Case study on Turbine Blade Internal Cooling

T. Ravi Teja[1], S. Krishna Chaitanya[2]

[1]*Assistant professor, Mechanical Dept, Narasaraopeta engineering college, Andhra Pradesh, India,*
[2]*Associate professor, Mechanical Dept,* St. Mary's Group of institutions, *Andhra Pradesh, India,*

Abstract

Gas turbines are used extensively for aircraft propulsion, land-based power generation, and industrial applications. The turbine inlet temperatures are far above the permissible metal temperatures. Therefore, there is a need to cool the blades for safe operation. Modern developments in turbine cooling technology play a critical role in increasing the thermal efficiency and power output of advanced gas turbine designs. Turbine blades and vanes are cooled internally and externally. This paper focuses on turbine blade internal cooling. Internal cooling is typically achieved by passing the coolant through several rib-enhanced serpentine passages inside the blades. Impinging jets and pin fins are also used for internal cooling. In the past 10 years there has been considerable progress in turbine blade internal cooling research and this paper is limited to reviewing a few selected publications to reflect recent developments in this area. In particular, this paper focuses on the effects of channel inlet geometry, sharp 180o turning, and channel cross-section aspect ratio on the coolant passages heat transfer at high rotation number conditions. Rotation effects on the blade leading-edge triangular-shaped channel and trailing-edge wedge shaped channel with coolant ejection are included

1. Introduction

The forced cooling of gas turbine blades began with crude designs that consisted of circular radial holes that allowed for forced air to convert heat from the blade walls. These simple designs were the first attempts to forcefully cool the turbine blade. As basic as these designs were, the benefits of increasing power and efficiency were quickly realized by the ability to increase the turbine entry temperature. Decades have passed since, and through research, revolutionary steps have been made to allow for ever increasing turbine entry temperatures.

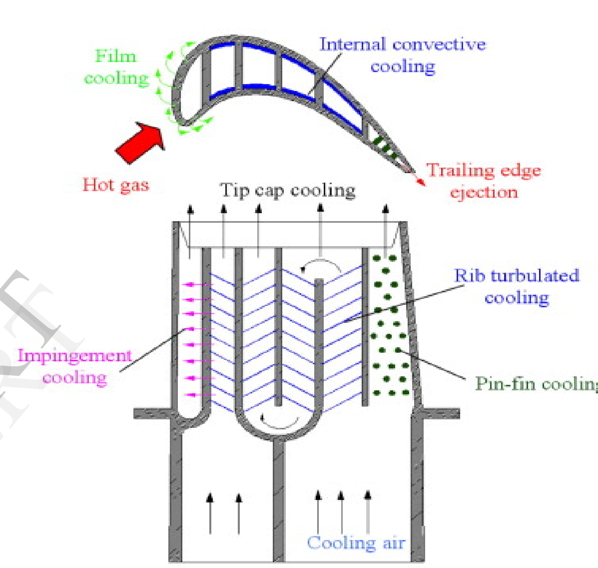

Fig. 1. Internal cooling techniques for gas turbine blades.

Advanced gas turbine blade designs now employ a variety of heat transfer enhancement methods and techniques as shown in Fig. 1.

2. Modeling Internal Cooling Passages

The gas turbine blade mid-region is cooled convectively with a compressor-bled air passing through the complex shaped internal cooling channels. These channels are specifically designed to fit the blade profile and have irregular cross sections. Since the design of these channels varies from blade to blade, and increased complexities of the flow field are introduced by irregular cross-sectional shapes, researchers have mostly used square and rectangular channels as models in the study of heat transfer. The square and rectangular channels are categorized by aspect ratio as seen in Fig. 2. At this point, it is important to make a comment in regard to aspect ratio definition. There is not a standard that is consistently used in publications for the definition of aspect ratio.

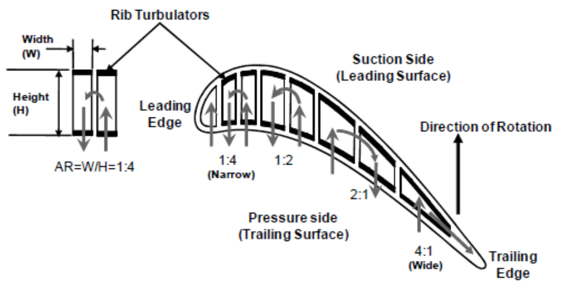

Fig. 2. The various aspect ratios of gas turbine blade internal cooling channels.

