International Journal of Modern Engineering Research (IJMER)
www.ijmer.com Vol. 3, Issue. 4, Jul - Aug. 2013 pp-2066-2072 ISSN: 2249-6645

CFD Simulation on Gas turbine blade and Effect of Hole Shape on leading edge Film Cooling Effectiveness

Shridhar Paregouda[1], Prof. Dr. T. Nageswara Rao
[1] *M. Tech Student, Oxford College of Engineering,Bangalore – 560068, Karnataka*
[2] *Professors, Department Of Mechanical Engineering, Oxford College of Engineering, Bangalore– 560068, Karnataka*

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. The highest thermal load usually occurs at the leading edge of the airfoil, and failure is likely to happen in this region. Film cooling is typically applied to the leading edge through an array of hole rows called showerhead. In this project initially benchmarks study the current state of heat transfer prediction for commonly used CFD software ANSYS Fluent. The predictions Reynolds-Averaged Navier-Stokes solutions for a baseline Flat film cooling geometry will be analyzed and compared with experimental data. The Fluent finite volume code will be used to perform the computations with the realizable k-ε turbulence model. The film hole is angled at 30° to the crossflow with a Reynolds number of 17,400. The focus of this investigation is to investigate advanced cooling hole geometries on film cooling effectiveness over flat surface. Three film-cooling holes with different hole geometries including a standard cylindrical hole and two holes with a diffuser shaped exit portion (i.e. a fanshaped and a laidback fanshaped hole) will be studied. Finally optimized shape of the hole configuration is included in NASA Mark II vane turbine geometry to study heat transfer characteristics of blade.*

I. INTRODUCTION

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 Fig 1. 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 [1-2].

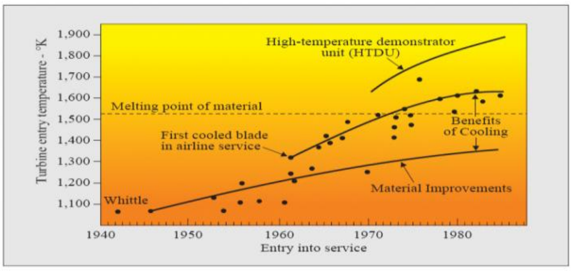


Fig.1 Development of Turbine Inlet Temperature [2]

Various internal and external cooling techniques are employed to bring down the temperature of the blade material below its melting point. As shown in Fig. 2 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.

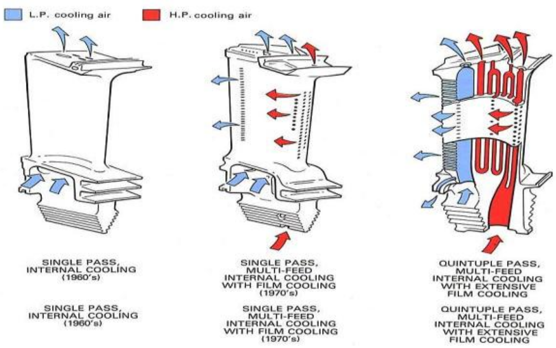


Fig.2 Cooling concepts of a modern multi-pass turbine blade [3]

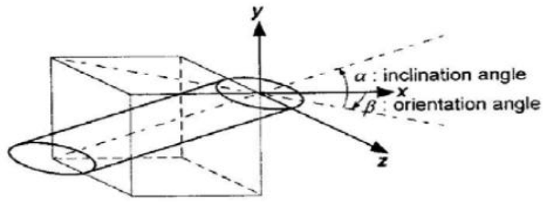


Fig.3 Compound Hole Configuration [6]

Flat surface models can be used to study the effects of individual parameters with relative ease and are less expensive. Early studies have proved that the results obtained on simple flat-surface models can be applied to real engine design with slight corrections. The effects of geometrical parameters (hole geometry, shape, size, spacing) and flow parameters (coolant-to-mainstream mass flux, temperature ratio, mainstream Reynolds number, velocity, etc.) have been studied on flat surface. Also, the effects of pressure gradient and curved surface have also been studied. Some studies have focused only on the heat transfer coefficient enhancement and others have presented only film effectiveness results. 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>15), 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. About 15-hole diameters downstream of injection, the film cooling effect disappears. 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 Fig 1.6. 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.

