Fifth South African Conference on Computational and Applied Mechanics
SACAM06
Cape Town, 16-18 January 2006

EXPERIMENTAL AND NUMERICAL STUDY OF NEAR BLEED HOLE HEAT TRANSFER ENHANCEMENT IN INTERNAL TURBINE BLADE COOLING CHANNELS

Scheepers G*, Morris RM, Visser JA, Roos TH and Snedden GC

Department of Mechanical and Aeronautical Engineering, University of Pretoria,
Pretoria, 0002, South Africa, * gersch@tuks.co.za

Key words: Transient Thin Film Measurement Technique, Gas Turbine Cooling

Abstract

An experimental and numerical investigation was conducted to determine local heat transfer enhancement near the entrance to a single film cooling hole. The purpose of the investigation was to determine and compare the effects of bleed hole suction ratio and extraction angle on the augmented heat transfer (near the hole entrance) to experimental results previously obtained with liquid crystal thermometry at lower temperature ratios. Steady state heat transfer results were acquired by using a transient measurement technique in an 80×actual size rectangular channel with thin film measuring gauges concentrated downstream of the bleed hole entrance. The Reynolds number, based on duct hydraulic diameter and bulk mean velocity, was kept constant at 2.5×10^4, while the hole suction ratio in terms of Reynolds number was varied from 0 to 5.

Nomenclature

c	Specific heat [j/gK]	V	Duct centerline velocity [m/s]
D	Hydraulic diameter [m]	V_{hole}	Average film cooling hole velocity [m/s]
h	Local heat transfer coefficient [W/m^{2}K]	T	Temperature [K]
h_o	Un-enhanced heat transfer coefficient [W/m^{2}K]	$\dot{q}$	Heat flux [W/m^2]
k	Thermal conductivity of coolant [W/mK]	t	Time [s]
m	Index for time	α	Thermal diffusivity [m^2/s]
Nu	Nusselt number, *hD/k*	ρ	Density [kg/m^3]
Re	Reynolds number, *ΔVD/:*	η	Weighting parameter
T	Temperature [K]		
T_{gas}	Temperature of coolant [K]		
T_{wall}	Temperature at substrate surface [K]		

1. Introduction

With the ever increasing demand for higher thermal efficiency, modern jet turbine blades are exposed to temperatures which comprehensively exceed the thermal capability of the blade material. For this reason sophisticated cooling techniques such as film cooling and augmented internal cooling have been employed in order to prolong the operating time of turbines under these extreme conditions. Now the main purpose of blade cooling channel bleed holes are to aid external film cooling, by bleeding air through the blade wall to create a protective film on the outer surface of the blade. This process causes enhanced heat transfer on the inner channel surface downstream of the bleed hole due to thermal boundary renewal, direct impingement, due to suction, and downstream vortex impingement.

This paper describes an experimental and numerical study of the heat transfer augmentation near the entrance to a gas turbine film cooling hole at different engine representative suction ratios (V_{hole}/V). For the experimental component the use of platinum thin film gauges and a transient testing technique were implemented to take measurements downstream of the film cooling hole entrance. Although this technique presents less detail than thermal crystals it however provides the capability of testing engine representative temperature ratios (T_{wall}/T_{gas}). The numerical component of the study makes use of a commercial CFD code to predict heat transfer and flow characteristics near the hole entrance.

2. Previous work

Previous work which relates directly to the current study are fairly limited, since most of the experimental and numerical work regarding cooling channels with bleed have focused mainly on the global relation between multiple holes and hole configurations in combination with various rib arrangements at a constant suction ratio.

The earliest work which can relate directly with this study was done by Ainsworth and Jones [1], who made use of a transient technique and thin film gauges. They studied the effect of mass removal from a circular duct with discreet bleed outlets and established that the augmentation increased proportionately with the bleed rate through the extraction hole. This was followed by Sparrow and Kemink [2] who investigated the influence of fluid withdrawal on turbulent heat transfer characteristics. They indicated that the upstream Reynolds number has a negligible effect on the near hole heat transfer augmentation and that the enhancement was ultimately affected by the suction ratio itself. This was confirmed by Byerley et al [3].

