2)$$

$$\frac{\partial(\rho h_{tot})}{\partial t}-\frac{\partial p}{\partial t}+\nabla\bullet(\rho U h_{tot})=\nabla\bullet(\lambda\nabla T)+\nabla\bullet(U\tau)+U\bullet S_M+S_E \quad (3)$$

where $h_{tot}=h+\frac{1}{2}U^2$ (4)

In this study, the k–ε model was chosen for turbulence modeling. This model is implemented in most general purpose CFD codes and is considered the industry standard model. It has been proven to be stable and numerically robust. For general-purpose simulations, this model offers a good compromise in terms of accuracy and robustness. The k–ε model assumes that the turbulence viscosity μ_t is linked to the turbulence kinetic energy k and dissipation rate ε via the relation:

$$\mu_t=\rho\,\mathbf{C}_\mu\frac{k^2}{\varepsilon} \quad (5)$$

C_μ is a constant (C_μ = 0.09)

The standard model uses the following transport equations:

$$\frac{\partial(\rho K)}{\partial t}+\nabla\bullet(\rho U K)=\nabla\bullet\left[(\mu+\frac{\mu_t}{\sigma_k})\nabla K\right]+P_k-\rho\varepsilon \quad (6)$$

P_k is the turbulence production due to viscous forces, which is modeled using:

$$P_K=\mu_t\nabla U\bullet(\nabla U+\nabla U^T)-\frac{2}{3}\nabla\bullet U(3\mu_t\nabla\bullet U+\rho K) \quad (7)$$

In this study, the flow is considered stationary incompressible which eliminates all time derivatives in the previous equations. The advection and turbulence terms are both solved with high-resolution schemes. The calculation is continued until the convergence criterion (10^{-5}) is satisfied.

2.1. Geometry and mesh generation

The baseline configuration considered in this study takes inspiration from the experiments by Walters et al[27]. Presented in Figure 1 is the computational domain geometry. The computational domain includes a mainstream channel, a plenum and the cooling injection hole. The dimensions of the mainstream channel are $75d \times 6d \times 5d$ in the x, y, z, coordinates, respectively, while the plenum dimensions are $30d \times 4d \times 5d$. The x, y, z coordinates are measured from the centre of the injection hole and are used in a dimensionless forms as x/d, y/d and z/d. The mainstream inlet plane is positioned at x/d=30 upstream from the centre of the hole, while the exit plane is at x/d=45 downstream. The cooling injection hole is 3d long with a nominal hole diameter of d=1mm. Four configurations of the injection hole were tested by changing the height of the lid placed on the hole. Figure 2 shows the film cooling hole geometries, of different heights, that were considered. The commercial mesh generator ICEM CFD was used to generate the computational grid. The computational domain is discretized using a structured multi-block grid. The mesh was significantly refined near the cooled wall and in the vicinity of the injection hole. Figure 3 shows the multi-block grid used for the present numerical simulation. The value of y^+ was set to be less than 1. The final grid adopted for calculations was obtained after a series of tests. A grid independence study was conducted as shown in Figure 4 using film cooling effectiveness (see Eq. 9) as the monitored variable. Three computational grids with hexahedral elements were generated and were composed of approximately 6×10^5 elements (coarse), 9×10^5 elements (moderate) and 13×10^5 elements (fine). It can be seen that the difference between the moderate and the coarse mesh is very small. The moderate mesh was chosen for all subsequent computations in the present paper.

2.2. Boundary conditions

The boundary conditions for all simulations conducted in the present study are similar to those in the experimental work of Walters et al.[27]. The plate and hole walls were defined as adiabatic walls with no-slip conditions. Symmetry conditions were used for the two lateral planes. The mainstream velocity is $U_\infty = 11$ m/s, mainstream temperature is $T_\infty = 330$ K and the jet temperature is $T_c = 300$ K. Three blowing ratios were considered for the simulations $M = 0.5$, 1, 1.5. The blowing ratio is defined as:

$$M = \frac{\rho_c U_c}{\rho_\infty U_\infty} \tag{8}$$

where, ρ_c and ρ_∞ are the densities of coolant jet and mainstream flow, respectively. U_c and U_∞ are the velocities of coolant jet and mainstream flow respectively.

