Proceedings of ASME Turbo Expo 2013: Turbine Technical Conference and Exposition
GT2013
June 3-7, 2013, San Antonio, Texas, USA

GT2013-94127

NUMERICAL MODELLING OF SLOT FILM COOLING USING A WALL FUNCTION

Karim M. Shalash
Optumatics
Cairo, Egypt

Lamyaa A. El-Gabry
Mechanical Engineering Department
The American University in Cairo
New Cairo, Egypt

Mohamed M. Abo El-Azm
Mechanical Engineering Department
Arab Academy for Science and Technology and Maritime Transport
Alexandria, Egypt

ABSTRACT

CFD modeling of gas turbine film cooling remains a challenge for the computational arena due to the lack of robust accurate turbulence model or numerical technique to solve this highly complex problem. Modeling the exact behavior of the coolant jet is computationally expensive due to the complexity of the jet mainstream interaction, such as vortex generation, and separation. This paper, validation progress is presented using experimental data executed by the second author [GT2011-46491] and Thurman et-al [GT2011-46498] for blowing ratios of 1.0 and 2.0, and density ratio of 1.0. A wall function approach is chosen for a robust computation, and aiming for CPU time reduction. The in-house CFD code EOS is used to solve the RANS equations. A simple flow over flat plate validation problem was executed using experimental data of Klebanoff and El-Tahry as a code validation evidence. The computational results of the flow field were in agreement with the experimental measurements, with a slight over estimation of the thermal field due to over prediction of dissipation, resulting in less diffusion and mixing between the coolant jet and the mainstream. The wall function approach is shown to have great potentials for a robust accurate solution. The use of the continuous slot injection is known to be the best film cooling technique when compared to the other conventional circular coolant tubes, with the critical drawback of lowering the mechanical integrity of the blade. One way to overcome this problem is the use of discrete slots, which will optimize for better film cooling effectiveness, while maintaining the structural strength of the blade. Rectangular slots showed an increase in film cooling effectiveness for the same mass flow rate of coolant and blowing ratio when compared to the standard circular hole, which is due to the minor counter rotating vortex pair effect. An oval slot design showed higher film cooling effectiveness and more spreading, covering more surface area.

INTRODUCTION

Increasing the temperature of gases flowing across a turbine increases the power generated by a gas turbine engine, a fact that is limited by the thermal strength of the blade. For that reason, the gas turbine cooling technology is recently taking giant leaps toward increasing the cooling effectiveness of the blades. Other than internal cooling passages, film cooling has been used to lower the surface temperature of the blade. For aerospace applications the injected coolant is air bled from the compressor stage, thus the effectiveness and efficiency of the cooling are of a high importance to minimize the coolant mass flow rate. A theoretical perfect film cooling system is one that uses a minimum mass flow rate of coolant, which does not affect the blade mechanical strength, nor create stress concentration points, and is also feasible to manufacture.

Computational fluid dynamics tools have been used extensively to simulate the complex flow behavior of the jet in crossflow problem, taking into account the counter rotating vortex pair, the separation bubble that might occur in some cases, the estimation of the thermal domain, as well as the localizing of hot uncooled spots. CFD has been used as a design and a research tool to solve film cooling problems. The major issue in the field of film cooling simulation nowadays is to find a robust accurate solver and turbulence model. Time is an important parameter for the industry. Consequently, a robust computational technique, which maintains the accuracy of the solution, would be a major breakthrough for the propulsion technology.

Azzi and Jubran [3] introduced a converging slot hole designed as a film cooling improvement concept. Direct Numerical Simulation was executed and compared to the standard cylindrical cooling tube as well as a shaped cylindrical layout. The slot increased the film cooling effectiveness, also the span coverage of the coolant jet when compared to the other two layouts for the same blowing ratio. Bogard and Thole [6] mentioned that the manufacturing of shaped film cooling holes is a highly significant parameter in blade design, since the difference in cost between manufacturing simple cylindrical holes and shaped is high. Liu et al. [10] executed an experimental study using liquid crystal thermography, comparing converging slots holes to standard cylindrical holes at a range of momentum flux ratios. The coolant jet of the slots was much more attached to the surface of the blade; as a result a much higher film cooling effectiveness has been achieved when compared to the standard cylindrical holes and shaped holes with different compound angles. The shaped hole with a 45º compound angle improved the film cooling effectiveness by 150% to 200% in some areas when compared with the baseline geometry. Haven [14] concluded in her experimental study on film cooling sensitivity to the hole shape, that the jet liftoff and hot gas entrainment toward the surface is highly affected by the proximity of the counter rotating vortex pair [CRVP] relative to one another. It was shown that large aspect ratio geometries tend to liftoff less than their smaller counterparts.