The motivation behind this study is to develop and test innovative film cooling hole geometries on film cooling heat transfer and cooling effectiveness over flat and turbine airfoil surfaces. The basic aim of these proposed geometries is to alter the mainstream to directly contact the test surface. Simulations will be conducted to study the effect of hole embedded in transverse slots and hole exit area. Several variations of geometry will be investigated to assist in future design changes. Also, the numerical prediction using FLUENT was performed to determine the jet mainstream interactions to better understand the interaction between the ejected coolant and the second flow and also the surface film effectiveness distributions.

Schmidt et al. [11] measured the film cooling effectiveness using a single row of inclined holes, which injected high-density, cryogenically cooled air. They reported that 60 deg 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.

Ekkad et al. [12] provided effectiveness results for two different density ratios. The adopted orientation angles were 0, 45, and 90 deg. using the transient liquid crystal technique, 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.

Ammari et al. [13] 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.

Bons et al.[14] 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. Several investigators have studied the slot film cooling. Blair [15] investigated the slot film cooling at the entry of a vane cascade endwall. Chyu [16] et al. provided film effectiveness measurements downstream of 2-D slots modeled as gasp leakages, both aligned and misaligned. Bunker [17] investigated film effectiveness for geometries wherein the coolant from discrete holes enters a slot before mixing with the mainstream. Basically, the angled holes are entrenched in a shallow trench. The holes embedded in the trench provided higher film effectiveness distributions than the ones on the plane surface. However, Bunker [17] provided only film effectiveness distributions and also the hole had a compound angle (radial injection) in the lateral direction. Bunker [17] based their study on an earlier study by Wang et al. [18]. Lu et al.[19] studied the effect of trench exit area and edge shape on film cooling performance using an IR thermography method. Their results showed that the film cooling holes provide higher film effectiveness when embedded in a trench. However, in some geometries when the trench began at the upstream edge of the hole, the film effectiveness diminished. The heat transfer coefficient enhancement due to the embedding was not significantly higher compared to the typical unembedded cylindrical hole. The overall heat flux ratio comparing film cooling with embedded holes to unembedded holes shows that the full trench and downstream trench spacing after the hole exit produce the highest heat flux reduction.

II. CFD SIMULATION OF FLAT PLATE FILM COOLING

In order to obtain better design in CFD, The mixing of a 3D cooling hole in a turbulent flow is more complicated to analyze than in the 2D slot. Downstream of the hole, the air flow usually form circular motions or vortices when a jet enters at high angle or when the jet velocity is higher the than main stream velocity. Figure below shows a schematic of this phenomenon. Note there are wake vortices between the jet and wall top surface that would affect heat transfer between them.

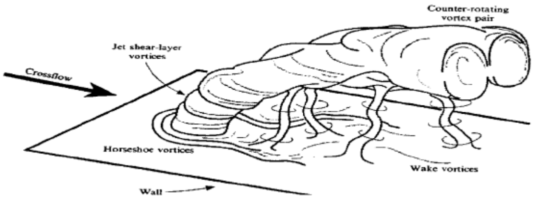


Fig.4 Flat plate Film cooling

There are many factors involved when designing an optimized cooling hole pattern such that the expense of provide cooling air flow to these holes is minimized while providing adequate reduction in metal temperature. The diameter, length, surface angle, and arrangement of holes against the hot gas flow direction are important geometric parameters in designing these holes. Besides, the clearance to internal cooling passage walls and practical manufacturing methods are also important considerations. However, this project will only study on the effect of the geometric parameters on the cooling holes effectiveness.

