Evaluation of Predicted Heat Transfer on a Transonic Vane Using ν^2 - f Turbulence Models*

Hong WU**, Jiao WANG** and Zhi TAO**

** National Key Laboratory of Science and Technology on Aero-engine Aero-thermodynamics
Beihang University, 37 Xueyuan Road, Beijing, China
E-mail: wangjiao@sjp.buaa.edu.cn

Abstract

In this study, three dimensional numerical simulations of fluid and heat transfer on the transonic vane have been performed using ν^2 - f turbulence model. The objective of this paper is to evaluate the ν^2 - f model's ability to predict the heat transfer characters on the 3D showerhead film cooled vane. Exit Mach number of $M_{ex}=0.76$, corresponding to exit Reynolds number based on vane chord of 1.1×10^6 , is tested with an inlet free stream turbulence intensity (Tu) of 16% and integral length scale normalized by vane pitch (Λ_x/P) of 0.23. For the showerhead film cooling simulations, a new three simulations technique is discussed to calculate the recovery temperature, adiabatic wall temperature, and surface Nusselt number. CFD predictions show an overall good agreement with experimental results.

Key words: Turbulence, Transonic Flow, Numerical Simulation, Heat Transfer, Film Cooling

1. Introduction

Film cooling of high pressure gas turbine components (blades and vanes) is a widely used technique that allows higher turbine inlet temperatures for improved engine cycle performance. In this cooling method, air is extracted from the compressor and forced through internal cooling passages within turbine blades and vanes before being ejected through discrete cooling holes on the surfaces of these airfoils. The air leaving these cooling holes forms a film of cool air on the component surface which protects the components from hot gas exiting the combustor.

CFD is an essential design tool to predict film cooling performance of gas turbine components. Several computational studies have been performed to explain and predict film cooling on turbine blades and vanes at engine realistic flow conditions. Garg and Ameri[1] used 3-D RANS code with Coakley's q - ω model, Chien's k - ϵ model and Wilcox's k - ω model to predict heat transfer coefficient at low freestream turbulence on the film cooled VKI rotor with six rows of cooling holes including three rows in the showerhead and the NASA C3X vane with nine rows of holes including five rows in the showerhead. Later, Garg[2] used a multi-block 3-D RANS code with Wilcox's k - ω model to study with-in hole and near-hole physics in relation to heat transfer on the showerhead film cooled VKI rotor blade. He found a reasonably good agreement with the experimental data over the whole surface but found that near the holes heat transfer coefficient distribution was dependent on velocity and temperature distribution profiles inside the cooling holes.

Heidmann et al.[3] performed 3-D coupled internal/external heat transfer simulation of Allied Signal's film cooled vane using the LeRC-HT code at engine representative exit Mach number condition of $M_{ex} = 0.876$ and freestream turbulence intensity of 8%. The

*Received 21 Apr., 2011 (No. 11-0259)
[DOI: 10.1299/jtst.6.424]

Copyright © 2011 by JSME

Nomenclature			
A	area	U	streamwise local velocity
A^+	effective sublayer thickness	v^2	wall normal Reynolds stress component
BR	blowing ratio	y^+	equivalent normal distance
C	vane chord	Greek	
d	cooling hole diameter	α	spanwise hole cooling angle
DR	density ratio	ε	turbulent dissipation
f	elliptic operator in V2F model	γ	compound injection angle
	heat transfer coefficient	η	adiabatic film cooling effectiveness
k	acceleration parameter / thermal conductivity/ turbulent kinetic energy	Λ_x	integral turbulence length scale
M	Mach number	ρ	local density of air
$\dot{m}$	mass flow rate	μ	dynamic viscosity of air
Nu	Nusselt number	ν	kinematic viscosity of air
P	vane pitch /Pressure	τ	shear stress
p	cooling hole pitch	ω	specific turbulent dissipation
Pr	Pandtl number	Subscripts	
q''	heat flux	0	no film cooling
Re	Reynolds number	∞, a	freestream, air
s	vane surface distance from stagnation point	aw, w	adiabatic wall, wall
T	temperature	c	coolant
Tu	streamwise freestream turbulence intensity	ex, I	exit, inlet
		o	stagnation
		r	recovery
		t	turbulent

