Spring 2014

EFFECTS OF CROSSFLOW IN AN INTERNAL-COOLING CHANNEL ON FILM COOLING OF A FLAT PLATE THROUGH COMPOUND-ANGLE HOLES

Zachary Taylor Stratton
Purdue University

Follow this and additional works at: https://docs.lib.purdue.edu/open_access_theses

Part of the [Aerospace Engineering Commons](#)

Recommended Citation

Stratton, Zachary Taylor, "EFFECTS OF CROSSFLOW IN AN INTERNAL-COOLING CHANNEL ON FILM COOLING OF A FLAT PLATE THROUGH COMPOUND-ANGLE HOLES" (2014). *Open Access Theses*. 264.
https://docs.lib.purdue.edu/open_access_theses/264

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

PURDUE UNIVERSITY
GRADUATE SCHOOL
Thesis/Dissertation Acceptance

This is to certify that the thesis/dissertation prepared

By Zachary T. Stratton

Entitled

EFFECTS OF CROSSFLOW IN AN INTERNAL-COOLING CHANNEL ON FILM COOLING OF
A FLAT PLATE THROUGH COMPOUND-ANGLE HOLES

For the degree of Master of Science in Aeronautics and Astronautics

Is approved by the final examining committee:

Tom I-P. Shih

Gregory Blaisdell

Jun Chen

To the best of my knowledge and as understood by the student in the *Thesis/Dissertation Agreement, Publication Delay, and Certification/Disclaimer (Graduate School Form 32)*, this thesis/dissertation adheres to the provisions of Purdue University's "Policy on Integrity in Research" and the use of copyrighted material.

Tom I-P. Shih

Approved by Major Professor(s): _____

Approved by: Weinong Chen

04/26/2014

Head of the Department Graduate Program

Date

EFFECTS OF CROSSFLOW IN AN INTERNAL-COOLING CHANNEL ON
FILM COOLING OF A FLAT PLATE THROUGH COMPOUND-ANGLE HOLES

A Thesis

Submitted to the Faculty

of

Purdue University

by

Zachary T. Stratton

In Partial Fulfillment of the

Requirements for the Degree

of

Master of Science in Aeronautics and Astronautics

May 2014

Purdue University

West Lafayette, Indiana

ACKNOWLEDGEMENTS

I would like to express my sincere gratitude to Professor Tom I-P. Shih under whose guidance I could successfully complete my Master's thesis. I would like to thank Professor Greg Blaisdell and Professor Jun Chen for their willingness to be on my thesis committee. I would like to thank Dr. David Bogard and his team at The University of Texas at Austin for providing the detailed setup of their rig as well as adiabatic effectiveness measurements. Also, the support from Dr. Greg Laskowski and Dr. Rob Briggs of GE Aviation and Dr. Brian Barr and Dr. Jason Dees of GE Global Research is gratefully acknowledged.

TABLE OF CONTENTS

	Page
LIST OF TABLES	v
LIST OF FIGURES	vii
NOMENCLATURE	viii
ABSTRACT	x
1 INTRODUCTION	1
1.1 Background	1
1.2 Previous Work on Film Cooling	3
1.3 Objective	6
2 PROBLEM DESCRIPTION	7
3 PROBLEM FORMULATION AND NUMERICAL METHOD OF SOLUTION	11
3.1 Reynolds-Averaged Navier-Stokes	11
3.2 Large-Eddy Simulation	12
4 RESULTS	14
4.1 Verification	14
4.1.1 RANS Verification	14
4.1.2 LES Verification	16
4.2 Validation	17
4.3 Effects of Operating Parameters on Adiabatic Effectiveness	21
4.4 Discharge Coefficient	25
4.5 Nature of Flow	27

	Page
4.5.1 Steady-State Behavior	27
4.5.2 Unsteady Jet Behavior	41
5 CONCLUSIONS	43
REFERENCES	45

