International Journal of LNCT

Volume-4 Issue-19 August -2020

ISSN: (Online):2456-9895

EFFECT OF BLOWING RATIO ON FILM COOLING FROM TRENCHED SHAPE HOLE FOR THE APPLICATION TO GAS TURBINE BLADES

Rohit Singh[1*], Dr. Shailendra Kumar Dwivedi[2] and Jitendra Raghuvanshi[3]
PG Scholar, Mechanical Engineering Department & LNCT, Bhopal, MP, India[1]
Professor (HOD), Mechanical Engineering Department & LNCT, Bhopal, MP, India[2]
Asst. Prof., Mechanical Engineering Department & LNCT, Bhopal, MP [3]

Abstract

In order to raise thermal efficiency of a gas turbine, higher turbine inlet temperature (TIT) is needed. However, higher TIT increases thermal load to its hot-section components and reducing their life span. Therefore, very complicated cooling technology such as film cooling and internal cooling is required especially for HP turbine blades. In film cooling, relatively cool air is injected onto the blade surface to form a protective layer between the surface and hot mainstream gas. Film cooling is one of the cooling systems investigated for the application to gas turbine blades. Gas turbines use film cooling in addition to turbulated internal cooling to protect the blades outer surface from hot gases. The present study concentrates on the numerical investigation of film cooling performance for a trenched shape hole in a modern turbine blade. The adiabatic film effectiveness and the heat transfer coefficient are determined numerically on a flat plate downstream of a single hole of inclined different blowing ratio cases by Computational Fluid Dynamics (CFD) analyses. The focus of this investigation is to investigate advanced cooling hole geometry on film cooling heat transfer and cooling effectiveness over flat and turbine airfoil surfaces.

In the present study, only inflow flow parameters are varied at a time, which leads to four different cases to achieve the desired blowing ratio (BLR). The geometry with trenched hole configuration has to be analysed for the different blowing ratios (0.5, 0.7, 1.0 and 1.5). The study is carried out for four cases using inlet and outlet boundary conditions. These cases are mainly divided into Case 1, Case 2, Case 3 and Case 4. Unstructured (tetrahedral) mesh is used with layers of prism and mesh is of ~4.5M elements. The present study flow field is solved by using k-omega turbulence model (Reynolds stress transport model) for simulation of turbulent flows in film cooling and the simulation is run using ANSYS FLUENT 13.0 computer code.

Keywords

Film cooling, Adiabatic cooling effectiveness (Ɛ), Blowing ratio (BLR), Trench, Numerical simulation, Computational Fluid Dynamics (CFD), Non-dimensioal axial distance (x/D) from the opening of jet and Surface angle (SA).

1.Introduction

Gas turbines are used for aircraft propulsion and in land-based power generation or industrial applications. In the 1930s, the development of high performance piston engines for aircraft propulsion had reached its peak, with further performance increases being unlikely. New technologies had to be developed in order to achieve the desired higher flight velocities, higher thrust to weight ratios and reduced engine sizes. In the following years, two new aircraft propulsion systems were developed and brought into service in military aircraft, namely the rocket and the gas turbine engines. The main three components of a gas turbine engine are compressor, combustor and turbine as shown in Figure 1. The compressor compresses the incoming air to high pressure, the combustor burns the fuel and produces high pressure, and high temperature, high velocity gas and the turbine extract the energy from the gas.

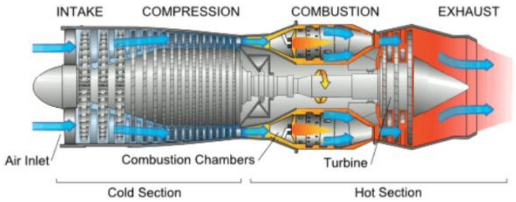

Figure 1: Components of gas turbine engine

The continuous improvement in the performance of air-breathing propulsion systems necessitates a continuous increase in the turbine inlet temperatures. This, coupled with the demands of reduced size of the combustors, has put a significant burden on turbine technology. Since the inlet temperatures of present generation gas turbines are much higher than the melting temperatures of the available alloys used to make the turbine blades, cooling of the blades is a critical issue in turbine technology. The development of turbine inlet temperatures is shown in Figure 2. Improvements in blade materials have allowed an increase of melting point around 200° and use of turbine cooling has allowed an increase of approximately another 250°, which allow turbine inlet gas temperature above the melting points of the materials used.