turbulence closure was done by means of modified Wilcox's k- ω model as implemented by Chima[4]. Two computations were performed for different isothermal wall temperatures to allow independent determination of heat transfer coefficient and film cooling effectiveness values. The numerical results were not validated with the experiments but showed that the shaped holes on the pressure surface provided a reduction in heat flux through the reduction in heat transfer coefficient and increase in film cooling effectiveness as compared to showerhead region circular shaped holes.

Ginibre et al.[5] computed wall heat flux on a 3D slice of high pressure film cooled nozzle guide vane (NGV) with multiple rows of injections to evaluate 3D Navier-Stokes solver developed by Snecma Moteurs. The turbulence closure was done by means of algebraic mixing length model and Spalart-Allmaras one-transport equation model. Ginibre et al. found that both the models predicted similar span averaged heat flux values but Spalart-Allmaras model was more appropriate for industrial use because of its generality and robustness.

Medic and Durbin[6] presented flow and heat transfer coefficient predictions for a high pressure film cooled VKI rotor blade with turbulent time-scale bound k- ε model, standard and modified k- ω model, and v^2 - f model. All the simulations were performed at exit Mach number condition of $M_{ex}=0.92$ and freestream turbulence of 5%. They found that for low blowing ratios, the modified k- ε model and v^2 - f model gave satisfactory predictions of overall heat transfer coefficient levels on the film cooled vane surface. For higher blowing ratios, on the suction surface, the modified k- ε model and v^2 - f model overpredicted film cooling in the vicinity of film cooling holes. For higher blowing ratios, on the pressure surface, the v^2 - f model predicted stronger jet lateral spreading as compared to the other models.

Montomoli et al.[7] upgraded the finite volume CFD solver Hybflow, to perform

conjugate heat transfer simulations of QinetiQ MT1 high pressure NGV at freestream turbulence of 5.5%. The heat transfer coefficient distribution over the vane surface showed good agreement with the experimental data of Chana and Mole [8].

Recently, Laskowski et al.[9] obtained coupled 3D CFD heat transfer simulations for a high pressure film cooled VKI rotor blade and high pressure film cooled C3X vane with the shear stress transport (SST) turbulence model of Menter[10]. Laskowski et al. found that film cooling simulations consistently underpredicted heat transfer coefficient downstream of the film cooling holes. They also noted better lateral film cooling coverage for the VKI rotor as compared to the C3X vane.

The unique contribution of this work is to numerically investigate the effects of coolant injection on film cooling performance of a showerhead film cooled first stage turbine vane at high freestream turbulence ($Tu = 16\%$) and realistic exit Mach number condition ($M_{ex} = 0.76$). Showerhead film cooling performance is characterized by Nusselt number and adiabatic film cooling effectiveness along the surface of the vane. The paper discusses a new three-simulations technique to calculate vane surface recovery temperature, adiabatic wall temperature, and surface Nusselt number to completely characterize film cooling performance in a high speed flow. All of the 3D film cooling simulations are compared to the experimental data investigated by Nasir S. et al.[11,12].