LIST OF TABLES

Table	Page
Table 1. Summary of cases	9

LIST OF FIGURES

Figure	Page
Figure 1. Turbine blade [22][23]	1
Figure 2. Film cooling illustration	2
Figure 3. Inside a turbine blade [24]	2
Figure 4. Experimental setup [21]	7
Figure 5. Computational model for RANS simulations (units: mm)	8
Figure 6. Computational model for LES	10
Figure 7. Grid system used in LES analysis.....	13
Figure 8. Grid system used in RANS analysis (Top: Coarse, Middle: Baseline, Bottom: Fine).....	15
Figure 9. Adiabatic effectiveness contours.....	15
Figure 10. Laterally-averaged adiabatic effectiveness	16
Figure 11. Top: Coarse side holes, Bottom: Refined side holes	17
Figure 12. Computed and experimentally measured laterally-averaged adiabatic effectiveness	18
Figure 13. Computed and experimentally measured adiabatic effectiveness contours for $BR = 0.5$	20
Figure 14. Computed and experimentally measured adiabatic effectiveness contours for $BR = 1.0$	20
Figure 15. Computed and experimentally measured adiabatic effectiveness contours for $BR = 1.5$	21
Figure 16. Adiabatic effectiveness for in-line crossflow direction.....	22

Figure	Page
Figure 17. Adiabatic effectiveness for counter crossflow direction	23
Figure 18. Comparison of in-line and counter crossflow directions for SST, k - ϵ , and experiment	24
Figure 19. Pressure drop in the internal-cooling channel for $BR = 0.5$	26
Figure 20. Computed discharge coefficients at each hole	26
Figure 21. Streamlines colored by velocity magnitude for SST and $BR = 1.0$ (Left: in-line. Right: counter)	28
Figure 22. Velocity magnitude in the hole for $BR = 1.0$ and counter crossflow ..	29
Figure 23. Iso-surfaces of the vortex core in the hole	30
Figure 24. Velocity magnitude in the hole for $BR = 1.0$	32
Figure 25. Comparison of velocity magnitude for cross-section 3	33
Figure 26. Streamlines for $BR = 1.0$ and counter crossflow from two viewpoints	34
Figure 27. Streamlines exiting the cooling hole for SST	36
Figure 28. Streamlines exiting the cooling hole for k - ϵ	36
Figure 29. Sketch of coolant streamlines exiting the hole	37
Figure 30. Non-dimensional temperature contours for $BR = 1.0$ and counter crossflow	38
Figure 31. Non-dimensional temperature contours for SST	39
Figure 32. Non-dimensional temperature contours for k - ϵ	39
Figure 33. Instantaneous flat plate adiabatic effectiveness contours and non-dimensional temperature contours at $x/d = 2$ for Case 2b	41

NOMENCLATURE

A_c	Cross-sectional area of the hole
BR	Blowing ratio $\rho_c U_c / \rho_\infty U_\infty$
$C_{d,i}$	Discharge coefficient at hole i
c_p	Specific heat at constant pressure
d	Film-cooling hole diameter
d_H	Hydraulic diameter
DR	Density ratio ρ_c / ρ_∞
L/d	Length-to-diameter ratio of the film-cooling hole
$\dot{m}_i$	Mass flow rate through hole i
P_i	Pressure at hole i
P_∞	Mainstream pressure
Re_{d_H}	Reynolds number based on hydraulic diameter
T_{aw}	Adiabatic wall temperature
T_c	Coolant temperature in internal-cooling channel
T_∞	Mainstream temperature
$U_{actual,i}$	Measured velocity in hole i
U_c	Coolant jet velocity
$U_{ideal,i}$	Theoretical velocity in hole i
u_τ	Friction velocity
U_∞	Mainstream velocity
VR	Velocity ratio U_c / U_∞
x	Streamwise coordinate measured from film-cooling hole trailing edge

y	Distance normal from film-cooled flat plate
y^+	Non-dimensional wall distance, $u_\tau y / \nu$
z	Spanwise coordinate measured from center of film-cooling hole

Greek

β	Compound angle of the film-cooling hole
δ	Boundary-layer thickness
η	Adiabatic effectiveness, $(T_{aw} - T_\infty) / (T_c - T_\infty)$
θ	Non-dimensional temperature, $(T - T_\infty) / (T_c - T_\infty)$
ν	Kinematic viscosity
ρ	Density
ϕ	Inclination angle of the film-cooling hole

ABSTRACT

Stratton, Zachary T. M.S.A.A., Purdue University, May 2014. Effects of Crossflow in an Internal-Cooling Channel on Film Cooling of a Flat Plate through Compound-Angle Holes. Major Professor: Tom I-P. Shih.

