Hole Configuration effect on Turbine vane leading edge Film Cooling effectiveness

Rahul.H.Kumar, K.S.Santhosh
Dept. Of Mechanical Engineering
College of Engineering, Trivandrum
Thiruvananthapuram, India
rahulhkumar02@gmail.com
santhosh.cet@gmail.com

***Abstract*— This study aims at numerical investigation of the effects of hole configurations for the film cooling on the leading edge of a C3X nozzle guide vane, under conjugate conditions. The analysis is carried out using the commercial CFD code FLUENT, with ANSYS Workbench 14.0. Since the study primarily focuses on the effects of hole position and geometry, the film cooling is carried out in 2-D. An internal convective cooling numerical study on the vane validates the use of turbulence model k-ω SST for the chosen problem. A classical flat plate case is also carried out for the film cooling numerical study validation. The stream-wise cooling effectiveness of five hole shapes at two different blow ratios, at three discrete locations centered at the stagnation region on the leading edge of the vane are compared and analyzed.**

Keywords—Turbine vane, Film cooling, Hole configurations, Cooling effectiveness.

I. INTRODUCTION

Film cooling technique is the most widely used technique for the cooling of turbine vanes and combustion chamber linings in propulsion systems. In film cooling method, the cool air taken from the last stage of compressor is ejected through holes on the surface of the blade, which creates a film of cool air on the blade surface. This film of cool air insulates the blade from the incoming hot flow. Hence a temperature gradient develops between the vane surface and the hot mainstream. The performance of film cooling depends upon many factors including hole shape and inclination, coolant to mainstream mass flux ratio or blow ratio, density ratio, temperature ratio, free stream turbulence etc. Film cooling effectiveness η is used to express the film cooling phenomena quantitatively.

The present study aims to numerically investigate the effects of hole geometry and location on the film cooling effectiveness on the leading edge of a C3X Nozzle Guide Vane in a two dimensional transonic cascade. The C3X vane is selected for the analysis, because the profile has been investigated in many of the previous studies, so that comparisons can be made easily. The commercial CFD solver FLUENT is used for the current study.

II. LITERATURE REVIEW

In the case of internal convective cooling, Grzegorz Nowak et.al [1], 2011 has conducted a numerical simulation of 3D Conjugate Heat Transfer problem on NASA C3X turbine vane which is cooled using ten radial cooling holes. For this they have used commercial CFD code ANSYS CFX as the solver and ICEM CFD for mesh generation. SST k-ω model was used for the purpose. Predicted results of wall temperature distribution and static pressures are validated with the experimental data from L.D Hylton et al [2], 1983.

Rajesh Kumar et.al [3], 2012 has done a comparative study of film cooling effectiveness on a flat plate. They have used k-ω SST turbulence model and Fluent solver for the purpose. The shaped hole geometries discussed in it are taken for the present study.

More recently, Falah et.al [4],2013 studied about the enhancement of flat plate adiabatic film cooling effectiveness when using conical shaped hole using ANSYS FLUENT. The film cooling effectiveness of conical hole is found better than that of its cylindrical counterpart particularly at high blowing ratio. The computational results of cylindrical holes are compared with the existing experimental results given by Durham et.al [5],2004, Kohli and Bogard [6],1997.

Shridhar et.al [7], 2013 numerically investigated 3D Conjugate simulations of C3X high pressure nozzle guide vane with showerhead leading edge film cooling. The computation model for numerical simulation is built based on the experimental study of E.R.Turner, et al.[8] in NASA,1985.

From the previous studies, the importance of shaped holes in the film cooling performance has been clearly understood. Many of the researchers have suggested the application of them on the turbine vanes, but only a few have tried the problem. Moreover,the act of injecting coolant into a stagnation region rather than a cross flow and the expectation of typically much larger heat loads, brings importance to the turbine leading edge film cooling. This hence being an active research area now and the same is taken for the present study.

III. GOVERNING EQUATIONS

The compressible flow governing equations solved for the present analysis are provided in the index notation as follows. The k-ω SST turbulence model, as found superior in similar studies [1], [3] is used for the simulations.