2. Computational approach

2.1 Turbulence model and governing equations

The v^2 - f turbulence model, originally suggested by Durbin[13], unlike conventional two-equation eddy viscosity models, solves two additional turbulence transport equations: one for the wall-normal Reynolds stress component (v^2) and the other for an elliptic relaxation function (f) – to incorporate near-wall turbulence anisotropic behavior and remove the necessity of wall damping functions. The kinetic energy and the dissipation rate equations are:

$$\partial_t(\rho k) + \nabla \cdot (\rho U k) = \rho P_k - \rho \varepsilon + \nabla \cdot ((\mu + \mu_t) \nabla k) \quad (1)$$

$$\partial_t(\rho \varepsilon) + \nabla \cdot (\rho U \varepsilon) = \frac{C_{\varepsilon 1} \rho P_k - C_{\varepsilon 2} \rho \varepsilon}{T} + \nabla \cdot \left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right) \quad (2)$$

where:

$$\rho P_k = -\frac{2}{3} \rho k (\nabla \cdot U) + 2 \mu_t |S|^2 - \frac{2}{3} \mu_t (\nabla \cdot U)^2 \quad (3)$$

$$|S|^2 = S_{ij} S_{ij}, S_{ij} = 1/2 (U_{i,j} + U_{j,i}), T = k / \varepsilon \quad (4)$$

The transport equations for v^2 and the f equations are:

$$\partial_t(\rho \bar{v}^2) + \nabla \cdot (\rho U \bar{v}^2) = \rho k f - \rho \frac{\bar{v}^2}{k} \varepsilon + \nabla \cdot ((\mu + \mu_t) \nabla \bar{v}^2) \quad (5)$$

$$f - L^2 \nabla^2 f = (C_1 - 1) \frac{2/3 - \bar{v}^2 / k}{T} + C_2 \frac{P_k}{k} \quad (6)$$

where

$$L = \min \left[C_L \max \left[\frac{k^{3/2}}{\varepsilon}, C_\eta \left(\frac{v^3}{\varepsilon} \right)^{1/4} \right], \frac{1}{\sqrt{6}} \frac{k^{3/2}}{C_\mu \bar{v} \sqrt{S^2}} \right] \quad (7)$$

In this paper, all the numerical investigations are performed using a commercially available software package (FLUENT 6.2.6). For the FLUENT solver, the constant number are $C_L = 0.22$, $C_\eta = 70$, $C_1 = 1.4$, $C_2 = 0.3$ and coupled implicit solution method with second-order upwind discretization scheme is chosen. The v^2 - f turbulence model is selected

here because of its proven ability to produce reasonable heat transfer predictions on airfoil surface with and without the presence of film cooling [6, 12, and 14-18].

2.2 Grid and boundary conditions

Figure 1. Computational domain

A side view of 3D computational domain for external and internal flow is shown in Fig. 1. The freestream inlet is located at 0.45C upstream of the vane leading edge. For all tests case, the total pressure, total temperature, freestream turbulence level, and integral length scale are specified at the freestream inlet and static pressure is specified at the domain outlet. Total pressure and temperature are set at the plenum inlet boundary depending on the particular injection rate being investigated. The exit end of the plenum is closed off. In addition, adiabatic no-slip wall boundary conditions are specified along the plenum and film cooling hole walls and also along the end walls of the 3D vane in the spanwise direction. Fluid specific heat, thermal conductivity and molecular viscosity are kept constant for all test cases. For turbulent Prandtl number, a constant value of 0.85 is used. Fig. 2 shows the leading edge region of the 3D computational domain. A 3D multi-block structured hexahedral mesh consisting of approximately 4.1 million cells for external and inlet flow is selected after grid independence study and is refined near the wall to better capture the wall heat transfer. This is achieved by adjusting the near-wall spacing $y^+ < 1$. Dense mesh with O-shape grid is used in the vicinity of filming cooling holes to improve grid quality and to better capture flow physics in the region as shown in Fig. 3. The convergence of the v^2-f model prediction is assumed to be reached when all normalized residuals are smaller than 10^{-5} , except energy smaller than 10^{-6} . All v^2-f model simulations are performed on Intel Core (TM) 2 CPU (2.66 GHz) and each simulation required approximately five days to reach convergence. In general 6500 iterations are required for full convergence.

