# Numerical simulation of blowing ratio effects on film cooling on a gas turbine blade

H. A. Daud[1], Q. Li[1], O. Anwar Bég[1] & S. A. A. AbdulGhani[2]
[1]*Department of Engineering and Mathematics, Sheffield Hallam University, UK*
[2]*Mechanical and Industrial Engineering, Qatar University, Qatar*

## Abstract

This article investigates the film cooling effectiveness and heat transfer in three regimes for a film-cooled gas turbine blade at the leading edge of the blade with $45^0$ angle of injection. A Rolls Royce blade has been used in this study as a solid body with the blade cross section from Hub to Shroud varying with a degree of skewness. A 3-D finite-volume method has been employed (FLUENT 6.3) with a $k-\varepsilon$ turbulence model. The numerical results show the effectiveness cooling and heat transfer behavior with increasing injection blowing ratio BR (1, 1.5 and 2). In terms of the film cooling performance, high BR enhances effectiveness cooling on pressure side and extends the protected area along the spanwise direction from hub to shroud. The influence of increased blade film cooling can be assessed via the values of Nusselt number in terms of reduced heat transfer to the blade.

*Keywords: turbine blade, film cooling, blowing ratio, CFD, heat transfer, effectiveness cooling.*

## 1 Introduction

Increasing the thrust, overall efficiency and reducing the fuel consumption as much as possible are major issues in modern gas turbine engineering, and this is generally achieved via increasing the turbine inlet temperature. These higher temperatures however have detrimental effects on the integrity of high pressure turbine components and materials composing the turbine blades. Film cooling technology is justified to protect blades surfaces from incoming hot gas and for increasing life time. Numerical and experimental studies of three-dimensional

WIT Transactions on Modelling and Simulation, Vol 51, © 2011 WIT Press
www.witpress.com, ISSN 1743-355X (on-line)
doi:10.2495/CMEM110251

film cooling for real turbine blade models are much less common than the flat plate models which deal with injection of a coolant fluid. In addition, rotating blade studies compared with the stationary blades are rare. The film cooling method is an efficient way to protect the blades of turbines from the hot gas and is widely used in contrast with Impingement Cooling, Transpiration cooling, and Convection Cooling. Numerous of the prior studies examined aerodynamics, heat transfer and film cooling over a flat, curved plate in two dimensions. For example, Kadja and Bergeles [1] presented numerical computations of blade turbine cooling via injection of a fluid into the main stream flow. The study was conducted via creating a slot along the flat surface. Typically, the local film cooling effectiveness increased with increasing blowing ratio and the Nusselt number was decreased near the slot. Hung *et al.* [2] studied 3-D effectiveness cooling and heat transfer over concave and convex plates using the liquid crystal thermograph technique. The performance of film cooling was investigated with one row of injection holes (angle of injection $35^0$) for various blowing ratios (BR=0.5, 1, 1.5 and 2). The concave model results showed that with increasing blowing ratio the heat transfer coefficient and film cooling effectiveness increase and the convex model surface results demonstrated that the compound angle ($0^0$, $45^0$ and $90^0$) increases both heat transfer and film cooling effectiveness at moderate and high blowing ratio. The thermal effect of the turbine blade has been studied for a NACA 0021 airfoil by Kassim *et al.* [3]. Good agreement was obtained between the experimental and computational results using 6.2 Fluent code for the different angles of attack, different jet angle and velocity ratio on the penetration area. Lakehal *et al.* [4], Theodoridis *et al.* [5] utilized a simple geometry in their study of a stationary blade (symmetrical turbine blade). Two dimensional and three dimensional finite volume simulations were employed to model film cooling holes at the leading edge of the blade with the FAST 3D CFD code. The computational results for different blowing ratios (BR = 0.3 to 1.1) were compared with experimental data including isentropic Mach number, velocity and pressure fields. Characterizations of the gas flow and heat transfer through a modern turbine are important engineering considerations, which represent very complex flow fields and high turbulence levels. Forest *et al.* [6] described the influence of hole shape on heat transfer coefficients and film cooling effectiveness through a series of experimental measurements compared with a heat transfer design code (HTDC). A computational method was used to simulate transonic flow in a Rolls Royce blade. Flared hole film cooling and heat transfer coefficient results were found to correlate poorly on the blade pressure side; on the suction side flared holes were found to provide increasing effectiveness cooling compared with cylindrical holes.