3. Results and discussion

In the present study, the results of adiabatic and laterally averaged film cooling effectiveness, η and $\bar{\eta}$ which are defined respectively by equations (9) and (10) were calculated for various blowing ratios depending on the operating conditions.

$$\eta = \frac{T_\infty - T}{T_\infty - T_c} \tag{9}$$

$$\bar{\eta} = \frac{1}{L}\int_L \eta dz \tag{10}$$

where L represents the spanwise dimension.

In addition to adiabatic and laterally averaged film cooling effectiveness results, the coefficient of pressure loss C_{PL} which is defined by equation (11) is calculated.

$$C_{PL} = \frac{P_{0,t} - P_{\infty,out}}{0.5 * \rho_{\infty,out} * U_{\infty,out}^2} \tag{11}$$

where $P_{0,t}$ is the total pressure, and $P_{\infty,out}$, $\rho_{\infty,out}$, and $U_{\infty,out}$ represent the total pressure, the density, and velocity at mainstream flow outlet, respectively.

$P_{0,t}$ is developed by Wilfert et al.[23] and is defined by equation (12) as follows:

$$P_{0,t} = \frac{\dot{m}_{c,in}}{\dot{m}_{\infty,in} + \dot{m}_{c,in}} P_c + \frac{\dot{m}_{\infty,in}}{\dot{m}_{\infty,in} + \dot{m}_{c,in}} P_\infty \tag{12}$$

where P_∞ and $\dot{m}_\infty$ represent the total pressure and mass flow rate at mainstream inlet and P_c and $\dot{m}_c$ are the total pressure of flow and mass flow rate at coolant inlet respectively.

In order to validate the calculation method, the distribution of centerline adiabatic film effectiveness obtained from the simulations is compared to the experimental data from the works of Walters et al.[27] and Schmidt et al.[28] as shown in Figure 5. We can clearly see that the simulation results are in good agreement with the experimental measurements, thus proving the good reliability of numerical prediction of film cooling by the ANSYS CFX software.

The analysis of the flow physics as a result of film cooling reveals one of the flaws of injection through a classic circular hole shape. The deficiency lies in the penetration of the secondary flow of the cooling jet into the hot main flow, favoring the mixing of the two fluids and thus reducing the cooling efficiency. This phenomenon becomes more important with the increase in the blowing ratio.

The solution proposed in this article is the injection of the heat transfer fluid through a novel shape of the injection hole called the lidded hole. The new configuration ensures a reduction in the penetration of the cold jet into the main flow, by allowing it to flow in a direction tangential to the wall and parallel to the main flow. In addition, the injection of the cooling jet through the lidded hole into the main hot flow will increase the pressure and reduce the temperature which will in turn enhance the film cooling performance.

Figure 6 illustrates the centreline adiabatic film cooling effectiveness obtained using various lid heights at a blowing ratio of $M = 0.5$. For lid heights H/d=0.25 and H/d=0.75, no significant change in the centreline film cooling effectiveness is observed compared to the simple injection hole configuration (H/d=0). However, for the medium lid height H/d=0.5, the centreline film cooling effectiveness is considerably improved compared to other cases.

For all the configurations studied, the maximum values of the cooling effectiveness are observed near the injection zone. The effectiveness of the film distribution on the centerline decreases away from the injection hole. Figure 6 shows the advantage of this new form of injection hole for a lid height of H/d=0.5 with a blowing ratio of M=0.5. The improvement in cooling effectiveness is the consequence of the orientation of the jet coming from the modified shape of the hole which promotes the formation of a cold layer near the wall. The choice of the optimal height of the lid depends essentially on the blowing ratio.

In order to analyse the influence of blowing ratio on film cooling effectiveness and based on the findings of Figure 6, Figure 7 shows the centreline adiabatic film cooling effectiveness at various blowing ratios (*M*=0.5, 1, 1.5) but for a constant lid height H/d=0.5. It is clear that the blowing ratio can significantly affect the centerline adiabatic film cooling effectiveness. More specifically, increasing the blowing ratio results in an increase of centerline film cooling effectiveness.

