CHAPTER 7

Innovative gas turbine cooling techniques

R.S. Bunker
GE Global Research Center, USA.

Abstract

Advanced heat transfer and cooling techniques form one of the major pillars supporting the continuing development of high efficiency, high power output gas turbine engines. Conventional gas turbine thermal management technology is composed of five main elements including internal convective cooling, external surface film cooling, materials selection, thermal-mechanical design at the component and system levels, and selection and/or pre-treatment of the coolant fluid. The present summary will examine specific cooling technologies representing cutting edge, innovative methods expected to further enhance the aero-thermal-mechanical performance of turbine engines. The techniques discussed will include forced convective cooling with unconventional turbulators and concavity surface arrays, swirl-cooling chambers, latticework cooling networks, augmentations of impingement heat transfer, synergistic approaches using mesh networks, and film cooling.

1 Introduction

The technology of cooling gas turbine components, primarily via internal convective flows of single-phase gases and external surface film cooling with air, has developed over the years into very complex geometries involving many differing surfaces, architectures, and fluid-surface interactions. The fundamental aim of this technology area is to obtain the highest overall cooling effectiveness with the lowest possible penalty on the thermodynamic cycle performance. As a thermodynamic Brayton cycle, the efficiency of the gas turbine engine can be raised substantially by increasing the firing temperature of the turbine. Modern gas turbine systems are fired at temperatures far in excess of the material melting temperature limits. This is made possible by the aggressive cooling of the hot gas path components using a portion of the compressor discharge air, as depicted in Fig. 1. The use of 15–25%

WIT Transactions on State of the Art in Science and Engineering, Vol 42, © 2008 WIT Press
www.witpress.com, ISSN 1755-8336 (on-line)
doi:10.2495/978-1-84564-062-0/07

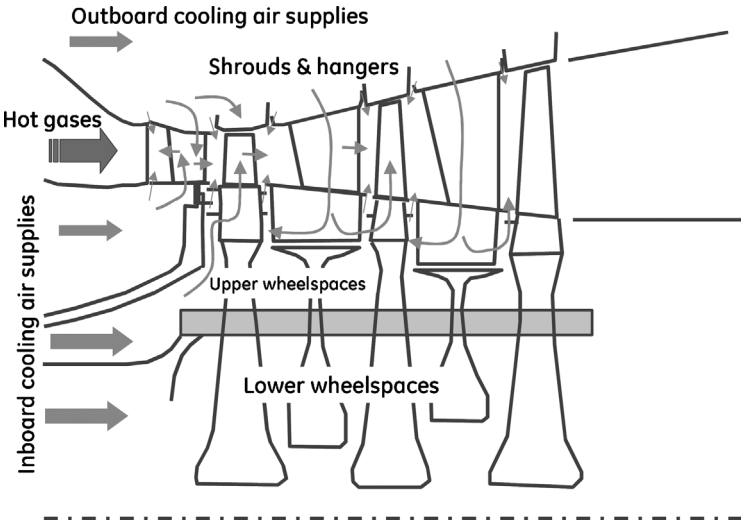

Figure 1: Schematic of turbine with cooling flows.

of this compressed air to cool the high-pressure portions of the turbine presents a severe penalty on the thermodynamic efficiency unless the firing temperature is sufficiently high for the gains to outweigh the losses. In all properly operating cooled turbine systems, the efficiency gain is significant enough to justify the added complexity and cost of the cooling technologies employed. Actively or passively cooled regions in power generating gas turbines include the stationary vanes or nozzles and the rotating blades or buckets of the high-pressure stages, the shrouds bounding the rotating blades, and the combustor liners and flame holding segments (fuel nozzles, splash plates). All such engines additionally cool the interfaces and secondary flow regions around the immediate hot gas path. A more detailed schematic of the cooling for an aircraft engine combustor and turbine first stage, which may be thought of also as an aero-derivative power turbine, is shown in Fig. 2.

Cooling technology, as applied to gas turbine components is composed of five main elements, (1) internal convective cooling, (2) external surface film cooling, (3) materials selection, (4) thermal-mechanical design, and (5) selection and/or conditioning of the coolant fluid. Cooled turbine components are merely highly specialized and complex heat exchangers that release the cold side fluid in a controlled fashion to maximize work extraction. The enhancement of internal convective flow surfaces for the augmentation of heat transfer was initially improved some 25–30 years ago through the introduction of rib-rougheners or turbulators, and also pin-banks or pin-fins. Figure 3 shows an example schematic of a blade cooling circuit that utilizes many turbulated passages, a pin bank in the trailing edge, and impingement in the leading edge (coolant is released via film holes, tip holes, and trailing edge). These surface enhancement methods continue to play a large role in today's turbine cooling designs. Film cooling is the practice of bleeding

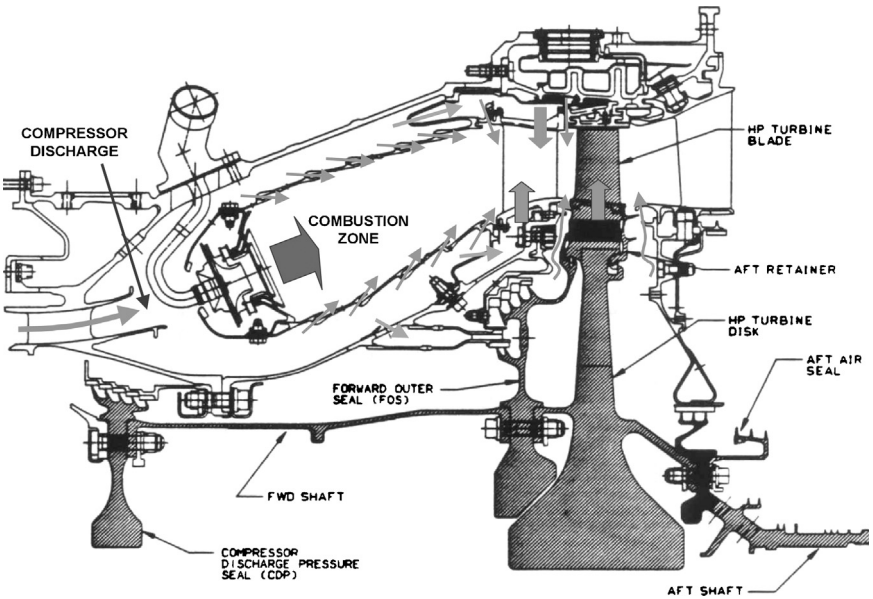

Figure 2: Cooling flows for a combustor and high-pressure turbine.

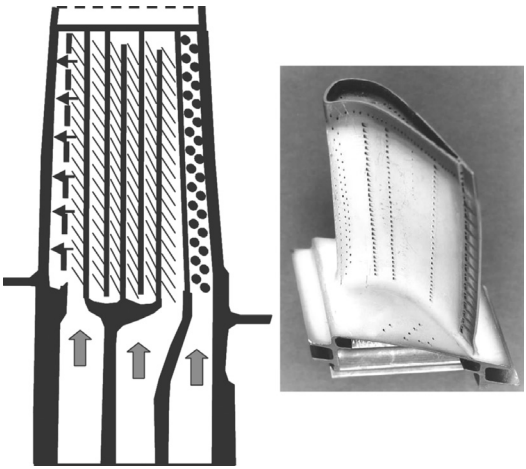

Figure 3: Schematic of a blade cooling circuit.

internal cooling flows onto the exterior skin of the components to provide a heat flux reducing cooling layer, as shown by the many holes placed over the airfoil in Fig. 3. Film cooling is intimately tied to the internal cooling technique used in that the local internal flow details will influence the flow characteristics of the film jets injected on the surface.

Several characteristics of gas turbine cooling are worth noting prior to describing any specific technologies. Almost all highly cooled regions of the high-pressure turbine components involve the use of turbulent convective flows and heat transfer. Very few, if any, cooling flows within the primary hot section are laminar or transitional. Moreover, the typical range of Reynolds numbers for cooling techniques, using traditional characteristic lengths and velocities, is from 10,000 to 60,000. This is true of both stationary and rotating components. The enhancement of heat transfer coefficients for turbine cooling makes full use of the turbulent flow nature by seeking to generate mixing mechanisms in the coolant flows that actively exchange cooler fluid for the heated fluid near the walls. These mechanisms include shear layers, boundary layer disruption, and vortex generation. In a marked difference from conventional heat exchangers, most turbine cooling means do not rely on an increase in cooling surface area, since the available surface area to volume ratios are very small. Surface area increases are beneficial, but are not the primary objective of enhancements. The use of various enhancement techniques typically results in at least 50% and as much as 300% increase in local heat transfer coefficients over that associated with fully developed turbulent flow in a smooth duct.

2 Turbulated channel cooling

One of the most common means for enhancing heat transfer coefficients within internal cooling passages, and especially the serpentine passages of many turbine blade designs, is the use of turbulators, also known as rib rougheners. Turbulators in the form of trip strips placed transverse to the bulk flow direction were one of the first improvements made to the cooling of blades, and hence many investigations have been made into the heat transfer and friction characteristics. Basic transverse turbulator research, which resulted in widely used data and correlations, was performed by Webb *et al*. [1], Burggraf [2], and Han *et al*. [3, 4]. With the advancements in materials and manufacturing technologies of the last decade, a drastically larger realm of surface enhancement techniques has become cost effective for use in the cooling of turbine airfoils. Turbulators may now be of varying shapes, orientations, segmentations, and sizes, essentially providing a continuous spectrum of possible geometries for achieving flow-surface interactions that serve to enhance local and global heat transfer coefficients. The bulk of researchers concentrated on turbulators of relative height $e/D < 0.2$, but greater than that relative roughness typically associated with uniform surface roughness. The general findings of all such research has been that surface averaged heat transfer coefficients within stationary, turbulated passages may be enhanced by factors from 1.8 to 2.8, while the friction factors or required pumping power are increased by factors of 3–10. Many parameters have been investigated in turbulated passages (angle to the bulk flow, P/e, e/D, shaping, taper, etc.), and many more in serpentine circuits, but the range of effects has remained much the same over the years.

Improvements over the use of transverse turbulators within channels have in the bulk of the literature focused on angled turbulators, segmented turbulators, and many other combinations of these in such forms as chevrons and segmented discrete

strips. Many researchers have noted significant heat transfer coefficient enhancements for surfaces with upstream pointing chevrons and staggered segments of chevrons. For example, Han and Zhang [5] investigated four cases of various segmented chevron geometry, as well as chevrons, segmented angled strips, and conventional turbulators in a simple duct flow. They found heat transfer coefficient enhancements as high as 3–4 compared to smooth surfaces for Re from 15,000 to 80,000. The friction coefficient enhancements were also generally higher than those for more conventional turbulators. Similar results at lower Re conditions were obtained by Taslim *et al.* [6] and Kiml *et al.* [7]. Cho *et al.* [8, 9] tested variations on segmented angled or transverse turbulators, finding average mass transfer coefficient augmentations in the range of 2.5–3 for Re up to about 70,000. While beneficial effects of turbulator angle were maintained, the number of segmentations did not show a consistent improvement trend. Chyu and Natarajan [10] presented detailed mass transfer data for transverse segmented turbulators where the segments were aligned or staggered. Alignment of the segments actually lowered the average mass transfer. While the segmentation gaps served to increase vertical mixing locally, these also disrupted the benefits of reattaching flows. Staggered segments, on the other hand, did produce higher average mass transfer coefficients by providing a more periodic nature to the disturbances. All of the foregoing studies examined enhancement features inside square or nearly square channels, so the features were not repeated laterally as would be the case between parallel plates or in ducts of higher aspect ratio.