Improving the arrangement of holes for the film cooling in rotating turbine blade has a strong influence on aero-thermodynamics. Burdet and Abhari [7] and also Garg and Abhari [8] predicted Nusselt numbers for such regimes. Their computations were based on an explicit finite volume code with an algebraic turbulence model and implemented in excess of 2.5 million grid points to calculate the flow. CFD results were compared with experimental data to predict transonic film flow. These studies highlighted the macro-flow structure for the interacting coolant jet process especially near the hole region, at different hole rows. On the pressure surface the results generated however were poor.

After reviewing the previous published studies, the majority of studies communicated hitherto have focused on 2-D or 3-D aerodynamic flow and heat transfer for a simple blade geometry, flat or curved plate, NACA 0021, symmetrical turbine blade and simple cross section blade from hub to tip. Consequently, this paper aims to extend these studied by focusing on:

- Using different cross section blade geometry (from hub to shroud) with angle of twist.
- The main flow (hot gas) and coolant system (cooled fluid) differing in temperature, pressure and chemical composition for the hot gas and cooled air.
- Solid body thermal properties which will be simulated by inserting the type of material type e.g. carbon steel (SAE4140) in the FLUENT software property specification pre-processor. The blade taken in the previous studies as a *shell surface* is shown in Fig. 1.
- Aerodynamic flow and heat transfer in modern gas turbine constitutes a very complex flow field with high turbulence levels; therefore we apply film cooling in this paper from the blade leading edge. This is additionally complicated due to the resulting interference between the main flow and injected coolant.

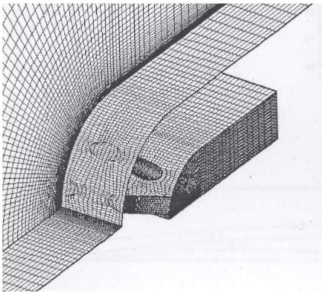


Figure 1: Previous studies defined blades as shell surface with cooling holes Azzi and Jubran [9].

The motivation of this study is to achieve robust numerical predictions of film cooling effectiveness, heat transfer, temperature distribution on a skewness solid blade (complex geometry), effect of coolant fluid property in the hub, mid and shroud area prior to further experimental tests which are currently being designed at the Sheffield Hallam University Engineering Laboratory, and which will be communicated imminently.

## 2 Computational flow model

Computation Fluid Dynamics (CFD) can analyze internal and external aerodynamic flow, heat and mass transfer and also multiphysical problems by solving three dimensional flows, steady state mathematical equations as continuity equation; Reynolds averaged Navier- Stokes (RANS) flow equations and the energy equation via numerical algorithms on a computer. The transport equation is additional term utilize to simulates a non reacting gas mixture when this gases jet in to a crossflow at different density (Renze *et al*. [10]). However, the ($k$ – $\varepsilon$) equation is the simplest turbulence model employed widely in engineering simulations, for which the initial boundary conditions need to be supplied. These involve algebraic approximations for turbulence kinetic energy ($k$) and its rate of dissipation ($\varepsilon$) via a semi-empirical model.

The numerical computations in this paper were completed via a series of steps, including grid checking, model solver definition (segregate implicit CFD technique) selection and turbulence simulation with the standard RNG ($k$ – $\varepsilon$) model. Eghlimi, *et al*. [11] applied Renormalization group (RNG) model in Fluent code to predict the turbulent gas flow and also Lakehal [12] has elucidated that there is little advantage to using different viscosity data for (RSM) Reynolds stress turbulence model as compared with the eddy viscosity model. In consistency with these we implement herein turbulence (or eddy) viscosity constants as follows: $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$, $C_{1\varepsilon}$ : 1.42, $C_{2\varepsilon}$ : 1.68, $C_\mu$ : 0.0845.

## 3 Turbine blade geometry and mesh

Fig. 2 illustrates the Rolls Royce blade model with a maximum chord length, $C_X$ of 52 mm, where ($C_X$) represents blade axial chord. The maximum width $C_Y$ is 27.3 mm ($C_Y$ is the blade true chord); in addition the film cooling geometry contains, on each side, one row of holes located near the leading edge of the blade. Lateral holes have a diameter of 4 mm, within inclined angle of 9.13$^o$ along the span and the holes diameter (d) is 1 mm with a lateral spacing of *5d*. Therefore, the number of holes on the pressure side and suction side are 42 with an angle of injection of 35$^o$.The incoming hot gas simulation box is represented in dimensions (4.8*2.11*3.85) Cx.