Lateral adiabatic film cooling effectiveness at various lid heights at constant blowing ratio *M*=1.5 is illustrated in Figure 8. The curves are plotted for a longitudinal position x/d=5 from the injection hole. It is clear that for lid heights H/d=0, H/d=0. 5 and H/d=0.75, the maximum values of lateral adiabatic film cooling effectiveness are on the central zone (z/d=0) with a decrease along the lateral direction, contrary to the case with lid height H/d=0.25. The optimum value was obtained at H/d=0.5, while the better lateral distribution of film cooling effectiveness is achieved at H/d=0.25.

Figure 9 shows the effects of blowing ratio on lateral adiabatic film cooling effectiveness for the lid height H/d=0.25 at x/d = 3. The figure indicates that lateral film cooling effectiveness can be highly improved by increasing the blowing ratio. Better spreading of the coolant on the cooled wall was obtained for the higher blowing ratio M=1.5. It is clear that for M=1 and M=1.5, the maximum values of lateral adiabatic film cooling effectiveness are at z/d=-1 and z/d=1 with a decrease along the lateral direction. This can be attributed to the appearance of contra-rotating vortex pairs (CRVP) on the y-z plane at x/d=3.

Based on the present simulations, Figure 10 shows the contours of film cooling effectiveness distributions for different lid heights at M=1 and M=1.5. Compared with the simple injection hole (H/d=0), an enhancement of film cooling performance is observed for the lower and medium hole lid heights (H/d=0.25, H/d=0. 5) due to the good lateral distribution of the coolant. However, the larger hole lid height (H/d=0.75) provokes a poorer coverage of the plate by the cooling fluid, thereby reducing the film cooling effectiveness. Better coverage of the coolant on the wall is obtained for H/d=0.25 at a high blowing ratio of M=1.5.

Temperature distribution contours on the central plane at various lidded hole cases for the blowing ratio M=1.5 are illustrated in the supplementary figure 11. For the simple coolant injection hole configuration (H/d=0), we can clearly observe a strong penetration of the cold fluid into the hot main flow. Such penetration promotes mixing of the two fluids and leads to a deterioration in the film cooling effectiveness. This issue can be resolved by replacing the simple injection hole with a lidded injection hole. As shown in the contours of Figure 11, the lidded injection hole configurations can keep the coolant attached to the surface, thus improving the film cooling effectiveness.

In order to better illustrate the influence of different studied configurations on the appearance of contra-rotating vortex pairs (CRVP), the supplementary figure 12 shows the velocity field at x/d=3 and M=1.5. For the simple injection hole case without lid (a), we can clearly see the appearance of a contra-rotating vortex pair. This flow physics can be explained by the separation of the coolant jet from the wall being cooled. This CRVP affects the film cooling performance negatively. The figure indicates also that the size of CRVP is reduced according to the injection hole lid (b). The intensity and shape of the CRVP decreases with the increase in lid heights (H/d=0.5, H/d=0.75).

The supplementary figure 13 illustrates the distribution of coefficient of pressure loss C_{PL} for all studied cases at different blowing ratios. From figure 13, it can be seen that the pressure loss coefficient is at the highest values for the simple case H/d=0 (i.e. case without lidded hole) for all blowing ratios. In addition, replacing the simple injection hole with a lidded injection hole can significantly reduce the coefficient of pressure loss especially for the lid height of H/d=0.25. For larger lid heights (H/d=0.5 and H/d=0.75), increasing the blowing ratio decreases slightly the coefficient of pressure loss. However, for the cases of H/d=0 and H/d=0.25, the increase of blowing ratio leads to an increase of C_{PL}.