The film-cooling holes in turbine blades are fed from an internal cooling channel. This channel imposes a crossflow at the entrance of the holes that can significantly affect the performance of the cooling jets that emanate from those holes. In this study, CFD simulations based on steady RANS with the shear-stress transport (SST) and the realizable k - ϵ turbulence models were performed to study film cooling of a flat plate with cooling jets issuing from eight round holes with a compound angle of 45 degrees, where the coolant channel that fed the cooling jets was oriented perpendicular to the direction of the hot-gas flow. One case was also performed by using large-eddy simulation (LES) to get a sense of the unsteady nature of the flow. Operating conditions were chosen to match the laboratory conditions, which maintained a density ratio of 1.5 between the coolant and the hot gas. Parameters studied include internal crossflow direction and blowing ratios of 0.5, 1.0, and 1.5. Results obtained showed an unsteady vortex forms inside the hole, causing a side-to-side shedding of the coolant jet. Values of adiabatic effectiveness predicted by the CFD simulations were compared with experimentally measured values. Steady RANS was found to be inconsistent in its ability to predict adiabatic effectiveness with relative error ranging from 10% to over 100%. LES was able to predict adiabatic effectiveness with reasonable accuracy.

1 INTRODUCTION

1.1 Background

Gas turbine engines can achieve greater thermal efficiency by increasing the temperature at the turbine inlet. The temperature sought far exceeds the melting point of the turbine's material. Therefore, the turbine parts must be cooled to keep the thermal loads on the material at acceptable levels for improved life of the components. Figure 1 shows the image of a turbine blade, the turbine component that is the subject of this study.

Figure 1. Turbine blade [22][23]

There are two main techniques used to cool the turbine material. The first is internal cooling, where cooler air bled off from the compressor is passed through hollow channels in the turbine blade. To ensure that the coolest possible air in the channel is near the wall, turbulators such as ribs, bumps, or dimples line the walls to mix the

flow or to create streamwise swirling flows that transport cooler fluid in the middle of the channel to the wall. The second technique is film cooling where some of the air in the channel is passed through small holes such that a thin insulating layer forms next to the external wall of the blade, which is illustrated in Figure 2. These holes are often angled or shaped in a way that maximizes the coverage of the cool film. Figure 3 shows the complexity of a modern turbine blade, and this complexity has led to a lack of understanding on how some of these components interact.

Figure 2. Film cooling illustration

Figure 3. Inside a turbine blade [24]

A common way to analyze the film cooling performance is to measure the adiabatic effectiveness. The adiabatic effectiveness is the non-dimensional temperature on the surface of the blade and is defined as follows:

$$\eta = \frac{T_{aw} - T_{\infty}}{T_c - T_{\infty}} \quad (1.1)$$

Everywhere on the surface where this measurements is greater than zero will reveal where cool air from the film cooling hole is acting, and its magnitude will reveal the degree to which it is cooling. The blowing ratio, which is the ratio of the coolant to hot gas mass flux, is another important parameter in film cooling. It is defined as:

$$BR = \frac{\rho_c U_c}{\rho_{\infty} U_{\infty}} \quad (1.2)$$

For low BR s there is less mass flow through the coolant holes relative to the hot gas moving over the hole and the coolant typically remains attached to the blades external surface. For high BR s there is more coolant exiting the holes and thus more cooling potential, but as BR continues to increase, the coolant jet becomes stronger and can penetrate into the hot-gas without touching the surface. This lift off of the coolant is detrimental to the cooling performance.

1.2 Previous Work on Film Cooling

Film cooling has been extensively researched as there are many parameters that affect the adiabatic effectiveness and heat transfer. On the hot gas side of the hole the Reynolds number, Mach number, turbulence intensity, boundary layer thickness, and geometry all affect the cooling. The relation between the coolant and the hot gas including blowing ratio, velocity ratio, density ratio, and momentum flux ratio will also affect performance. The hole shape, angle, and length will affect the flow pattern

in the hole and on the coolant side the Reynolds number, geometry and presence of internal crossflow via a coolant channel or stagnant plenum all affect how the flow approaches the hole leading to different cooling results.

As the current study is primarily interested in the effect of internal crossflow on compound angle film cooling some past studies that focus on these concepts are reviewed.