Figure 2. Leading edge region of 3D CFD domain

Figure 3. O-shape mesh in the vicinity of film cooling holes

2.3 Three simulations

Commonly, the cooling performance of a showerhead cooled configuration can be characterized in term of the cooling effectiveness which is defined as

$$\eta = \frac{T_r - T_{aw}}{T_r - T_c}, \quad (8)$$

and the definition of heat transfer coefficient is

$$h = \frac{q''}{T_{aw} - T_w}, \quad (9)$$

where the q'' is the local heat flux. The Nusselt number is defined as

$$Nu = \frac{hC}{k}, \quad (10)$$

where C is the vane chord length, and k is the thermal conductivity of the flim.

Three independent simulations are performed for a particular blowing ratio condition to completely characterize the film cooling performance of showerhead film cooled vane in a transonic flow. The notion of three-simulations technique to calculate adiabatic effectiveness and heat transfer coefficient in a film cooling environment is an extension of two-simulations technique document in literature such as [3,19 and 20] by adding a third simulation to predict freestream recovery temperature on an airfoil surface in a high speed flow. First simulation is performed with no blowing (BR=0) and no-slip adiabatic vane surface boundary condition to find local freestream recovery temperature (T_r). The local freestream recovery temperature is actually a local adiabatic wall temperature for a two-temperature problem (freestream and surface). Second simulation is performed for a particular blowing ratio (BR>0) with no-slip adiabatic vane surface boundary condition to find adiabatic wall temperature (T_{aw}) for a three temperature (freestream, coolant and surface). Once recovery and adiabatic wall temperatures are known, film cooling adiabatic effectiveness is found using Eq. (8). A third simulation with no-slip constant vane surface temperature boundary condition was performed to predict vane surface heat transfer coefficient (h) for a particular blowing ratio (BR>0) using Eq. (9). Heat transfer coefficient for no blowing (BR=0) is also predicted with no-slip constant vane surface temperature boundary condition and is defined on the basic of freestream recovery temperature.

3. Results and discussion

3.1 Mach number distribution comparison

Fig. 4 shows the comparison of calculated and measured Mach number distribution along the vane midspan at exit Mach 0.76. The local Mach number is plotted against non-dimensional surface distance, s/C . Positive and negative values of s/C correspond to

Figure 4. Midspan Mach number distribution comparison

suction and pressure sides, respectively. Overall, the v^2-f model calculation compare very well with the experimental data on the vane surface except for slight underprediction in the geometric throat region on the suction side. The Mach number distribution varies smoothly along the pressure side, and does not exhibit any velocity peak downstream of the stagnation region. The flow on the suction side continuously accelerates up to the geometric throat area ($s/C = 0.51$) and then decelerates up to the measurement extent ($s/C = 1.3$).

3.2 Recovery and adiabatic wall temperature

Figure 5. T_r contours at $M_{ex}=0.76$ (BR=0)

Figure 6. T_r contours at $M_{ex}=0.76$ (BR=1.5)

Figure 5 and 6 present 3-D vane recovery and adiabatic wall temperatures contours for $BR = 0$ and 1.5 , respectively. The vane recovery temperature contours ($BR = 0$) are affected by flow acceleration on the vane surface and show slight fluid ejection from the extreme film cooling rows on both surfaces due to small pressure variation between rows. The vane adiabatic wall temperature contours ($BR = 1.5$) are also affected by flow acceleration on the vane surface and show coolant accumulation in the jet exit direction.