Recent research into further turbulated channel or turbulated surface cooling is taking the direction of even more complex geometries such as perforated turbulators, turbulators detached from the wall, and turbulators with other vortex generators added on top. Care must be taken, however, to maintain robustness and high yield manufacturing also. Amongst the most promising geometries are variations on chevron turbulators, including broken or segmented chevrons, as well as the staggering of these features to increase surface flow interactions. Figure 4 shows an example of heat transfer coefficient data from several surfaces placed in a parallel plate channel flow at a Reynolds number of 100,000. Among the geometries tested are a smooth surface, transverse turbulators, unbroken chevron arrays, and broken/staggered counter-angled strips formed by removing the apex of each chevron, as shown in Bunker *et al.* [11]. The parallel plate channel is large enough in width to have about a dozen such pairs of features in a full adjacent surface array. The laterally averaged smooth surface Nusselt number distribution agrees with literature values for a developing flow between parallel plates. The Nusselt number distribution for the transverse turbulators is elevated by a factor of about 1.75, and exhibits the characteristic trait of decay with axial distance from the channel entry. The somewhat surprising behavior of the chevrons is that no decay characteristic is present; the heat transfer coefficient is enhanced to a level of more than two times the developed smooth surface value and remains there throughout. The broken and staggered chevron strip heat transfer coefficients improve a bit more, which might be attributed to the increased free edges generating additional mixing. Friction coefficients also increase with these changes.

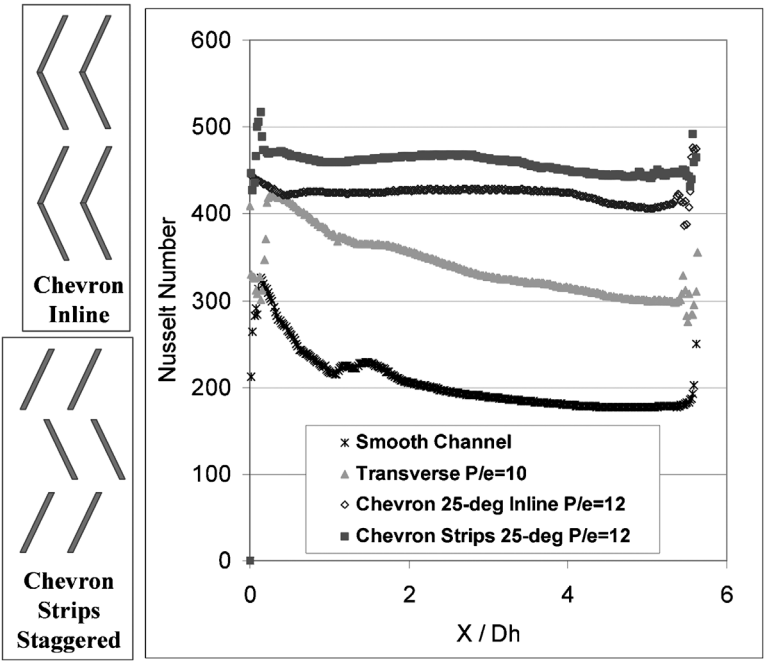

Figure 4: Turbulated heat transfer with forms of chevrons.

In a second example of the potential for such new turbulated surfaces, Fig. 5 presents laterally averaged heat transfer coefficients for staggered and counter-angled strips inside a duct of curved cross sectional geometry at Reynolds number of 100,000. The smooth surface behavior is as expected, with characteristic entry region decay. The transverse turbulators (90°) also exhibit this decay, but clearly show the periodic trend associated with flow separation and reattachment at each turbulator; the heat transfer coefficients are quite non-uniform. The angled 45° turbulators serve to eliminate this periodic behavior, but at the price of a somewhat lower average heat transfer coefficient, and with no benefit to friction coefficient (not shown). The counter-angled strips result in the best of both parameters by not only maintaining an elevated and uniform heat transfer coefficient, but also leading to almost a 50% lower friction coefficient compared with the other turbulated cases. It is speculated that the interaction of surface curvature effects with the counter-angled turbulator strips results in this very beneficial behavior.

3 Mesh network and micro cooling

As turbine blade cooling advances beyond the conventional methods, cooling will migrate from a largely internal bulk flow medium within hollow airfoils (e.g. Fig. 3) to one that may be built into or "within" the exterior walls of an airfoil.

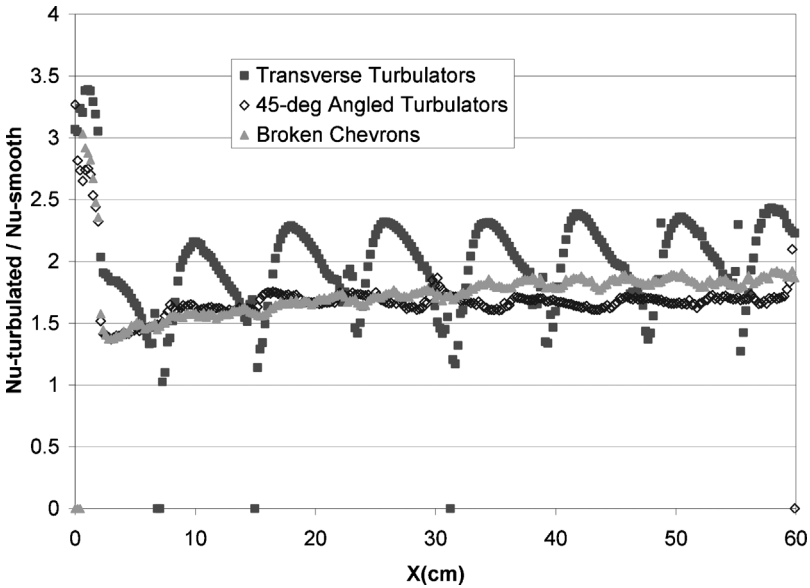

Figure 5: Heat transfer enhancement distribution comparison.

In essence, this change seeks to move the heat sink (the coolant and its wetted surface area) closer to the heat source (the hot gas path surfaces), thereby producing a more effective heat exchanger. This approach has the benefit of retaining a cooler internal bulk structure for the blade, which results in a longer rupture life. In the extreme, this type of design might be conceived as a micro cooling method. The use of many small cooling passages near the surface has potential benefits, but must also consider the risks associated with dirt plugging, manufacturing tolerances, and cost. The nearer term version of in-wall cooling is sometimes referred to as double-walled cooling, or here as mesh network cooling, in which the passages are of sizes amenable to investment casting. A greater degree of uniformity in internal cooling is key to many current thermal stress–strain limitations. Turbine airfoils can generally withstand higher bulk temperatures as the local thermal gradients are reduced. Zhang *et al*. [12] addressed this type of cooling network for different open flow areas, or alternately different solidity of channel meshes, using straight channels that intersect with included angle of 60 degrees, showing 2–2.5 times local heat transfer enhancement compared to a smooth channel. Heat transfer in small pre-film impingement chambers within airfoil wall sections was studied by Gillespie *et al*. [13] showing the contributions to cooling of both wall chamber interior surfaces.

The concept of micro cooling for airfoils is the natural extension of the more macroscopic mesh network cooling to reach the limits of thermodynamics and heat transfer. Micro cooling spreads out the cooling network in a series of smaller and highly distributed channels, or sub-channels, providing better uniformity of

cooling and lesser in-plane thermal gradients. This is analogous to the distribution of blood vessels in the human body. Micro cooling may be thought of as a complete airfoil cooling solution, or as a regional cooling device (e.g. leading edges or trailing edges). The most notable developments of micro cooling are those of the Allison Advanced Development Corporation, known by the trade names of Lamilloy® and CastCool® [14]. The airfoil walls in Lamilloy are formed as two or more sheets of metal bonded together with distributed pins, while those of CastCool are investment cast and limited to a double-wall type. The cooling flow is introduced via impingement jets that are staggered with respect to the exiting film holes. Full cooling distribution is obtained, and a form of transpiration-like film cooling results from the normal holes in the outer layer. Examples of overall cooling effectiveness and film cooling are shown in Nakamata *et al.* [15] and Hale *et al.* [16], respectively. The challenges of micro cooling include hole plugging, wall strength, film cooling, manufacturing, and cost. No commercial use of these micro cooled solutions has yet appeared however. Other forms of proposed micro cooling include fabricated diffusion surfaces formed by electro-galvanic metal deposition, such as that described by Battisti [17]. Russian and Ukrainian turbine airfoil research [18] developed a shell-and-spar approach to manufacturing. In this method, the interior portion of the airfoil is cast in a simple format, and cooling channels machined in the exposed surface. The channels are then filled with a leachable material, a thin outer airfoil skin is bonded to the non-channel metal regions, and the filler is leached out. The result is a distribution of small cooling channels close to the outer surface. Again, these micro cooling forms face significant material and processing challenges, as well as unexplored thermal and fluid design domains.

A demonstration of the capability of mesh network cooling is provided in the research of Bunker *et al.* [19]. Cooling meshes were tested using round pins and rounded diamond shaped pins with height-to-diameter (H/D) ratios of 0.2 and center spacing (S/D) ratios of 1.5, as well as less dense rounded diamond pins of smaller H/D of 0.3 and S/D of 2.14. These geometries differ substantially from conventional pin fin arrays in which $H/D > 1$ and S/D ratios are typically about 2.5. Special attention was paid to the combination of techniques including pin meshes, turbulators, and dimple/concavity arrays that are shown to provide multiple design solutions for heat transfer and pressure loss objectives. Figure 6 shows one such combined geometry using pins, concavities, and turbulators. Figure 6 also shows the potential use for these new in-wall mesh networks as improvements in cooling designs for turbine blades per Lee and Bunker [20]. Figure 7 shows that average channel area-corrected heat transfer capabilities exceeding three times that of smooth channels have been demonstrated using actual surface wetted area enhancements of no more than 20%. This cooling capability increase is also realized with a large decrease in channel material solidity, up to 30%, compared to conventional pin fin arrays. For both heat transfer and friction coefficients, some effects are additive, while others are synergistic in exceeding strictly additive behavior. The use of concavity surfaces within other structural geometries is particularly important for minimizing increases in friction.

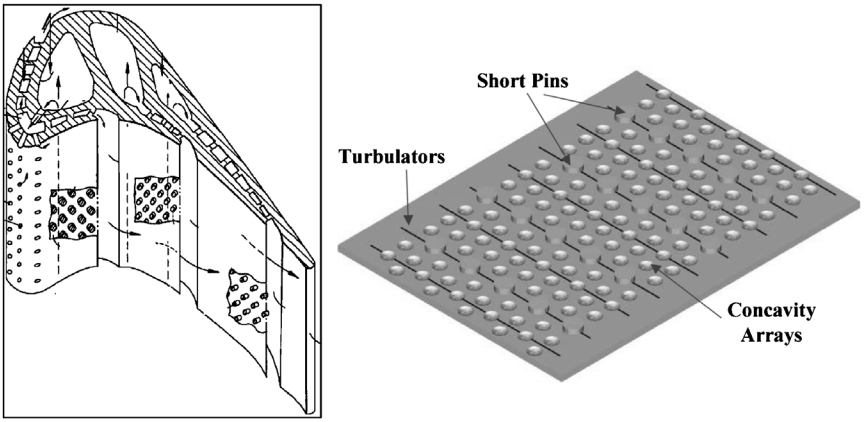

Figure 6: Mesh network geometry and turbine blade design.