Mass conservation:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \tag{2}$$

Energy equation:

$$\frac{\partial}{\partial x_i}\left[u_i(\rho E + p)\right] = \frac{\partial}{\partial x_j}\left[\left(k + \frac{c_p \mu_t}{Pr_t}\right)\frac{\partial T}{\partial x_j} + u_i\left(\tau_{ij}\right)_{eff}\right] + S_h \tag{3}$$

Equation of state: $P = \rho RT$ (4)

k-ω SST Transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + \widetilde{G_k} - Y_k + S_k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega + S_\omega \tag{6}$$

A. Film Cooling Parameters

- Blow ratio, B.R: It is defined as the coolant to mainstream mass flux ratio.

$$B.R = (\rho_c U_c)/(\rho_\infty U_\infty) \tag{7}$$

where ρ_c, ρ_∞ are the coolant and mainstream densities respectively. U_c and U_∞ are the coolant and mainstream velocities. In the present study, two blow ratios 1.0 and 4.0 are taken for the study.

- Density ratio, D.R: It is defined as the coolant to mainstream density ratio. The value of density ratio is kept at 1.0 for all the cases in this study.

$$D.R = \rho_c/\rho_\infty \tag{8}$$

The density ratio is kept equal to 1.0 for all cases in the present study.

- Temperature ratio, T.R: It is defined as the coolant to mainstream temperature ratio. A temperature ratio of 0.5 is used for the study.

$$T.R = T_c/T_\infty \tag{9}$$

where T_c is the coolant inlet temperature and T_∞ is the mainstream inlet temperature. A temperature ratio of 0.5 is used in the present study.

- Film cooling effectiveness, η : It is an important parameter that quantifies the film cooling performance in each case. It is defined as follows:

$$\eta = \frac{(T\infty - Tw)}{(T\infty - Tc)} \tag{10}$$

where T_w is the wall temperature obtained. Its value lies within 0 and 1, where value of 1 corresponds to the maximum cooling effectiveness when T_w becomes equal to T_c.

IV. COMPUTATIONAL DOMAIN

The C3X vane coordinates are obtained from L.D Hylton et.al [2] and modeled using AutoCAD. To limit the dependency of the solution on the inlet and outlet positions, the domain extends up to 2.5axial chord lengths C, upstream and downstream the vane.

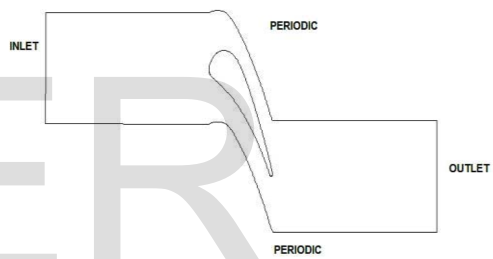


Fig. 1. C3X Computational domain

The computation domain is divided into main flow, secondary or coolant flow and the solid domains. Total pressure and temperature values are given at the inlet and static pressure condition at the outlet. Periodic conditions are provided to replicate the multiple vane passages.

V. VALIDATION STUDIES

A. Internal convective cooling

The 3-D model consists of ten radial internal cooling passages distributed over the C3X vane profile, consistent with the work referred in [1] with boundary conditions taken from experimental run no.158 of L.D Hylton et.al [2]. Steady compressible flow simulation with k-ω SST turbulence model is used for the purpose.

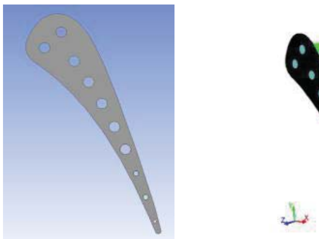


Fig. 2. 3D model with internal cooling channels

The mesh was created in the ANSYS Design Modeler. A hybrid mesh with prismatic cells in the boundary layer near the walls and tetragonal cells in the rest of the domain is made. The maximum y+ was set to less than 2 for all the wall boundaries.

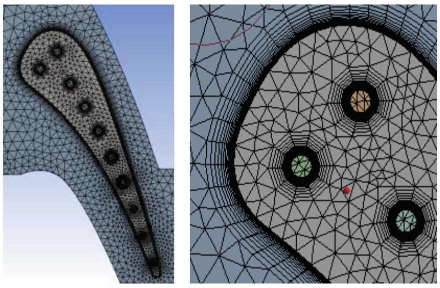


Fig. 3. Hybrid mesh created in Design Modeler

The test case data taken from [2] are given below.

