Proceedings of National Conference on Technological Developments in Civil and Mechanical Engineering (NCTDCME-18)
SPVP,S.B. Patil College of Engineering, Indapur
JournalNX- A Multidisciplinary Peer Reviewed Journal (ISSN No: 2581-4230)
15th -16th March- 2018

NUMERICAL STUDY OF TURBINE BLADE FILM COOLING TECHNOLOGY

Niranjan B. Ghule (Author)
M. Tech Student, Mechanical Engineering Department,
Sardar Vallabhbhai National Institute of Technology,
Surat, Gujrat, India, 395007
niranjanbghule@gmail.com

Prayass Rai (Author)
MTech Student, Mechanical Engineering Department,
Sardar Vallabhbhai National Institute of Technology,
Surat, Gujrat, India, 395007
prayass.rai007@gmail.com

Abstract— In order to increase thermal efficiency of a gas turbine, higher turbine inlet temperature (TIT) is needed. However, higher TIT increases thermal loading to its hot-section components and reducing their life. Therefore, one of the very complicated cooling technology such as film cooling and internal cooling is required especially for HP turbine blades. In film cooling, relatively cool air is injected onto the turbine blade surface to form a protective layer between the surface and hot mainstream gas. The highest thermal load usually occurs at the leading edge of the airfoil, and failure is probably to happen in this region. Film cooling is typically applied to the leading edge through an array of hole rows called showerhead. In this work firstly prediction of blowing ratio is done on a flat plate for film cooling then to get the idea of flow physics analysis is extended over the actual turbine blade using ANSYS Workbench.

Keywords—Effectiveness, film, blowing ratio,turbine blade.

I. INTRODUCTION

The inlet gas temperature has a limit due to the physical properties of the turbine blades. By sophisticated cooling technique and introduction of high temperature withstanding materials, this limitation can be overcome to a great extent. It is a common practice to have multiple cooling passages inside the blade through which coolant fluid is bled in the radial direction. Cooling techniques include jet impingement, enhanced cooling with rib tabulators, and pin fin method. Due to the nature of its working, the power generated by a turbine increases with increasing the temperature at which the gas enters, called the turbine inlet temperature. An increased power output results in a higher efficiency. However, the turbine inlet temperature cannot be increased arbitrarily because of the limits imposed due to the temperature at which the blade material melts. Although advances have been made in material science to make new alloys having high melting points that can withstand operation at such high temperatures without failing, these materials are expensive and are difficult to machine. As the blade material melts at a lower temperature than the operating conditions of the turbine, a cooling method must be incorporated into the blade design to ensure the safe and smooth running of the turbine. It is important, while devising a cooling scheme, to have knowledge about the boundary conditions of the blade during turbine operation, so that large gradients can be avoided. This is because large gradients cause thermal stress cutting the component life short significantly.

Fig.1 shows the history of different techniques developed and its respective improvement in inlet temperature since 1950 to 2010.

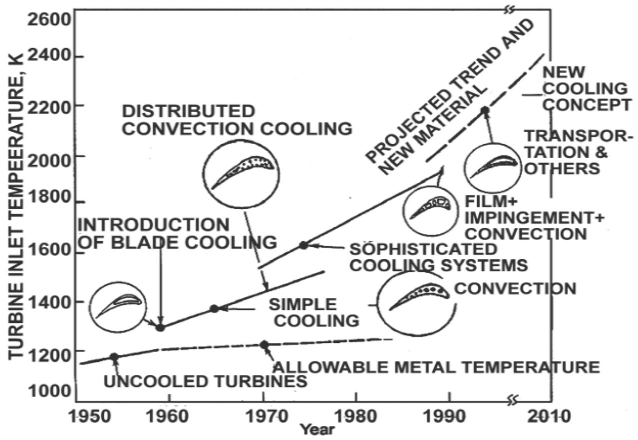

Figure 1 Year wise improvement in Inlet temperature respective with the different technologies

Advanced gas turbine engines operate at high temperatures (1200-1400°C) to improve thermal efficiency and power output. As the turbine inlet temperature increases, the heat transferred to the blades in the turbine also increases. The level and variation in the temperature within the blade material (which causes thermal stresses) must be limited to achieve reasonable durability goals.