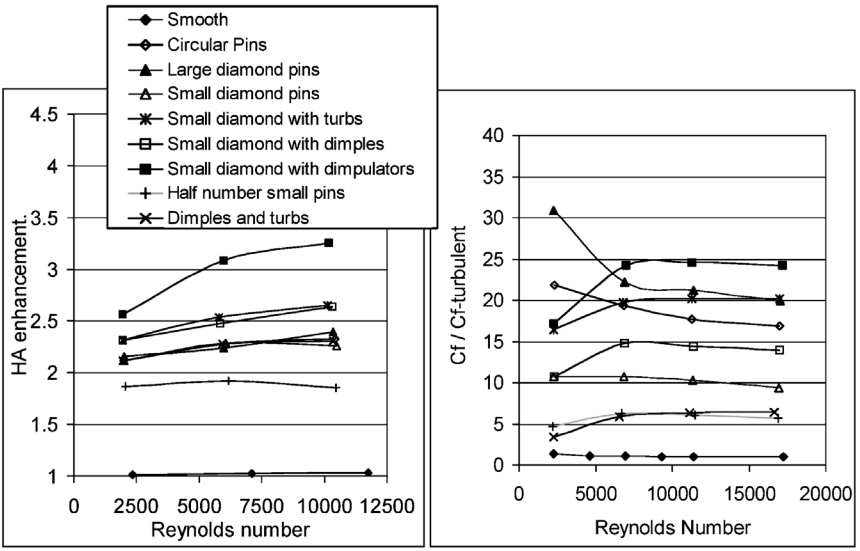

Figure 7: Mesh network heat transfer and friction enhancements (reproduced with permission from ASME).

4 Latticework (vortex) cooling

Latticework cooling, also known as vortex cooling or bounded vortical duct cooling, in its application to high-temperature gas turbine components originated within the former Soviet design bureau engineering system 25 years ago [21]. This method of vane and blade structure and cooling developed into the most recognizable standard within the many Soviet designs (Russian and Ukrainian designs today). Its application history parallels the Western use of serpentine cooling for

turbine blades. Latticework cooling can most simply be described in the radial cooling channel format shown in Fig. 8. The very name is descriptive of the geometry which forms the basic sub-element of the design, "lattice cooling" by means of coplanar crossing channels. As shown, the upper and lower halves of an overall channel or cooling region each has a series of unidirectional sub-channels oriented at some angle β to the radial direction (radial is used in this example, but is not a limitation). The upper and lower walls may be thought of as the pressure and suction sides of an airfoil, for example. The two portions of sub-channels are oriented so as to oppose each other, or cross as in a latticework design. When cooling flow enters the main network, such as at a blade root section, essentially half proceeds in the upper sub-channels, and half in the lower sub-channels, with little or no mixing between upper and lower sub-channels [21]. The main action within this cooling design comes at the edges of the lattice network where the sub-channels encounter seemingly "dead ends" or "bounding" walls, these would be the interior rib or the external wall of an airfoil design. Upon encountering the side wall, the flow must turn by the angle 2β as it enters the upper (or lower) crossing channel, i.e. it switches from a pressure side sub-channel to a suction side sub-channel, or vice versa. The flow makes its way in this "switchback" fashion until it leaves the lattice channel by way of film holes or routing to another portion of the airfoil interior, e.g. another lattice, or trailing edge holes. The overall motion of flow in the length of lattice channel is then that of a flattened "vortex".

A summary of the turbine cooling applications for this technology is presented by Nagoga [22]. The main advantages of this technology include (1) a robust architecture for investment casting with ceramic cores, (2) overall heat transfer coefficient enhancement levels comparable to those of turbulated serpentines, (3) similar overall pressure losses to turbulated serpentines, and (4) a potentially higher blade strength. The manufacturing aspect of vortex airfoils was more apparent as

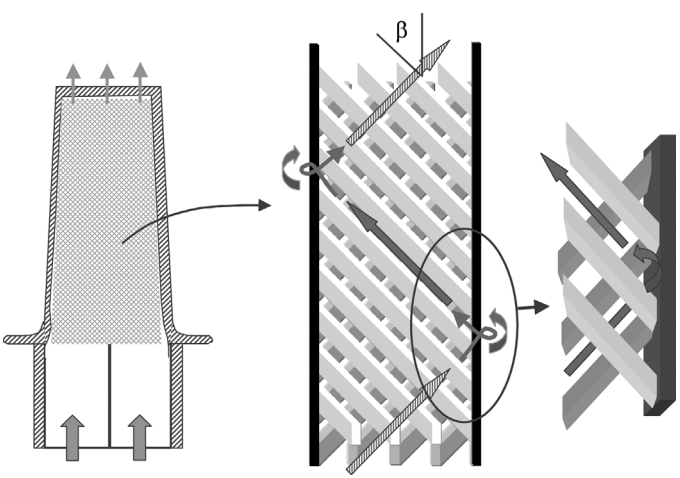

Figure 8: Schematic of latticework cooling.

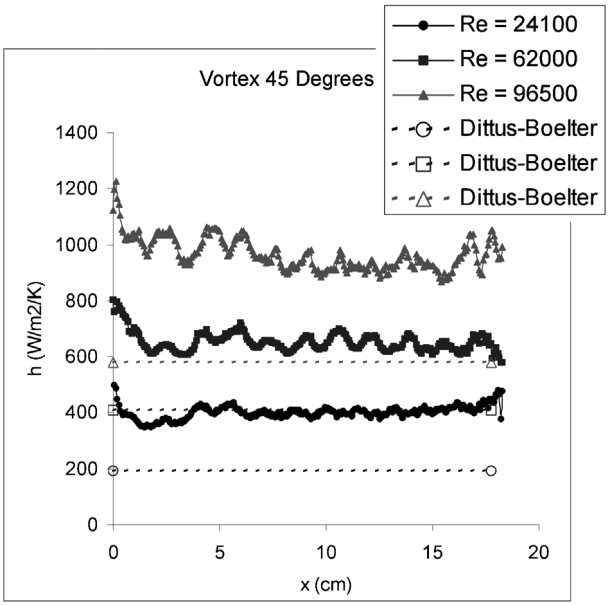

Figure 9: Latticework laterally averaged heat transfer coefficients (reproduced with permission from ASME).

an advantage in the initial years of development when ceramic core material strength required higher core connectivity, but may be diminished today with the advances of investment casting technology. The blade strength advantages are due to the use of distributed internal ribs of full or partial extent with substantial radial orientation components, but may be countered by other designs and materials. The overall cooling effectiveness of a turbine blade with leading and trailing edge latticework cooling is shown in Goreloff *et al*. [23].

The study of Bunker [24] provides detailed information concerning the heat transfer coefficients and pressures in latticework cooling channels. This study used two test methods to determine the local and overall heat transfer coefficients for a vortex channel with β of 40–45°. Both liquid crystal and infrared thermographic methods were used on acrylic and metallic models, respectively, to discern the heat transfer coefficients without and with the important effects of internal rib fin effectiveness. Tests with insulating ribs determined the heat transfer on the primary surfaces representing the pressure and suction walls of an airfoil. Tests with integral metal ribs determined the additional impact of the fin effectiveness provided by the lattice ribs. A simple radial vortex channel design was employed throughout with sub-channel aspect ratios near unity and sub-channel Reynolds numbers from 20,000 to 100,000. Figure 9 shows an example of the laterally averaged primary surface heat transfer coefficients, varying in the radial (x) direction for a narrow, 45° channel latticework model. Primary surface enhancements average about 1.5 over fully developed, smooth duct behavior (Dittus–Boelter), but

reach local values of about 3 immediately after each turn. Pressure distributions show high turning losses on the order of those associated with serpentine 180° turn circuits (virtually all 180° turns in real blades contribute a major portion of the circuit total loss). Heat transfer coefficient distributions are remarkably uniform throughout the channels excepting the turns themselves, because turn enhancements are retained for relatively long distances. Figure 10 provides the overall vortex channel heat transfer coefficient enhancement levels, including internal rib effectiveness, which are shown to be 2.5–3. In narrow vortex channels, overall enhancements are about 3, while in the wider vortex channels, turn effects are less of the total effect producing overall enhancements levels of about 2.8. The effects of sub-channel internal ribs, which act as fins, are shown to be very important in the overall thermal picture. Acharya *et al*. [25] studied the rotational effects on heat transfer for one of these latticework models for a range of Rotational numbers; obtaining data on all inner and outer main surfaces. Their results indicate very uniform heat transfer, but more importantly that the effects of rotation are not at all significant as they are in conventional serpentine cooling channels.

5 Augmented surface impingement cooling

The use of impingement jets for the cooling of various regions of modern gas turbine engines is widespread, most especially within the high-pressure turbine. Since the cooling effectiveness of impingement jets is very high, this method of cooling provides an efficient means of component heat load management given sufficient available pressure head and geometrical space for implementation. Regular arrays of impingement jets are used within turbine airfoils and endwalls to

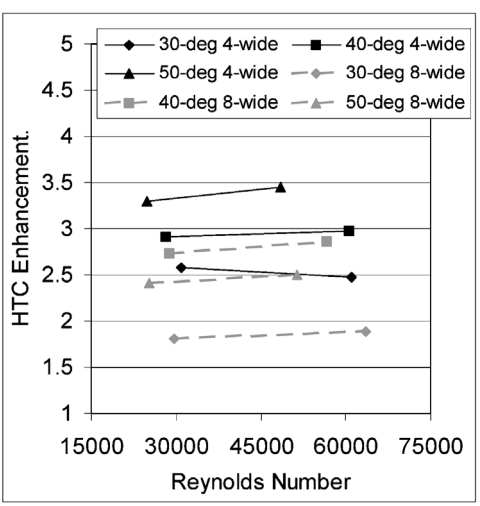

Figure 10: Latticework overall heat transfer enhancements (reproduced with permission from ASME).

provide relatively uniform and controlled cooling of fairly open internal surface regions. Such regular impingement arrays are generally directed against the target surfaces by using sheet metal baffle plates, inserts, or covers which are fixed in position relative to the target surface. These arrangements allow for the design of a wide range of impingement geometries, including in-line, staggered, or arbitrary patterns of jets. In more confined regions of airfoils such as the leading edge or trailing edge, spanwise lines of impingement jets are sometimes used to focus cooling on one primary location of high external heat load like the airfoil aerodynamic stagnation region. There also exist many other applications for individual impingement jets on selected stationary and rotating surfaces. Vane endwalls, blade platforms, unattached shrouds, and combustor liners may all have specific local cooling requirements well suited to the use of individual jet cooling. Impingement jets are also used on rotor disk cavity faces and in some applications may provide additional functions of sealing. Summaries of applicable impingement heat transfer research may be found in Martin [26] and Han and Goldstein [27].