In past studies of compound-angle holes the coolant is fed from a stagnant plenum. Ligrani et al. [1] showed that a row of holes with $\beta=30^\circ$ had higher adiabatic effectiveness for blowing ratios ranging from 0.5 to 1.5. Schmidt et al. [2] showed that a hole with $\beta = 60^\circ$ had improved the lateral spreading of the coolant jet and significantly improved adiabatic effectiveness at high momentum flux ratios over a streamwise injected hole ($\beta = 0^\circ$). Goldstein and Jin [3] used the heat/mass transfer analogy to study holes with $\beta=45^\circ$ noting that liftoff of the jet decreased compared to streamwise injection and again showing an increase in adiabatic effectiveness. Honami [4] looked at the behavior of the coolant jet for $\beta=90^\circ$ and showed that the laterally injected jet has an asymmetric structure with a large-scale vortex motion promoted by the primary stream on one side of the jet, but suppressed on the other side. Jung and Lee [5] studied holes at $\beta=0^\circ, 30^\circ, 60^\circ$, and 90° with varying velocity ratio and showed that as the compound angle increased, the vortex that forms as the coolant exits the hole becomes stronger and entrains more hot gas from the free-stream. They also noted that the impact of the compound angle on adiabatic effectiveness is more significant for higher velocity ratios. Aga et al. [6] used 3D PIV measurements to investigate the flow structure downstream of a compound-angle hole and confirmed that one side of the classic counter rotating vortex pair associated with a jet in crossflow is diminished significantly leading to a single dominant vortex. Lee et al. [7] studied aerodynamic losses associated with different compound angles and found

that when the velocity ratio is less than one there is a small increase in losses as the compound angle gets closer to 90° , with the highest losses occurring at $\beta=90^\circ$ at $VR = 2.0$. McGovern and Leylek [8] did a numerical study for $\beta=60^\circ$ using standard $k-\epsilon$ to look inside the hole to show that the hole entrance has a large separation region and is essentially unchanged from the streamwise injected hole except for the point of separation.

Another group of studies have been focused primarily on the effects of an internal crossflow on film cooling performance. Thole et al. [9] studied coolant flowing parallel to the mainstream, finding that the change in separation location at the hole entrance affected the velocity distribution at the hole exit decreasing the cooling capability. Kholi and Thole [10] did a numerical study using standard $k-\epsilon$, they found for a cooling channel perpendicular to the mainstream there is a single swirl motion in the hole as opposed to the separation region seen in a plenum fed hole, however this had little effect on the laterally-averaged adiabatic effectiveness for $BR = 0.5$. Wilfert and Wolff [11] found their internal coolant channel to increase adiabatic effectiveness by over 40% compared to a plenum fed case, they also found hole length-to-diameter ratio to play a significant role in cooling performance. Gritsch et al. [12] found the near-hole adiabatic effectiveness to change by over 100% when an internal crossflow was applied versus a plenum. Their study also showed that internal crossflow improved the performance of cylindrical holes, but reduced the performance of shaped holes. Kissel et al. [13] varied the coolant channel Re and found the heat transfer coefficient decreasing with increasing Re , while the adiabatic effectiveness is nearly unaffected. Saumweber and Schulz [14] studied internal crossflow oriented perpendicular to the mainstream and found, as [10] did, a single helical motion in the hole. They noted the intensity of this motion was linked to the coolant channel Mach number and can lead to large differences in adiabatic effectiveness.

1.3 Objective

The previous research highlights some important physical consequences of compound angles and internal crossflow on film cooling. Notably, compound angles lead to a dominant vortex downstream of the hole and internal crossflow creates a vortex in the cooling hole. To date, no studies have looked at the consequence of internal crossflow on compound-angle film cooling holes. Also, there has been little effort to validate the CFD, the notable exception being [13], but they do not match the experimental data. The effect of turbulence model choice on film cooling, especially for complex geometries is not well understood.

Thus the objective of this numerical study is twofold. The first objective is to understand the flow features for a compound-angle film-cooling hole subjected to an internal coolant crossflow with varying blowing ratios and coolant flow direction. Of particular interest is how the flow enters, moves through, and exits the hole and the impact of this flow structure on adiabatic effectiveness. The second objective is to understand the physical differences between turbulence models including the shear-stress-transport formulation of $k-\omega$, realizable $k-\epsilon$, and large-eddy simulation and how well they compare with the experimental data.