II. TYPES OF TURBINE BLADE COOLING

There are two broad categories of cooling used in gas turbine blades:

1) Internal Cooling
2) External Cooling

In internal cooling, the cool compressed air flows internally inside the passages of the turbine blade and thus heat transfer occurs between the cold air in the passage and the adjacent hot surface of the turbine blade. In internal cooling techniques Impingement, Pin-Fin, Rib Turbulated, Dimple cooling techniques are involved. In external cooling, the cool compressed air is ejected from holes on the surface of the turbine blade or the vane and creates a thin film between the surroundings and the blade surface thus preventing contact between the hot air and the blade surface, enhancing heat transfer. While, External cooling technique involves Film cooling, Transpiration cooling and Thermal Barrier Coating (TBC).

Film Cooling

Film cooling is one of the major technologies allowing today's gas turbines to obtain extremely high turbine working temperatures, high efficiencies, and extended life parts. In turbine blade film cooling, relatively cool air is injected from the inside of the blade body to the outside surface, which forms a protective layer between the blade surface and hot mainstream. Which resists the transfer of heat from hot gas to turbine blade through the contacting surface.

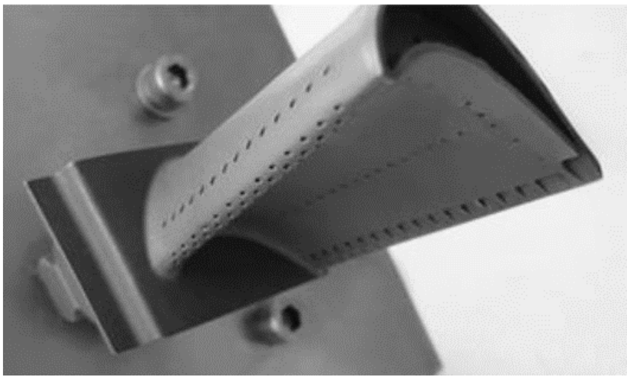

Figure 2 Film cooling

Figure 3 shows the schematic of film cooling concept. Typically, the heat load to the surface without film cooling is represented as heat flux q_0"= h_0 (T_g- T_w) , where h_0 is the heat transfer coefficient on the surface with wall temperature Tw and oncoming gas temperature (T_g). When coolant is injected on the surface, the driving temperature is T_f, film temperature, which is a mixture of gas (T_g) and coolant temperature (T_c), q"= h(T_f - T_w), where h is the heat transfer coefficient on the surface with film injection. Also, a new term film Effectiveness η) is introduced, where η= (T_g - T_f)/ (T_g - T_c). The η values vary between 0 and 1, with 1 as the best film cooling effectiveness. Therefore, the heat flux ratio can be written as: q"/ q_0"= (h/h_o) [(T_f - T_w)/(T_g -T_w)]=(h/h_o)[1- η(T_g - T_c)/(T_g - T_w)]. To obtain any benefit from film cooling, the heat load ratio, q"/ q_0", should be below 1.0. The heat transfer coefficient ratio (h/h_o) is enhanced due to turbulent mixing of the jets with the mainstream and is normally greater than 1.0. The temperature ratio) [(T_f - T_w)/(T_g - T_w)], which is related to the film effectiveness should be much lower than 1.0 such that the heat load ratio is lower than 1.0.