Some of the more recent work has been done by Shen et al. [4], who did an assessment of the effect of upstream holes and hole diameter. The results indicated that the heat transfer coefficient level, around the entrance to the holes, was insensitive to the number of holes upstream.

3. Experimental apparatus and method

Byerley [3] was the first to do a detailed study of near hole augmentation with the use of thermal crystals. The study conveyed detailed results, and also better understanding of the flow structure near and downstream of the hole. Unfortunately the experimental technique only allowed testing at fairly low wall to fluid temperature ratios. Therefore the decision was made to heat up the air which enters the measurement channel, thus transferring thermal energy towards the wall, resulting in a higher density fluid in the boundary layer. Byerley also wanted to investigate purely the influence of suction; therefore the channel sidewalls were widened to neglect any sidewall interaction with the bleed influenced flow field.

With the above mentioned aspects in mind, the rationale was to test within the same scope as Byerley, but with higher and multiple wall temperature ratios, and also to take into consideration the effect of sidewall flow field interaction. An overall layout of the main experimental channel is displayed in figure 1. Regulated compressed air flows through the main channel section, from where a percentage of air is bled off through a bleed hole. The remaining air flows out of the main channel through an orifice plate.

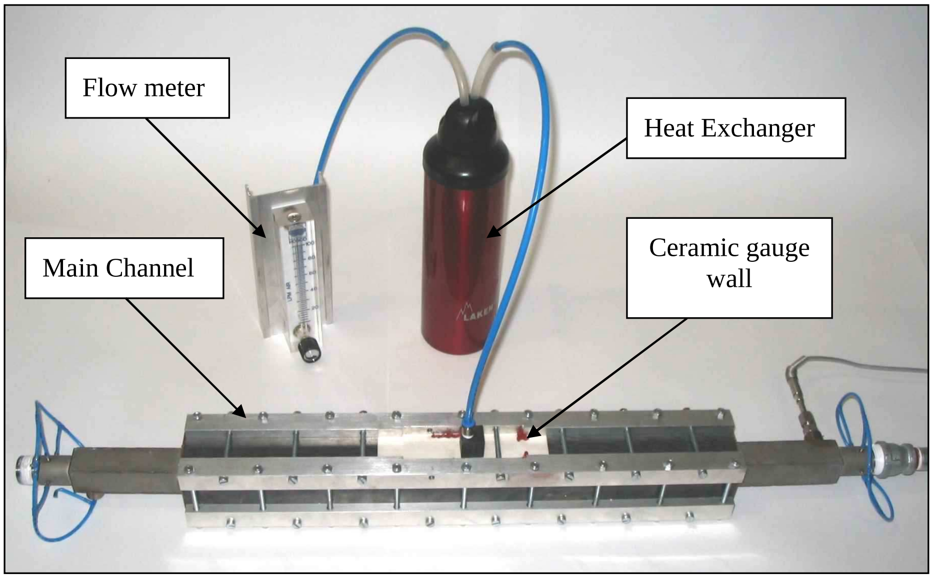

Figure 1 . Main channel section with heat exchanger and flow meter.

The bleed hole provides the capability of adjustable hole size and bleed angle, leading towards a heat exchanger and flow meter with variable back pressure to measure the amount of air bled off. The main channel wall is a combination of erthalyte and ceramic plates (Macor), with platinum thin film gauges fired on the ceramic wall near the hole entrance and at a section upstream of the bleed hole. The measurement point upstream (approximately 10 bleed hole diameters in length) is situated in an unenhanced region which will be used to calculate the enhancement factor. The enhancement factor is defined as the local value of heat transfer, downstream of the hole, divided by the local unenhanced value of heat transfer in the fully developed, turbulent channel flow upstream of the hole[3] (h/h_o). The channel section is rectangular and approximately 80 times actual size, with a duct-to-hole hydraulic diameter ratio of 5, falling within the range of actual blades. Two bleed angles are tested: The first connecting perpendicular with the main channel, thus at 90°, while the second hole angle connected at 150°.