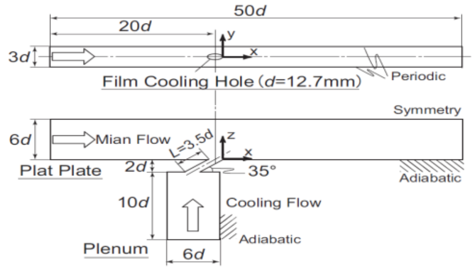
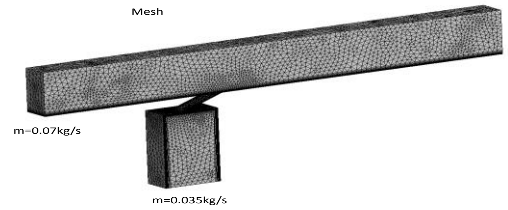


Fig.5 Computational domain

The geometry for the annulus is created in ANSYS Design Modeler and meshing is done in ANSYS Meshing, with grid size 150x150 as shown in Fig 1.4. The mesh nearby to walls is fine meshed to cope-up the thermal and velocity boundary layer formation and at the centre it is coursed meshed.



Blowing ratio, BR	0.5
Main stream Reynolds number, Re_m [-]	16400
Main stream static temperature, T_m [K]	298
Cooling air static temperature, T_c [K]	188

CFD-Mesh
No of elements-0.5 millions

Fig.6 Boundary conditions and CFD Meshing

A tetrahedral grid is used in this study to allow the highest quality in all regions with the prism boundary layers. In this way, the computational domain can be partitioned into several subsections. Jet Cooling hole, mainstream flow duct and other three in the channel. Each section is meshed with appropriate topology. In order to resolve the mean velocity, mean temperature, heat flux and turbulent quantities in the viscosity-affected near wall region (Rey <200) accurately, the near wall turbulence model requests that y+ value at the wall-adjacent cell should be the order of one and there are at least 4 ten cells in this region. So in the hole and the region near the test wall the density of cells is identified to satisfy this requirement. After a series of tests and adjustments the final adopted grid for calculations is obtained. Total no of elements is 0.5 millions.

III. RESULTS AND DISCUSSIONS

Figure 7 shows the jet at the hole exit by means of contours of velocity at XY-plane going through the first row hole. Indeed, they are utilized to illustrate how the jet will interact with freestream flow for 0.5 blowing ratio. At the hole exit there is also strong recirculation zone for low BR. By increasing the coolant mass flow this zone starts to be dissipated, see recirculation zones. Below this zone higher temperature compared with the temperature of recirculation can be seen, which can be seen from contours of temperature, figure 9 at the hole exit. In addition, as BR raises and the jet will become stronger, the lift is increased and the coolant will be detached from the surface. As a result at this region, close to the hole for high BR higher temperature can be seen increase further. Fig. 6 shows the Temperature contours at 45 and 90 min. Fig 7, 8 and 9 shows the velocity vectors in evaporator, adiabatic and condenser section. It shows the in adiabatic section the velocity constant and compared to other section.

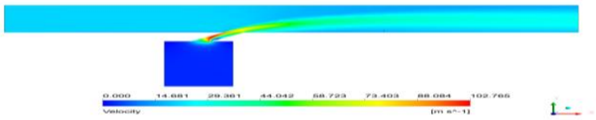


Fig. 7 Pressure contours of film cooling

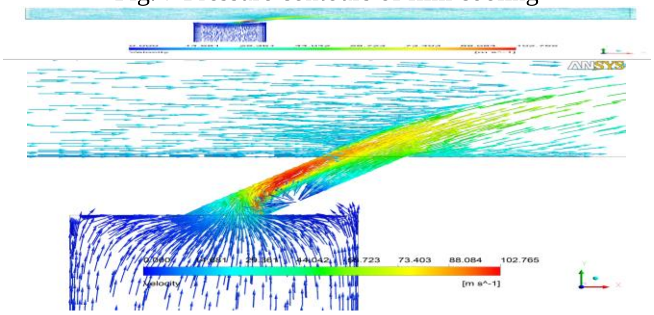


Fig.8 Velocity vectors contours of film cooling

Fig.8 velocity vector map at the slot exit is indicates. After the jet leaves the slot, air particles have high velocity and are projected deep into the main stream, especially those near the right wall of the slot. This band of high velocity sub-stream creates a boundary between the hot main stream and the cooling jet stream. Below this boundary, the mixing air moves relatively slow and circulates around creating a stagnation region.