The meshing process of these complex volumes is performed using an unstructured mesh and the best way to control size mesh intervals for edges and

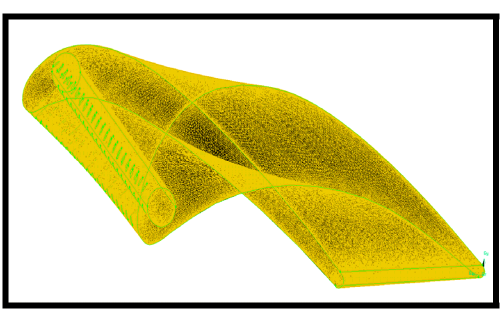


Figure 2: Local mesh of blade model as a solid body with cooling holes system.

mesh elements for face and volumes are the selection option size function from the tool mesh.

In this paper a high quality of mesh for the blade, hot gas and fluid coolant is achieved via the multi-block method with fewer cells. Therefore, a size function is employed for meshing the volumes to control the size, growth rate and (owing to the geometric complexity in the Rolls Royce blade) the holes geometries. Tetrahedral elements were used to complete the volumetric mesh and the final mesh generated for the three volumes contains in excess of 10 million cells (elements). All simulations were executed until the solution convergence factor $1*10^{-5}$and $1*10^{-7}$ were satisfied on the energy equation. The solution has been controlled by the select SIMPLIE algorithm to solve velocity-pressure coupling with an implicit procedure, such that more than 3500 iterations are sustained to convergence (solution stability).

## 4 Boundary conditions

The initial boundary condition details are inserted in the code the inlet main velocity at 163.4 m/sec and the mass flux (blowing) ratio values for the plenum are 1, 1.5 and 2. In addition, the ratio of the hot gas temperature to the cooled air temperature was specified as ($T_\infty / T_C = 1.6$) where; ($T_C$) is the coolant temperature and ($T_\infty$) designates the incoming hot gas temperature ($T_\infty = 460$ K). In the numerical simulation the outlet flow was defined as a static pressure and the turbulence flow ($T_u$) was calculated as a function of the Reynolds number. The Reynolds number ($R_e$) is approximately $2.57*10^5$ based on the maximum axial chord of the blade model. So, $T_u$ can be written as defined as:

$$T_u=0.16*Re^{(-1/8)} \tag{1}$$

The Rolls Royce blade is simulated as a solid body. The blade material properties are demonstrated in table 1.

Table 1: The blade material properties.

| Material type (low carbon steel ) | Density ( $\rho$ ) kg/m$^3$ | Thermal conductivity (k) W/m-K | specific heat ($C_P$) J/kg-K |
|---|---|---|---|
| SAE 4140 | 8030 | 42.7 | 473 |

## 5 Result and discussion

Fig. 3(a), 3(b) and 3(c) illustrate the distribution of the static temperature for the solid blade from the hub to shroud regions and from the blade leading edge to the trailing edge. The effects of blowing ratio (BR) on the distribution of blade temperature (T) via injecting coolant fluid into main hot gas are very clear. Blowing ratio (BR) in this paper can be calculated by:

$$BR = \rho_C * V_C / \rho_\infty * V_\infty \tag{2}$$

Fig. 3(a), 3(b) and 3(c) depict contour temperature distributions for blowing ratio BR=1, 1.5 and 2, respectively. The colour temperatures have been graduated from low to high temperature (leading edge to trailing edge area). Consequently, with increasing of mass flux ratio (BR) the blade surface temperature will be reduced on both the pressure side and the suction side and also from the hub to shroud. Thereby, the protected area of the blade from incoming hot gas will be increased with increasing blowing ratio. In this study film cooling technology will be studied firstly- convection cooling though blowing coolant fluid in to the lateral hole from hub to shroud (internal cooling). Secondly, coolant fluid will be injected for both the pressure and suction side as a secondary fluid to create a blanket above the blade surface (external cooling) simultaneously. According to Fig. 3, the drops in predicted blade temperature