An experimental study has been executed by Baldauf et al. [4] comparing film cooling effectiveness for several blowing ratios under engine like conditions, it was shown that for blowing ratios from 0.2 to 0.85 the film cooling effectiveness increases with the increase of the blowing ratio, the top film cooling effectiveness is for a blowing ratio between 0.85 and 1.2, beyond this region the film cooling effectiveness starts decreasing with the blowing ratio increase. Thole et al. [5] studied the effect of the momentum flux ratio I and the jet behavior. It was shown that there are three main jet behaviors with the mainstream. For $I < 0.4$ the jet is fully attached to the blade surface, and for $I > 0.8$ the jet is fully detached from the surface. As for the $0.4 < I < 0.8$ the jet detach and reattach soon to the blade surface.

Kadotani and Goldstein [6] performed various studies on the effect of mainstream variable on film cooling; it was shown that a thin mainstream boundary layer would give higher film cooling effectiveness when compared with large boundary layer thickness, due to the stronger momentum of the thin boundary layer impacting the jet. It thus, improves the jet attachment to the surface.

Leylek and Zerkle [7] executed the first numerical simulation of a three dimensional film cooling jet in cross flow problem using the standard k-epsilon turbulence model with wall function; the authors slightly over predicted the center line film cooling effectiveness for a fully attached jet. While Ferguson et al. [9] showed that it is more difficult to predict the film cooling effectiveness for detached jets using a variety of different turbulence models versions, with both wall function, and a two layer wall treatment algorithm. D. Lakehal et al. [11] adopted a wall function to model lateral injection film cooling, using the standard k-epsilon model. The model predicted the counter rotating vortex pair and the jet trajectory, as well as the temperature gradient between the core of the jet and the mainstream, but could not well predict the lateral spreading of the coolant.

NOMENCLATURE

DR	*Density ratio*
BR	*Blowing ratio*
I	*Momentum flux ratio*
ρ	*Density*
σ_k	*Turbulent Prandtl number for kinetic energy*
σ_ϵ	*Turbulent Prandtl number for rate of turbulence dissipation*
P_k	*Generation of turbulent kinetic energy that arises due to velocity gradient*
P_b	*Generation of turbulent kinetic energy that arises due to buoyancy*
ϵ	*Rate of dissipation of kinetic energy*
Y_m	*Dilatation dissipation term*
S_k	*Kinetic energy source term*
C_1	*Model constant*
S_ϵ	*Rate of dissipation of kinetic energy source term*
$C_{1\epsilon}$	*Model constant*
$C_{3\epsilon}$	*Model constant*
C_μ	*Turbulent viscosity parameter*
u^+	*Dimensionless velocity*
y^+	*Dimensionless distance to wall*
K	*Von* Kármán *constant (K=0.41)*
E	*Empirical wall roughness constant (E=1.0)*
T^+	*Dimensionless temperature*
Pr_t	*Turbulent Prandtl number*
Pr	*Prandtl number*
T_∞	*Mainstream inlet temperature*
T_c	*Coolant temperature*
Re	*Reynolds number*
C_2	*Model constant*
ν	*Kinematic fluid viscosity*
k	*Kinetic energy*
μ_t	*Eddy viscosity*
μ	*Dynamic fluid viscosity*
D	*Hole diameter*
X	*Horizontal coordinate along plate*
Y	*Horizontal coordinate normal to X*
Z	*Vertical coordinate*

EXPERIMENTAL DATA

The experimental data used for validation were executed by El-Gabry et-al [GT2011-46491] and Thurman et-al [GT2011-46498]. The papers are two parts of an experimental study done using a large scale model of 30º inclined row of three film cooling holes.

The hole diameter is 19 mm, the distance between each hole is 3 hole diameters. The length of the tube is more than 23 diameters, in an attempt to generate a fully developed flow. The test rig was built inside an open loop wind tunnel. Data were collected at BR of 1.0 and 2.0, while maintaining a DR of approximately 1.0.

El-Gabry et-al. [1] executed velocity and turbulence measurements at different planes in the domain using hot wire anemometry, with a calculated uncertainty of 4% for mean velocities and 5% for fluctuations. The Reynolds stress uncertainties were typically less than 15% but were as much as 25% in some of the high turbulence regions. For these measurements both the mainstream and the injected air were kept at ambient temperature.

The results showed the well-known CRVP, one of the major causes of the reduction of film cooling effectiveness. The jet is noted to deflect more toward the surface at the lower blowing ratio, due to the weaker jet momentum relative to the higher blowing ratio; still the jet did not completely remained attached to the surface.