4. Conclusion

In this paper, a novel shape of film cooling injection hole (lidded hole) that can be used for improving film cooling performance has been proposed and investigated via three-dimensional numerical simulations in the commercial software ANSYS CFX. Four configurations are considered by changing lid height (H/d=0, 0.25, 0.5, 0.75). Film cooling effectiveness is presented for different blowing ratios (M=0.5, 1 and 1.5). An acceptable agreement with previous experimental measurements had been obtained. The main conclusions are as follows:

1) Lidded hole configuration reduces the separation of the coolant fluid from the surface to be cooled and keeps the cold fluid attached to the wall. Significant improvement in film cooling effectiveness compared to the simple case (injection hole without lid) is obtained.
2) The maximum enhancement in overall film cooling effectiveness using the lidded hole case H/d=0.25 is about 210% achieved at a blowing ratio of M = 1.5. This percent is calculated as the difference between the overall film cooling effectiveness of case 2 (H/d=0.25) and case 1 (hole without lid, H/d=0) divided by that of case 1.
3) Better lateral distribution of the coolant was observed for lower and medium heights of the hole lid (H/d=0.25, H/d=0. 5) compared to the simple case (H/d=0). However, a longer hole lid height (H/d=0.75) leads to poorer coverage on the plate by the cooling fluid, thus reducing film cooling effectiveness.

4) For all cases, optimum film cooling effectiveness and coolant coverage was obtained for the lidded injection hole case H/d=0.25 at high blowing ratio M=1.5.
5) The blowing ratio can significantly influence the film cooling performance. Increasing the blowing ratio results in an increase of lateral and centreline film cooling effectiveness.

References

1. Ammari HD and Lampard D. The effect of density ratio on the heat transfer coefficient from a film cooled flat plate. *ASME Journal of Turbomachinery* 1990; 112: 444-450.
2. Sinha AK, Bogard DG and Crawford ME. Film-Cooling Effectiveness Downstream of a Single Row of Holes With Variable Density Ratio. *ASME Journal of Turbomachinery* 1991; 113:442-449.
3. Drost U and Bölcs A. Investigation of detailed film cooling effectiveness and heat transfer distributions on a gas turbine airfoil. *ASME Journal of Turbomachinery* 1999; 121: 233-242.
4. Liu CL, Zhu HR, Zhang ZW, et al. Experimental Investigation on the Leading Edge Film Cooling of Cylindrical and Laid-Back Holes with Different Hole Pitches. *Int. J. Heat Mass Transfer* 2012; 55: 6832–6845.
5. Guelailia A, Khorsi A and Hamidou MK. Computation of leading edge film cooling from a console geometry (converging slot hole). *Thermophysics and Aeromechanics* 2016; 23: 33-42.
6. WANG J, Tian K, Luo J, et al. Effect of hole configurations on film cooling performance. *Numerical Heat Transfer, Part A: Applications* 2019; 75: 725-738.
7. Cao N, Li X, Wu Z, et al. Effect of film hole geometry and blowing ratio on film cooling performance. *Applied Thermal Engineering* 2020; 165: 114578.
8. Tian K, Wang J, Zhanxiu C, et al. Effect of hole blockage configurations on film cooling in gas turbine components. *Chemical Engineering Transactions* 2017; 61: 229-234.
9. Goldstein RJ, Eckert ERG, Chiang HD, et al. Effect of surface roughness on film cooling performance. *ASME J. Eng. Gas Turbines Power* 1985; 107: 111−116.
10. Schmidt DL and Bogard DG. Effects of free-stream turbulence and surface roughness on film cooling. *International Gas Turbine and Aeroengine Congress and Exhibition*, Birmingham, UK, 10–13 June 1996, paper no. 96-GT-462, V004T09A049; 7. New York: ASME.

11. Schmidt DL, Sen B and Bogard DG. Effects of surface roughness on film cooling. *International Gas Turbine and Aeroengine Congress and Exhibition*, Birmingham, UK, 10–13 June 1996, paper no. 96-GT-299, V004T09A035; 7. New York: ASME.

12. Guelailia, A, Khorsi A, Slimane SA, et al. Numerical investigation on the effect of deposits height and position on the film cooling efficiency. *Journal of applied mechanics and technical physics* 2020; 61: 70−71.

13. Yuen CHN and Martinez-Botas RF. Film cooling characteristics of a single round hole at various streamwise angles in a crossflow Part I Effectiveness. *Int J Heat Mass Transfer* 2003; 46: 221−235.