TABLE I. MAIN STREAM CONDITIONS FOR THE INTERNAL COOLING

Inlet Total pressure	Inlet total temperature	Inlet Mach No.	Inlet turbulent intensity	Inlet Reynolds No	Outlet Reynolds No.	Outlet Mach No.	Outlet static pressure
243700 Pa	808 K	0.17	8.3%	$0.38 x 10^6$	$1.47 x 10^6$	0.91	142529.92 Pa

TABLE II. COOLANT AIR CONDITIONS

Inlet Total Pressure	Inlet total Temperature	Inlet turbulent intensity	Outlet static pressure
300000 Pa	375 K	10%	275000 Pa

For all the cooling passages equal boundary conditions were assumed. The vane material is ASTM310 stainless steel, same used in the experiment [2] with the following properties:

Thermal conductivity, k = 0.020176 x T + 6.811
Density, ρ = 7900 kg/m^3
Specific heat, Cp = 586.15 J/kg.K.

The obtained numerical results are compared with centerline experimental distribution of static temperature along pressure and suction surfaces of the vane. The static pressure (P_S) is non-dimensionalized with total pressure (P_t) and wall distance is expressed as a percentage of axial chord C. The predicted pressure distribution matches well with the experimental data as shown in Fig. 4. Wall temperature distribution is validated with the experimental data. Temperature distribution also shows the same trend, but some discrepancies are found especially in the suction side of the vane. Though some limitations are there in modeling the transition at the suction side, the study validates the use of k-ω turbulence model for the chosen problem.

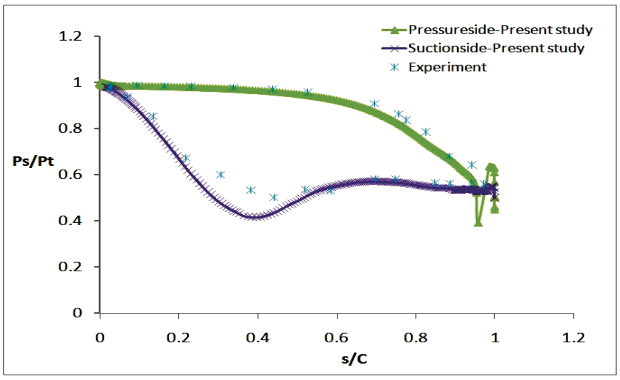


Fig. 4. Static pressure distribution comparison.