Figure 1 - Photograph of the test section

Thurman et-al. [2] presented the second part, executing high resolution heat transfer, and temperature measurements using the same large scaled model and rig. The same blowing ratios were tested. The mainstream was inducted via the vacuum exhaust system of the wind tunnel at room temperature. The coolant was chilled by passing it through a heat exchanger consisting of copper tubes submerged in a tank of iced water; the entire path of the coolant downstream of the heat exchanger is insulated. The coolant is cooled 20ºC below the mainstream temperature. The mainstream and the coolant temperature were measured using an open ball thermocouple. Type-E thermocouples are used for temperature measurements, the probes are connected to an actuator above the test section. Liquid crystal thermography was used for the heat transfer test, with a replica of the test section supplied with a small sheet of Inconel and bus bars to ensure a constant heat flux condition.

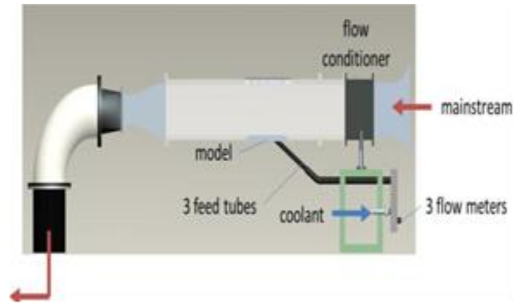

Figure 2 - Test section layout

At the blowing ratio 1.0 the film cooling effectiveness measured along the jet centerline is of a value around 40%. For the higher blowing ratio 2.0 and due to the jet detachment the centerline film cooling effectiveness measured is nearly 15%.

NUMERICAL METHODOLOGY

A hexahedral grid was adopted to increase the solution accuracy; a blocking technique was used, putting a higher density grid block in the jet-crossflow interaction region, to capture all the details. The four blocks of the domain are connected via shearing interfaces. The y+ value was fixed around 30 along the plate. A grid sensitivity study was executed, starting with a 500,000 cell grid, and enlarging it by 100,000 cells steps, while maintaining the y+ value. The 1,111,234 cells grid results coincided with the results of 929,120 cells grid, therefore the later grid was used, inside which 503,496 cells were located inside the mixing block.

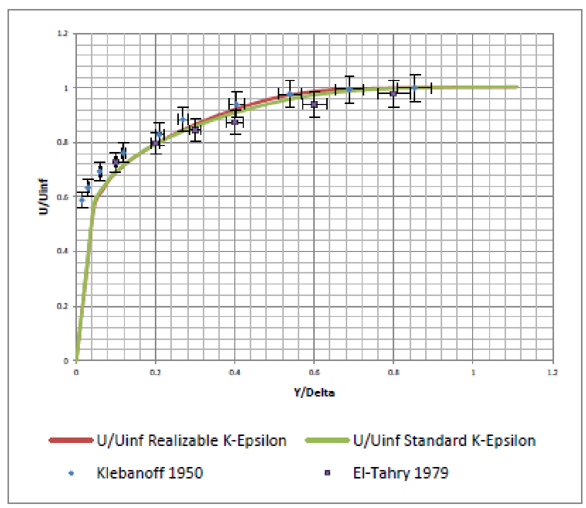

Figure 3 - Flow over flat plate comparing standard and realizable k-epsilon turbulence models compared to experimental data of El-Tahry and Klebanoff.

The in house CFD code EOS [property of Optumatics] is used for the numerical simulation. EOS solves the RANS equations using a finite volume method, and a pressure based closure. The differencing scheme used for the momentum equation was a second order central. As for the energy and turbulence equations only first order upwind was used. SIMPLE segregated algorithm was used to deal with the pressure velocity coupling. For a given solution iteration the three momentum equations are solved, then a pressure correction is made in the continuity equation. After the flux updating, scalars such as temperature and turbulence are updated. The aggression of the solution was controlled by relaxation factors for each governing equation; these factors were kept minimum at the beginning of the solution to avoid divergence, and then increased as the solution starts converging toward a lower scaled residual. The convergence criterion is reaching a scaled residual of 10^{-6}.

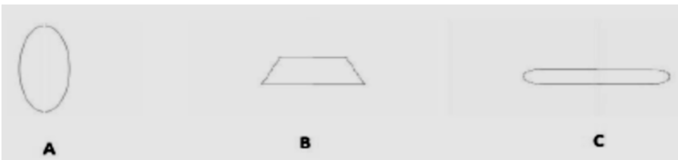

Figure 4 - Different coolant holes layouts, A) Standard cylindrical, B) Rectangular slot, and C) Oval slot.