The internal cooling channels near the blade leading edge have been modeled as narrow rectangular channels with AR = 1:4 and 1:2. The cross section of the cooling channels changes along the cord length of the blade due to the blade profile. In the middle of the blade, the channels become squarer in shape, and studies have modeled this region with square cross sections. Square channels also serve to provide for a more fundamental study by eliminating a parameter since all sides of the channel are of the same dimension. Towards the trailing edge, the channels have wider aspect ratios of AR = 2:1 and 4:1. An experimental study of the effects of the buoyancy parameter in various aspect ratio channels was performed by Fuel at all. The study considered five different aspect ratio channels (AR = 1:4, 1:2, 1:1, 2:1, and 4:1) with a fully developed flow inlet condition. The results showed that the overall levels of heat transfer enhancement for all the ribbed channels were comparable. However, significant inferences arose in the pressure losses incurred in each of the channels. The 1:4 channel incurred the lowest pressure penalty; therefore, the thermal performance (TP) of the 1:4 channel was superior to the 1:2, 1:1, and 2:1 channels. It is worth noting that the thermal performance takes into account the pressure penalty (*f*/*f*0) and the heat transfer enhancement, and for a constant pumping power,

TP = (Nu/Nu0)/(*f*/*f*0)1/3.

2.1. Rotational Effects on Internal Passage Flow and Heat Transfer

The flowing fluid inside of rotating turbine blade cooling channels is subjected to an inertial force, rotation induced centrifugal force, and the Coriolis force. The inertial force is coincident with the mainstream flow direction, while the centrifugal force continuously acts in the radially outward sense. The Coriolis force acts perpendicular to the mainstream flow direction. Figure 3 shows that with a radically outward flow stream (first pass), the Coriolis force pushes the core fluid mass towards the trailing wall.

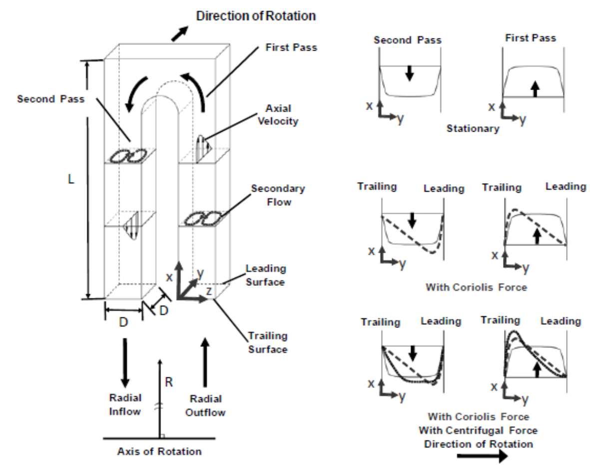

Fig. 3. Coriolis and centrifugal force effects on rotating channel flow field.

with the trailing wall. Due to the continuous replenishment of new fluid, the crossstream flow will split, reverse its direction, and travel along the side walls of the channel until it once again is turned 180o back towards the trialing wall. This continuous cross-stream fluid motion results in a pair of cross-stream vortices that, on average, cause the mass of coolant near the trailing wall to be greater than that near the leading wall. Due to mass continuity, the local axial velocity near the trailing wall is increased. Conversely, the local axial velocity near the leading wall decreases. Furthermore, the hot walls of the blade create a temperature gradient throughout the coolant, resulting in a variation of fluid density. The fluid mass near the trailing wall is of lower temperature since the coolant bulk flow velocity is skewed as previously mentioned. The density of the coolant will tend to be greater near the trailing wall. The centrifugal force acts strongly near the trailing wall since the fluid is heavier and accelerates the heavier fluid towards the tip of the channel. The result is a further increase in the local axial velocity near the trailing wall and flow stabilization near the leading wall.

It is worthwhile then to develop nondimensional parameters that may be used to correlate rotating effects to heat transfer. The rotation number (Ro) has been widely accepted to establish the strength of rotation by considering the relative strength of the Coriolis force compared to the bulk inertial force. As such, the rotation number is defined as Ro = Ω*D*h/*V*.