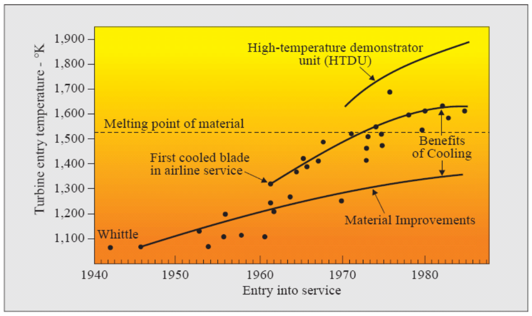

Figure 2: Development of Turbine Inlet Temperature [1]

Various internal and external cooling techniques are employed to bring down the temperature of the blade material below its melting point. As shown in Figure 3 in internal cooling, relatively cold air is bypassed from the compressor and passed through the hollow passages inside the turbine blade. In external cooling, the bypassed air is exited out through small holes at discrete locations of the turbine blade. This relatively cold air creates a protective blanket that saves the turbine blade from the harsh environment. This type of cooling is called film cooling.

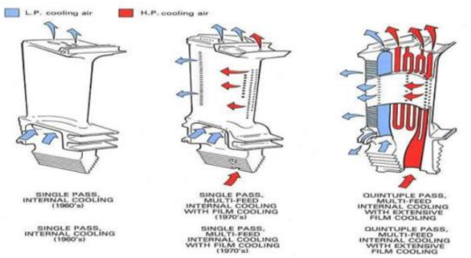

Figure 3: Cooling concepts of a modern multi-pass turbine blade [2]

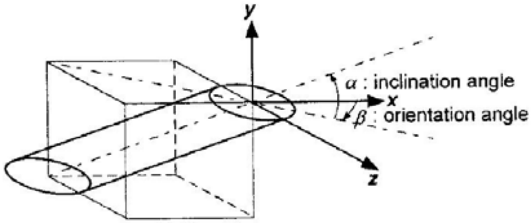

Figure 4: Compound Hole Configuration [3]

Heat transfer coefficient downstream of the film injection is enhanced due to increased turbulence produced by mixing if the coolant jets with the mainstream boundary layer. This increased turbulence locally enhances the heat transfer coefficients. The effect of the coolant jet decreases downstream of injection as the jet structure dissipates and the mainstream dominates the coolant film completely. The high heat transfer coefficient in the near injection region is due to the three-dimensional nature of the jet, and far downstream ($x/D>26$), the jet structure is completely absent and is two-dimensional in nature. The heat transfer coefficient ratio decreases with increasing axial distance from the injection hole. The heat transfer coefficient ratio is almost equal to unity. There is a large body of existing literature on film cooling efficiency with the effect of hole geometry. Some of these studies compared simple angle holes with compound angle holes. The compound angle injection hole has two injection angles, as shown in Figure 4. The inclination angle (α) is defined as the angle between the injection vector and its projection on the x–z plane, whereas the orientation angle (β) is defined as the angle between the streamwise direction and the projection of the injection vector on the x–z plane. In the compound angle orientation system, the coolant is injected with a spanwise momentum, which provides more uniform film coverage and shows higher heat transfer coefficient enhancement.