The domain consists of a coolant feeding tube, and a box modeling the wind tunnel and the flat plate to be cooled. The test section dimensions are 655x57x200-mm, the tube diameter is 19 mm, the angle between the plate and the tube is 30º. Since the problem is symmetric, only half the domain is modeled, and a symmetry plane is used to mirror the results. The top and the side wall are modeled as slip walls, to eliminate the boundary layer effect; the flat plate and the tube are no slip walls. The inlets of the mainstream and the coolant are velocity inlet boundaries, while the outlet is an inlet/outlet boundary, which is a pressure boundary without an assigned flow direction, to enhance the solution stability, and to show any reversed flow. The whole domain is considered adiabatic, to model the insulations on the test rig. The difference in temperature between the mainstream and coolant jet is 22ºC. The mainstream velocity is 9.1 m/s, the turbulence intensity is 4%. The coolant inlet velocity was 9.1 m/s at the BR=1.0 case, and 18.2 m/s at the BR=2.0 case. The running fluid is air with a turbulent Prandtl number of 0.9. The mainstream inlet velocity profile was assigned to the mainstream inlet orifice. Two other coolant tubes were proposed, with the same cooling hole cross- sectional area, one is a rectangular divergent slot shape, another is a conceptual oval shape hole.

A simple flow over flat plate problem was executed in figure 3 as code validation evidence using the standard and the realizable versions of the k-epsilon turbulence models, and compared to experimental data of Klebanoff and El-Tahry [13] and [12].

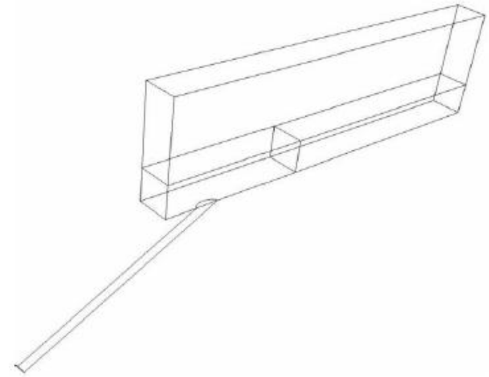

Figure 5 - Circular Tube Domain Geometry

Different k-epsilon turbulence model versions have been tried, standard, RNG, and the realizable which have been adopted due to the better turbulent viscosity modeling shown by it. Below are the transport equations for turbulence kinetic energy, and dissipation:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho k u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k+P_b+\rho\epsilon-Y_m+S_k$$

$$\frac{\partial}{\partial t}(\rho\epsilon)+\frac{\partial}{\partial x_j}(\rho k\epsilon)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right]+\rho C_1 S_\epsilon-\rho C_2\frac{\epsilon^2}{k+\sqrt{\nu\epsilon}}+C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon}P_b+S_\epsilon$$

$$C_1=\max[0.43,\frac{\eta}{\eta+5}],\qquad \eta=S\frac{k}{\epsilon},\qquad S=\sqrt{2S_{ij}S_{ij}}$$

And the turbulent viscosity is modeled as follows:

$$\mu_t=\rho C_\mu\frac{k^2}{\epsilon}$$

$$C_\mu=\frac{1}{A_0+A_s\frac{kU^*}{\epsilon}}$$

$$A_0=4.0,\qquad A_s=\sqrt{6}\cos\phi$$

$$\phi=\frac{1}{3}\cos^{-1}\left(\sqrt{6}W\right),\quad W=\frac{S_{ij}S_{ik}S_{ki}}{S^{`3}},\quad S^{`3}=\sqrt{S_{ij}S_{ij}}$$

$$S_{ij}=\frac{1}{2}\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)$$

$$C_{1\epsilon}=1.44,\quad C_2=1.92,\quad \rho_k=1.0,\quad \rho_\epsilon=1.2$$

The law of the wall theory states that the dimensionless tangential velocity $u^+$is calculated based on the y^+value. For the viscous sublayer region $u^+$is computed as follows:

$$u^+=y^+,\quad for\ y^+\le 5$$

On the other hand the velocity profile in the logarithmic region is given by:

$$u^+=\frac{1}{k}lnEy^+,\ for\ y^+\ge 30$$

The algorithm implemented in the CFD code uses, a critical y^+value of 12, below which the linear equation is used, and above it the law of the wall is used. The y^+value was kept around 30 over the plate to be cooled, thus the law of the wall is used as a wall function. For heat transfer the code computes T^+as follows:

$$T^+ = Pr_t u^+ + 9Pr_t(\frac{Pr}{Pr_t} - 1)(\frac{Pr_t}{Pr})^{1/4}$$

RESULTS AND DISCUSSION

The CFD results have been compared to the experimental results for both high and low blowing ratios. The dimensionless temperature theta has been compared along the jet centerline, for the BR = 1.0 case the numerical results were in good agreement with the thermocouple data collected.