A wealth of information exists on the basic cases of individual and array jet impingement heat transfer. The heat transfer coefficient distributions due to a single axisymmetric jet impinging normally on a smooth flat plate with free spent air discharge were investigated by Gardon and Cobonpue [28]. This study determined the basic effects of jet-to-target spacing and jet Reynolds number on stagnation region and radial heat transfer. Perry [29] measured single air jet impingement heat transfer characteristics on a free surface for normal and oblique impingement angles, determining the local and averaged effects of jet angling. Mass transfer normal jet impingement experiments of Schluender and Gnielinski [30] and Petzold [31], as summarized in Martin [26], show similar stagnation and non-stagnation region Sherwood number behavior for very high jet Re conditions of as much as 375,000.

While jet Reynolds number, target distance, and impingement angle all have some effect on stagnation region heat transfer coefficient, as well as the overall surface heat transfer distribution, target surface roughness and texturing can have a major impact on the impingement dominated heat transfer portion of the surface. Moreover, roughness can frequently be tailored to address cooling needs in specific localities. Effects that tend to thin the boundary layer relative to the roughness element heights, such as increased Reynolds number or decreased jet diameter, lead to increased heat transfer. Chakroun *et al.* [32] examined impingement heat transfer for a single normal air jet on a deterministic patterned rough surface. Heat transfer augmentations of up to 28% were noted, with a measured increase in the turbulence intensity of the flow around the roughness elements. El-Gabry and Kaminski [33] measured peak and average heat transfer coefficients for an array of normal or angled jets impinging on a randomly rough surface. Figure 11 shows a photo of the close-packed particles forming the rough surface (average roughness Ra value of 33 μm), as well as the basic heat transfer data for normal and angled jet arrays spaced 2 jet diameters from the surface. Heat transfer coefficients were observed to increase by as much as 25% in this study, much more than the roughly 8% surface area increase. Also of note, due to the use of a rough surface, the ratio

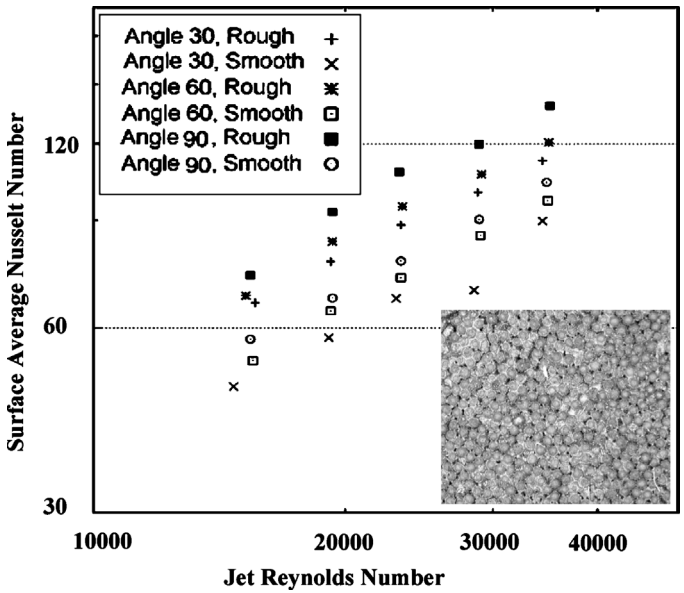

Figure 11: Rough surface impingement jet array heat transfer (reproduced with permission from ASME).

of peak-to-average heat transfer coefficient was reduced to half that of the smooth surface. As an example of the very localized enhancement due to roughness, Fig. 12 provides the smooth and rough surface heat transfer coefficient distributions for the turbine blade platform cooling model study of Bunker and Bailey [34]. Also shown is a photo of the random roughness surface, which, in this case, is not close-packed (average roughness Ra value of 30 μm). In this study, a single nearly normal impingement jet is used with Reynolds number of 130,000, spaced 2.5 jet diameters from the surface. The immediate stagnation region of the jet exhibits a 50% increase in heat transfer coefficient, while this benefit is very quickly lost as flow transitions to a convective condition over the surface.

Several forms of patterned surface augmentation, more amenable to investment casting, have also received increasing attention for use under impingement jets. These include pin arrays in the form of discrete shaped bumps, regularly spaced turbulators, and even arrays of concavities. The intent of these surface augmentation methods is to derive a greater benefit than simple wetted surface area increase by also creating additional vortices and turbulence. The study of Son *et al.* [35] examined the use of cylindrical surface pins under impingement jets for a flat surface, while that of Taslim *et al.* [36] investigated conical bumps to augment airfoil leading edge impingement cooling. Both studies found heat transfer to be augmented by roughly an amount equal to the increased surface area ratio factor. Kanokjaruvijit and Martinez-Botas [37] tested several configurations of impingement jet array parameters with dimpled target surfaces of various forms. They found that shallow dimples could provide as much as 50% improved heat flux

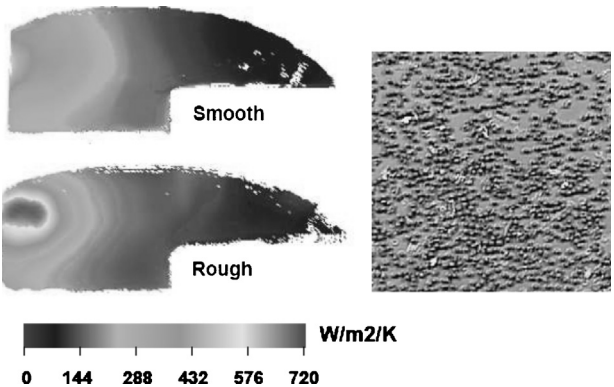

Figure 12: Rough surface single impingement jet heat transfer augmentation (reproduced with permission from ASME).

(including area factor), while deeper dimples could actually serve to create poor recirculation zones. Gau and Lee [38] studied various forms of two-dimensional jet impingement on rib-roughened surfaces, also showing that with proper relative geometry control heat flux can be significantly increased, but creation of trapped flows can decrease heat flux below the non-ribbed condition. Of great significance throughout these studies was the finding that pressure losses were negligibly affected by the use of such surface methods with impingement flows.

As noted above, the correct relative size and location of impingement jets and surface augmentations is very important for obtaining desired results. Most research has focused on the modification of the surfaces, but the jet arrays may also be modified from the conventional uniform in-line or staggered spacing. Bailey and Bunker [39] examined heat transfer with jet array spacings outside that of prior literature, both smaller and larger. Significant deviations from past data were noted, most especially in cases where jet arrays become very dense, leading to large variations in jet Reynolds numbers. As an extension to this work, Gao *et al.* [40] studied the effects of linearly stretched jet arrays in which the row-to-row spacing is increased with distance inside the channel. Fig. 13 shows an example of the Nusselt number distribution for average jet Reynolds number of 10,000. The streamwise averaged heat transfer coefficients decline as the jets become less numerous. Fig. 13 also shows a second modification to compensate for this decline by increasing the jet diameter on each row. The combined effect serves to make the streamwise distribution much more uniform.

6 Concavity surfaces cooling

Another class of surface enhancements results from the depression of features into the cooling channel or surface walls, forming recesses rather than projections. Generically, such features are known as concavities (or dimples), and may

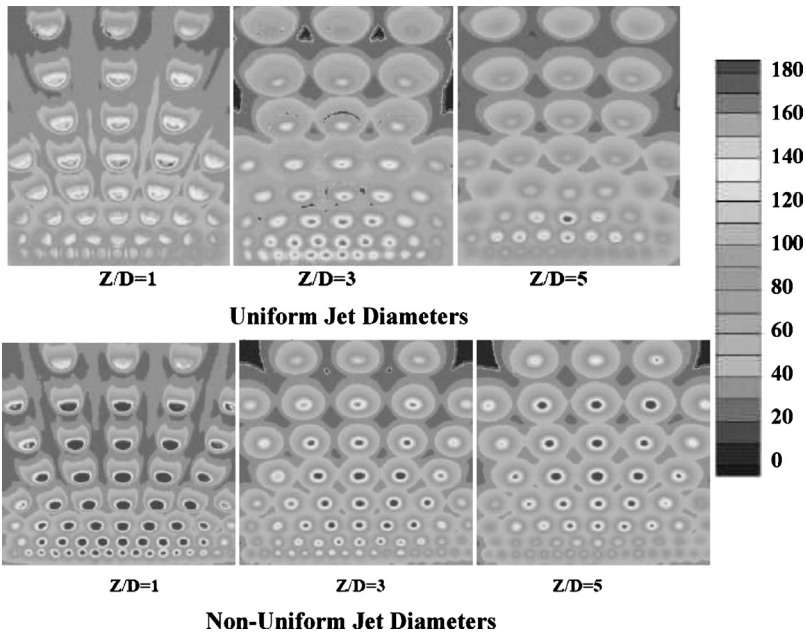

Figure 13: Stretched and variable jet diameter impingement Nusselt distributions (reproduced with permission from AIAA).

be formed in an infinite variation of geometries with various resulting heat transfer and friction characteristics. Concavity surfaces are commonly known for their drag reduction characteristics in external flows over bodies. The most famous example being golf balls, where they serve to delay the point of boundary layer separation, thereby reducing overall drag for the sphere, as shown by Bearman and Harvey [41]. Application of such concavity surfaces to external flows for marine vessels and airframes has been suggested by Kiknadze *et al*. [42], in which defined arrays of shallow concavities may cover the main hull or fuselage surfaces to prevent a thick boundary layer from forming, thereby reducing the drag.

Applications of concavity surfaces involving heat transfer remained largely unknown until the recent dissolution of the Soviet Union, at which time Russian research from the 1980s began to surface. The basic fluid dynamic condition for flow over concavities of spherical or cylindrical shape is well described in the study of Afanas'yev *et al*. [43]. A flow with boundary layer thickness less than the concavity surface diameter reacts with the cavity by flowing into the "bowl", experiencing a separated region of some extent on the entry side as depicted in Fig. 14. The spherical shape, or one nearly approximating it, creates a pressure field within the bowl acting to collapse or concentrate the flow in the downstream portion of the recess, creating a vortex structure. In a steady flow over a symmetric spherical dimple, a pair of symmetric, counter-rotating vortices is ideally formed, as shown by the computational work of Isaev *et al*. [44] in Fig. 15. In most real flow cases,

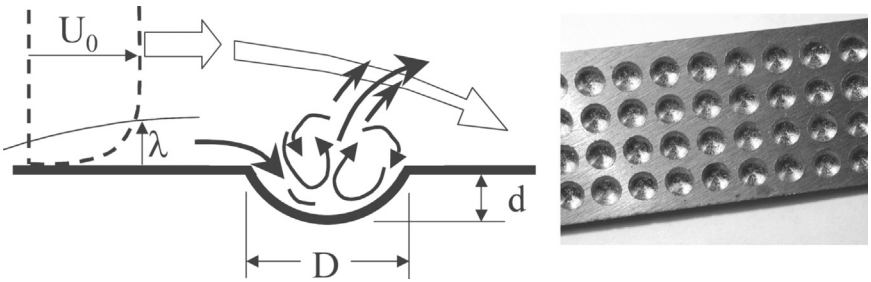

Figure 14: Schematic of concavity flow and sample surface array.