There are some researches about the multihole film cooling. The configuration of single hole film cooling is illustrated in Figure 5, a large number of holes are drilled on the blade wall, normal or inclined to the wall surface, arranged in a specific pattern. This cooling scheme functions through three processes. First, when the coolant flows through the annular passage, heat is absorbed by the coolant from the hot wall by convection. Second, the coolant passes through the small holes, leading to the so-called internal heat convection to further cool the wall. Third, when the coolant is discharged from the holes, it forms a cooling film. The second process accounts for the higher level of overall cooling effectiveness of this cooling scheme than a conventional slot film cooling scheme. The second cooling process contributes to the same degree provided the same hole diameter and inclination angle as well as the same number of holes per unit area (hole opening density). Therefore, the potential to further enhance cooling lies in the third process as different hole patterns could lead to different adiabatic film cooling effectiveness (Ɛ).

The design practice in the heat transfer community for film cooling has been mostly empirical in nature, relying heavily on a large experimental database. However, there are many parameters and factors that need to be considered. In general, it is important to optimize the amount of coolant for airfoil film cooling under engine operating conditions. It is also important to determine the effects of free-stream turbulence and unsteady wakes on the airfoil film-cooling performance. The turbine-cooling system designers need to know where the heat is transferred from the hot mainstream to the airfoil in order to design better film-cooling patterns for airfoils. These film-hole patterns (i.e., film-hole location, distribution, angle and shape) affect film cooling performance.

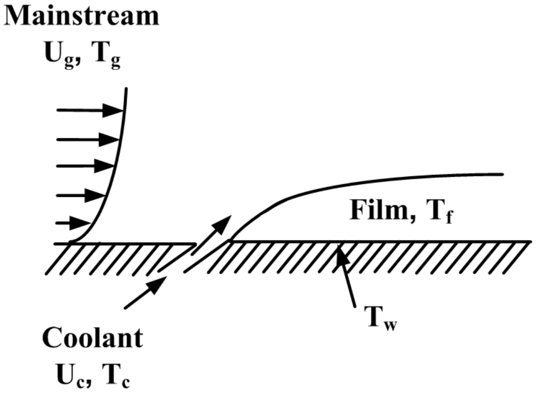

Figure 5: Schematic of film cooling concept

Ammari et al. [3] also presented the effect of density ratio on heat transfer coefficient contours downstream of a film hole inclined 35° along the streamwise direction for two different coolant-to-mainstream density ratios of 1.0 and 1.52 for a coolant blowing ratio of M=1.46. Differences of 10% occurred when coolant densities were changed. It was observed that lower-density injectant provides higher heat transfer coefficient at the same blowing ratio due to higher momentum.

Ekkad et al. [4] provided effectiveness results for two different density ratios. The adopted orientation angles were 0, 45, and 90º. using the transient liquid crystal technique, 10 they reported that compound angle injection produces higher film effectiveness than simple angle injection for both density ratios. They concluded that the highest effectiveness was obtained at a mass flux ratio of 1.0 for compound angle injection.

Bons et al. [5] studied the effect of high stream turbulence on film cooling effectiveness. At high free stream turbulence, heat transfer coefficients with film cooling are not as significantly as the film effectiveness. Film injection by itself produces high heat transfer coefficient enhancement due to high turbulent mixing between jet mainstream.

Lin et al. [6] computed the leading edge film cooling with injection through rows of compound angle holes. They used a low-Reynolds number $k - \omega$ turbulence model in a cell-centered finite volume code called CFL3D. He et al. [10] computed film cooling at the leading edge region. Irmisch [11] simulated film-cooling aerodynamics with unstructured mesh. In a series of publications Garg and Gaugler [12] predicted different aspects of film cooling and Walters et al. [13] predicted jet in cross-flow.

Hole supply geometry typically indicates how the coolant enters the film hole and what happens to the coolant within the film hole. Burd et al. [9] made flow measurement son a flat surface with different hole lengths feeding the film coolant. Leylek and Zerkle [8] indicated that short holes are subjected to a “jetting” effect.

Schmidt et al. [7] measured the film cooling effectiveness using a single row of inclined holes, which injected high-density, cryogenically cooled air. They reported that 60º orientation angle injection at a high momentum flux ratio results in higher effectiveness values than streamwise-directed holes. The forward expansion hole with compound angle orientation showed significantly improved effectiveness.