$$Theta = \frac{T_\infty - T}{T_\infty - T_c}$$

The colored contours in figure 6 show the difference in diffusion at the core of the jet. While the non-filled contours in figure 8 show the slight lift in the jet core in the numerical simulation. The centerline film cooling effectiveness in figure 7 was predicted well, with an overall over estimation, especially in the recirculation area between X/D 2.5 to 3.5.

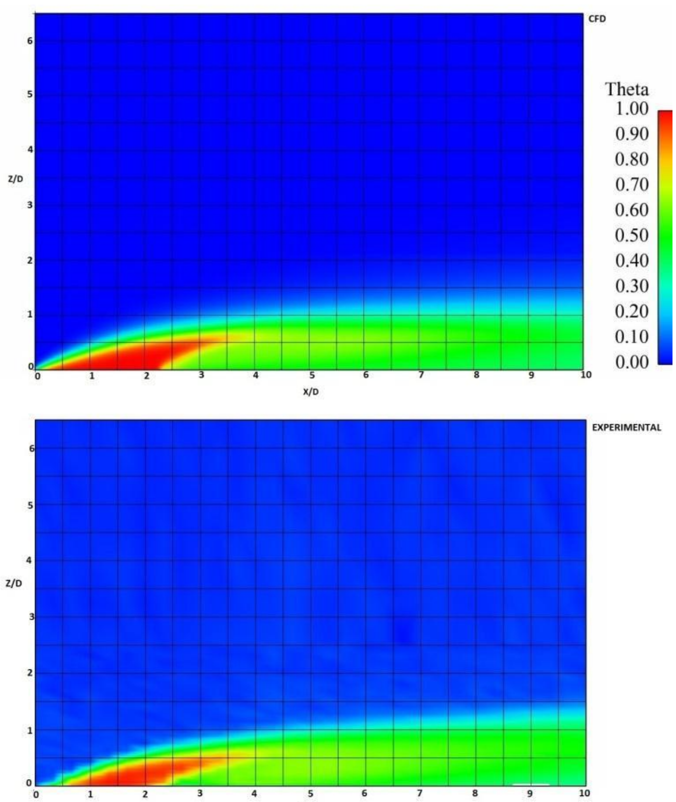

Figure 6 - Numerical and Experimental Dimensionless Filled Temperature Contours "Theta" at Centerline BR = 1.0

The higher blowing ratio case was more difficult to predict, because of the coolant jet separation. The centerline film effectiveness was under estimated. The lee side of the jet was not modeled properly causing the jet not to reattach, resulting the under estimation of the centerline film effectiveness. Once more, the jet diffusion is over estimated as shown in figure 9 and figure 11; the core of the jet is very well preserved when compared to the experimental data, which reveals more mixing with the mainstream. Figure 10 clearly shows the reattachment problem which was reflected on the centerline film effectiveness, the full detachment of the jet lowers the cooling effectiveness, thus the trend does not coincide with the experimental reattached jet results.