Figure 7. Calculated midspan T_r and T_{aw}

Figure 7 presents calculated non-dimensional vane recovery and adiabatic wall temperatures along the vane midspan for $BR = 0$ and 1.5 . The non-dimensional temperatures are plotted against non-dimensional surface distance (s/C and s/d). On the pressure surface, the vane recovery temperature ($BR = 0$) decreases slowly downstream of the stagnation region because of low favorable pressure gradient (Mach number) along the surface. The pressure surface clearly shows the effect of coolant injection ($BR = 1.5$) on adiabatic wall temperature reduction downstream of the stagnation region. The effect of coolant on adiabatic wall temperature remains up to $s/C = -0.50$ on the pressure surface, but then the adiabatic wall temperature follows the trend of local recovery temperature due to high favorable pressure gradient (Mach number) beyond $s/C = -0.50$. On the suction surface, the vane recovery temperature decreases downstream of the stagnation region up to the geometric throat due to high favorable pressure gradient (Mach number) but then increases downstream of throat due to the adverse pressure gradient. It is also important to mention that the calculated recovery temperature was considerably different from assumed recovery temperature based on local Mach number and turbulent boundary layer recovery factor ($r = Pr^{1/3}$) as shown in Fig. 7. An assumed recovery temperature was unable to provide reasonable effectiveness distribution on vane suction surface. A possible reason is a compressibility effect on recovery factor for gases with Prandtl number $Pr < 1$, as hinted by Oudheusden[21]. The suction surface also shows the effect of coolant injection ($BR = 1.5$) on adiabatic wall temperature reduction downstream of the stagnation region. The adiabatic wall temperature increases due to jet dissipation from $s/C = 0.15$ to $s/C = 0.28$ but then responds to high flow acceleration downstream of $s/C = 0.28$ and follows the trend of local recovery temperature up to $s/C = 0.80$.

3.3 Adiabatic effectiveness comparison

Figure 8. Midspan η comparison at BR=1.5

Comparison of vane midspan calculated and measured adiabatic wall effectiveness at $BR = 1.5$ and exit Mach 0.76 is shown in Fig. 8. For the experimental data, the adiabatic effectiveness decays downstream of coolant injection on both surfaces due to jet dissipation. The suction surface effectiveness does not exhibit a continuous decay downstream of the coolant injection as does the pressure surface and is influenced by a favorable pressure gradient due to Mach number in the region $s/C = 0.28$ to $s/C = 0.40$. On the pressure surface, the v^2-f model shows an overall very reasonable adiabatic effectiveness distribution comparison with the experimental data. On the suction surface, the v^2-f model overpredicts adiabatic film cooling effectiveness but captures the experimental data trend on the entire surface. The effect of favorable pressure gradient due to Mach number on freestream recovery and adiabatic wall temperature results in nearly steady adiabatic effectiveness data from $s/C = 0.28$ to $s/C = 0.40$ but then it decays through the geometric throat. Unlike pressure surface, the overprediction of adiabatic film cooling effectiveness is possibly due to the overprediction of local recovery temperature and adiabatic wall temperature in a highly favorable pressure gradient flow. The authors are most interested in the trend of vane surface adiabatic effectiveness at this time and would like to research and discuss the differences between measured and calculated levels of vane surface adiabatic effectiveness in detail in future.

3.4 Nusselt Number Comparison

Figure 9 provide the comparison of measured and calculated Nusselt number distribution along the vane midspan for $BR = 0$ and 1.5 at exit Mach 0.76. For the experimental data, the primary effect of coolant injection at $BR = 1.5$ is to augment Nusselt number downstream of injection on the vane surface as compared to no film injection case ($BR = 0$). The Nusselt number augmentation is due to more mixing and local increase in turbulence levels within the boundary layer. The effect of blowing ratio on Nusselt number augmentation is clearly more noticeable on the pressure surface as compared to the suction surface. Though augmented heat transfer distributions follow the same trend as the $BR = 0$ case on the suction side. Airfoil surface heat transfer augmentation due to film cooling injection has been documented widely in literature such as Refs. [2, 22-25]. The v^2-f model overpredicts Nusselt number on the vane pressure surface for $BR = 0$ and 1.5. The vane pressure surface predictions appear to be fully turbulent for both blowing ratio cases and show negligible effect of coolant injection. The Nusselt number overprediction on the vane pressure surface can be attributed to the difficulty of v^2-f model to accurately calculate the effects of large scale high freestream turbulence on a pressure surface (low favorable pressure gradient) Nusselt number and has also been observed by Luo and Razinsky[18].