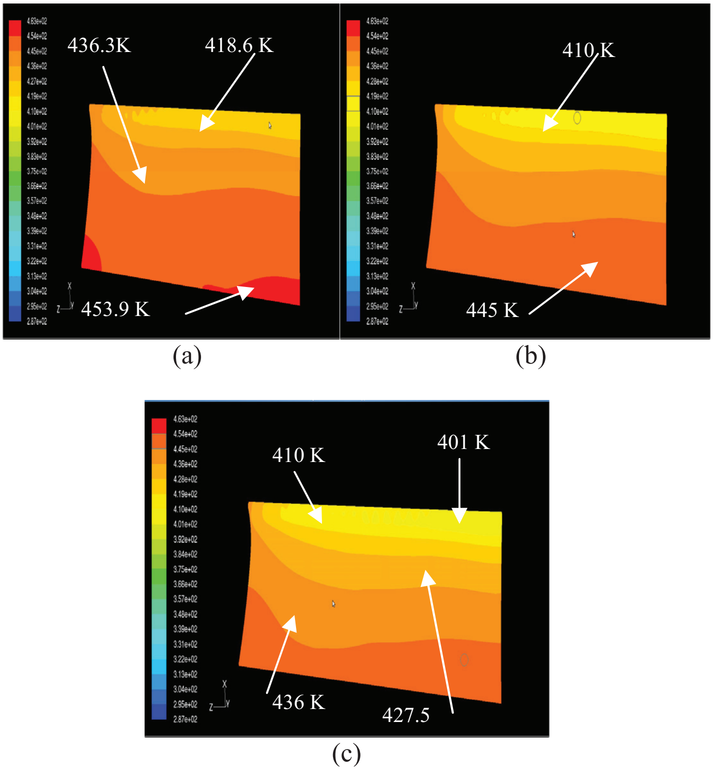



Figure 3: Predicted counters temperatures of the blade model at different blowing ratio (a) BR=1, (b) BR=1.5, (c) BR=2.

with rise of blowing ratio (BR) indicate that the effectiveness cooling will be enhanced from the hub to the shroud and from the leading to trailing edge, a result which correlates very well with the computations of Kadja and Bergeles [1].

A unique feature of our study is to demonstrate the effect of cooling on the blade as a solid body. In Fig. 4 therefore presented the trajectory of temperature and effectiveness cooling distribution for both pressure side and suction side on the solid body blade (always the effectiveness will be reversed trajectory compares with Temperature trajectory). In addition the hub, mid and shroud area temperature at blowing ratio BR=2 are also illustrated. The temperature along the span of the blade can be detected from distribution of temperature at the hub area to mid and from the mid to shroud area. Invariably the hub area temperatures will be lower than the mid and shroud area temperatures at the leading edge region. This is a result of the coolant fluid being blowing (injected) from the blade base (hub area). In the midspan section the predicted temperatures curve descends much more than the hub temperature profile at X/Cx>0.25 on the pressure surface and at X/Cx< -0.02 on the suction side Fig. 4. This pattern is due to the camber of blade and the angle of twist- the blade cross section with the span and axial cord at each section changes from hub to shroud (blade design shape).

Local film cooling effectiveness (η) is analyzed in this paper for both blade sides (pressure and suction side) as a function of ($T_\infty$), ($T_W$) wall temperature and ($T_C$) using this equation:

$$\eta = (T_\infty - T_W)/(T_\infty - T_C) \tag{3}$$

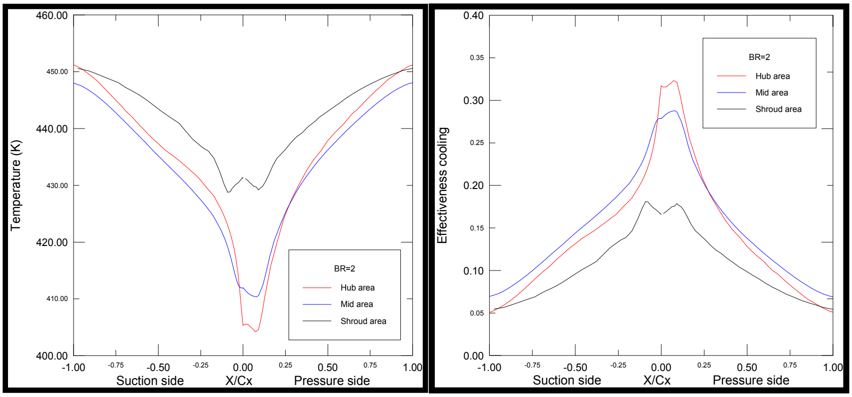
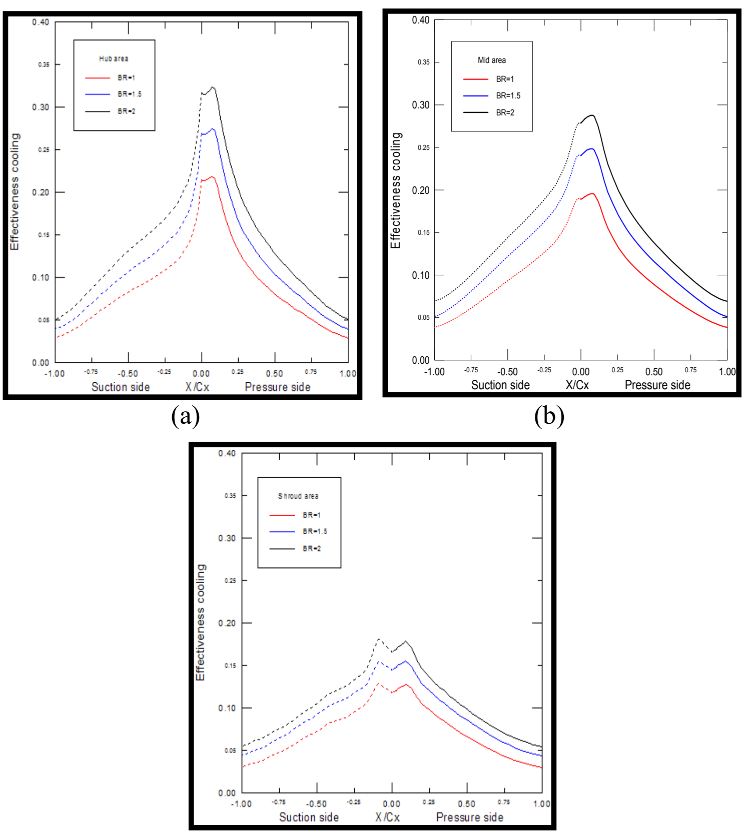



Figure 4: Temperature and the effectiveness cooling distributions difference in hub, mid and shroud area on the blade model by injecting coolant air at BR=2.

Fig. 5 shows the effectiveness cooling ($\eta$) in three regimes (hub, midspan and shroud), for different blowing ratios, BR=1, 1.5 and 2. Fig. 5(a) displays the effects of increasing blowing ratio (BR) from 1 to 2 at the hub area. Significantly, with increasing blowing ratio from BR=1 to 1.5 the effectiveness cooling will be increased by about 20.1% and 15.2% through respective increases in BR to 1.5 and 2 (near the leading edge of blade). The pressure side effectiveness cooling is higher than suction side at this regime (high BR enhanced film cooling on pressure side), a trend in agreement with Burdet and Abhari [7], and Tao *et al.* [13] attributable to the similar angle of injection ($45^0$) and lateral holes inclined ($9.13^0$) design.

In the midspan area the effectiveness cooling is illustrated in Fig. 5(b) and it also increases corresponding to bowling ratio by about 21.3% and 13.6% through BR=1 to 1.5 and from BR=1.5 to 2. At a blowing ratio BR=1 the effectiveness cooling near the injection zone (leading edge zone) is higher on the



(c)

Figure 5: Effects of blowing ratio (BR) on the blade effectiveness cooling- (a) hub, (b) mid and (c) shroud.

suction side than that on the pressure side, but with increasing BR, the film cooling effectiveness on the pressure side will gradually increase compared with the suction side.

Fig. 5(c) shows variation of effectiveness cooling with blowing ratio in the shroud regime. The trajectory of the pressure and suction side is nearly analogous (matched). About 18.3% and 12.7% film cooling effectiveness will be enhanced with increase blowing ratio changing from BR=1 to 1.5 and from BR=1.5 to 2 respectively. The film cooling effectiveness contours for the pressure and suction side on the blade model at blowing ratio BR = 2 is shown in Fig.6.

The values of effectiveness cooling change gradually form the leading edge to trailing edge (colder to hottest place). Briefly, when the values of blowing ratio (BR) are reduced the hottest area will be increased on both sides and the blade will be exposed to incoming hot gas.