2 PROBLEM DESCRIPTION

The problem studied is shown in Figure 4 and Figure 5. It consists of a flat plate to be film cooled with cooling jets issuing from an internal-cooling channel through eight compound-angle holes with $d = 5$ mm. The cooling holes are spaced $6.25d$ apart and have a length-to-diameter of $L/d = 6$, an injection angle of $\phi = 30^\circ$, and a compound angle of $\beta = 45^\circ$. The internal-cooling channel has an aspect ratio of 2.8 and hydraulic diameter of 36.8 mm with a flow development length of $10d_H$ upstream and downstream of the holes; it is oriented perpendicular to the hot-gas path. The long development length on both sides of the holes allows for coolant to flow in either direction for the same geometry. These internal crossflow directions are referred to as “in-line” and “counter.”

Figure 4. Experimental setup [21]

Figure 5. Computational model for RANS simulations (units: mm)

The operating conditions were chosen to match the laboratory values which allows the study to be validated against the experimental data. The coolant is nitrogen at 202 K with $Re_{d_H} = 35,000$, the back pressure at the exit of the channel is adjusted to give rise to the desired blowing ratio for the cooling jets through the holes. For the flat plate side, the hot gas is air at $T_\infty = 303$ K and $P_\infty = 1$ atm with a turbulence intensity of 6%. The mainstream has a velocity of $U_\infty = 13.8$ m/s and a boundary-layer thickness of $\delta = 2.8d$ at the holes' leading edge. The coolant and hot gas temperatures give rise to a density ratio of 1.5. The density ratio is defined as follows:

$$DR = \frac{\rho_c}{\rho_\infty} \quad (2.1)$$

Blowing ratios of 0.5, 1.0, and 1.5 are studied which allow for the analysis of attached and detached jets. A summary of cases is seen in Table 1.

Table 1. Summary of cases

Case	Flow Direction	Blowing Ratio
1a	In-Line	0.5
1b	In-Line	1.0
1c	In-Line	1.5
2a	Counter	0.5
2b	Counter	1.0
2c	Counter	1.5

The computational domain consists of the following boundaries. A mass flow is imposed on the cool-gas inlet, and a back pressure is imposed on the cool-gas outlet. The coolant channel walls, film-cooling hole walls, and the flat plate are modeled as

adiabatic no-slip walls. Symmetry is imposed on the side walls of the hot-gas path, and a free-slip wall is imposed on the top wall. A velocity profile is imposed on the hot-gas inlet and an atmospheric back pressure is enforced on the hot-gas outlet.

Figure 6 shows the computational domain for the large-eddy simulation (LES) study. To reduce computational costs, the number of holes were reduced from eight to three. A trip wire with diameter 1.5875 mm is placed 150 mm upstream of the holes, this matches the laboratory setup. An internal trip wire is also used to generate a turbulent boundary layer in the cooling channel. Case 2b is studied for this setup. The operating conditions match those previously specified, except that the coolant used is air at 202 K, the density ratio is unchanged.

The boundary conditions imposed for the LES case are as follows. The coolant channel is bounded by a velocity inlet, and a pressure outlet. The hot-gas path has a uniform velocity inlet, a pressure outlet, periodic side walls, and a free-slip ceiling. The flat plate, trip wires, holes, and cooling-channel walls are adiabatic no-slip walls.

Figure 6. Computational model for LES

3 PROBLEM FORMULATION AND NUMERICAL METHOD OF SOLUTION

3.1 Reynolds-Averaged Navier-Stokes

For the operating conditions described in the previous section, the maximum Mach number is less than 0.2, thus density variations due to velocity is negligible. However, the temperature difference between the coolant and hot-gas give rise to a density ratio of 1.5. This is a significant density variation in the flow, therefore the fluid must be modeled as a compressible gas. In this study the flow is modeled by the ensemble averaged continuity, species balance, momentum (compressible Navier-Stokes), and energy equations with Sutherland's model for viscosity and temperature-dependent c_p and k . The gas is assumed to be thermally perfect.