The buoyancy parameter (Bo) is useful to include the effects of density variation (centrifugal effects) and is defined as the ratio of the Grashof number to the square of the Reynolds number; both of which are based on the channel hydraulic diameter. Thus Bo = $(\Delta\rho / \rho)(Ro2)(R/Dh)$. Typical rotation numbers for aircraft engines are near 0.25 with Reynolds numbers in the range of up to 50,000. One method to achieve conditions similar to a real gas turbine engine in the laboratory is to use air at high pressures. As the pressure of the air increases so will the density. For a fixed Reynolds number, dynamic viscosity, and hydraulic diameter, an increase in density will proportionately decrease the bulk velocity. A lower bulk velocity will in turn increase the rotation number since the rotation number is the ratio of the Coriolis force to bulk inertial force. Increasing the range of the rotation number and buoyancy parameter is very important since gas turbine engineers can utilize these parameters in their analysis of heat transfer under rotating
conditions.

Liou et al. [1] investigated the heat transfer in a rectangular channel (AR = 1:2) with 45o angled ribs under high rotation numbers of up to 2.0 with a corresponding Reynolds number of 5000. Zhou et al. [2] and Zhou and Acharya [26] studied a 4:1 aspect ratio channel with a rotation number of 0.6 at a Reynolds number of 10,000. Chang et al. [3] experimentally studied a square duct with ribs at a rotation number of 1.8 with a corresponding Reynolds number of 7500. increased the range of the rotation number by a factor of 4 for the AR = 2:1 channel. Liu et al. [4] and Huh et al. [5] studied heat transfer in a 1:4 aspect ratio channel. They conducted experiments with air at a pressure of approximately 620 kPa absolute. In their studies, a rotation number of 0.65 was achieved at Reynolds numbers of 10,000. Figure 4 shows that heat transfer on the trailing surface with radially outward flow does indeed increase under rotating conditions due to the flow phenomena previously described. Both the smooth and ribbed channel experience degradation in heat transfer on the leading surface. Rotation reduces the heat transfer in the ribbed channel by a very significant 50%. However, due to buoyancy effects, the leading surface heat transfer trends reverse after a critical rotation number is reached. Physically this implies that the Coriolis force and centrifugal force have skewed the bulk flow so severally towards the trailing surface that reverse flow is occurring on the leading wall.With radially inward flow, the heat transfer in the smooth channel shows the expected behavior on the leading wall. Surprisingly however, due to the aspect ratio of the channel, the heat transfer on the trailing wall also increases.

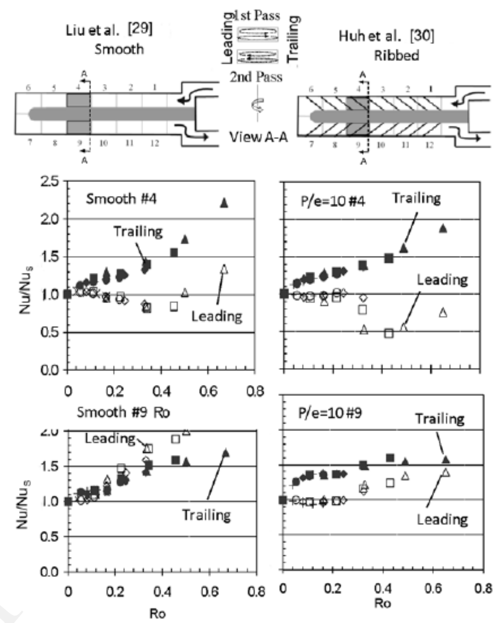

Fig. 4. Rotation number effects on heat transfer in smooth and ribbed 1:4 AR channels.

3. COOLING THE TURBINE BLADE LEADING EDGE

The mainstream flow of hot gases exits the combustor and is directed through the nozzle guide vanes and then impinges on the leading edge of the first stage rotor blades. The temperature of these gases may exceed 1500oC, which is clearly well in excess of allowable metal temperatures. These high heat loads require clever cooling schemes to be used at the leading edge of the blade. A commonly employed internal cooling technique is to use jet impingement on the internal surface of the cooling passage at the blade leading edge. Air from the compressor of the gas turbine system is bled off and passed through the internal cooling channels of the gas turbine blade. The internal channel at the blade leading edge utilizes this cooling air for impingement on the blade leading edge inner surface. Selectively placed jet-holes (or nozzles) are located on the leading edge pass inner wall that divides the leading edge channel from other serpentine passages. Flow from the neighboring internal passage is forced through these jet-holes so that impingement occurs on the leading edge inner surface of the blade. The design application of this type of cooling scheme must consider several different factors; namely, the shape of the jet nozzle, the layout of the jet holes, the shape of confinement chambers, the shape of the target surface, the jet-to target spacing, and cross-flow effects are just a few.