2.Numerical CFD Simulation

The present Trench CFD studies on a simplified geometry. This is required to understand the sensitivities of features likely to constrain the core/casting definition, and to understand the robustness of the current trench design for geometric (Surface Angle) condition. The effectiveness following single hole of the surface inclination angle of 25° was investigated to achieve the blowing ratios, while maintain pressure ratio is 1.07 across the film duct and plenum inlet Mach number is 0.16.

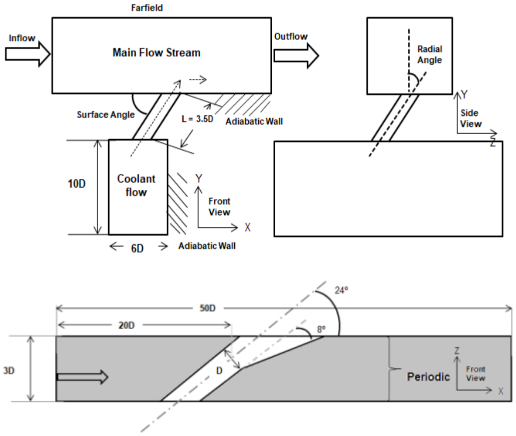

Figure 6: Computational Domain

Study involves the first order sensitivity study of the cooling performance with respect to variation in geometric parameter i.e. surface angle. In total four cases are analysed, each of them with only one variation with respect to the hole diameter. The following dimensions set to prepare the all models are assumed same Hole diameter (D = 0.4 mm), Radial angle (50°), Trench depth (0.8D), Hole length to diameter (3.5D), Layback angle (8°) and Diffuser angle (24°). The mainstream channel wall surface is mainly divided into region, it is separated with trench shaped hole entry region, along x direction: (a) the upstream region, $-20 \leq x \leq 0$ in which only hot mainstream gases are flowing and (b) the downstream region, $0 \leq x \leq 30$, where the film cooling influence has be seen i.e. effective wall. All these models are made using ANSYS Design Modeler. The geometry contains the mainstream channel and plenum channel with the inclined trenched holes. The boundary conditions to run the CFD analysis to be referred in given below Figure 6.

The geometry is a divided into two domain Mainstream channel and Coolant feed plenum and connected with trenched shape hole. The trenched shape hole is inclined at given surface angle and radial angle. The surface angle is between the axis of the film cooling hole and the mainstream surface, projected in the XY plane and The Radial Angle is also shown in figure 6, and it is defined as the angle between the axis of the film cooling hole and the normal to mainstream surface, projected in the plane YZ.

The main stream channel has one inlet and one outlet and entry channel of trench and top surface is defined by free stream wall and both side walls are considered as symmetric and bottom walls are split in three pieces to see the impact of blowing ratio and film effectiveness. Similarly the plenum channel modelled with inlet, outlet and walls with an opening to the trench. There are in total two inlets and outlets namely – Mainstream Inlet, Plenum Outlet and Mainstream Outlet, Plenum Outlet and other surfaces are considered as wall, interface and symmetry. Both the zone separated by interface boundary.

The 3D CFD model shown in Figure 7 is meshed using ANSYS meshing generation suites, to create an unstructured tetrahedral mesh. The model is meshed with ~4.5 million elements for all the cases. The near wall spacing of the element is chosen (0.0075mm) so as to obtain near wall y+ ranging from 1 to 3 and 12 prism layers are used to resolve the boundary layers.

Figure 7: Mesh Generated on Geometry Model

Boundary Conditions and Computation Methods

In this paper, four trench shaped hole with surface angle configuration with different targeted blowing ratio cases have been taken for simulations. All the numerical simulations are performed at fixed surface angle configuration for each targeted blowing ratio so total four cases are studied.

The boundary conditions applied to the problem are extracted from several advanced gas turbine engines to simulate the actual situation of turbine guide vane cooling.