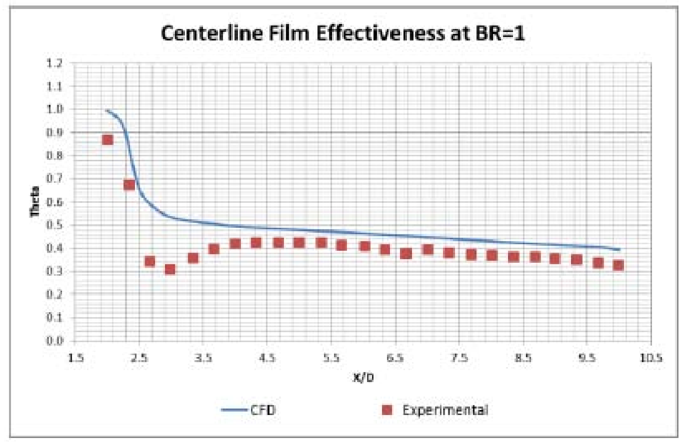

Figure 7 - Centerline Film Effectiveness at BR = 1.0

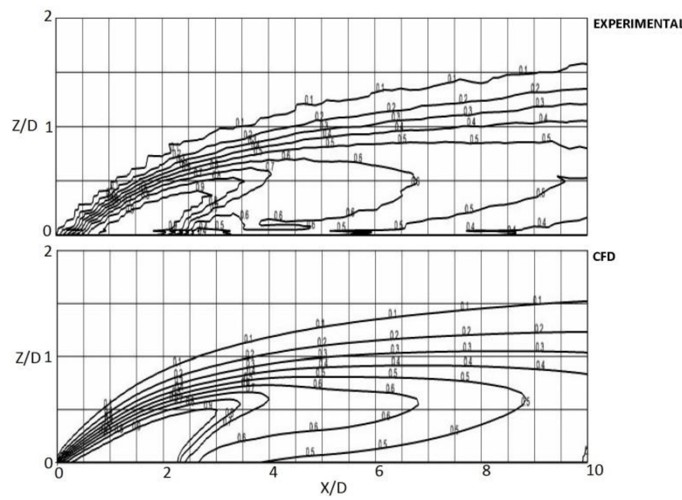

Figure 8 - Numerical and Experimental Dimensionless Non-filled Temperature Contours "Theta" at Centerline BR = 1.0

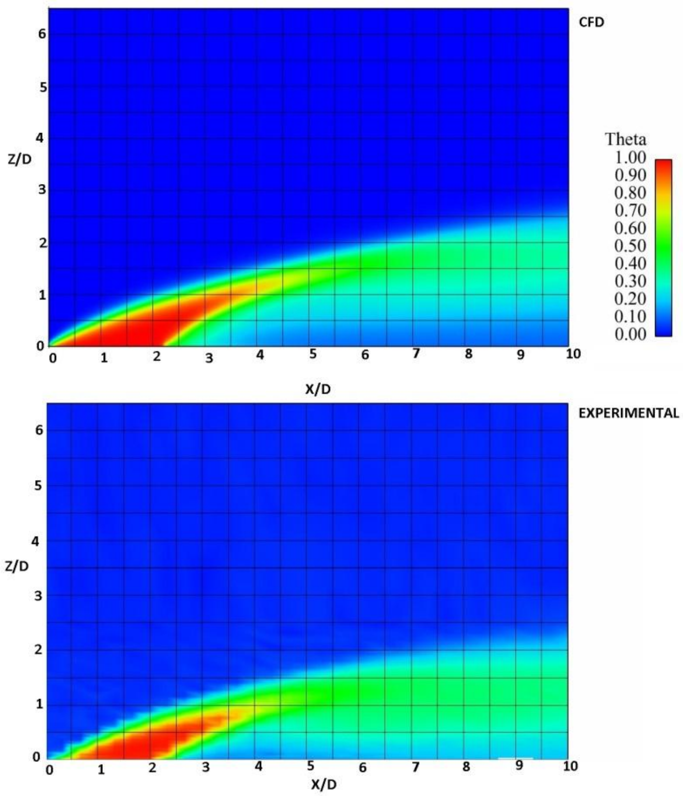

Figure 9 - Numerical and Experimental Dimensionless Filled Temperature Contours "Theta" at Centerline BR = 2.0

The two other slot tube designs showed improvement in the centerline film cooling effectiveness, as well as the surface coverage. The design philosophy was to cover more area with the same mass flow rate of coolant, by keeping the hole area and the jet velocity the same.