3.1 Film Cooling Hole Effects on Impingement

Since the leading edge of the gas turbine blade incorporates a showerhead film cooling design, studies have also included film-cooling holes on the target plate. When an initial cross flow is present, the jet impingement on an effusion target plate has been shown to provide higher heat transfer than impingement on a solid plate alone. Rhee et al. [6] explain that the cross flow effect is reduced due to the film cooling holes. Studies by Cho et al. [7] and Taslim and Khanicheh [8] also have shown that the heat transfer can be significantly increased by including the film cooling holes on the target plate.

3.2 Mach Number Effects on Impingement

Most studies involving impingement heat transfer have varied the jet Reynolds number with the Mach number. Brevet et al. [9] considered the effects of changing Mach number at a constant Reynolds number for a single jet. The highest Mach number in the study was 0.69. From their results, they concluded that at low Mach numbers (<0.2) the influence on heat transfer, by the Mach number, could be neglected. However, at higher Mach numbers, compressibility effects must be considered and it was shown that heat transfer could be increased on the target surface. More recently, Park et al. [9] considered the separate effects of Mach number and Reynolds number of a jet array. They showed that for higher Reynolds numbers at high Mach numbers, previous correlations were inadequate. A new correlation was proposed for the extended range of Reynolds and Mach numbers.

4. TRAILING EDGE HEAT TRANSFER

Effective cooling of the trailing edge is required to prevent burnout. The internal cooling passage has been represented with wide aspect rectangular channels [3] However, the cross-sectional shape is best represented with a wedge or trapezoid. To enhance heat transfer in this region of the blade, the leading and trailing surfaces are roughened with ribs or pin-fins. Further protection is provided with coolant ejection from the narrow portion of the channel. Hwang and Lu [5] studied the effects of lateral-flow ejection, pin shapes, and flow Reynolds number in a trapezoidal duct. Carcasci et al. [6] also investigated heat transfer and pressure drop inside wedgeshaped cooling channels with pedestals (long ribs) and pin-fins. Cunha and Chyu [7] used a liquid crystal technique to obtain heat transfer data inside the wedge-shaped channel with and without discharge through slots or holes. Wright and Gohardani [8] used a traditional copper plate method to investigate the heat transfer in a rectangular (AR = 3:1) and trapezoidal channel. The study considered fully developed and developing flow conditions. The experiments were conducted with and without coolant ejection. Other studies [8] have also considered different aspects of heat transfer and fluid in the trapezoidal/wedge shaped channel.

4.1 Rotating Effects on Trailing Edge Cooling Channels

The aforementioned studies on trapezoidal and wedge shaped cooling passages were all conducted under stationary conditions. However, the additional effects of Coriolis induced secondary flows and centrifugal driven buoyancy alter the heat transfer characteristics. Chang et al. [8] studied heat transfer in rib roughened trapezoidal duct with bleed holes. Wright el al. [8] considered heat transfer in a trailing edge cooling passage with smooth walls. The channel was placed at an angle of 135^{o} relative to the direction of rotation. Wright et al. [8] showed that the data in a smooth wedge-shaped channel can be correlated as a function of buoyancy parameter. Liu et al. [10] extended the study to include slot ejection. The rotation number in their study was 1.0 based on inlet velocity. Due to ejection from the channel, heat transfer analysis utilized a local rotation number, Reynolds number, and buoyancy parameter. Figure 5 presents their streamwise and spanwise averaged Nusselt number ratios. The data is correlated with the buoyancy parameter by a power law function. Over the range of Reynolds numbers and rotational speeds tested, it is seen that each of the three surfaces follows very distinct trends. The data for each surface collapse on to common curve. Most notably, for all three surfaces, the Nusselt number ratios increase as the rotation number increases.