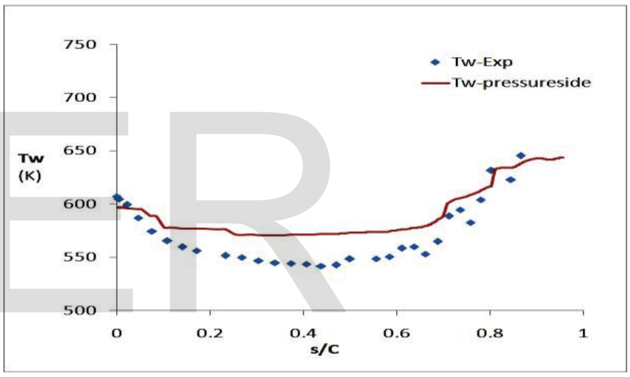


Fig. 5. Pressure side static wall temperature distribution

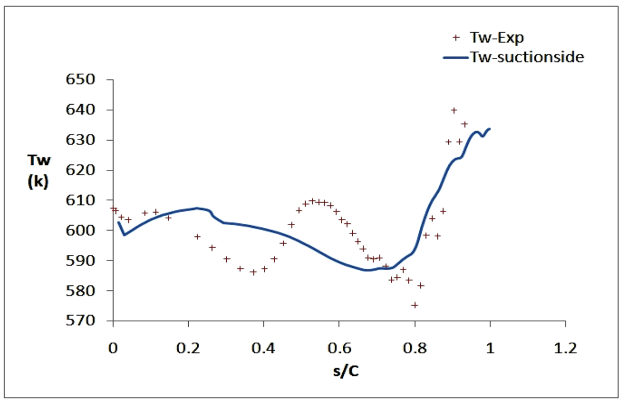


Fig.6. Suction side static wall temperature distribution

B. *Flat plate film cooling*

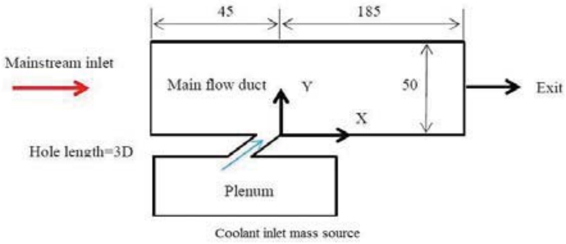


Fig.7. Flat plate model schematic

TABLE III. TEST CASE DETAILS OF THE FLAT PLATE CASE

Mainstream inlet velocity	Mainstream Temperature	Mainstream inlet turbulence intensity	Coolant inlet Temperature
20 m/s	323 K	2%	298 K.

The geometry is taken from [4], hole diameter is 5mm and a length of 15mm, which is inclined at 35° to the stream-wise direction. A mass source is used instead of velocity-inlet condition for the coolant. A steady incompressible flow simulation with standard k-e turbulence model is carried out at two blow ratios 0.5 and 1.0. The numerical results are compared on the basis of centerline-averaged film cooling effectiveness and matched well with experimental results.

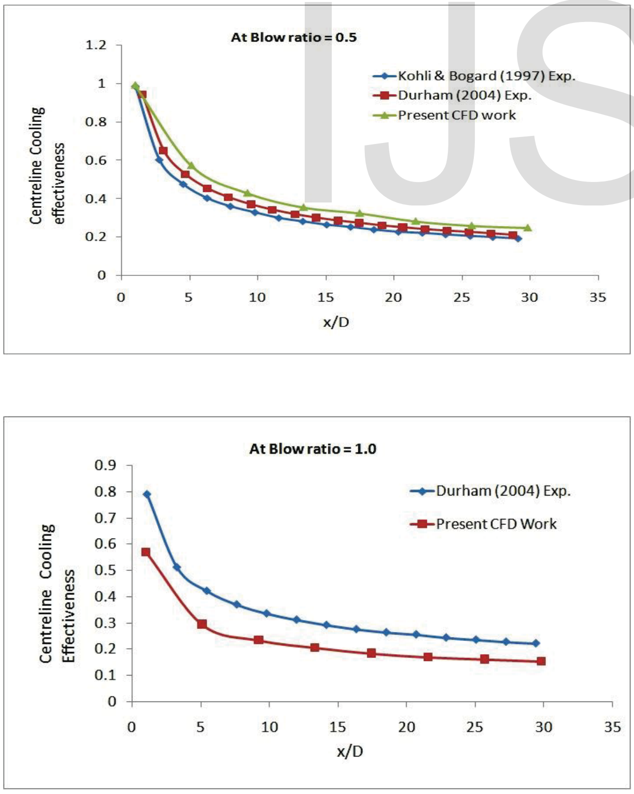


Fig. 8. Flate plate centre-line cooling effectiveness comparison at blow ratios 0.5 and 1.0 with experimental results.

VI. C3X LEADING EDGE FILM COOLING

The present study focuses on the film cooling at the leading edge region of the C3X turbine nozzle guide vane. For this, a pressure-side, a suction-side and the stagnation positions are selected as shown in the Figure 9, below. Holes are positioned eight hole-diameters apart from each other. Separate circular coolant chambers are provided for each of the hole locations.

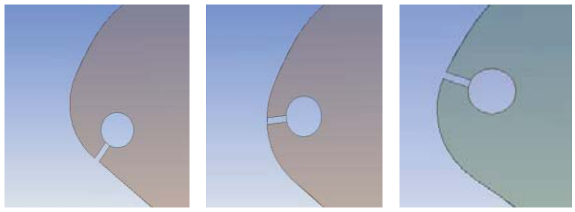


Fig.9. Hole positions studied: (from left) pressure-side, stagnation and suction-side positions

Film hole dimensions and the test case for the study are taken from the experimental study of E.R.Turner et al [8].

Hole diameter, D = 0.099cm.
Hole length, L = 3.4 D = 0.335cm.
Coolant chamber diameter, Dc = 6 D = 0.6cm

A. *Hole shapes used in the study*

Five different hole shapes that have gained recent attention in researches, from [3] are taken for this study. They include:

1. Cylindrical hole oriented at 90deg.
2. Cylindrical hole oriented at 35deg in the stream-wise direction.
3. Shaped hole.
4. Trenched Cylindrical.
5. Shaped & Trenched Cylindrical as shown in Fig.10 below.