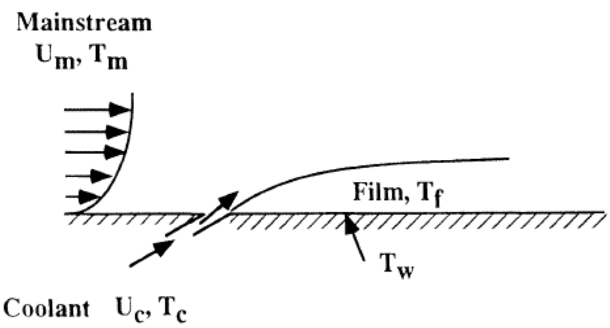

Figure 3 Schematic of film cooling concept

The best film cooling design is to reduce the heat load ratio (i.e., smaller h/ho enhancement with greater η) for a minimum amount of coolant available for a film cooled airfoil.

II. COMPUTATION OF FLAT PLATE FILM COOLING

It is common in literature to use a flat plate to perform fundamental studies on various effects on film cooling. Also, results on flat plates have been used to calibrate and standardize experimental techniques to measure film cooling effectiveness and heat transfer coefficients. While the best film cooling coverage can be obtained by injecting the fluid parallel to mainstream, manufacturing constraints dictate that holes be angled. Using film cooling holes perpendicular (90°) to the mainstream results in very low film cooling effectiveness is observed. The use of holes inclined at 40°-30° typically strikes a balance between film cooling performance and manufacturing ease.

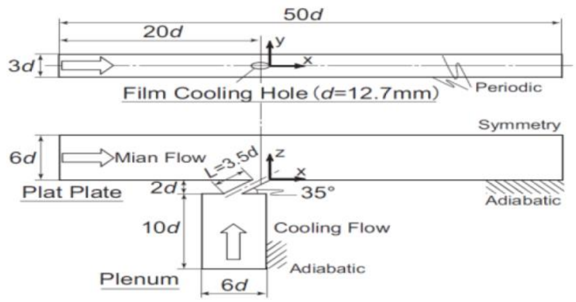

Figure 4 Computational Domain

Computational domain is referred from W Vickery and H Iacovides [2] to analyze Blowing Ratio variation results. The non-uniform grid density was used for various areas, particularly fine mesh near hole outlet and inflation layers are provided near the flat surface where the real flow is to be analyzed. All the components of computational domain are meshed with unstructured grid. A tetrahedral grid is used in this study to allow the highest quality in all regions with the prism boundary layers' application. Jet Cooling hole, mainstream flow duct and other three in the channel. Each section is meshed with appropriate topology. In order to resolve the mean velocity, mean temperature, heat flux and turbulent quantities in the viscosity-affected near wall region (Rey <150) accurately, the near wall turbulence model requests that y+ value at the wall-adjacent cell should be the order of one and there are at least 2 ten cells in this region. So in the hole and the region near the test wall the density of cells is identified to satisfy the requirement. After a multiple of tests and adjustments the final adopted grid for calculations is obtained.

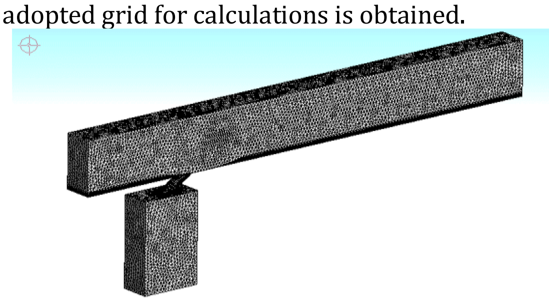

Figure 5 Meshing of a model

A. Methodology

A pressure-based approach is used to solve the governing equations for the conservation of mass, momentum and energy apart from turbulence transport equations. The solver employs finite-volume discretization and an algorithm, which involves a general class of the projection method. The Green-Gauss node based approach is used to quantify the fluxes at the cell faces preserving second-order spatial accuracy. All the computations are carried out till the residuals converged. The shear stress transport (SST) k-ω turbulence model [4] a two-equation eddy-viscosity model, which has become very popular and is used here.

B. Boundary condition

Mainstream Reynolds Number kept, Re = 16400 With the mainstream hot gas inlet temperature is 1600K and cooling air temperature is 900K and other boundary conditions are shown in below table.