Even though the 150° bleed angle may not be frequently employed, its influence, however, differs enough that results obtained for it should be reported here. The main channel flow is kept at a Reynolds number of 2.5×10^4 and the hole suction ratio is varied from 0 to 5, which is representative of actual engine flow characteristics. Air at 15 ºC was used as test gas with the initial wall temperature at 100 ºC and 130 ºC, giving a temperature ratio of 1.3 and 1.4. The above mentioned flow characteristics and bleed angles were chosen to compare the current results with that obtained by Byerley et al. [3].

Once the mainstream and bleed flow regulators were adjusted to the correct flow rate, the model is heated up in an oven to a uniform temperature. After the required wall temperature has been reached, data acquisitioning is initialized. The solenoid inlet valve is then opened, initiating cold air flow through the heated channel section, thus introducing a step in heat flux. An experimental uncertainty analysis, based upon the technique used by Kline and McClintock [5] for single sample experiments, estimated a Nusselt number uncertainty of less than 7 %.

4. Data reduction

To determine the heat flux a numerical data reduction technique is used which was previously validated against the original analog technique by Denos et al. [6]. With this technique the wall heat flux is obtained by solving equation 1, the one-dimensional unsteady conduction equation, in the successive substrate by means of the temperature histories measured at the surface and at a given depth as boundary condition.

$$\frac{\partial T(x,t)}{\partial x^2} = \frac{1}{\alpha(x)} \frac{\partial T(x,t)}{\partial t} \tag{1}$$

This provides the capability of determining the temperature distribution history within the substrate itself, which can then be used to determine the surface heat flux by means of equation 2.

$$\dot{q}_{wall}(t) = -k\left(\frac{\partial T}{\partial x}\right)_{x=0} \tag{2}$$

The initial temperature distribution within the substrate can be assumed to be uniform since the substrate is under thermal equilibrium before blow down. To solve equation 1 an implicit Crank-Nicholson scheme, exhibited by equation 3, was implemented (η = ½).

$$\frac{T_i^{n+1} - T_i^n}{\Delta t} = \frac{\left[\eta\left(T_{i+1}^{n+1} - 2T_i^{n+1} + T_{i-1}^{n+1}\right) + (1-\eta)\left(T_{i+1}^n - 2T_i^n + T_{i-1}^n\right)\right]}{\Delta x^2} \tag{3}$$

By expressing the unknowns at time *n+1* as a function of the computed solution at time *n*, a tri-diagonal system is obtained which can easily be solved to attain the distribution of temperature in the substrate at time step *n+1*. The temporal domain displayed in equation 3 is obviously the sampling rate while the spatial domain is the discretized substrate, with nodes clustered towards the surface boundary where the highest thermal gradient is expected. This data reduction technique has been implemented in a program and validated against an analytical test case with constant heat flux.

Whatever technique is chosen to derive heat flux from surface temperature measurements, the knowledge of the thermal product ($\sqrt{\rho c k}$) is of utmost importance. As mention by Denos et al. [6] and Dunn [7] the thermal product of the Macor ceramic had to be measured after the thin film gauges were fired on, since the gauge firing process may cause diffusion of platinum into the substrate which alters the ceramic's original thermal characteristics. For this a technique suggested by Dunn [7] was implemented with which the gauge is subjected to a pulse of known constant heat flux. The surface temperature can be measured from which the thermal product is then determined.

5. Numerical model

The computational domain of the CFD analysis was representative of the experimental geometry and its pressure and mass flow boundary conditions were implemented. The geometric configuration of the model and its boundary conditions are displayed in figure 2 and the near hole mesh configuration in figure 3.