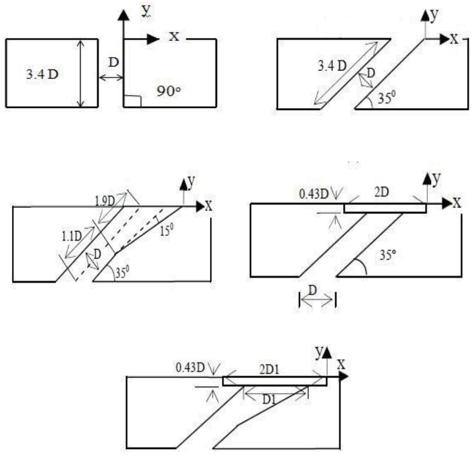


Fig.10. Schematic of the hole shapes used in the study

These hole shapes are created at each of the three positions studied for the comparison of their performance on the basis of streamwise cooling effectiveness. The implementation of the five holes at the stagnation position is shown in the Fig.11.

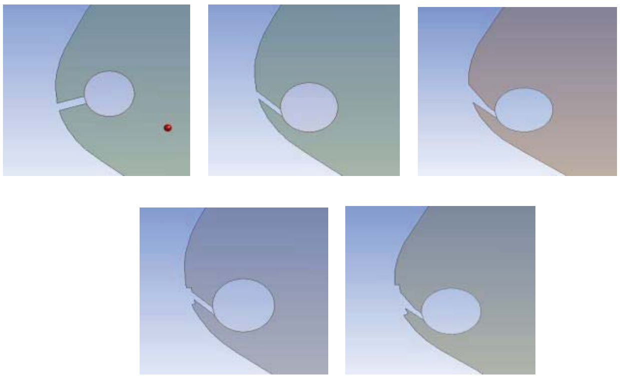


Fig.11.The hole shapes created at the stagnation position.

A mass source is defined for the coolant inlet (plenum), where coolant mass flux in $kg/m^3.s$ is provided for different blow ratios. Coolant temperature is taken as 345 K.

TABLE IV: MAINSTREAM FLOW CONDITIONS FOR C3X FILM COOLING.

Inlet total pressure	Inlet total Temp.	Inlet Mach No.	Outlet Mach No.	Inlet Turb. intensity	Inlet Re.	Outlet Re.	Outlet static pressure
207360 Pa	690 K	0.16	0.89	6.5%	$0.38x10^6$	$1.5x 10^6$	123934.9 Pa

Vane material is the same used in internal cooling validation case. The hybrid mesh created in ANSYS Design Modeler, that produced a total of 52000 nodes in the entire domain after a mesh independence study. Wall y+ is ensured to be less than 1 for all the vane walls and for the coolant channels less than 2. All iterations were run until a steady-state converged status with a residual of 10^{-4}.

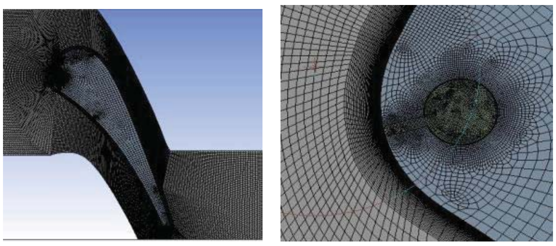


Fig.12. The mesh generated for the stagnation position.

B. Results and Discussions

Two dimensional steady state compressible flow simulations are carried out for each of the shape cases with SST k-w turbulence model, by varying the blow ratio. The film cooling performance of the five hole shapes at each of the hole positions are quantified and compared based on their stream wise cooling effectiveness. The effectiveness at two different blow ratios 1.0 and 4.0 is compared along the downstream direction of each film hole positions. It is calculated up to a distance of 20 hole diameters in each of the cases for uniformity.

1) At the pressure-side position

At the pressure-side position, the mainstream flow expands in a uniform fashion, showing a uniform trend in the cooling effectiveness downstream. From the effectiveness curves as shown in the Fig.13 below, it is observed that at low blow ratio 1.0, the cylindrical holes shows higher cooling effectiveness than the shaped and trenched holes. As the blow ratio increases to 4.0, the near hole cooling effectiveness of the shaped hole gets improved, followed by the shaped and trenched hole.