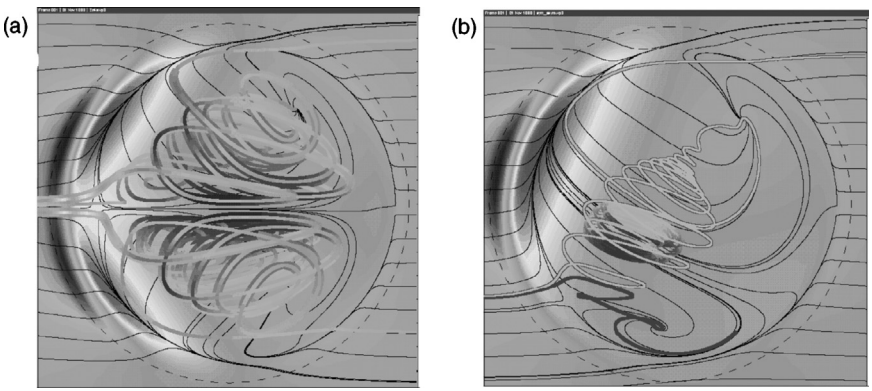

Figure 15: CFD showing jet-vortical structures in a deep spherical dimple in turbulent flow: (a) symmetric vortex pair; (b) asymmetrical single vortex (graphic courtesy of S. Isaev).

only one vortex is created, and this vortex may move side-to-side with some frequency. This condition is said to "expel" flow from the cavity as an organized vortical structure. As the vortex penetrates into and interacts with the mainstream flow, it provides a scrubbing action, which brings fresh core fluid to the surface for enhanced heat transfer. Because the motion is organized, rather than the more dissipative effect of shearing layers, the pressure loss is less than that observed with projecting obstructions such as turbulators. In fact, some cases of concavity surfaces have friction nearly the same as smooth surfaces. Studies focusing on the heat transfer coefficients on a surface with a single hemispherical concavity show the fundamental potential of this method for thermal enhancement. Kesarev and Kozlov [45] show the detailed shear stress and heat transfer coefficient distributions relative to a flat surface for a single dimple, observing overall heat transfer augmentations of up to 1.5.

The full potential for concavity surface heat transfer comes in the application of full-surface arrays of ordered concavities. Wind tunnel surface studies of Afanas'yev and Chudnovskiy [46] and also Afanas'yev *et al*. [47] for arrays of

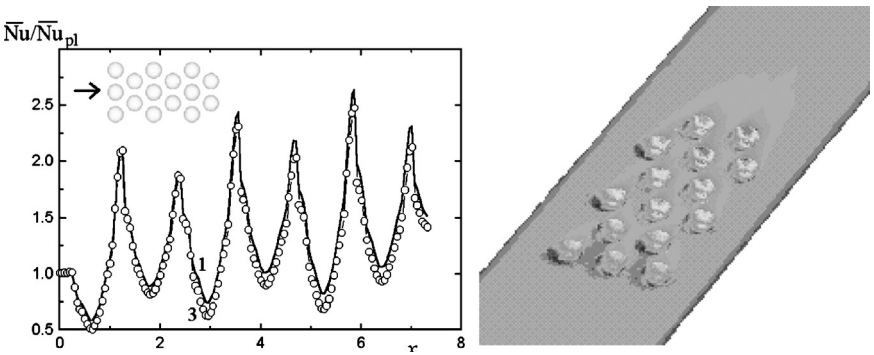

Figure 16: Computed heat transfer enhancement with dimple array.

spacings from about 1 to 1.7 dimple diameters showed heat transfer enhancements of 30–40% with no increase in friction. Figure 16 shows the typical development and streamwise increase in heat transfer coefficients for an array of dimples at fairly low Reynolds number, as predicted numerically by Bunker *et al*. [48], and confirmed in parallel experiments of Bunker *et al*. [49]. Heat exchanger studies of Belen'kiy *et al*. [50, 51] investigated the use of dimple arrays on the inner walls of annular passages and on tube surfaces of crossflow bundles, respectively. In some cases, as much as a 2.4 heat transfer enhancement was obtained with friction increases on the order of 2–4, relative to smooth surfaces. Recent studies using arrays of hemispherical dimples in more confined channels have been reported by Chyu *et al*. [52], Moon *et al*. [53], and Mahmood *et al*. [54]. In these studies, heat transfer enhancements of 2 to 2.5 have been demonstrated with friction factor increases from about 1.5 to 4. For application in turbine cooling, the summary by Nagoga [22] provides some insights into the effects of concavity array geometric parameters, namely the dimple depth-to-diameter ratio, the channel height-to-dimple diameter ratio, and the dimple spacing or surface density "f". An important feature to note about dimpled surfaces is the analogous form of the friction coefficient to that seen for rough surfaces, i.e. a decreasing magnitude of coefficient as Reynolds number increases, and a limiting value for a "fully rough" zone. A critical Reynolds number is demonstrated above which the friction coefficient does not change.

As with many other cooling technologies, care must be taken to examine the specific use and application of the augmentation means. For example, Bunker and Donnellan [55] investigated the use of dimpled surface arrays in circular cooling passages with Reynolds numbers up to 90,000. The resulting behavior of Nusselt number enhancement versus friction factor enhancement relative to a smooth passage is shown in Fig. 17, along with a photo of a sample dimpled tube (split for viewing). In this instance, the dimpled surfaces obtained as much as two times smooth surface heat transfer with moderate friction factor increases, though not matching a Reynolds analogy ideal. Still, the performance is much better than many conventional forms of turbulated channels.

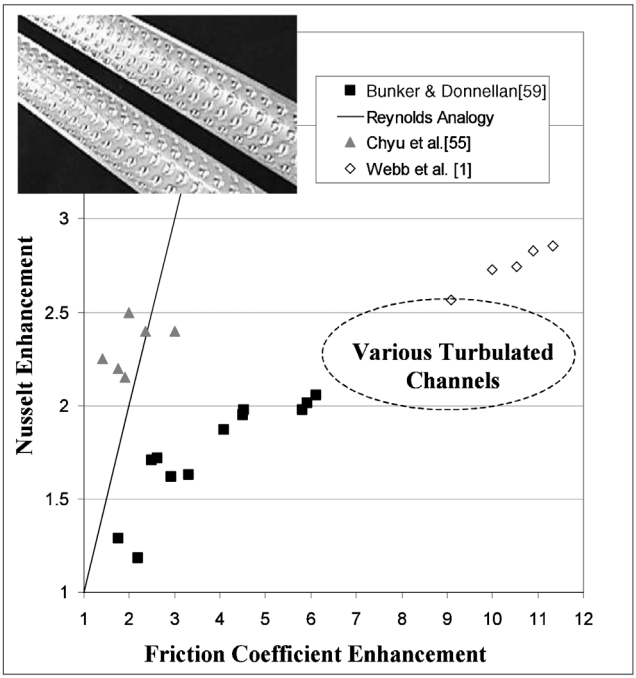

Figure 17: Concavity array heat transfer inside circular passage (reproduced with permission from ASME).

7 Swirl (cyclone) cooling

In the broadest sense, the concavity surface flows are one of a larger category known as "vortex" technologies, as described in the summary of Khalatov [56], which include various means of the formation of organized vortical or swirling flows in turbines. The latticework cooling presented earlier is one of these vortex technologies, utilizing the angled turning regions to generate bulk swirl. Another emerging vortex technology is the use of discrete wall jets injected into concave cooling passages, or along concave internal wall sections, to induce a bulk swirl motion. This cooling technique is generally known as swirl cooling, and also cyclone cooling from the original Russian studies. The studies of Glezer *et al.* [57], Hedlund *et al.* [58], Ligrani *et al.* [59], and Glezer *et al.* [60] provide the fundamentals of swirl cooling for blade leading edge passages.

Figure 18 shows two sketches of swirl cooling chambers implemented inside a blade leading edge, one without film extraction and the other with film extraction. Rather than using direct impingement jets aimed at the apex of the concave region, racetrack shaped wall jets are injected tangential to the surface at intervals along the blade height. The jets provide high heat transfer locally, but also serve to refresh the local coolant nearest to the surface at each new injection site. These

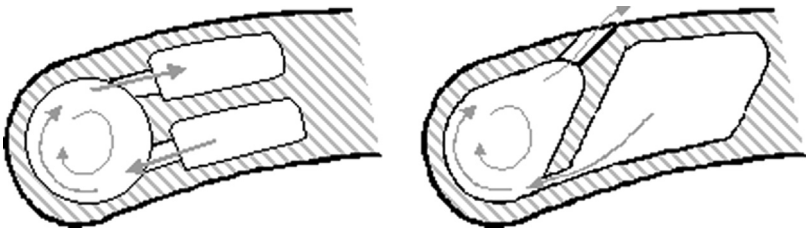

Figure 18: Swirl (cyclone) cooling passages.

studies have shown that swirl cooling can provide equivalent overall heat transfer to that of direct impingement. Channel average heat transfer coefficient enhancement factors of about 3 have been measured relative to a smooth surface, turbulent channel flow for Reynolds numbers from 5000 to 80,000. In an analogous manner to the decay of heat transfer coefficients downstream of abrupt entry regions, the swirl heat transfer is highest at the location of injection and decreases in a regular fashion along the bulk flow direction until the next injection location is reached. Other studies, such as that of Khalatov *et al*. [61] demonstrate the use of swirl cooling as an enhancement to 180° turn regions in serpentine channels. In this form, the coolant at the end of one circular passage is injected into the next circular passage via the wall jet mechanism (referred to as a tangential swirl generator in this case), thereby forming an initial cyclone effect that is allowed to decay along the remainder of the channel with no addition injections.

8 Film cooling

Film cooling is one of the major technologies allowing today's gas turbines to obtain extremely high turbine firing temperatures, subsequent high efficiencies, and longer life parts. The art and science of film cooling concerns the bleeding of internal component cooling air through the external walls to form a protective layer of cooling between the hot gases and the component external surfaces. The application of effective film cooling techniques provides the first and best line of defense for hot gas path surfaces against the onslaught of extreme heat fluxes, serving to directly reduce the incident convective heat flux on the surface. Fundamental early research concentrated on nearly ideal film cooling formed by a two-dimensional layer as depicted in Fig. 19a. Goldstein [62] provides a thorough summary of the theory, modeling, and experimental studies surrounding such idealized film cooling layers. Because of its high importance and widespread application, research into the many aspects of film cooling has seen a tremendous increase in the last ten to fifteen years. The publications relating directly or indirectly to film cooling deal with the major effects of film hole internal fluid dynamics, interactions with the mainstream gas flow, turbulence and vorticity production,

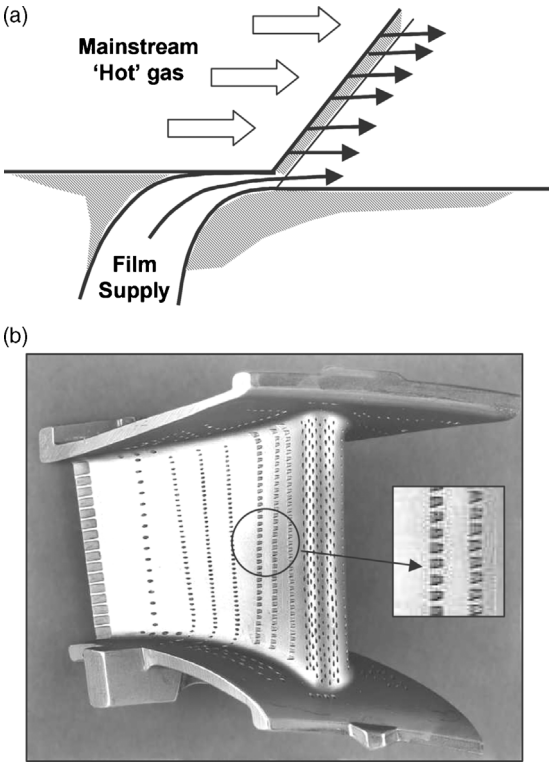

Figure 19: (a) Idealized two-dimensional film cooling; (b) typical film cooled turbine inlet guide vane.

effects of approach flows prior to the hole entry, hole shaping, orientation, and spacing, hole length-to-diameter ratio, density ratio, blowing strength, momentum flux ratio, effects of mainstream turbulence intensity, mainstream acceleration, external surface curvature, and external surface roughness.