Figure 9. Midspan Nu comparison at BR=0, 1.5

On the suction surface, the ν^2 - f model also overpredicts Nusselt number downstream of the stagnation region for BR = 0 and 1.5, but unlike pressure surface, it does show Nusselt number augmentation downstream of stagnation region due to coolant injection (BR = 1.5). Also on the suction surface, the measured and calculated Nusselt number show a peak and a valley downstream of the film cooling rows for BR = 0 and 1.5. The reasons for this peak and valley can be attributed to the effect of favorable pressure gradient due to Mach number on a pseudo-turbulent (highly perturbed and augmented) boundary layer to possibly relaminarize or the effect of favorable pressure gradient on the inner sublayer thickness of the pseudo-turbulent boundary accompanied by a bypass transition again to a fully turbulent boundary layer. The authors are encouraged to see that the ν^2 - f model captures the Nusselt number trend on the suction surface for BR = 0 and 1.5 and would like to explain the effect of favorable pressure gradient due to Mach number on a pseudo-turbulent boundary layer downstream of film cooling rows by a boundary layer analysis below.

3.5 Boundary Layer CFD Analysis on suction surface

According to Weigand et al.[26], high favorable pressure gradient induces a thicker laminar sublayer by delaying the critical momentum thickness Reynolds number for laminar to turbulent transition. A thick sublayer results in an increase in heat transfer resistance and a decrease in Nusselt number. An effective sublayer thickness (A^+) was estimated from the CFD solution for BR = 1.5 case as a critical value of y^+ where turbulent eddy diffusivity (ϵ_t) is of the same scale as that of molecular diffusivity (ν). Fig. 10 shows the effective sublayer thickness along the suction side for BR = 1.5 case. The plot shows that the effective sublayer thickness increases along the suction side in a favorable pressure gradient region ($s/C = 0.23 - 0.37$), and then decreases due to increase in turbulent eddy transport and possible local breakdown of sublayer near critical momentum thickness Reynolds number. This increase in turbulent eddy transport and local instability in the sublayer leads to boundary layer transition to a fully turbulent boundary layer in the geometric throat region ($s/C = 0.51$). This type of peak and valley on vane suction side heat transfer distribution with showerhead film cooling has also been noted by the experimental study of Turner et al. [27]. The authors believe that the effect of high favorable pressure gradient on effective sublayer thickness also impacts the adiabatic effectiveness through changes in local adiabatic wall and recovery temperatures.

Regarding suction side boundary layer transition, the experimental data shows that the boundary layer transition location ($s/C = 0.34$) is unaffected by coolant injection (BR = 1.5). The experimental data also shows that once the boundary layer becomes turbulent on the suction side, the effect of coolant injection on the heat transfer diminishes. The ν^2 - f model predicts a slight delay in the boundary layer transition on the suction surface for both blowing ratio cases, and also shows an augmentation in a fully turbulent boundary layer for the BR = 1.5 case as compared to BR = 0 case.

Figure 10. Suction side midspan A^+ calculation at $BR=1.5$

4. Conclusion

This paper presents 3D numerical investigation of the heat transfer characters on the transonic vane surface and the effects of coolant injection on film cooling performance of a showerhead film cooled first stage turbine vane at high freestream turbulence and realistic exit Mach number condition. A new three-simulations technique is discussed to calculate vane surface recovery temperature, adiabatic wall temperature, and surface Nusselt number to completely characterize film cooling performance in a high speed flow. For the 3D showerhead film cooling simulations, the primary effects of coolant injection ($BR = 1.5$) are to augment Nusselt number and reduce adiabatic wall temperature downstream of the injection on the vane surface as compared to no film injection case ($BR = 0$) at high freestream turbulence of $Tu = 16\%$ and exit Mach number of $M_{ex} = 0.76$.