Fig. 9 Velocity vectors at adiabatic section

Figure 9 shows the temperature contours for the 35 degree slot model with refined mesh. Due to it vertical entrance, the cooling jet disturbs the hot stream strongly and creates a large mixing region high above the alloy surface. So the cooling effect is contained in the air and leaves the heat transfer into the alloy to a minimal level. The cooling jet doesn't create a film after it exits the slot so the effective cooling film length is zero. However, the temperature contours show that the jet does reduce the alloy top surface temperature in some amount, about 100K after a distance 1D.

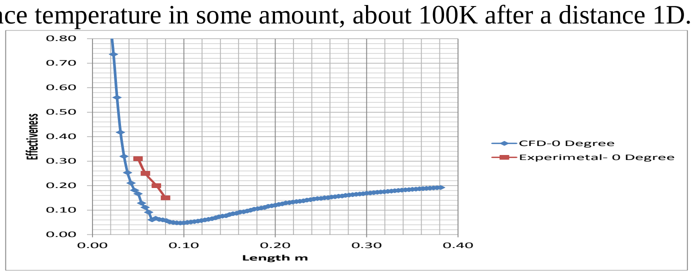


Fig. 10 Comparison of film cooling effectiveness

Averaged film cooling effectiveness at BR=0.5. The k-ω SST model was adopted in the RANS simulations Fig.5.7. The results of previous experiments [39] are also plotted in the figure. Experimental conditions of the reference [39] are almost identical to the present computation. Good agreement is shown among the numerical results except just downstream of film cooling hole.

IV. OPTIMIZATION OF ANGLE

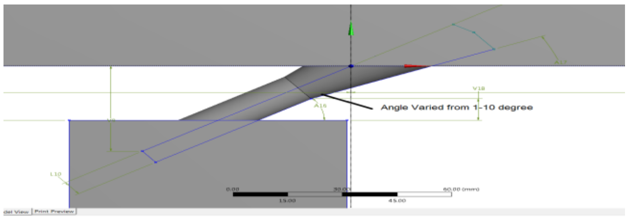


Fig. 11 Optimization of Angle

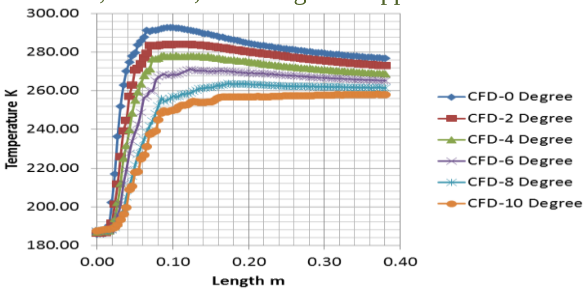


Fig.12 Comparison of Temperature contours of film cooling-0-10^0

The Tw is evaluated by taking the average of all temperature values at nodes along the top surface of the alloy after the slot exit, 0.381m downstream, the same for all models to get a consistent comparison. Table 3 lists the alloy surface average temperature reduction. This temperature reduction is the difference between the Tw and the averaged temperature values along the top the alloy top surface before the slot. And finally, the average cooling film effectiveness or □is calculated per the formula above. The higher □means the cooling film is more effective, with the ideal max value of 1. As expected, the model with 10 degree exit angle gives the best film effectiveness. Also, the table shows the difference in results between coarse and fine meshes. In general, fine mesh produces slightly lower values except the case of 10 degree exit angle. The 10 degree exit angle, in practice, has been used widely in practical applications, such as turbine airfoils. Figure 18 shows temperature contours for the 10 degree exit angle. The cooling film is thin and attaches right after the jet exits the slot, producing a long cooling thermal layer above the alloy surface. The film length where the cooling jet effectively maintains its coolant temperature, shown in dark blue, is approximately 7D. In the case of 8 degree exit angle, this length is 4.5D while the 0 degree exit angle produces no length because the cooling jet detaches from the alloy surface as soon as it leaves the slot. The temperature contours in the alloy shows most of the alloy benefits from the temperature reduction due to the cooling film, with the average temperature reduction of 249K on top surface