Over the past 30+ years, investigations have been performed by a broad spectrum of researchers to understand the fundamental physics of film cooling, and to improve the state-of-the-art. The primary focus of most research has been on the use of discrete film holes, or rows of film holes, on the hot gas path surfaces of the turbine, since mechanical constraints dictate this format. Fig. 19b shows an example of a typical high-pressure turbine inlet guide vane with many film cooling rows of varying spacing and orientation. Only one primary advance in this technology has been put into widespread practice over these many years. That single improvement has been the change from round film holes to shaped film holes. Furthermore, the use of the term "shaped", while allowing a potentially vast number of geometries, is actually limited again to a single general class of geometry. Shaped holes are composed of round metering or throat sections with a uniform and symmetric

expanded exit region on the hot gas surface. Most commonly, all shaped holes applied in practice have fan diffuser exits with divergence angles between 10 and 15 degrees on each lateral side as well as on the side into the surface, as shown in the inset photo of Fig. 19b. Goldstein *et al*. [63] provided the seminal study on film cooling with discrete shaped holes. A recent review of shaped film hole cooling technology is provided in Bunker [64].

Several alternative geometries of film cooling holes have been proposed within the last few years, which have, in some form, demonstrated at least equivalent film effectiveness performance to the shaped holes noted above. These differing film holes may have specific form and function, either of limited or widespread potential, but each must also ultimately face the challenges of manufacturing, operability, and cost effectiveness. Moser *et al*. [65] analyzed a transonic wall jet geometry in which the film hole transitions from some interior cross section, such as circular, to a surface slit, as shown in Fig. 20. This transition is such that the hole converges in both the axial and lateral directions. The basic principle of this design is the creation of a choked flow and under-expanded jet that will conform to the exterior curved surface via the Prandtl–Meyer effect for expansion waves. Sargison *et al*. [66] demonstrated a converging slot-hole geometry, also shown in Fig. 20, in which the hole transitions from circular to slot with convergence in the axial direction and divergence laterally. The hole area does diminish to cause the flow to accelerate, though not necessarily to a choked condition. The exit is the metering section. This accelerated flow is speculated to have lower jet turbulence and more stability. Nasir *et al*. [67] tested round film holes with the addition of various triangular tabs covering the upstream edge of the holes, as drawn in Fig. 20. The tabs in some cases modified the exit flow and vortex structure in a manner that kept the coolant from lifting off. Fric and Campbell [68] investigated a so-called cratered film hole in which the circular hole exits into a shallow right circular surface cup or depression. The flow actually impinges on the edge of this depression causing it to deflect and fill the depression prior to issuing onto the external surface. Figure 21 shows water tunnel planar laser induced

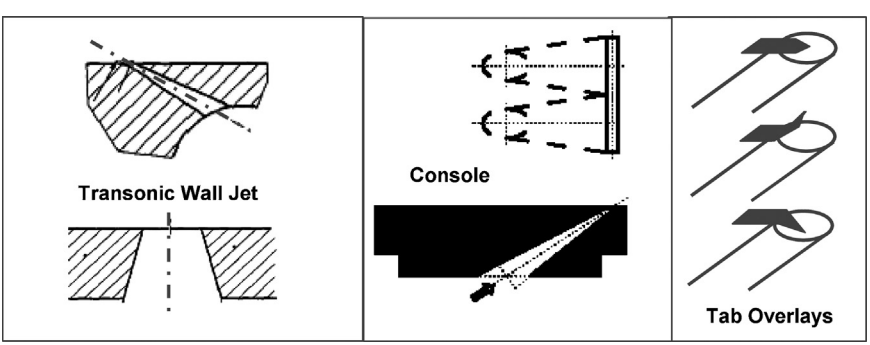

Figure 20: Innovations in film cooling holes.

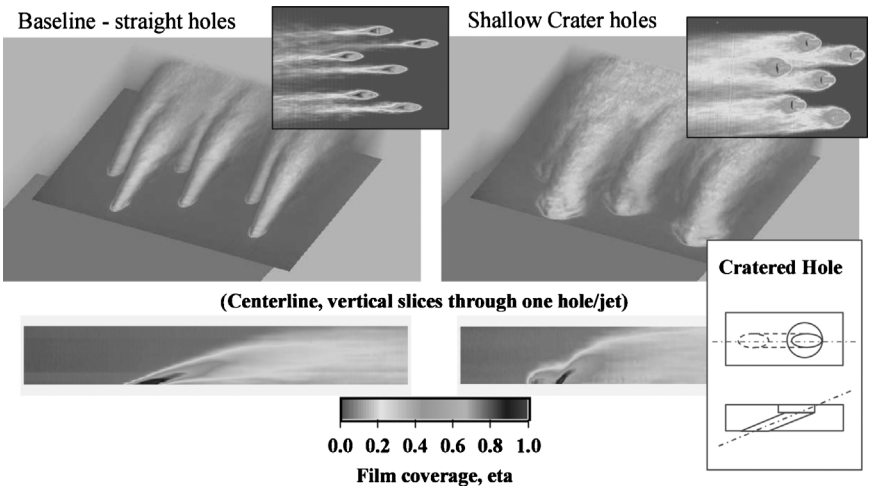

Figure 21: Crater film hole modification of injected coolant.

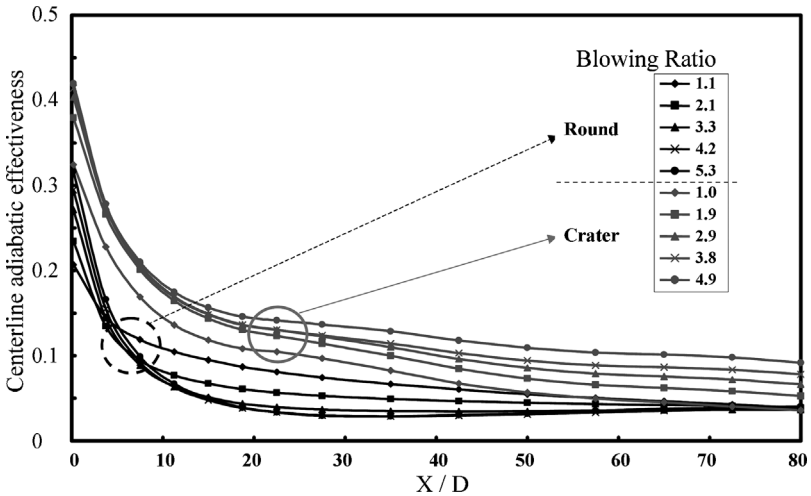

Figure 22: Improved centerline adiabatic effectiveness using crater film holes.

fluorescence data comparing the crater hole effect to that of a round film hole at a blowing ratio of 0.8. Three-dimensional, near-surface, and centerline cross-sectional views all show the effect of film jet lateral spreading due to the crater. Flat plate high-speed wind tunnel data shown in Fig. 22 result in 50–100% improved effectiveness over round holes, with better performance as blowing ratio is increased.

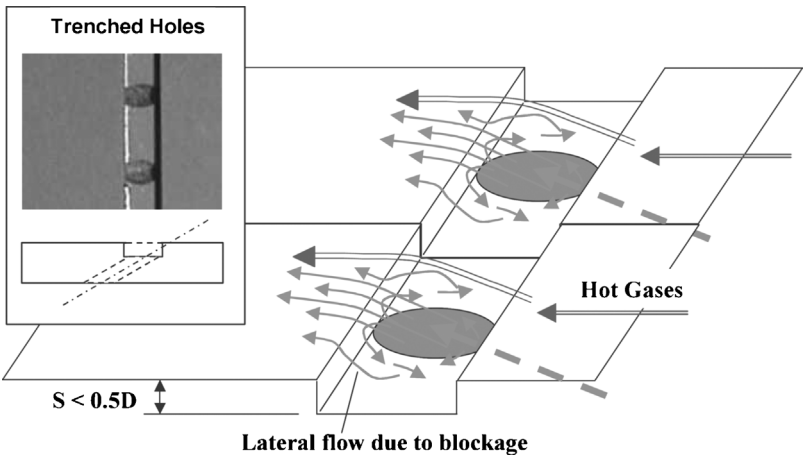

Figure 23: Flow interaction effect using trench film row (reproduced with permission from ASME).

Bunker [69] extended the work of [68] to a two-dimensional version, or shallow trench film geometry. The intention of this geometry is the same, to cause the film cooling flow to spread into the trench prior to issuing on the surface, as depicted in Fig. 23. The particular advantage of this geometry is that it can be formed using the protective coatings applied to the surface, without machining the trench into the substrate. Film effectiveness improvements of 50–75% over round holes were measured in flat plate tests, putting this on par with shaped holes. The studies of Ekkad *et al*. [70] and Waye and Bogard [71] both continued this research with additional parameter variations. Figure 24 is adapted from the results of [71] in which several variants of upstream and downstream trench edges were investigated at a blowing ratio of unity. The clearly superior geometries are those of the higher film effectiveness curves, all of which involve sharp trench edges immediately blocking the film hole downstream exit as shown in the inset sketch. Figure 24 also shows the full surface distributions for round holes and trench holes indicating the much greater lateral spreading of the coolant for the latter case.