In order to compare the computational results of each case with other cases, the main boundary conditions for all four cases are set to be the same. For the Mainstream, the temperature and Mach number at the entrance of the fluid domain are assumed to be uniform and are set as $T\infty = 1600K$ and $M\infty = 0.4$, respectively. Coolant flow to mainstream flow density ratio is set as 1.54. For the coolant flow, the coolant is ejected at a stagnation temperature that is

400K lower than that of the mainstream to allow the coolant to be traced and the velocity Uc at the entrance is set based on targeted coolant-to-mainstream density ratio of 1.54 and a coolant blowing ratio 0.7.

The exit of mixing flow, i.e. the downstream end face of the fluid domain treated as a “pressure outlet” with pressure set to operating pressure. The inlet boundary conditions for the turbulent quantities are difficult to prescribe. In this study, a turbulence intensity of 2% and a turbulence length scale of 10% hole diameter are specified at the mainstream inlet, and a turbulence intensity of 1% and a turbulence length scale of 10% hole diameter are specified at the plenum inlet.

The computational domain is solved using CFD solver. The flow is assumed to be steady and compressible. The pressure, density and temperature are related by using the ideal gas law. The walls of the model were defined as no-slip, specified shear, hydraulically smooth and adiabatic walls. The extended cavity inlet and outlet surfaces were modelled as free-slip walls, which has zero shears. The k−omega turbulence model is applied to simulate the flow field, which treated the very well near wall region. The discretization formats of the flow and turbulence equations are all set to second order upwind. Each equation is solved using SIMPLEC scheme with under-relaxation. A summary of the convergence parameters detailed below are provided for each case at least 4 orders of magnitude reduction for all residuals, scaled energy residual below 10-6 and the net mass flux imbalance and energy imbalance should be less than 0.001% and 0.01% respectively.

Results and Discussion

The objective of this investigation is to understand the usefulness of placing film duct at right surface angle of a film cooling hole in improving cooling effectiveness, Ɛ. In this study, cooling effectiveness obtained by four cases having fixed surface angle film duct configurations is compared for the blowing ratio cases themselves. The main factors that influence the value of Ɛ are flow parameter such as blowing ratio.

The adiabatic cooling effectiveness, Ɛ of the film cooled plate on the top hot side surface of the plate is defined as Fill Effectiveness

$$Ɛ = ((T_\infty - T_w))/((T_\infty - T_c)) \qquad (1)$$

The film cooling effectiveness, ε is primarily defined by the adiabatic wall temperature, Tw and T∞ and Tc is the hot gas temperature and cooling flow temperature. This wall temperature is the driving temperature potential for heat transfer to the wall and reducing this temperature is the objective of film cooling.

Figure 8: Comparison of Film Effectiveness plots for all cases

Figure 9: Comparison of film cooling contour plots for all cases

Figure 9 show the distribution of lateral averaged adiabatic cooling effectiveness, Ɛ and contours for Case 2 with surface angle, SA=25º for blowing ratio equal to 0.7 and three other blowing ratio cases (Case 1, Case 3 and Case 4) at defined surface angle, SA=25 º respectively. From all these figures it can be seen that case2 for BLR =0.7 trench shaped hole showed improved cooling effectiveness compared to the other blowing ratio cases. The main reason behind this is “jet lift-off” phenomenon.

In Case 2, the cooling effectiveness is quite high near the hole region but as x/d increases, rapid decrease in effectiveness has been observed compared to other cases, except Case 1. In other words, the rate of decrease in effectiveness is quite high for Case 2 compared to other cases. This is because of the fact that, in Case2 when coolant jet comes out of the hole, a pair of kidney vortices set up in the jet. This pair of kidney vortices tries to lift-off the coolant jet from the surface. As jet lift-off occurs, the surrounding hot mainstream air occupies the space underneath coolant jet and pushes the coolant jet further away from the blade surface. Due to this coolant jet detaches from the surface and cooling effectiveness rapidly decreases.