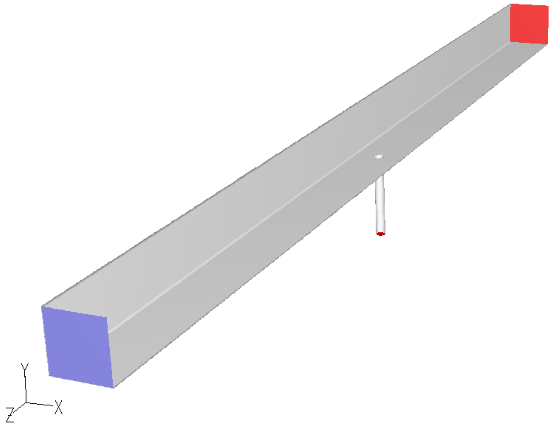

Figure 2. Geometrical representation of numerical domain.

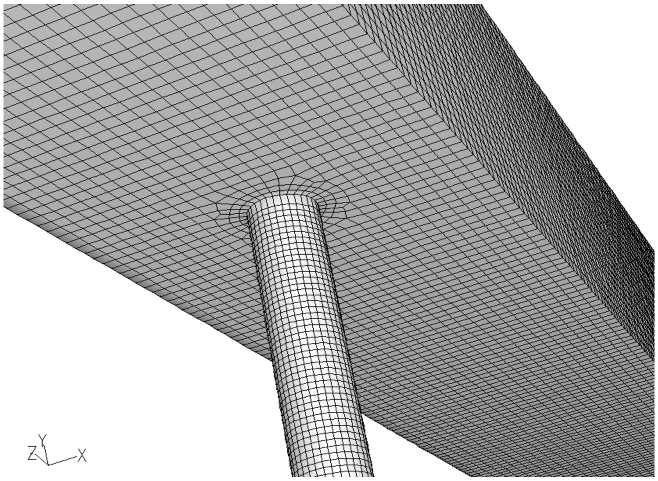

Figure 3. Near hole mesh configuration.

The standard k-T turbulence model was implemented. A preliminary mesh independence study of the given channel was done where uncertainties like y^+ sensitivity were adhered to. The following boundary conditions were implemented:

- Mass flow inlet.
- Pressure outlet applied on the bleed and channel outlet.
- Constant wall temperature

The fluid was modelled as compressible, although the maximum Mach number value in the channel was just below 0.3.

6. Results

To validate the experimental channel, initial tests were done without suction and the acquired heat transfer values were compared with an empirically calculated Nusselt value for the given Reynolds number of 2.5×10^4. The experimentally obtained Nusselt number was within 5 % of the empirically estimated value of 59 [8], within the 7 % error estimate.

The results displayed in figure 4 are a comparison between the current experimental results and the results obtained by Byerley et al. [3] for a 90° suction hole at a suction ratio of 5 and temperature ratio of 1.3. The original detailed results of Byerley were represented as contour plots. To compare the two data sets, Byerley's contour enhancement distributions were processed by taking line averages, having the same width as obtained by the thin film gauges used in the current study. Figure 4 indicates enhancement factors downstream of the suction hole, with the x-axis representing the distance downstream of the hole divided by the bleed hole diameter. As can be seen the results' profile compare fairly well downstream although near the hole an increase of 14 % was noted.

Figure 5 displays a comparison between the CFD solution and the experimental results in this study. As can be seen heat transfer near the hole is predicted fairly accurately, although divergence further downstream can clearly be seen. This can be attributed to the fact that downstream enhancement, caused by vortex downwash, seems to be fairly poorly predicted by the current mesh.