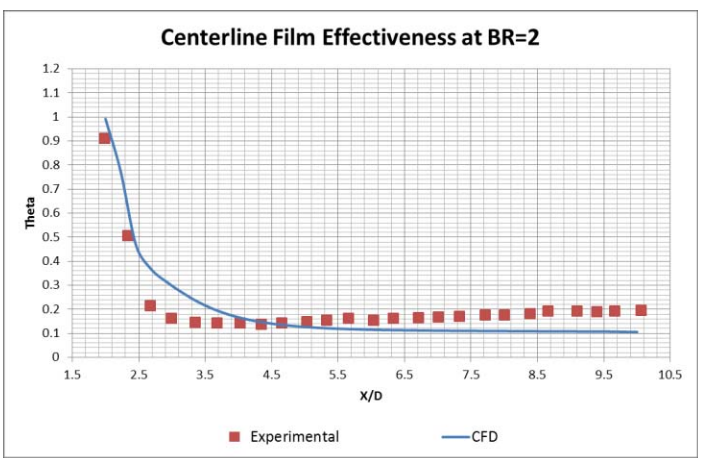

Figure 10 - Centerline Film Effectiveness at BR = 2.0

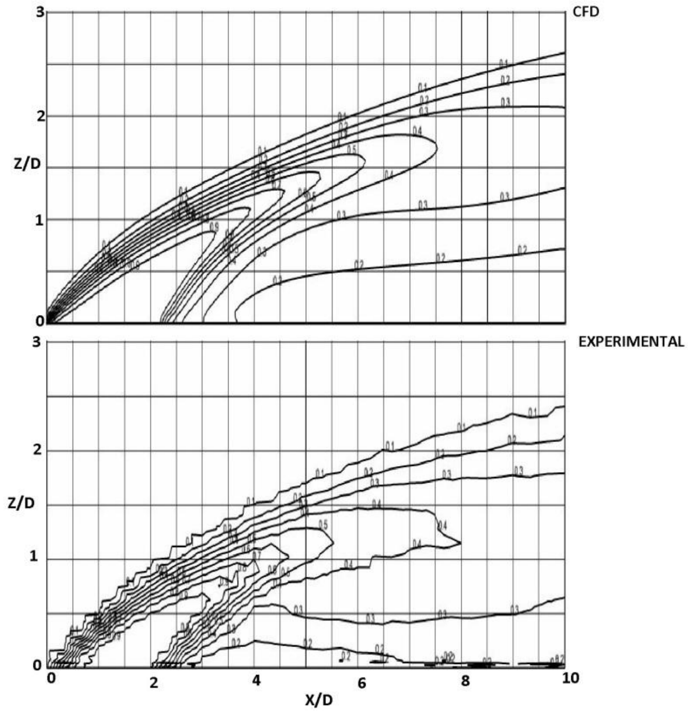

Figure 11 - Numerical and Experimental Dimensionless Non-filled Temperature Contours "Theta" at Centerline BR = 2.0

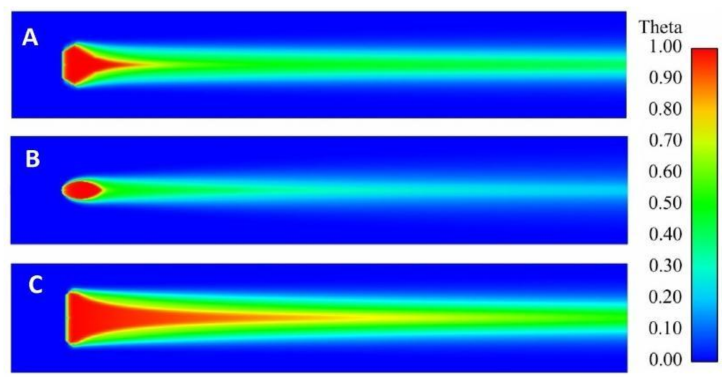

Figure 12 - Film Cooling Effectiveness Comparison between three different hole shapes; A) Rectangular divergent slot, B) Standard circular hole, C) Oval slot

The rectangular divergent slot showed good improvements. An expected problem was the stress concentration over the corners. The design was consequently taken to the next level by introducing the oval slot which eliminates the stress concentration problem. It is also easier to manufacture because of the lack of sharp edges inside the slot. The rectangular divergent slot resulted in higher film cooling effectiveness as shown in figure 13 and covered more plate surface area as shown in figure 12. The oval tube hole showed even higher film cooling effectiveness maintaining a value between 0.8 and 1.0

till 10 cylindrical whole base diameters downstream of the hole.

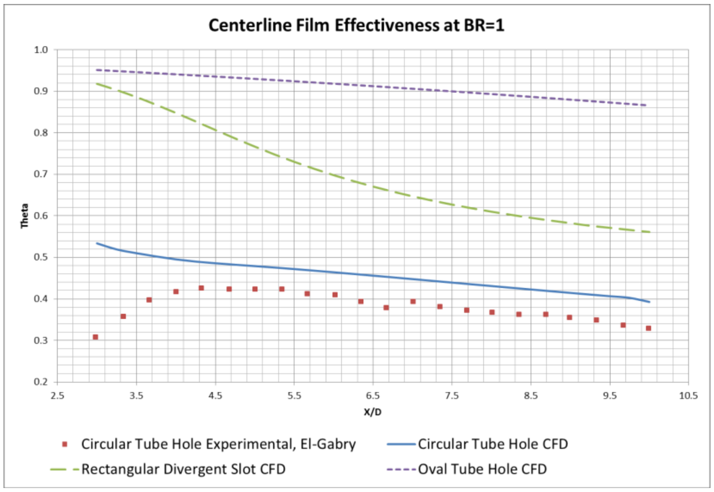

Figure 13 - Centerline Film Cooling Effectiveness at BR = 1.0 Comparing between three different hole shapes; A) Rectangular divergent slot, B) Standard circular hole, C) Oval slot