Adiabatic Effectiveness

On the pressure surface, at $BR = 1.5$, the v^2-f model shows an overall good adiabatic effectiveness distribution comparison with the experimental data. On the suction surface, at $BR = 1.5$, the v^2-f model overpredicts adiabatic effectiveness but captures the experimental data trend on the entire surface. Also on the suction surface, the adiabatic effectiveness calculation on the suction surface is found to be influenced by favorable pressure gradient due to Mach number through changes in local adiabatic wall and recovery temperature.

Nusselt Number

On the pressure surface, at $BR = 0$ and 1.5 , the v^2-f model overpredicts Nusselt number and show negligible effect of coolant injection. On the suction surface, the v^2-f model also overpredicts Nusselt number downstream of the stagnation region for $BR = 0$ and 1.5 , but unlike pressure surface, it does show Nusselt number augmentation downstream of stagnation region due to coolant injection ($BR = 1.5$). Similar to experimental results, the Nusselt number calculation on the suction surface shows a peak and a valley downstream of the film cooling rows in a favorable pressure gradient region for both tested blowing ratio conditions. The reason for this peak and valley is attributed to the effect of favorable pressure gradient due to Mach number on the inner sublayer thickness of the pseudo-turbulent boundary accompanied by a bypass transition again to a fully turbulent boundary layer.

Acknowledgment

We would like to express our gratitude to Dr. Nasir S. and Dr. Wing Ng. of Virginia Tech for their help and support of this paper. This work is supported by National Natural Science Foundation of China (NSFC No. 50976008).

References

- [1] Garg, V.K., and Ameri, A.A., 1997, "Comparison of Two-Equation Turbulence Models for Prediction of Heat Transfer on Film-Cooled Turbine Blades," *Numerical Heat Transfer, Part A* 32, 347-371.
- [2] Garg, V.K., 2001, "Modeling Film Coolant Flow Characteristics at the Exit of Shower Head Holes," *Intl. J. Heat and Fluid Flow*, 22, pp. 134-142.
- [3] Heidmann, J.D., Rigby, D.L., and Ameri, A.A., 2000, "A Three-Dimensional Coupled Internal/External Simulation of a Film-Cooled Turbine Vane," *ASME J. Turbomach.*, 122, pp. 348-359.
- [4] Chima, R. V., 1996, "A $k-\omega$ Turbulence Model for Quasi-Three-Dimensional Turbomachinery Flows," NASA TM-107051.
- [5] Ginibre, P., Lefebvre, and M., Liamis, N., 2001, "Numerical Investigation of Heat Transfer on Film-Cooled Turbine Blades," *Annals New York Academy of Sciences*, 934, pp. 377-384.
- [6] Medic, G., and Durbin, P.A., 2002, "Toward Improved Film Cooling Prediction", *ASME J. Turbomach.*, 124, pp. 193-199.
- [7] Montomoli, F., Adami, P., Della Gatta, S., and Martelli, F., 2004, "Conjugate Heat Transfer Modeling in Film Cooled Blades," *ASME GT2004-53177*.
- [8] Chana, K.S., and Mole, A.H., 2002, "Summary of Cooled NGV and Uncooled Rotor Measurements from the MT1 Single Stage High Pressure Turbine in the DERA Isentropic Light Piston Facility", Brite-EuRam Turbine Aero-Thermal External Flow project (BRPR-CT97-0519).
- [9] Laskowski, G.M., Tolpadi, A.K., and Ostrowski, M.C., 2007, "Heat Transfer Predictions of Film Cooled Stationary Turbine Airfoils," *ASME GT2007-27497*.
- [10] Menter, F. R., 1994, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA-Journal*, 32, pp.1598-1605.
- [11] Nasir, S., Carullo, J.S., Ng, W.F., Thole, K.A., Wu, H., Zhang, L.J., and Moon, H.K., 2007, "Effects of Large Scale High Freestream Turbulence, and Exit Reynolds Number on Turbine Vane Heat Transfer in a Transonic Cascade," *ASME IMECE-2008-44098*.
- [12] Nasir, S., Bolchoz, T., Ng, W.F., Zhang, L.J., Moon, H.K., and Anthony, R.J., 2008, "Showerhead Film Cooling Performance of a Turbine Vane at High Freestream Turbulence in a Transonic Cascade," *ASME IM-ECE-2008-66528*.
- [13] Durbin, P., 1991, "Near-Wall Turbulence Closure Modeling Without "damping functions"," *Theor. Comput. Fluid Dyn.*, 3, pp. 1-13.
- [14] Medic, G., and Durbin, P. A., 2002, "Toward Improved Prediction of Heat Transfer on Turbine Blades," *ASME J. Turbomach.*, 124, pp. 187-192.
- [15] Ameri, A. A., and Ajmani, K., 2004, "Evaluation of Predicted Heat Transfer on a Transonic Blade Using v^2 -f models," *ASME GT-2004-53572*.
- [16] Pecnik, R., Pieringer, P., and Sanz, W., 2005, "Numerical Investigation of the Secondary Flow of a Transonic Turbine Stage Using Various Turbulence Closures," *ASME GT-2005-68754*.
- [17] Luo, J., and Razinsky, E.H., 2007, "Conjugate Heat Transfer Analysis of a Cooled Turbine Vane Using the v^2 -f Turbulence Model," *ASME J. Turbomach.*, 129, pp. 773-781.
- [18] Luo, J., and Razinsky, E.H., 2008, "Prediction of Heat Transfer and Flow Transition on Transonic Turbine Air-foils under High Freestream Turbulence," *ASME GT-2008-50868*.
- [19] Heidmann, J.D., and Ekkad, S., 2007, "A Novel Anti-Vortex Turbine Film Cooling Hole Concept," *ASME GT-2007-27528*.
- [20] Rozati, A., and Tafti, D.K., 2007, "Large Eddy Simulation of Leading Edge Film