TABLE I. BOUNDARY CONDITIONS

Zone name	Zone type	Gauge Pressure	
Hot Gas Air Inlet (1600K)	Mass Flux Inlet	3.1 Bar	0.07 Kg/m^2.s
Cooling Air inlet (900K)	Mass Flux Inlet	3.1 Bar	0.014 to 0.14 (5 steps) Kg/m^2.s
Test section side walls	Wall		-
Outlet	Pressure Outlet	1.6 Bar	-

C. Results and Discussion

In fig. 6 results are obtained on various BRs (M) varying from 0.2 to 1 whereas results obtained are quite good at M above 0.6 to 1 and it can be observed from figure and graph followed.

The four profiles presented in figure 6 represent samples of four states for the coolant jets: (i) fully attached coolant jets shown in fig. 6(a) but it is incapable to cover the sufficient region, (ii) fully attached coolant jets shown in fig. 6(b), (iii) coolant jets detachment starts shown in fig. 6(c), and (iv) coolant jets detachment propagates

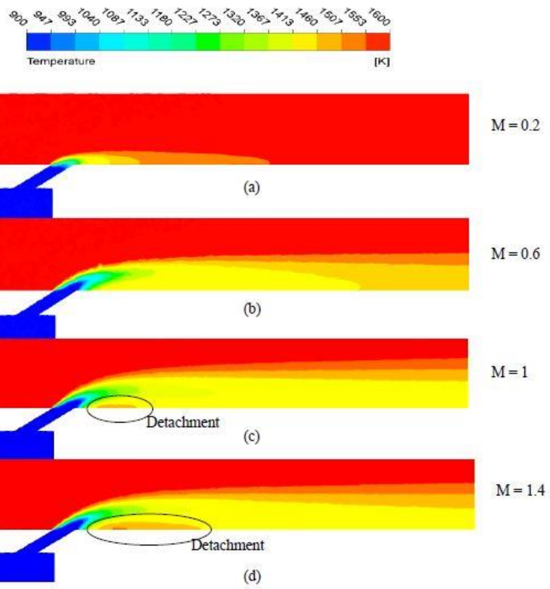

Figure 6 Film flow at various blowing ratios

however, increase in M value above 1 shown in fig. 5.1.4d, clearly as the coolant jets begin to detach the coolant temperature at the wall decreases as the core of the coolant jet travels above the surface. The range of flux ratios for each of these flow states was found to be M< 0.6 for fully attached jets, 0.6<M< 1.4 for detached/reattached jets, and M> 1.4 for fully detached jets for flat surface flows.

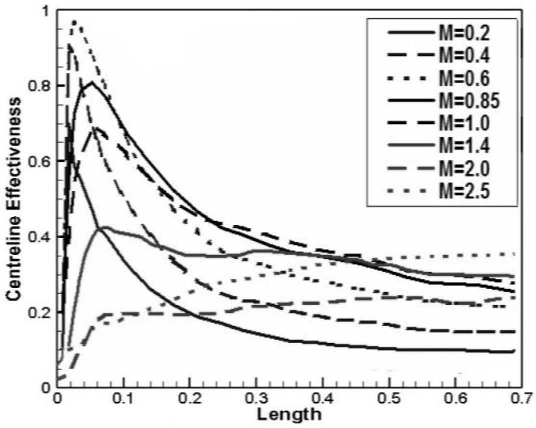

Figure 7 Plot of Centreline Effectiveness to the length in flow direction for various M.

Clearly, whether or not the coolant jets are attached strongly affects the cooling performance and that can be observed in plot of Centerline effectiveness to the length in flow direction for various M in figure 7.

III. COMPUTATION OF FILM COOLING ON TURBINE BLADE

In this case a trial for film cooling is done on a blade for this the blade configuration is referred from S. Sarkar and P. Gupta [4]. This case giving the idea of flow physics on turbine blade it may help to configure accurate location, number of holes, geometry size and shape. It further also helps to predict the BR (M) for the suction and pressure side.