A good correlation has been found between previous studies e.g. Kadja and Bergeles [1] and Guangchao *et al.* [14] and other published paper results on film cooling effectiveness studies with cylindrical hole shapes. With increases in values of BR the blade surface temperature drops and the effectiveness cooling is enhanced. BR is proportional to temperature difference ($\Delta T$) and inversely proportional to effectiveness cooling (η).

Fig.7 shows the comparison between the CFD prediction of Burdet *et al.* [15] and our computed film cooling effectiveness (η) on the blade model near hub position for BR=1 at angle of injection $35^0$. At first glance, the values of effectiveness cooling (η) at X/d < 0.3 on the pressure side are significantly higher than Burdet *et al.* [15], which is beneficial. On the suction side at X/d < 0.18 the values of (η) seems to coincide with Burdet *et al.* [15] while, the profile of the effectiveness cooling (η) descend after this region. The plummet in values of effectiveness cooling for both sides is attributed to the blade being modeled as a solid body which is much more realistic than the previous studies (where blades were analyzed as a shell) and also the effects of blade design, holes

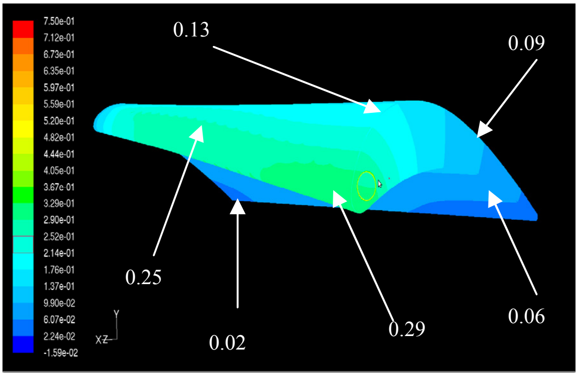



Figure 6: Film cooling effectiveness couture at blowing ratio BR= 2 on the blade model.

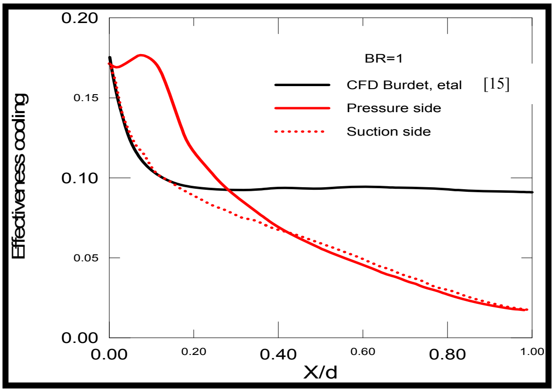



Figure 7: Distributions of film cooling effectiveness at blowing ratio BR=1 at angle of injection $35^0$: comparison between CFD results of Burdet *et al.* [15], the blade model calculation.

diameter and number of holes along the span which are different from previous investigations.

Fig. 8 illustrates the effects of coolant fluid on the distribution of film cooling effectiveness for the pressure side and suction side at the midspan location with BR=1, 1.5. In this article two different coolant fluids (air at T=287.5 and T=153 K) have been injected to resolve the best way to achieve improved blade protection. High effectiveness cooling was obtained through injected air as coolant fluid at temperature 287.5K. Evidently, from equation (3) η depends on $T_\infty$ , $T_W$ and $T_C$ , even with a decrease $T_C$ there is not necessarily an increase in effectiveness cooling (η) since the blowing ratio (BR) will be also affected by the temperature property as indicated by equation (2).

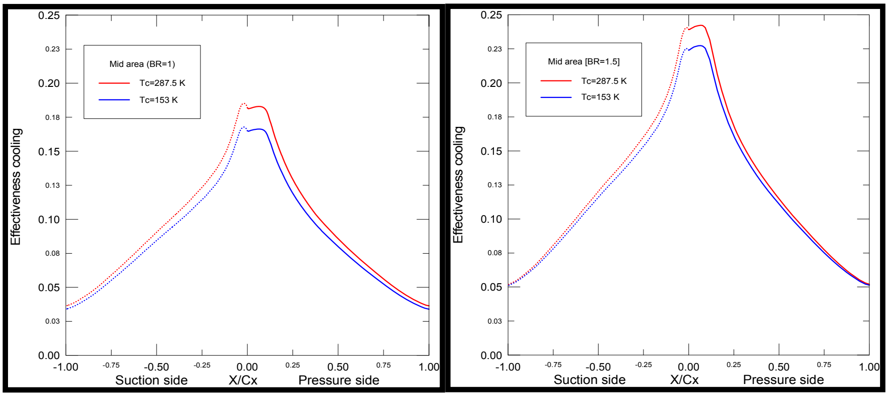



Figure 8: Effects of coolant fluid temperature on the film cooling effectiveness at BR=1, 1.5.