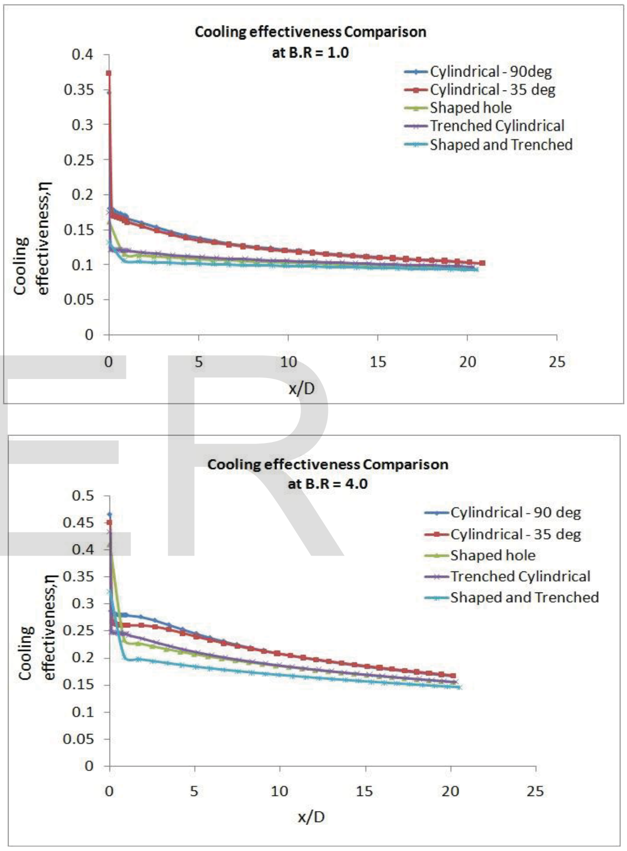
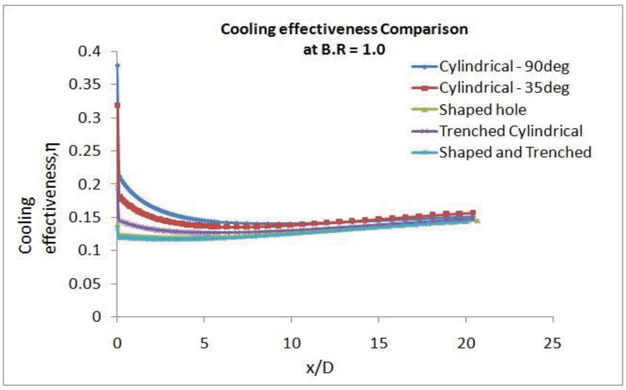


Fig.13. Pressure-side Cooling effectiveness comparison

2) At the suction-side position

From the effectiveness curves as shown in the Fig.14, it is clear that, at all the blow ratios studied, the cylindrical holes exhibit superior performance than others. The trenched cylindrical holes, unlike at the pressure-side position shows considerable improvement next to the cylindrical holes with more uniform cooling downstream at the suction-side position.



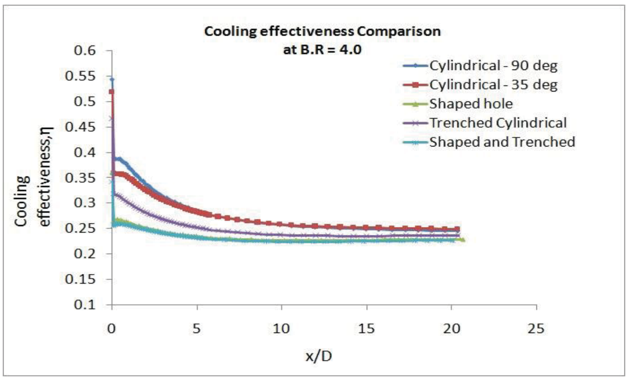
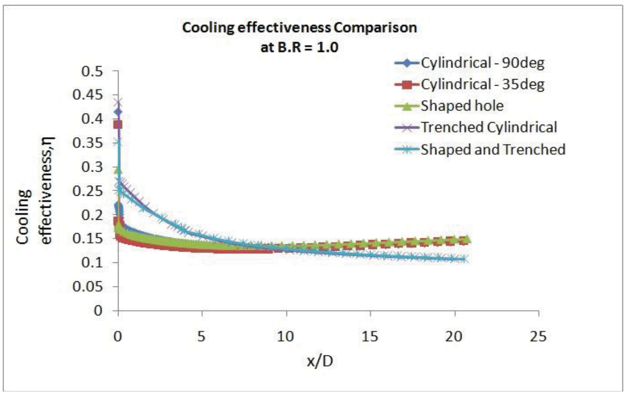