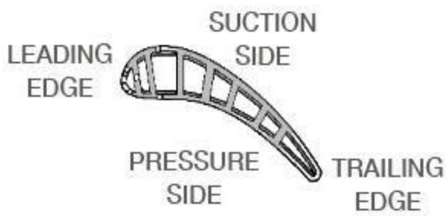

Figure 8 Regions on a turbine blade

Computational domain is referred from W S. Sarkar and P. Gupta [4] to analyze the flow physics over a blade at M = 0.5. For modelling ANSYS BladeGen and Design Modeler is used. Primarily 10 holes of 15mm diameter were provided with the pitch of 20mm and then it is improved up to 19 numbers and 25 mm pitch with same diameter. Holes are angled at 145^o for first row of suction side and 220^o for second row in pressure side.

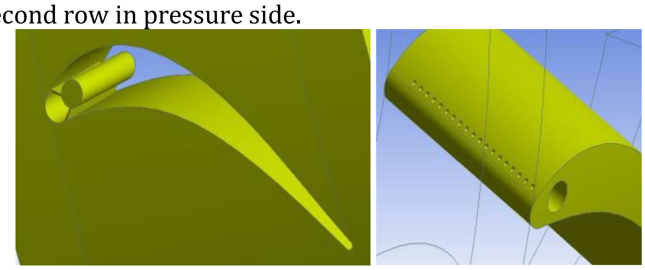

Figure 9 Location and orientation of holes 19 x 15 x 25mm

The non-uniform grid density was used for various areas, particularly fine mesh near hole outlet and inflation layers are provided near the flat surface where the real flow is to be analyzed. All the components of computational domain are meshed with unstructured grid.

TABLE II. MESH DETAILS

Sizing	
Min Size	2.e-3 m
Max Face Size	0.01 m
Max Size	0.01 m
Inflation	
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
Statistics	
Nodes	107906
Elements	475147

adjustments the final adopted grid for calculations is obtained.

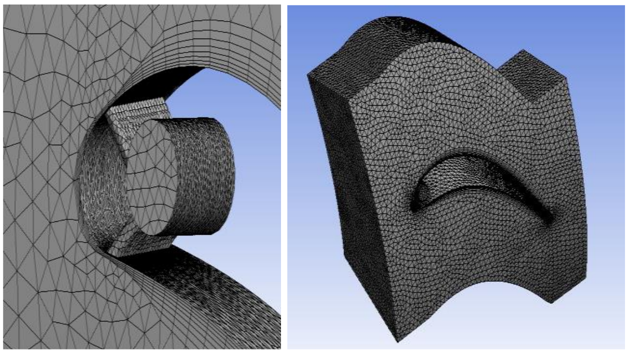

Figure 10 Meshing of a model

A. **Methodology**

Same methodology is followed as previous flat plate case

B. **Boundary condition**

Mainstream Reynolds Number kept, Re = 16400 With the mainstream hot gas inlet temperature is 1600K and cooling air temperature is 900K and other boundary conditions are shown in below table.