14. Yuen CHN and Martinez-Botas RF. Film cooling characteristics of a single round hole at various streamwise angles in a crossflow Part II Heat transfer coefficients. *Int J Heat Mass Transfer* 2003; 46: 237−249.

15. Zhuang W and Hui-Ren Z. Showerhead film cooling injection orientation design on the turbine vane leading edge considering representative lean burn combustor outflow. *Part G-Journal Of Aerospace Engineering* 2021; 235: 2342-2356.

16. Kouchih FBA and Boualem K. The Effect of an Upstream Dune-Shaped Shells on Forward and Backward Injection Hole Film Cooling. *ASME Journal of Heat Transfer* 2020; 142: 122302.

17. Guelailia A, Khorsi A and Boudjemai AM. Thermal protection of rocket nozzle by using film cooling technology-effect of lateral curvature. *Int J Heat Technology* 2018; 36:1070−1074.

18. Zhu R, Terrence S, et al. Film cooling performance and flow structure of single-hole and double-holes with swirling jet. *Chinese Journal of Aeronautics* 2022; 35: 201-213.

19. Yang X, Zhang K, Junmei W, et al. Numerical analysis of vane endwall film cooling and heat transfer with different mainstream turbulence intensities and blowing ratios. *International Journal of Thermal Sciences* 2022; 175: 107482.

20. Khorsi A, Guelailia A and Hamidou MK. Improvement of film cooling effectiveness with a small downstream block body *Journal of applied mechanics and technical physics* 2016; 57:1-6.

21. Zhang C and Wang Z. Influence of streamwise position of crescent-shaped block on flat-plate film cooling characteristics. *Journal of the Brazilian Society of Mechanical Sciences and Engineering* 2019; 41:1-14.

22. Hefang D, jinfang T, Mingmin Z, et al. Overall cooling performance evaluation for film cooling with different winglet pairs vortex generators. *Applied Thermal Engineering* 2022; 201: 117731.
23. Wilfert G and Fottner L. The Aerodynamic Mixing Effect of Discrete Cooling Jets With Mainstream Flow on a Highly Loaded Turbine Blade. *ASME Journal of Turbomachinery* 1996; 118: 468-478.
24. Thole KA, Sinha A, Bogard DG, et al. Mean temperature measurements of jets with a crossflow for gas turbine film cooling application. *Rotating machinery transport phenomena* 1992. New York; p. 69–85.
25. Coulthard SM, Volino RJ and Flack KA. Effect of jet pulsing on film cooling—Part I: effectiveness and flow-field temperature results. *ASME Journal of Turbomachinery* 2007; 129: 232–46.
26. Saumweber C, Schulz A and Wittig S. Free-stream turbulence effects on film cooling with shaped holes. *ASME Journal of Turbomachinery* 2003; 125: 65-73.
27. Walters DK and Leylek JH. A Detailed Analysis of Film-Cooling Physics: Part I—Streamwise Injection with Cylindrical Holes. *ASME Journal of Turbomachinery* 1997; 122: 102–112.
28. Schmidt DL, Sen B and Bogard DG. Film Cooling with Compound Angle Holes: Adiabatic Effectiveness. *ASME Journal of Turbomachinery* 1996; 118: 807–813.
29. Versteeg HK and Malalasekera W. *An Introduction to Computational Fluid Dynamics, The Finite Volume Method*. 1th ed. England: Longman Scientific and Technical, 1995, p.520.

Nomenclature

d	coolant hole diameter (mm)
h	enthalpy (J.kg^{-1})
H	hole lid height (mm)
M	blowing ratio
p/d	hole pitch
p	pressure (Pa)
t	Time (s)
T	temperature (K)

U	velocity (m.s^{-1})
x	axial distance along surface (mm)
y^+	boundary layer thickness
z	lateral distance (mm)

Greek Symbols

α	injection angle (°)
ρ	density (kg.m^{-3})
η	film cooling effectiveness
λ	Thermal conductivity (W.m^{-1}.k^{-1})
τ	Reynolds stress tensor
μ	Dynamic viscosity (Pa.s)

Subscripts and superscripts

∞	free stream conditions
c	Injection conditions
tot	total
—	average value