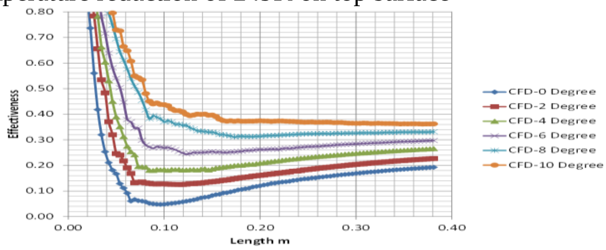


Fig. 13 Comparison of Effectiveness contours of film cooling-0-10^0

V. CFD ANALYSIS OF C3X VANE

The experimental data for validation of the conjugate simulation of flow and heat conduction need to provide clear boundary conditions and cooling air flow rates of high accuracy. The C3X transonic turbine guide vane was selected as the test case, for this work provides detailed measurement of the external and internal convection and the metal surface temperature. The experimental facility consisted of a linear cascade of three C3X turbine vanes.

The computational domain included one vane in the middle of the flow field, with periodic boundary conditions employed to simulate cascade flow as shown in Fig.14. The computational domain inlet is located one chord length upstream of the leading edge, where the turbulence level was measured in the experiments. The outlet is located one chord length downstream of the trailing edge. Meshes are created for the hot gas path and the solid vane. Geometry is created in DM create computational domain. The Extracted fluid domain of C3X vane as shown in Fig.14

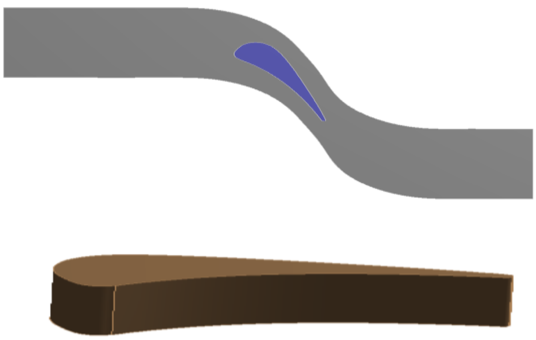


Fig. 14 C3X Turbine Vane

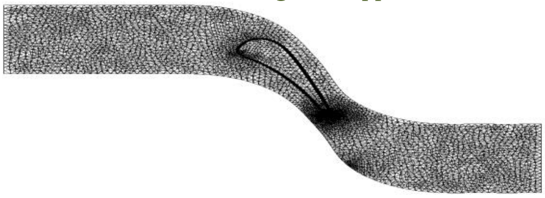


Fig.15 Turbine Blade Meshing

The 3D simulation used unstructured grid (shown in figure 15), increasing the grid density around wall in the fluid domain and the solid domain. Total number of elements was 0.6. Total number of elements in the fluid domain was 3121480 and total number of elements in the solid domain was 2laks. The boundary conditions were shown in table 3. The boundary conditions of cooling channel were given cooling temperature and total pressure boundary conditions.