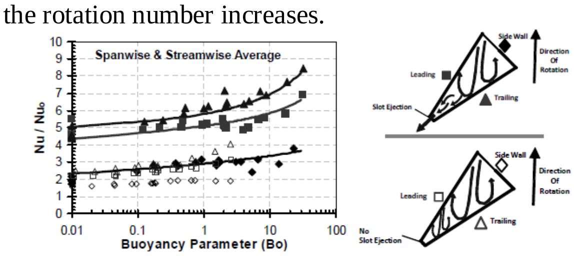

Fig. 5. Rotation number and buoyancy parameter effects on heat transfer in the trapezoidal channel with and without ejection.

5. Conclusion

Over the past decade, research on gas turbine blade heat transfer has progressed considerably. Many aspects of internal cooling have been considered. Advanced jet impingement studies have coupled the benefits of high heat transfer resulting for impinging jets with surface enhancement techniques. The additional complications associated with showerhead film cooling ejection have also been taken into account. However, the full realization of the effects of rotation on leading edge jet impingement needs progression as is evidenced by limited literature. Heat transfer studies on the internal serpentine cooling passages found in the mid-portions of the blade have evolved from experiments at low rotation numbers to urbane experimental methods that provide more realistic engine conditions. Yet still, progression is needed to provide more information on even higher buoyancy parameter ranges at higher Reynolds numbers. This information will be of great interest to the land based gas turbine designer. Other points of focus involve heat transfer inside cooling channels with bleed hole effects, tip internal surface enhancement methods along with ejection, all under rotating conditions. With advancements being made on alternative fuel sources for turbines (i.e., hydrogen), a step change in the capability of blades to handle higher heating loads is a must. Compound and new cooling concepts need to be developed and explored, such as heat pipe and micro-channel applications for blade tip, leading, and trailing edge cooling. Fundamental studies need to consider the effects of rotation on these new cooling concepts. Development of this technology will ensure that the blade design is not the limiting factor for increased efficiency and the move to other fuel sources.

6. References

1. Han, J. C., Dutta, S., and Ekkad, S. V. *Gas Turbine Heat Transfer and Cooling Technology*, Taylor and Francis, New York, 2000.
2. Han, J. C. and Wright, L. M. Enhanced internal cooling of turbine blades and vanes. In: *The Gas Turbine Handbook*, U.S. Department of Energy, Office of Fossil Energy, National Energy Technology Laboratory (NETL), 2006, pp. 321– 354.
3. Fu, W. F., Wright, L. M., and Han, J. C. Heat transfer in two-pass rotating rectangular channels (AR = 1:2 and AR = 1:4) with 45o angled rib tabulators, ASME Paper No. GT2004-53261, 2004.
4. Fu, L. W., Wright, L. M., and Han, J. C. Buoyancy effects on heat transfer in five different aspect-ratio rectangular channels with smooth walls and 45 degree ribbed walls, *J. Heat Transfer*, 2006, Vol. 128, No. 11, pp. 1130– 1141.
5. Bailey, J. C. and Bunker, R. S. Heat transfer and friction in channels with very high blockage 45-deg staggered turbulators, ASME Paper No. GT2003-38611, 2003.
6. Casarsa, L., Cakan, M., and Arts, T. Characterization of the velocity and heat transfer fields in an internal cooling channel with high blockage ratio, ASME Paper No. GT2002-30207, 2002.
7. Kiml, R., Mochizuki, S., and Murata, A. Effects of rib arrangements on heat transfer and flow behavior in a rectangular rib-roughened passage: application to cooling of gas turbine blade trailing edge, *J. Heat Transfer*, 2001, Vol. 123, pp.675–681.
8. Gao, X. and Suden, B. Heat transfer and pressure drop measurements in ribroughened rectangular ducts, *Exp. Thermal Fluid Sci*., 2001, Vol. 24, pp. 25–34.
9. Rhee, D. H., Lee, D. H., Cho, H. H., and Moon, H. K. Effects of duct aspect ratios on heat/mass transfer with discrete V-shaped ribs, ASME Paper No.GT2003-38622, 2003.
10. Cho, H. H., Wu, S. J., and Kwon, H. J. Local heat/mass transfer measurements in a rectangular duct with discrete ribs, *J. Turbomachinery*, 2000, Vol. 122, pp. 579–586.