Prediction of heat transfer from the hot gas towards the blade can be achieved via the Nusselt number ($N_U$) based on the axial chord Cx and this is defined as:

$$N_U = (q^*Cx)/k(T_\infty - T_W) \tag{4}$$

where, $q$ is the blade wall heat transfer rate. Fig. 9 shows the predicted profiles of Nusselt number ($N_U$) at the mid span for BR=1 1.5 and 2.

Near the leading edge along the span (from hub to shroud) the heat transfer attains the maximum level and this is in excellent agreement with the results of Burdet and Abhari [7], Garg and Abhari [8] and Burdet *et al.* [15]. On the pressure side ($N_U$) suddenly drops after the leading edge. Certainly, the effects of film cooling are manifested via the creation of a layer of protection from the incoming hot gas on this side. At X/Cx>0.15 the BR influence is clearly observed as reducing ($N_U$) – again this trend concurs with the studies of Kadja and Bergeles [1] and Guangchao *et al.* [14]. On the suction side, the Nusselt number ($N_U$) suddenly falls after the leading edge and in the region X/Cx=-0.25 the values of ($N_U$) slightly increase owing to the effects of turbulence and the camber of blade and gradually falls; the better protection of blade from hot gas through increases effectiveness cooling, η and reduced blade Nusselt number ($N_U$).

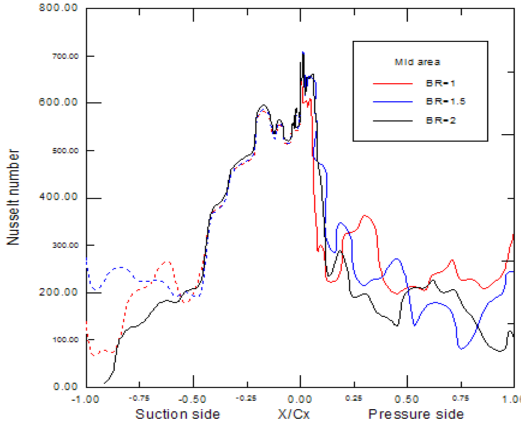



Figure 9: Predicted profile of Nusselt number (Nu) at midspan for BR=1, 1.5, 2.

Fig. 10 shows Nusselt number contours at the leading edge and film cooling regions at BR=2. The maximum value of Nusselt number ($N_U$) is determined at the leading edge along the blade span. This implies heat transfer from the hot gas *in the direction of the blade*. Near the holes there is no heat transfer to the blade due to coolant fluid injection. Thus, the Nusselt number ($N_U$) drops to negative values (heat transfer will be in the opposite direction).

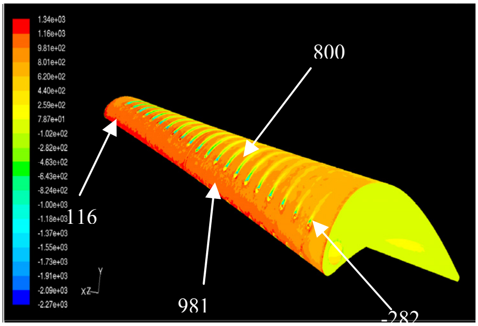



Figure 10: Nusselt number (Nu) counter on the blade model for BR=2.

## 6 Conclusions

The film cooling performance for a complex gas turbine blade at the leading edge with three blowing ratios has been studied numerically. The main findings of this investigation are as follows:

1) Film cooling effectiveness near the leading edge significantly increases with blowing ratio (η is proportional with BR). The influence of increasing BR on the film cooling effectiveness appeared on the pressure and suction side; the pressures side effectiveness cooling is much more enhanced than the suction side (high BR enhanced film cooling on pressure side).
2) The response of film cooling effectiveness along the span from hub to shroud section is enhanced with increasing blowing ratio. However, if there is dissimilarity between the published paper and the previous studies certainly this is due to the number of holes, holes diameter, blade section from hub to shroud and skewness.
3) There is no benefit in injection of coolant fluid at low temperature to enhance blade cooling at constant BR (see equation (5)).
4) The heat load on the blade represented by Nusselt number ($N_U$) is strongly influenced by increasing BR.