- Cooling Part-II: Heat Transfer and Effect of Blowing Ratio,” ASME GT-2007-27690
- [21] van Oudheusden, B.W., 2004, “Compressibility Effects on the Extended Crocco Relation and the Thermal Recovery Factor in Laminar Boundary Layer Flow,” ASME J. Turbomach., 126, pp. 32-41
- [22] Arts, T., and Bourguignon, A.E., 1990, “Behavior of a Coolant Film with Two rows of Holes along the Pressure Side of a High Pressure Nozzle Guide Vane,” ASME J. Turbomach., 112, pp. 512-520
- [23] Ou, S., Han, J.-C., Mehendale, A.B., and Lee, C.P., 1994, “Unsteady Wake Over a Linear Turbine Blade Cas-cade with Air and CO₂ Film Injection: Part I—Effect on Heat Transfer Coefficients,” ASME J. Turbomach., 116, pp. 721-729
- [24] Ekkad, S.V., Mehendale, A.B., Han, J.-C., and Lee, C.P., 1997, “Combined Effect of Grid Turbulence and Un-steady Wake on Film Effectiveness and Heat Transfer Coefficient of a Gas Turbine Blade with Air and CO₂ Film Injection,” ASME J. Turbomach., 119, pp. 594-600
- [25] Ames, F.E., 1996, “Experimental Study of Vane Heat Transfer and Film Cooling at Elevated Levels of Turbulence,” NASA CR-198525.
- [26] Weigand, B., Crawford, M.E., and Kays, W.M., 2004, “Convective Heat and Mass Transfer,” 4th Ed., McGraw-Hill Inc., New York
- [27] Turner, E.R., Wilson, M.D., Hylton, L.D., and Kaufman, R.M., 1985, “Turbine Vane External Heat Transfer: Vol. 1: Analytical and Experimental Evaluation of Surface Heat Transfer Distributions with Leading Edge Showerhead Film Cooling,” NASA CR-174827