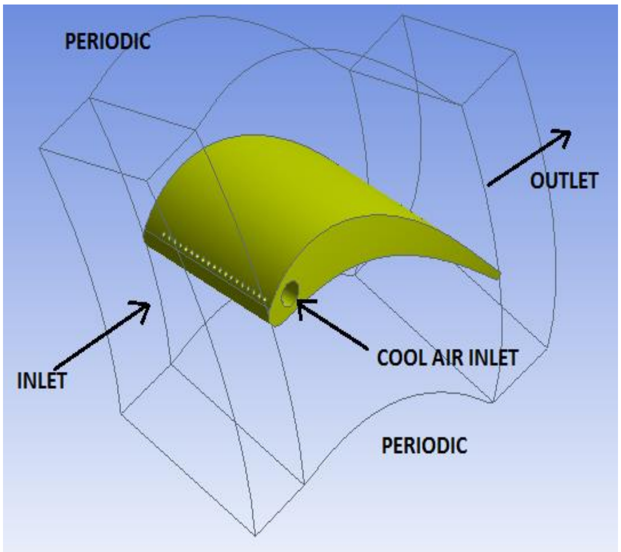

Figure 11 Boundary Types

TABLE III. BOUNDARY CONDITIONS FOR 2ND CASE

Zone name	Zone type	Gauge Pressure	
Hot Gas Air Inlet (1600K)	Mass Flux Inlet	3.1 Bar	0.07 Kg/m².s
Cooling Air inlet (900K)	Mass Flux Inlet	3.1 Bar	0.035 Kg/m².s
Test section side walls	Wall		-
Outlet	Pressure Outlet	1.6 Bar	-

C. **Results and Discussion**

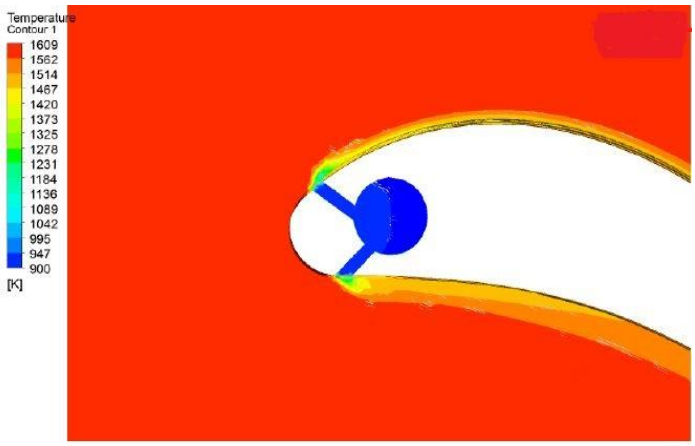

Figure 12 Film of cooling air over a Blade surface

The coolant layer formation over the both suction and pressure side surface with variation of thickness of film in both sides because of suction and pressure side. Blowing ratio (M) in this numerical analysis is kept equal to 0.5. From figure 12, it can be observed by qualitative analysis that the detachment region is developed in case of pressure side is more while no lift or detachment region is observed at suction side. Film is flowing at constant thickness over a longer distance at suction side while the detached film at pressure side is more prone to mix with the hot gas and cause to less effective.

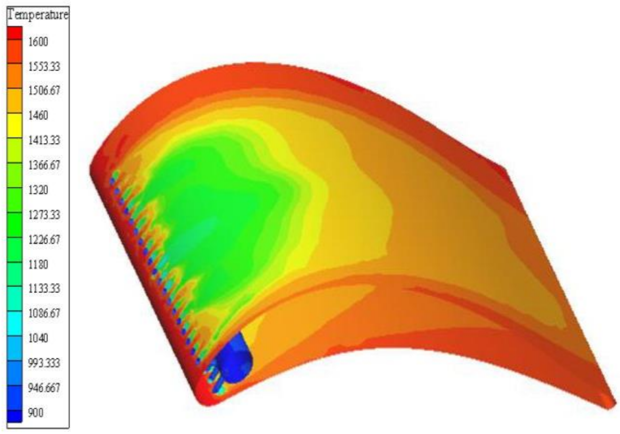

Figure 13 Temperature contour on Blade surface at Suction Side

Figure 13 and 14 are showing obtained results at blowing ratio 0.5. Here, it can be observed that the same blowing ratio is not suitable on both sides of blade due to different value of pressure while it can also have observed the lateral spreading of coolant over the blade is less in case of pressure side.