## Acknowledgement

The authors would like to thank the Government of Iraq, Ministry of Education for their generous financial support during Harbi A. Daud's studies at Sheffield Hallam University.

## References


[1] Kadja M. & Bergeles G., Computational study of turbine blade cooling by slot- injection of a gas. *Applied Thermal Engineering*, **17(12)**, pp. 1141-1149, 1997.

[2] Hung M.S, Ding P.P & Chen P.H, Effect of injection angle orientation on concave and convex surface film cooling, *Experimental Thermal and Fluid Science*, **33**, pp.292-305, 2009.

[3] Kassim, M.S, Yoosif, A.H. & Al-Khishali K.J.M., Investigation into flow interaction between jet and cross mainstream flows, *PhD thesis, University of Technology, Mechanical Engineering Department, Iraq, Baghdad*, 2007.

[4] Lakehal D., Theodoridis G.S. & Rodi, W., Three-dimensional flow and heat transfer calculations of film cooling at the leading edge of a symmetrical turbine blade model *Int. J. Heat and Fluid Flow*, 22, pp.113-122, 2001.

[5] Theodoridis, G.S., Lakehal, D., & Rodi, W., Three dimensional calculation of the flow field around a turbine Blade with film cooling injection near the leading edge, *Flow, Turbulence and Combustion*, **66**, pp. 57-83, 2001.

[6] Forest, A.E., White, A.J., Lai, C.C., Guo, S.M., Oldfield, M. L. G. & Lock, G. D., Experimentally aided development of a turbine heat transfer prediction method, *Int. J. Heat and Fluid Flow*, **25**, pp. 606-617, 2004, available online at www.ScienceDirect.com.

[7] Burdet, A. & Abhari, R.S., Three-dimensional flow prediction and Improvement of Holes Arrangement of a Film-Cooled Turbine Blade Using Feature Based Jet Model, *ASME J. Turbomachinery*, **129**, pp. 258-268, 2007.

[8] Garg, V. K. & Abhari, R. S., Comparson of predicted and experimental Nusselt number for a film cooled rotating blade, *Int. J. Heat Fluid Flow*, **18**, pp. 452-460, 1997.

[9] Azzi, A. & Jubran B.A., Influence of leading edge lateral injection angles on the film cooling effectiveness of a gas turbine blade, *Heat and Mass Transfer*, **40**, pp. 501-508, 2004, available online at www.ScienceDirect.com.

[10] Renze, P., Schroder, W. & Meinke, M., Large- eddy simulation of film cooling flow at density gradients, *Int. J. Heat Fluid Flow*, **29**, pp.18-34, 2008, available online at www.ScienceDirect.com.

[11] Eghlimi, A., Kouzoubov, A. & Fletcher, C.A.J., Anew RNG-Based tow equation model for predicting turbulent gas particles flow, *Int. Conference on CFD in Mineral & Metal Processing and Power Generation CSIRO, Sydney Australia*, 1997.

[12] Lakehal, D., Near wall modeling of turbulent convective heat transport in film cooling of turbine blades with the aid of direct numerical simulation data, *ASME J. Turbomachinery*, **124**, pp.458-498, 2002.

[13] Tao, Z., Yang, X., Ding, S., Xu, G., Wu, H., Deng, H.& Luo, X., Experimental Study of Rotation effect on Film Cooling over the Flat wall with a Single Hole, *Experimental Thermal and Fluid Science*, **32**, pp.1081-1089, 2008, available online at www.ScienceDirect.com.

[14] Guangchao, L., Huiren, Z. & Huiming, F., Influences of Hole Shape on Film Cooling Characteristics with $CO_2$ Injection, *Chinese J. Aeronautics*, **2**, pp.393-401, 2008, available online at www.ScienceDirect.com.
[15] Burdet, A., Abhari R.S & Martin, G.R., Modeling of Film Cooling – part II: Model for use in Three Dimensional Computational Fluid Dynamics, *ASME J. Turbomachinery*, **129**, pp.221-231. 2007.

 © 2011 WIT Press
www.witpress.com, ISSN 1743-355X (on-line)