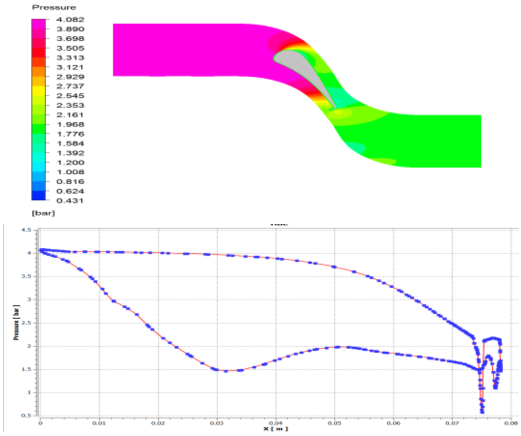


Fig.16 Contours of static Pressure at mid plane

Fig 16 shows the pressure distribution on mid plane of Gas turbine vane. On the suction side (SS), the gas flow accelerates rapidly from the stagnation point toward the throat, reaching the maximum speed around 500m/s. The flow then decelerates until the location around 65 % chord, before resuming a mild acceleration toward the trailing edge (TE). The flow is under a favorable pressure gradient on the entire pressure side (PS). The pressure stays almost constant near P0 from the LE to about 50% chord. and then falls off with further distance toward the TE as shown in Fig.16.

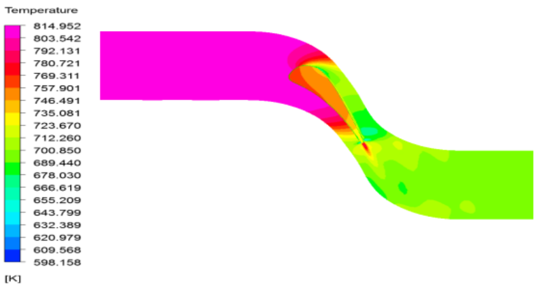


Fig.16 Contours of Temperature at mid plane

7. Shower head cooling

In the drive to increase cycle efficiency, gas turbine designers have increased turbine inlet temperatures well beyond the metallurgical limits of engine components. In order to prevent failure and meet life requirements, turbine components must be cooled well below these hot gas temperatures. Film cooling is a widely employed cooling technique whereby air is extracted from the compressor and ejected through discrete holes drilled in the surface of turbine airfoils, tips, and end walls. The air leaving these holes forms a film of cool air on the component surface which protects the part from hot gases exiting the combustor.

The showerhead-cooled vane used in this study had five rows of staggered cylindrical holes. The center row was placed at the geometric stagnation point while the remaining rows were spaced at four and eight hole diameters on either side. A cylindrical plenum supplied air to all five rows, and each row was spaced three hole diameters apart within the plenum as seen in Figure.17. The analysis is carried out for below conditions. The colling hole mass flow rate