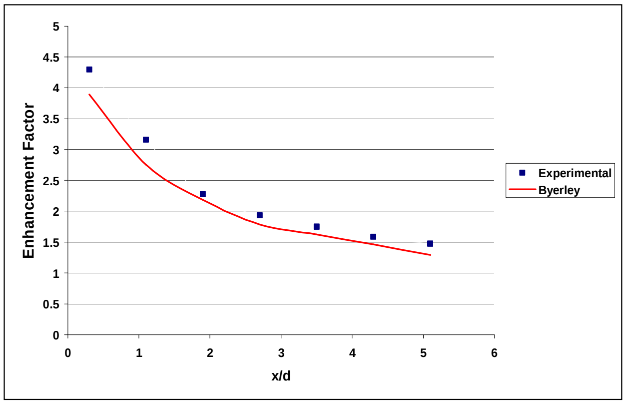

Figure 4. Enhancement factors downstream of the bleed hole.

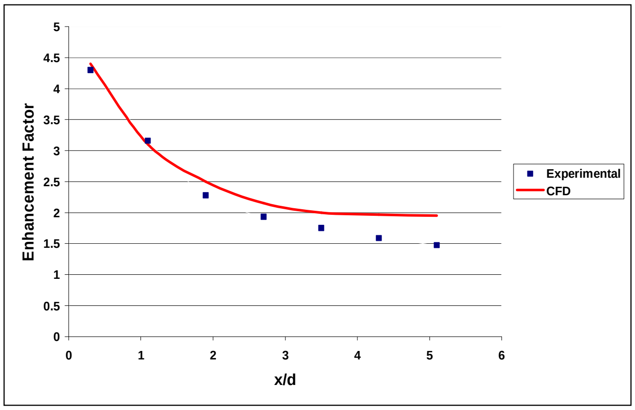

Figure 5. Comparison between experimental and numerical enhancement factors.

7. Conclusion

Heat transfer enhancement near a film cooling hole was examined with experimental and numerical methods. Initial validation runs on the test bed provided results which were within 5 % of the empirically predicted values for fully developed flow through a square channel. Various bleed hole angles, wall temperatures and bleed-hole-channel ratios were tested at suction ratios varying from 0 to 5. The experimental enhancement profile downstream of the hole showed a good comparison between the previously obtained results at lower wall temperature ratios. An increase in enhancement, in the region of 10-15 % was noted near the suction hole edge.

Enhancement predicted near the suction hole with CFD compared well with the experimental results at higher suction ratios, although divergence further downstream was noted. Initial results indicate that the vortex formation downstream, induced by suction, is poorly predicted by the current mesh and turbulence model.

References

1. Ainsworth, R.W, and Jones, T.V., Measurement of heat transfer in circular, rectangular and triangular ducts representing typical turbine blade internal cooling passages using transient techniques, *ASME Paper* 79-GT-40, 1979.
2. Sparrow, E.M., and Kemink, R.G., Heat transfer downstream of a fluid withdrawal branch in a tube, *ASME Journal of Heat Transfer*, 101, 1979.
3. Byerley, A.R., Jones, T.V. and Ireland, P.T., Internal cooling passage heat transfer near the entrance to a film cooling hole: Experimental and computational Results, *International Gas Turbine and Aerospace Congress and Exposition*, Cologne Germany, June 1-4, 1992.
4. Shen, J.R., Wang, Z., Ireland, P.T., Jones, T.V. and Byerley, A.R., Heat transfer enhancement within a turbine blade cooling passage using ribs and combinations of ribs with film Cooling holes, *ASME Journal of Turbomachinery*, 118, 428-433, 1996.
5. Kline, S.J., and McClintock, F.A., Describing uncertainties in single sample experiments, *Mech. Eng. (ASME)*, 75,3-8, 1953.
6. Denos, R., Billiard, N., Iliopoulou, V., and Ferrara, F., Data reduction and thermal product determination for single and multi-layered substrates thin film gauges, *16th Symposium on Measuring Techniques in Transonic and Supersonic Flow in Cascades and Turbomachines,* 2002.
7. Dunn, M.G., *Measurement Techniques: Thin Film Gauges,* Von Karman Institute for Fluid Dynamics, Lecture Series 1995-01, 1995.
8. Mills, A.F., *Basic Heat And Mass Transfer*, IRWIN, 1995.