On the other hand, in Case 2 the film cooling effectiveness is found to be higher compared to other cases. The main reason behind this is reduced jet lift-off. In Case 2, jet lift-off is reduced to the great extent because of the fact that, majority volume of the coolant is flowing within the trench shape, so it is properly guided and protected by the shape of hole. This reduced the turbulent mixing between the mainstream and coolant flow. Due to this reason coolant jet did not lose its coolness and retained its coolness throughout the length of the effective wall. This reduced the jet lift-off and improved the cooling effectiveness of Case 2. Compared to other cases, in Case 2, minimum and maximum improvement in film cooling effectiveness has been observed

Hence it can be stated that “Incorporation of Trench shaped hole at surface angle, SA=25º into film cooling system is advantageous” among the cases.

On the other hand, in Case 2 the film cooling effectiveness is found to be higher compared to other cases. The main reason behind this is reduced jet lift-off. In Case 2, jet lift-off is reduced to the great extent because of the fact that, majority volume of the coolant is flowing within the trench shape, so it is properly guided and protected by the shape of hole. This reduced the turbulent mixing between the mainstream and coolant flow. Due to this reason coolant jet did not lose its coolness and retained its coolness throughout the length of the effective wall. This reduced the jet lift-off and improved the cooling effectiveness of Case 2. Compared to other cases, in Case 2, minimum 4% and maximum 30% improvement in film cooling effectiveness has been observed

Hence it can be stated that “Incorporation of Trench shaped hole at surface angle, SA=25º into film cooling system is advantageous” among the cases.

With reference to Figure 8, when comparing lateral averaged cooling effectiveness curves amongst Case 1, Case 2, Case3 and Case 4 with surface angle at SA=25º, it can be seen that cooling effectiveness obtained by Case 2 is higher over a wide range of plate length ($2 \leq x/D \leq 26$) compared to other cases. Even though the highest effectiveness value is obtained for Case 1 near the hole region, Case 2 showed better effectiveness over the wide range of plate. This is because of the fact that, in Case 1, there is a rapid decrease in effectiveness compared to Case 2. In other words, it can be stated that effectiveness values for Case 4 are nearly same as Case 1 & 3 with much variation. Hence it can be concluded that for less inclined hole i.e. hole having average medium surface angle, cooling effectiveness distribution is poor across the effective wall length whereas for minimum surface angle i.e. Case 2, distribution of cooling effectiveness is better than others. From Figure 8 it has been observed that there is a strong influence of angle variation on cooling effectiveness compared to angle variation. Since blowing ratio variation shows significant variation in cooling effectiveness curve.

Conclusion

The present numerical study has highlighted the potential of using trench shaped with varying blowing ratio for film holes for cooling purposes in gas turbines. In this study, influence of trench shaped hole geometry on film cooling effectiveness have also been investigated by varying blowing ratio of trench shaped hole at four different geometry configurations Case1, Case 2, Case 3 & Case 4 for given boundary condition with given surface angle for targeted, BLR. The lateral averaged cooling effectiveness curves obtained by four trench shaped hole for different blowing ratio cases were analyzed and compared among the cases.

After analyzing all the results and graphs, the key conclusions drawn are as follows:

Varying the blowing ratio of trench shaped hole into film cooling system has improved the cooling effectiveness at surface angle, SA=25º over the effective wall, which is Case 2 among the cases at given condition. Compared to other cases, in Case 2, minimum 4% and maximum 30% improvement in film cooling effectiveness has been observed.

In Case 2, it has been observed that cooling effectiveness is less over the trench wall but influence is quite strong over the effective wall compared to other cases due to blowing ratio variations.

At given blowing ratio, BLR=0.7 trench shaped hole with surface angle, SA=25º (i.e. Case 2) showed best overall cooling effectiveness distribution across the length of the effective wall surface.

Highest cooling effectiveness near trench shaped hole region is achieved at targeted blowing ratio. However their effectiveness rapidly decreases with increase in x/d but Case 2 is predicting slightly higher as compared to other cases.