The complete picture of film cooling must however also include the effect of the film injection on the augmentation of heat transfer coefficients due to the disruption of the boundary layer. Typically, any film cooling injection will increase heat transfer coefficients at least locally just downstream of the injection location, followed by a return to non-disrupted conditions further downstream (though not a return to the original boundary layer thickness). The augmentation of heat transfer coefficients can in some cases actually outweigh the benefit of film cooling effectiveness and cause a net heat flux increase to the surface. The goal of film cooling is always to obtain the highest possible net heat flux reduction, taking both factors into account. Using the same vane cascade test apparatus as [71], Harrison *et al*. [72] measured the heat transfer

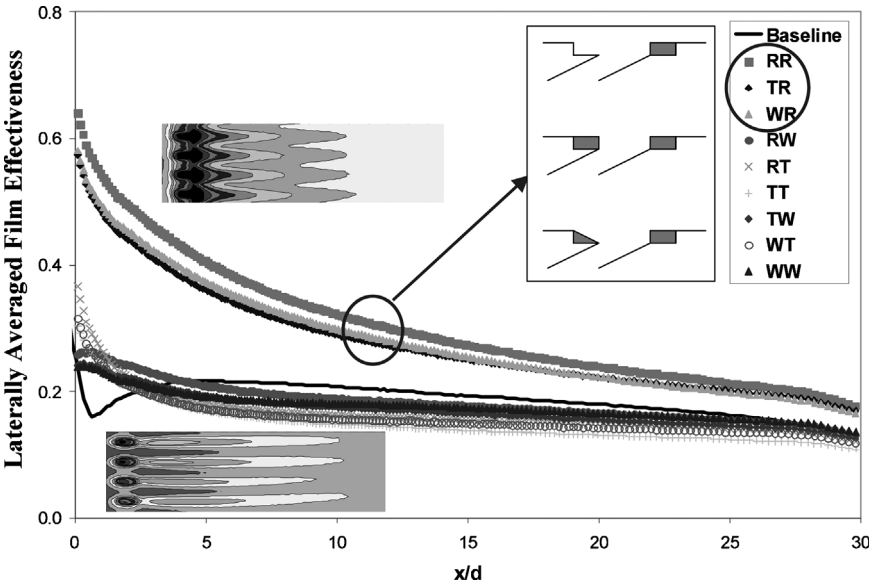

Figure 24: Laterally averaged film effectiveness for sharp edged trench (reproduced with permission from ASME).

coefficients for both round film holes and shallow trench film cooling under several imposed upstream conditions of the approaching boundary layer. With both the film effectiveness and heat transfer coefficients, and assuming a typical value for an engine wall-to-gas temperature ratio of 0.6, the laterally averaged net heat flux reductions were determined for several blowing ratios. Figure 25 shows these distributions for the representative conditions of upstream heating (a) without and (b) with a tripped boundary layer prior to the film holes. In all cases, the film effectiveness of the shallow trench holes greatly outweighs any increase in heat transfer coefficients, resulting in very good net heat flux reductions that increase with blowing ratio. In contrast, the effectiveness of the round film holes is so poor as to lead to some negative net heat flux ratios, especially at higher blowing ratios.

9 Conclusion

This chapter has presented several of the cutting edge, innovative cooling methods expected to further enhance the aerothermal–mechanical performance of turbine engines used for power systems. However, these methods are by no means an exhaustive or comprehensive summary. Many other variations and combinations of these techniques are anticipated as manufacturing advances become a reality. Further improvements and new techniques may become feasible as materials, systems integration and controls also advance.

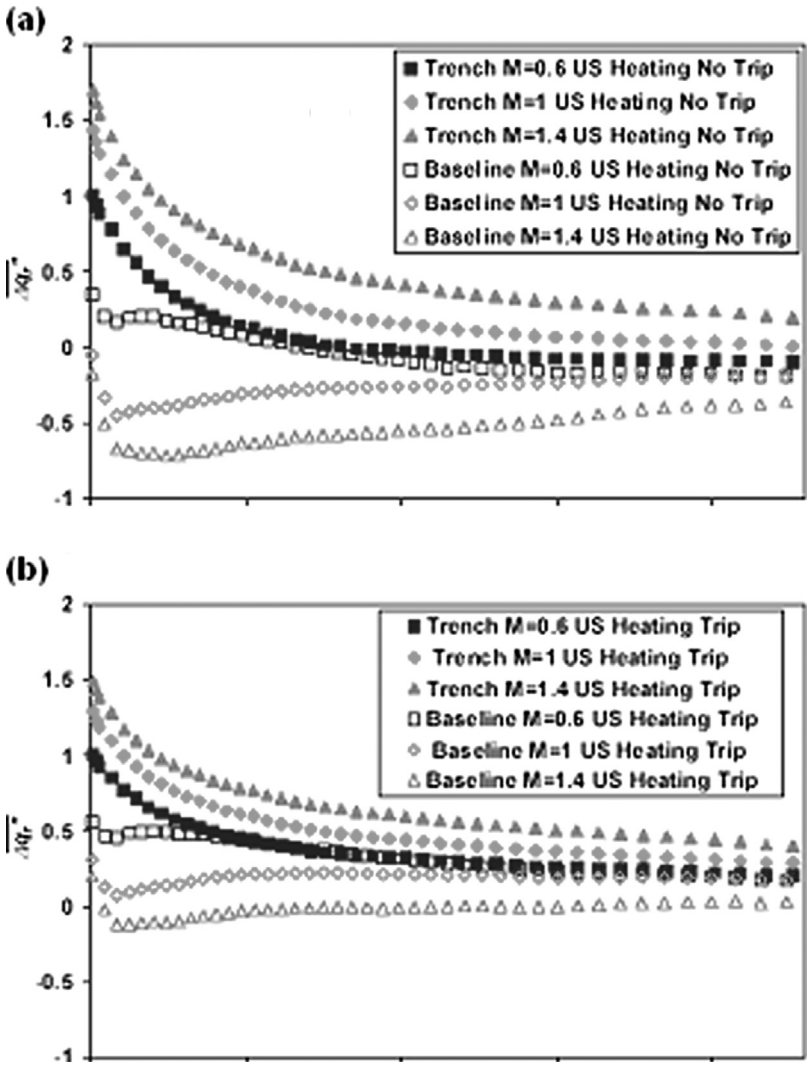

Figure 25: Net heat flux reduction comparison for round and trench film (a) without and (b) with trip (reproduced with permission from ASME).

References

[1] Webb, R.L., Eckert, E.R.G. & Goldstein, R.J., Heat transfer and friction in tubes with repeated-rib roughness. *International Journal of Heat and Mass Transfer*, **14**, pp. 601–617, 1971.

[2] Burggraf, F., Experimental heat transfer and pressure drop with two-dimensional turbulence promoter applied to two opposite walls of a square

tube. *Augmentation of Convective Heat and Mass Transfer*, eds A.E. Bergles & R.L. Webb, ASME: New York, pp. 70–79, 1970.

[3] Han, J.C., Glicksman, L.R. & Rohsenow, W.M., An investigation of heat transfer and friction for rib-roughened surfaces. *International Journal of Heat and Mass Transfer*, **21**, pp. 1143–1156, 1978.

[4] Han, J.C., Park, J.S. & Lei, C.K., Heat transfer enhancement in channels with turbulence promoters. *Journal of Engineering for Gas Turbines and Power*, **107**, pp. 628–635, 1985.

[5] Han, J.C. & Zhang, Y.M., High performance heat transfer ducts with parallel broken and V-shaped broken ribs. *International Journal of Heat and Mass Transfer*, **35(2)**, pp. 513–523, 1992.

[6] Taslim, M.E., Li, T. & Kercher, D.M., Experimental heat transfer and friction in channels roughened with angled, V-shaped and discrete ribs on two opposite walls. *Journal of Turbomachinery*, **118**, pp. 20–28, 1996.

[7] Kiml, R., Mochizuki, S. & Murata, A., Heat transfer enhancement mechanism in a rectangular passage with V- and Λ-shaped ribs. *Journal of Flow Visualization and Image Processing*, **8**, pp. 51–68, 2001.

[8] Cho, H.H., Wu, S.J. & Kwon, H.J., Local heat/mass transfer measurements in a rectangular duct with discrete ribs. *Journal of Turbomachinery*, **122**, pp. 579–586, 2000.

[9] Cho, H.H., Lee, S.Y. & Wu, S.J., The combined effects of rib arrangements and discrete ribs on local heat/mass transfer in a square duct, Paper No. 2001-GT-175, IGTI Turbo Expo, New Orleans, USA, 2001.

[10] Chyu, M.K. & Natarajan, V., Effects of a slit in a rib on the heat transfer from a rib-mounted wall. *Journal of Heat Transfer*, **115**, pp. 792–796, 1993.

[11] Bunker, R.S., Bailey, J.C., Widener, S.K. & Johnson, T.E., Combustor cooling with angled segmented surfaces, US Patent Application US2006/0042255A1, 2006.

[12] Zhang, N., Yang, W.J. & Lee, C.P., Heat transfer and friction loss performance in flow networks with multiple intersections. *Experimental Heat Transfer*, **6**, pp. 243–257, 1993.

[13] Gillespie, D., Wang, Z., Ireland, P. & Kohler, S.T., Full surface local heat transfer coefficient measurements in a model of an integrally cast impingement cooling geometry. *Journal of Turbomachinery*, **120**, pp. 92–99, 1996.

[14] Sweeney, P.C. & Rhodes, J.P., An infrared technique for evaluating turbine airfoil cooling designs. *Journal of Turbomachinery*, **122**, pp. 170–177, 1999.

[15] Nakamata, C., Okita, Y., Matsuno, S., Mimura, F., Matsushita, M., Yamana, T. & Yoshida, T., Spatial arrangement dependence of cooling performance of an integrated impingement and pin fin cooling configuration, Paper No. GT-2005-68348, IGTI Turbo Expo, Reno-Tahoe, Nevada, USA, 2005.

[16] Hale, C.A., Plesniak, M.W. & Ramadhyani, S., Film cooling effectiveness for short film cooling holes fed by a narrow plenum. *Journal of Turbomachinery*, **122**, pp. 553–557, 1999.

[17] Battisti, L., Cerri, G. & Fedrizzi, R., Novel technology for gas turbine blade effusion cooling, Paper No. GT2006-90516, IGTI Turbo Expo, Barcelona, Spain, 2006.
[18] Novikov, A.S., Meshkov, S.A. & Sabaev, G.V., Creation of high efficiency turbine cooled blades with structural electron beam coatings, in collection of papers Electron Beam and Gas-Thermal Coatings, Paton IEW, Kiev, pp. 87–96, 1988.
[19] Bunker, R.S., Bailey, J.C., Lee, C.P. & Stevens, C.W., In-wall network (Mesh) cooling augmentation for turbine airfoils, Paper No. GT-2004-54260, IGTI Turbo Expo, Vienna, Austria, 2004.
[20] Lee, C.P. & Bunker, R.S., Thermal shield turbine airfoil, US Patent 7011502, 2006.
[21] Nagoga, G.P., Personal communication, Moscow, 2000.
[22] Nagoga, G.P., Effective methods of cooling of blades of high temperature gas turbines. Publishing House of Moscow Aerospace Institute (Russian language), p. 100, 1996.
[23] Goreloff, V., Goychengerg, M. & Malkoff, V., The investigation of heat transfer in cooled blades of gas turbines. *26th Joint Propulsion Conference*, AIAA Paper No. 90-2144, 1990.
[24] Bunker, R.S., Latticework (vortex) cooling effectiveness. Part 1: Stationary channel experiments, Paper No. GT-2004-54157, IGTI Turbo Expo, Vienna, Austria, 2004.
[25] Acharya, S., Zhou, F., Lagrone, J., Mahmood, G. & Bunker, R.S., Latticework (Vortex) cooling effectiveness: rotating channel experiments. *Journal of Turbomachinery*, **127**, pp. 471–478, 2005.
[26] Martin, H., Heat and mass transfer between impinging gas jets and solid surfaces. *Advances in Heat Transfer*, **13**, pp. 1–60, 1977.
[27] Han, B. & Goldstein, R.J., Jet impingement heat transfer in gas turbine systems. *Annals of the New York Academy of Sciences*, 934 (Heat Transfer in Gas Turbine Systems), pp. 147–161, 2001.
[28] Gardon, R. & Cobonpue, J., Heat transfer between a flat plate and jets of air impinging on it. *International Developments in Heat Transfer*, *Proceedings 2nd International Heat Transfer Conference*, ASME: New York, pp. 454–460, 1962.
[29] Perry, K.P., Heat transfer by convection from a hot gas jet to a plane surface. *Proceedings of the Institute of Mechanical Engineers (London)*, **168**, pp. 775–784, 1954.
[30] Schluender, E.U. & Gnielinski, V., Warme und Stoffubertragung zwischen Gut und aufprallendem Dusenstrahl (Heat and mass transfer between a surface and impinging jet). *Chem.-Ing.-Tech.*, **39**, p. 578, 1967.
[31] Petzold, K., Heat transfer on a perpendicularly impinged plate. *Wiss. Z. Tech. Univ., Dresden*, **13**, pp. 1157–1161, 1964.
[32] Chakroun, W.M., Al-Fahed, S.F. & Abdel-Rehman, A.A., Heat transfer augmentation for air impinged on rough surface, Paper 97-GT-436, IGTI Turbo Expo, Orlando, Florida, USA, 1997.