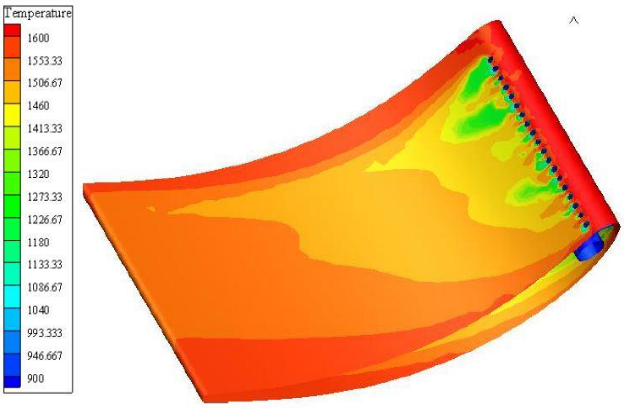

Figure 14 Temperature contour on Blade surface at Pressure Side

At the same time reduction in temperature by film cooling is up to 300°C covered near 30% of the blade upper surface downstream length and the resulting effectiveness is near 0.5 and on the pressure side surface length covered towards downstream side is about 10% only with the resulting effectiveness equal to 0.5. However, this results are obtained at Re=16400 so the flow is purely laminar type instead of this if Reynolds number is increased up to the range of 10^5 the film may be effective for shorter length only Because of this more number of holes needs to be provided which may give the better results.

IV. CONCLUSION

Prediction of blowing ratio (M) can be helpful in practical cases in this case it can be observed that the value of M is quite suitable in the range of 0.5 to 1 otherwise the detachment of film cause less effective cooling.

This study is limited to Re = 16400 but in actual Reynolds number may vary in the range of 105. Nevertheless, this study may helpful for low range of speed and gives the idea of right shape, orientation, numbers and location of holes. This case will also helpful in prediction of Blowing Ratio (M) values. Due to different pressure on both sides of turbine blade i.e. Suction and pressure sides it can be observed here that it is suitable to adjust different value of M on both sides by providing separate coolant inlet slots to achieve different values of Blowing Ratios (M).

Numerical investigations have been applied on the flow and film cooling for these critical regions. The compressible Navier-Stokes equations, K-ω turbulence model together have been solved by Fluent in these Case studies. The main interest in other experiments has been the flow field of the cooling jet mixing with mainstream while heat transfer is secondary. However, many results of this study are in good agreement with published work.

REFERENCES

1) Je-Chin Han and Srinath Ekkad, "Recent Development in Turbine Blade Film Cooling"International Journal of Rotating Machinery 2001, Vol. 7, No. 1, pp. 21-40
2) W Vickery and H Iacovides "Computation Of Gas Turbine Blade Film Cooling" Turbulence Mechanics Group School of Mechanical, Aerospace and Civil Engineering, The University of Manchester.
3) Karsten Kusterer et al. , "Highest-Efficient Film Cooling By Improved Nekomimi Film Cooling Holes — Part 1: Ambient Air Flow Conditions" Proceedings of ASME Turbo Expo 2013, USA, GT2013-95027
4) S. Sarkar and P. Gupta, "Thermal Analysis of a Turbine Blade: Effect of Film Cooling and Internal Convective Cooling", ASME GTINDIA2015-1296
5) Schmidt, D. L., Sen., B., 1996, "Film Cooling With Compound Angle Holes: Adiabatic Effectiveness," ASME Journal of Turbo machinery, Vol.118, pp. 807–813.
6) Karsten Kusterer et al. , "Double-Jet Ejection Of Cooling Air For Improved Film-Cooling," ASME, GT2006-90854.
7) Honami, S., Shizawa, T., and Uchiyama, A., 1994, "Behavior of the Laterally Injected Jet in Film Cooling: Measurements of Surface Temperature and Velocity/ Temperature Field Within the Jet," ASME Journal of Turbo machinery, Vol. 116, pp. 106–112.
8) Mehendale, A.B., Han, J.C. and Ou, S., "Influence of High Mainstream Turbulence on Leading Edge Heat Transfer," ASME Journal of Heat Transfer, Vol. 113, November 1991,pp. 843-850.
9) Je-Chin Han* and Akhilesh P. Rallabandi, "Turbine blade film cooling using psp technique," Frontiers in Heat and Mass Transfer (FHMT), 1, 013001 (2010), DOI: 10.5098/hmt.v1.1.3001