From this conclusion it can be stated that, in order to get maximum cooling effectiveness distribution across the length of the plate using trench shaped at given surface angle, keep the targeted blowing ratio of hole is slightly higher than the minimum.

References

1. Friedrichs, S. “Endwall Film-Cooling in Axial Flow Turbines” Cambridge University Dissertation, 1997.

2. Han, J.C., Park, J.S., and Lie, C.K., 1984. "Heat Transfer and Pressure Drop in Blade Cooling Channels with Turbulence Promoters." Texas A&M University (NASA CR-3837).
3. Ammari,H.D., Hay, N., and Lampard, D., 1990 "The Effect of Density Ratio on the Heat Transfer Coefficient from a Film Cooled Flat Plate." ASME Journal of Turbomachinery, Vol. 112, pp444-450.
4. Ekkad, S. V., Zapata, D., and Han, J. C., 1997, "Film Effectiveness Over a Flat Surface With Air and CO2 Injection Through Compound Angle Holes Using a Transient Liquid Crystal Image Method," ASME Journal of Turbomachinery, Vol.119, pp. 587–593.

5. Bons, J.P., MacArthur, C.D., and Rivir, R.B., 1996, "The Effect of High Free-Stream Turbulence on Film Cooling Effectiveness." ASME Journal of Turbomachinery, Vol.118, pp.814-825.
6. Lin, Y.L., Stephens, M.A., and Shih, T.I., 1997 "Computation of Leading-Edge Film Cooling with Injection through Rows of Compound Angle Holes," International Gas Turbine and Aeroengine Congress & Exhibition, June 1997, Orlando, Fla. ASME Paper 97-GT-298.
7. Schmidt, D. L., Sen., B., 1996, "Film Cooling With Compound Angle Holes: Adiabatic Effectiveness," ASME Journal of Turbo machinery, Vol.118, pp. 807–813.
8. Leylek, J.H., and Zerkle, R.D., 1994. "Discrete-jet Film Cooling: A Comparison of Computational Results with Experiments." ASME Journal of Turbomachinery, Vol. 116, pp358-368.
9. Burd, S.W., and Simon, T.W., 1997, "The Influence of Coolant Supply Geometry on Film coolant Exit Flow and Surface Adiabatic Effectiveness." ASME Paper 97-GT-25.
10. He, P., Salcudean, M., and Gartshore, I.S.,1995, "Computations of Film Cooling for the Leading Edge Region of a Turbine Blade Model," International Gas Turbine and Aeroengine Congress & Exhibition, June 1994, Houston, Tex. ASME Paper 95-GT-20.
11. Irmisch, S., 1995, "Simulation of Film-Cooling Aerodynamics with a 2D Nvier-Stokes Solver Using Unstructured Meshes," International Gas Turbine and Aeroengine Congress & Exhibition, June 1995, Houston, Tex. ASME Paper 95-GT-24.
12. Garg, V.K., Gaugler, R.E., 1994, "Prediction of Film Cooling on Gas Turbine Airfoils," International Gas Turbine and Aeroengine Congress & Exhibition, June 1994, The Hague, Netherlands. ASME Paper 94-GT-16.
13. Walters, D.B., Mcgovern, K.T., Butkiewicz, J.J, and Laylek, J.H., 1995 "A Systematic Computational Methodology Applied to Jet-In-Crossflow. Part 2; Unstructured/Adaptive Grid Approach," ASME International Mechanical Engineering Congress & Exhibition, Nov 1997, San Francisco, Calif. ASME Paper 97-WA/HT-2.
14. Kelso, R. M., Lim, T. T., and Perry, A. E., "An Experimental Study of Round Jets in Cross-Flow," J. Fluid Mech., 306, pp. 111–144, 1996.
15. Han, J.C., Park, J.S., and Lie, C.K., 1984. "Heat Transfer and Pressure Drop in Blade Cooling Channels with Turbulence Promoters." Texas A&M University (NASA CR-3837).