Fig.14. Suction-side Cooling effectiveness comparison

3) *At the stagnation position*

The vane and the coolant hole geometries greatly influence the coolant flow downstream direction and the effectiveness here. When comparing the near hole effectiveness of the geometries ,it is clearly observed that the trenched cylindrical hole followed by the shaped and trenched hole exhibit superior cooling effectiveness at all blow ratios.



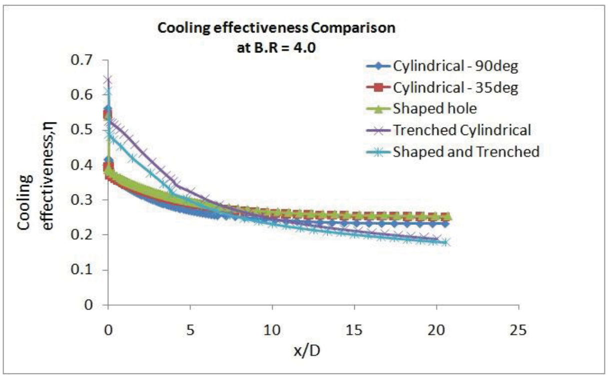


Figure 15: Stagnation position Cooling effectiveness comparison

VII. CONCLUSIONS

The present study compared the stream-wise cooling effectiveness of five different hole configurations at three positions on the leading edge of a C3X nozzle guide vane. The validation studies matched well with the experimental results.

The salient conclusions drawn from the study are:

- The mainstream conditions, hole geometry, shape and orientations have considerable influence in the turbine vane film cooling effectiveness.
- In general, cylindrical holes perform well at low blow ratios and shaped or trenched holes are efficient at higher blow ratios.
- Jet lift-off is an undesirable phenomenon at higher blow ratios, therefore shapes that result in more attached jets are required.
- Shaped and trenched holes exhibit more uniform film effectiveness at all blow ratios than the cylindrical holes.
- The film cooling studies are mostly carried out on flat plate cases. These studies when extended to a real turbine vane, especially on its leading edge is challenging on the basis of changing main flow conditions and the design of optimum hole configurations.

References

[1] Grzegorz Nowak and Wróblewski. “Optimization of blade cooling system with use of conjugate heat transfer approach”. International Journal of Thermal Sciences, pp.770-1781, 2011.

[2] L.D.Hylton and M.S.Mihelc. “Analytical and Experimental Evaluation of the Heat Transfer Distribution Over the Surfaces of Turbine Vanes”. NASA CR-168015:pp.24 – 690, 1983.

[3] Rajesh Kumar Panda and Paresh Halder. “A comparative study of film cooling effectiveness on a flat plate”. Proceedings of ASME Turbo Expo, Denmark: pp.1-10, 2012.

[4] Falah.F.Hatem, and Assim.Yoosif. “Enhancement of Adiabatic Film Cooling Effectiveness by Using Conical Shape Hole”, Anbar Journal for Engineering Sciences.pp.465-477,2013.

[5] Durham.M.G.”Comparison of Square-hole and Round-hole of Film Cooling : A Computational Study”, Msc.Thesis, University. of central Florida, 2004

[6] Kohli,A.,Bogard, D.G.”Adiabatic Effectiveness for Film Cooling”, ASME Journal of Turbo machinery. pp.352–35,1997.
[7] Shridhar and T.Nageshwara Rao.”CFD Simulation on Gas turbine blade and Effect of Hole Shape on leading edge Film Cooling Effectiveness”, IJMER, Vol.3.pp.2066-2072,2013.
[8] E.R.Turner and M.DWilson.”Analytical and Experimental Evaluation for Leading Edge Showerhead Film Cooling”. NASA CR-174827:pp.24-850, 1985.