CONCLUSION AND RECOMMENDATIONS

The wall function showed potentials in estimating film cooling heat transfer. It can be used to estimate the cooling trend for non-separated jets, with a relatively smaller number of cells, thus a lower computational time. The model could not properly predict the lateral coverage of the jet. The wall function approach has robust potentials in modeling film cooling over complete blade geometries, and with relatively good accuracy. Both the wall function and the turbulence model could be used to predict the behavior and the overall trends of film cooling systems. The higher blowing ratio remains difficult to model using the wall function because of the aggressive recirculation on the lee side of the jet. At the point of separation the shear stress at the wall is equal to zero, thus the standard wall function is not suitable to predict the behavior of the jet.

The discrete slots showed a better capability to cover more blade surface, significantly decreased the strength of the CRVP, and increased the film cooling effectiveness thereby. The oval tube hole showed better film cooling performance, and also eliminates strong stress concentration points when compared to the rectangular divergent slot. The manufacturing, the mechanical integrity, as well as the operational reliability of the slots remain an open subject for further research.

ACKNOWLEDGMENTS

The authors would like to thank the technical team at Optumatics, specifically Dr. Sherif El-Tahry the Chief Technical Officer at Optumatics for his continuous support and help throughout the duration of the project.

REFERENCES

[1] El-Gabry, L.A., Thurman, D.G., Poinsatte, P.E., and Heidman, J.D., 2011,"Turbulence and Heat Transfer Measurements in an Inclined Large Scale Film Cooling Array – Part I, Velocity and Turbulence Measurements," ASME paper GT2011-46491, ASME Turbo Expo 2011, Vancouver.

[2] Thurman, D.G., El-Gabry, L.A., Poinsatte, P.E., and Heidman, J.D., 2011,"Turbulence and Heat Transfer Measurements in an Inclined Large Scale Film Cooling Array – Part II, Temperature and Heat Transfer Measurements," ASME paper GT2011-46498, ASME Turbo Expo 2011, Vancouver.

[3] Azzi, A. and Jubran, B.A., 2007,"Numerical Modeling of Film Cooling from Converging Slot-hole," Heat and Mass Transfer, Vol. 43, pp. 381-388.

[4] Baldauf, S., Scheurlen, M., Schulz, A., and Wittig, S., 2002, "Correlation of Film-Cooling Effectiveness from Thermographic Measurements at Engine like Conditions," ASME Journal of Turbomachinery, 124, pp. 686–698.

[5] Thole, K. A., Sinha, A., Bogard, D. G., and Crawford, M. E., 1990, Mean Temperature Measurements of Jets with a Crossflow for Gas Turbine Film Cooling Application, Rotating Machinery Transport Phenomena, J. H. Kim and W. J. Yang, ed. Hemisphere Publishing Corporation, New York, New York.

[6] Bogard, D. G. and Thole, K. A., 2006, Gas Turbine Film Cooling, AIAA J of Propulsion (invited review paper), vol. 22, pp. 249-271.

[7] Leylek, J. H., and Zerkle, R. D., "Discrete-Jet Film Cooling: A Comparison of Computational Results with Experiments," Journal of Turbomachinery, Vol. 116, 1994, pp. 358–368.

[8] EOS Developer Manual, Optumatics, 2011, pp.35-143.

[9] Ferguson, J., Walters, D., and Leylek, J., "Performance of Turbulence Models and Near-Wall Treatments in Discrete Jet Film Cooling Simulations," American Society of Mechanical Engineers, ASME Paper 98 - GT - 438, 1998.

[10] Cun-liang Liu, Hui-ren Zhu, Jiang-tao Bai, and Du-chun Xu, "Film cooling performance of converging slot-hole rows on a gas turbine blade," International Journal of Heat and Mass Transfer, Volume 53, Issues 23–24, November 2010, Pages 5232-524.

[11] D. Lakehal, G.S. Theodoridis, and W. Rodi, "Computation of film cooling of a flat plate by lateral injection from a row of holes,"

International Journal of Heat and Fluid Flow, Volume 19, Issue 5, October 1998, Pages 418-430.

[12] S. El-Tahry, (1979). Turbulent Plume Dispersal. Ph.D. Thesis. Imperial College of Science and Technology. London – UK.

[13] Klebanoff, P.S. National Advisory Committee Aeronautical, Technical Notes 3133, 1954.

[14] B. A. Haven, (1996). The Effect of Hole Geometry on the Near Field Character of Crossflow Jets. Ph.D. Thesis. University of Washington. Washington – USA.