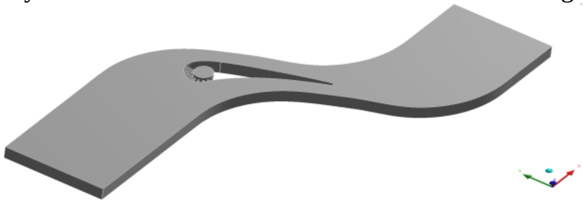


Fig.17 C3X vane with shower head cooling Hole

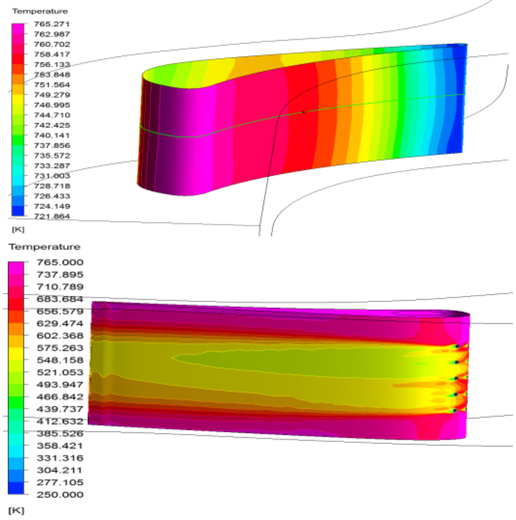


Fig.18 Temperature contours-With and without showerhead cooling holes

VI. CONCLUSIONS

Turbine blade edge regions, including blade tip, leading edge and the platform, experience a high heat load. The earliest film-cooling correlations were made for continuous slot cooling configurations. The slot correlations were then extrapolated to discrete hole film-cooling with cylindrical holes. Advances in film-cooling technology have led to the extensive use of shaped non-cylindrical exit holes in turbine applications. Numerical investigations have been applied on the flow and film cooling for these critical regions. The compressible Navier-Stokes equations, K-epsilon turbulence model together have been solved by Fluent in these studies. . The film hole is angled at 30° to the cross-flow with a Reynolds number of 17,400. It shows the film cooling effectiveness tends to be under-predicted in the value. 3D CFD models of the film cooling processes have shown that by increasing the exit angle of the cooling hole would increase the film effectiveness and in turn reduce the alloy surface temperature. Outside this study, many research and experiment papers analyzed a 30 degree angle with possible lateral compound angle to optimize film coverage. The main interest in other experiments has been the flow field of the cooling jet mixing with main stream while heat transfer is secondary. However, many results of this study are in good agreement with published work.

In all models studied, the 35 degree exit angle in 2D slot model reduced the alloy surface temperature by 15% while in 3D discrete hole model the film coverage length is 9D for 100K (6%) temperature reduction. In reality, the machine used for hole drilling (usually laser) and complex 3D surfaces of the part are the factors limiting the exit angle to be around 30 degrees. For additional heat transfer benefits, a lateral angle in combination with incline (exit) angle and a diffuser at hole exit should be used to give more film coverage.

Then the the showerhead-cooled vane used in this study had five rows of staggered cylindrical holes. The center row was placed at the geometric stagnation point while the remaining rows were spaced at four and eight hole diameters on either side. A cylindrical plenum supplied air to all five rows, and each row was spaced three hole diameters apart within the plenum. This shows 200K reduction in temperature compared to without cooling hole model.

REFERENCES

[1]. http://www.andrew.cmu.edu/
[2]. Friedrichs, S. "Endwall Film-Cooling in Axial Flow Turbines" Cambridge University Dissertation, 1997
[3]. 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 (prepared for
[4]. NASA CR-3837)
[5]. Hay, N. and Lampard, D., 1996, "Discharge Coefficient of Turbine Cooling Holes: A Review." ASME paper 96-GT-492
[6]. Goldstein, R.J., 1971. "Film Cooling" In *Advancement in Heat Transfer.* Academic Press, New York, Vol.7, pp321-379
[7]. Lin, Y.Z., Song, B., 2006, "Measured Film Cooling Effectiveness of Three Multihole Patterns", *Journal of Heat Transfer*, Vol. 128, pp.192-197
[8]. Jung, I.S., Lee, J.S., 2000, "Effects of Orientation Angles on Film Cooling Over a Flat Plate: Boundary Layer Temperature Distributions and Adiabatic Film Cooling Effectiveness", *Journal of Turbo machinery*, Vol.122, pp. 153-160
[9]. Goldstein, R. J., Eckert, E. R. G., Eriksen, V. L., and Ramsey, J. W., 1970, "Film Cooling Following Injection Through Inclined Circular Tubes," *Israel Journal of Technology*, Vol.8, No. 1–2, pp. 145–154.
[10]. Mehendale, A.B., Han, J.C. and Ou, S., "Influence of High Mainstream Turbulence on Leading Edge Heat Transfer," *ASME Journal of Heat Transfer*, Vol. 113, November 1991,pp. 843-850.
[11]. Honami, S., Shizawa, T., and Uchiyama, A., 1994, "Behavior of the Laterally Injected Jet in Film Cooling: Measurements of Surface Temperature and Velocity/ Temperature Field Within the Jet," *ASME Journal of Turbo machinery*, Vol. 116, pp. 106–112.
[12]. Schmidt, D. L., Sen., B., 1996, "Film Cooling With Compound Angle Holes: Adiabatic Effectiveness," *ASME Journal of Turbo machinery*, Vol.118, pp. 807–813.
[13]. 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 Turbo machinery*, Vol.119, pp. 587–593.103.