[33] El-Gabry, L.A. & Kaminski, D.A., Experimental investigation of local heat transfer distribution on smooth and roughened surfaces under an array of angled impinging jets. *Journal of Turbomachinery*, **127(3)**, pp. 532–544, 2004.
[34] Bunker, R.S. & Bailey, J.C., Turbine blade platform impingement cooling with effects of film extraction and roughness, Paper No. GT-2005-68415, IGTI Turbo Expo, Reno-Tahoe, Nevada, USA, 2005.
[35] Son, C., Gillespie, D., Ireland, P. & Dailey, G.M., Heat transfer enhancement strategy for an impingement cooling system. *Proceedings of the 8th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery*, Vol. 2, Hemisphere Pub. Co., pp. 721–729, 2000.
[36] Taslim, M.E., Setayeshgar, L. & Spring, S.D., An experimental evaluation of advanced leading edge impingement cooling concepts. *Journal of Turbomachinery*, **123**, pp. 147–153, 2002.
[37] Kanokjaruvijit, K. & Martinez-Botas, R.F., Heat transfer and pressure investigation of dimple impingement, Paper No. GT-2005-68823, IGTI Turbo Expo, Reno-Tahoe, Nevada, USA, 2005.
[38] Gau, C. & Lee, C.C., Impingement cooling flow structure and heat transfer along rib-roughened walls. *International Journal of Heat and Mass Transfer*, **35(11)**, pp. 3009–3020, 1992.
[39] Bailey, J.C. & Bunker, R.S., Local heat transfer and flow distributions for impinging jet arrays of both sparse and dense extent, GT-2002-30473, IGTI Turbo Expo, Amsterdam, 2002.
[40] Gao, L., Ekkad, S.V. & Bunker, R.S., Impingement heat transfer. Part I: linearly stretched arrays of holes. *Journal of Thermophysics and Heat Transfer*, **19(1)**, pp. 57–65, 2005.
[41] Bearman, P.W. & Harvey, J.K., Golf ball aerodynamics. *Aeronautical Quarterly*, pp. 112–122, May 1976.
[42] Kiknadze, G.I., Gachechiladze, I.A. & Oleinikov, V.G., Method and apparatus for controlling the boundary or wall layer of a continuous medium, US Patent No. 6119987, 2000.
[43] Afanas'yev, V.N., Veselkin, V.Yu., Leont'ev, A.I., Skibin, A.P. & Chudnovskiy, Ya. P., Thermohydraulics of flow over isolated depressions (pits, grooves) in a smooth wall. *Heat Transfer Research*, **25(1)**, pp. 22–56, 1993.
[44] Isaev, S.A., Leont'ev, A.I. & Baranov, P.A., Identification of self-organized vortexlike structures in numerically simulated turbulent flow of a viscous incompressible liquid streaming around a well on a plane. *Technical Physics Letters*, **26(1)**, pp. 15–18, 2000.
[45] Kesarev, V.S. & Kozlov, A.P., Convective heat transfer in turbulized flow past a hemispherical cavity. *Heat Transfer Research*, **25**, pp. 156–160, 1993.
[46] Afanas'yev, V.N. & Chudnovskiy, Ya.P., Heat transfer and friction on surfaces contoured by spherical depressions. *Heat Transfer Research*, **24**, pp. 24–104, 1992.
[47] Afanas'yev, V.N., Chudnovsky, Ya.P., Leont'ev, A.I. & Roganov, P.S., Turbulent flow friction and heat transfer characteristics for spherical cavities on a flat plate. *Experimental Thermal and Fluid Science*, **7**, pp. 1–8, 1993.

[48] Bunker, R., Belen'kiy, M., Gotovskii, M., Isaev, S. & Fokin, B., Experimental and numerical investigation of heat transfer intensification in duct of non-constant area with smooth and concavity surfaces., *3rd Russian National Heat Transfer Conference*, 2002.

[49] Bunker, R., Gotovskii, M., Belen'kiy, M. & Fokin, B., Heat transfer and pressure loss for flows inside converging and diverging channels with surface concavity shape effects. *Proceedings 4th International Conference on Compact Heat Exchangers and Enhancement Technology*, September. 29–October 3, Crete Island, Greece, Begell House Inc.: New York, 2003.

[50] Belen'kiy, M.Ya., Gotovskii, M.A., Lekakh, B.M., Fokin, B.S. & Khabenskii, V.B., Experimental study of the thermal and hydraulic characteristics of heat transfer surfaces formed by spherical cavities. *Teplofizika Vysokikh Temperatur*, **29(6)**, pp. 1142–1147, 1992.

[51] Belen'kiy, M.Ya., Gotovskii, M.A., Lekakh, B.M., Fokin, B.S. & Dolgushin, K.S., Heat transfer augmentation using surfaces formed by a system of spherical cavities. *Heat Transfer Research*, **25**, pp. 196–202, 1993.

[52] Chyu, M.K., Yu, Y., Ding, H., Downs, J.P. & Soechting, F.O., Concavity enhanced heat transfer in an internal cooling passage, IGTI Turbo Expo, Paper No. 97-GT-437, Orlando, 1997.

[53] Moon, H.K., O'Connell, T. & Glezer, B., Channel height effect on heat transfer and friction in a dimpled passage, IGTI Turbo Expo, Paper No. 99-GT-163, Indianapolis, 1999.

[54] Mahmood, G.I., Hill, M.L., Nelson, D.L., Ligrani, P.M., Moon, H.K. & Glezer, B., Local heat transfer and flow structure on and above a dimpled surface in a channel, IGTI Turbo Expo, Paper No. 2000-GT-230, Munich, 2000.

[55] Bunker, R.S. & Donnellan, K., Heat transfer and friction factors for flows inside circular tubes with concavity surfaces. *Journal of Turbomachinery*, **125**, pp. 665–672, 2003.

[56] Khalatov, A.A., Vortex technologies in aerospace engineering. *Proceedings of the U.S.–Ukrainian Workshop on Innovative Combustion and Aerothermal Technologies in Energy and Power Systems*, May 20–25, Kiev, 2001.

[57] Glezer, B., Moon, H.K. & O'Connell, T., A novel technique for the internal blade cooling, IGTI Turbo Expo, Paper No. 96-GT-181, Birmingham, UK, 1996.

[58] Hedlund, C.R., Ligrani, P.M., Moon, H.K. & Glezer, B., Heat transfer and flow phenomena in a swirl chamber simulating turbine blade internal cooling, Paper No. 98-GT-466, IGTI Turbo Expo, Stockholm, Sweden, 1998.

[59] Ligrani, P.M., Hedlund, C.R., Thambu, R., Babinchak, B.T., Moon, H.K. & Glezer, B., Flow phenomena in swirl chambers, IGTI Turbo Expo, Paper No. 97-GT-530, Orlando, 1997.

[60] Glezer, B., Moon, H.K., Kerrebrock, J., Bons, J. & Guenette, G., Heat transfer in a rotating radial channel with swirling internal flow, Paper No. 98-GT-214, IGTI Turbo Expo, Stockholm, Sweden, 1998.

[61] Khalatov, A., Syred, N., Bowen, P., Al-Ajmi, R., Kozlov, A. & Schukin, A., Innovative cyclone cooling scheme for gas turbine blade: thermal-hydraulic

performance evaluation, Paper 2000-GT-237, IGTI Turbo Expo, Munich, Germany, 2000.

[62] Goldstein, R.J., Film cooling. *Advances in Heat Transfer*, **7**, pp. 321–379, 1971.

[63] Goldstein, R.J., Eckert, E.R.G. & Burggraf, F., Effects of hole geometry and density on three-dimensional film cooling. *International Journal of Heat and Mass Transfer*, **17**, pp. 595–607, 1974.

[64] Bunker, R.S., A review of turbine shaped film cooling technology. *Journal of Heat Transfer*, **127**, pp. 441–453, 2005.

[65] Moser, S., Ivanisin, M., Woisetschlaeger, J. & Jericha, H., Novel blade cooling engineering solution, Paper 2000-GT-242, IGTI Turbo Expo, Munich, Germany, 2000.

[66] Sargison, J.E., Guo, S.M., Oldfield, M.L.G., Lock, G.D. & Rawlinson, A.J., A converging slot-hole film cooling geometry. Part 1: Low-speed flat plate heat transfer and loss, Paper 2001-GT-0126, IGTI Turbo Expo, New Orleans, Louisiana, USA, 2001.

[67] Nasir, H., Acharya, S. & Ekkad, S., Film cooling from a single row of cylindrical angled holes with triangular tabs having different orientations, Paper 2001-GT-0124, IGTI Turbo Expo, New Orleans, Louisiana, USA, 2001.

[68] Fric, T.F. & Campbell, R.P., Method for improving the cooling effectiveness of a gaseous coolant stream which flows through a substrate, and related articles of manufacture, US Patent No. 6383602, 2002.

[69] Bunker, R.S., Film cooling effectiveness due to discrete holes within a transverse surface slot, Paper GT-2002-30178, IGTI Turbo Expo, Amsterdam, Netherlands, 2002.

[70] Ekkad, S., Lu, Y. & Nasir, H., Film cooling from a row of holes embedded in transverse slots, Paper No. GT-2005-68598, IGTI Turbo Expo, Reno-Tahoe, Nevada, USA, 2005.

[71] Waye, S. & Bogard, D., High resolution film cooling effectiveness measurements of axial holes embedded in a transverse trench with various trench configurations, Paper GT-2006-90226, IGTI Turbo Expo, Barcelona, Spain, 2006.

[72] Harrison, K.L., Dorrington, J.R., Dees, J.E., Bogard, D.G. & Bunker, R.S., Turbine airfoil net heat flux reduction with cylindrical holes embedded in a transverse trench, Paper No. GT 2007-27996, IGTI Turbo Expo, Montreal, Canada, 2007.