All cases model the effects of turbulence with two separate methods. The first method closes the governing equations with the shear-stress transport (SST) version of k - ω by Menter [15]. With this model all equations can be integrated to the wall, through the viscous sub-layer. The second method closes the governing equations with the realizable k - ϵ model [16]. This model uses the two-layer model of Chen and Patel [17] in the near-wall region. This divides the turbulent portion of the boundary layer, where the two-equation k - ϵ model is used, from the viscous sub-layer where the one-equation model of Wolfshtein [18] is used. Wall functions were not used for either turbulence model.

Solutions to the governing equations were obtained by using the ANSYS® Fluent Version 14.0 code [19]. Since only steady-state solutions were sought, the SIMPLE pressure-velocity coupling scheme is used to generate solutions; this is a pressure based segregated solver. The fluxes for all equations at the cell faces were interpolated by using the second-order upwind scheme. Pressure was computed using second-order accuracy. For all computations, iterations were continued until all residuals for all equations plateau to ensure convergence to steady-state has been reached. When it plateaued the residual is less than 10^{-5} for continuity, less than 10^{-7} for energy, less than 10^{-6} for turbulent kinetic energy, and less than 10^{-5} for dissipation rate of turbulent kinetic energy.

The grid system is described in the verification section of the results.

3.2 Large-Eddy Simulation

Large-eddy simulation resolves the large scale turbulent eddies in the flow and models the smaller scales of the solution. This differs from the RANS approach which models all scales of the turbulence from very large to very small. In general, this means LES should yield more accurate results, but at much higher computational costs. The specific LES method employed in this study is based on Lattice-Boltzmann, which simulates a discrete fluid represented by particle densities in discrete space and time rather than solving the discretized Navier-Stokes equations [20]. The assumptions of compressibility, thermally perfect gas, and temperature-dependent properties are maintained.

LES was performed for Case 2b and solutions are obtained using the PowerFLOW® code [20]. PowerFLOW® performs very-large-eddy simulation (VLES) with the method outlined above. To model turbulence in the near wall region, PowerFLOW® incorporates a renormalization group form of the k - ϵ equations with

proprietary extensions. The sub-grid scales, or unresolved turbulence properties, are passed to the resolved large eddies by turbulent viscosity and turbulent Prandtl numbers.

The solver is inherently time-accurate and the time step used was 4.463×10^{-7} seconds. The grid system, seen in Figure 7, is generated automatically based on user-defined resolution using cubic elements called voxels. Three resolution levels were used, the side lengths of the voxels comprising these three levels were 0.25 mm, 0.5 mm, and 1 mm, for a total grid size of 37 million voxels.

Figure 7. Grid system used in LES analysis

4 RESULTS

4.1 Verification

4.1.1 RANS Verification

To ensure that the solutions presented are grid independent, a grid sensitivity study was performed. Three successively finer grids were used, seen in Figure 8. The coarse mesh, baseline mesh, and fine mesh have 5.5 million cells, 13 million cells, and 17.3 million cells, respectively. The y^+ is less than unity next to all walls on all grids. The adiabatic effectiveness on the flat plate was used as the criteria to judge grid independence. The study was conducted using SST for Case 1a. Figure 9 shows the lateral spreading of the coolant remains relatively unchanged as the grid is refined, except the magnitude increases where the adiabatic effectiveness is maximum. Figure 10 shows the laterally-averaged adiabatic effectiveness values for the three meshes. Since the baseline grid shows less than a 1% difference from the fine grid, it is used for the remainder of the study.

Figure 8. Grid system used in RANS analysis
(Top: Coarse, Middle: Baseline, Bottom: Fine)

Figure 9. Adiabatic effectiveness contours

Figure 10. Laterally-averaged adiabatic effectiveness

4.1.2 LES Verification

The LES verification examines the consequence of voxel size on the solution. The levels of resolution in PowerFLOW[®] are generated using the h-refinement technique, where a cubic element is subdivided into four smaller cubic elements. A y^+ of less than unity next to the walls is not achieved. Two grids were studied, the first refined only the region near the middle hole and the other refined the region near all three holes. The comparison of grids is seen in Figure 11. Both meshes contain the same size voxels, but the distribution was changed. The adiabatic effectiveness contours show a large difference between the different voxel densities. Since the effect of the holes at $z/d = \pm 6$ on the middle hole was unknown, they were refined as seen on the bottom grid. However, this had little effect on the middle hole. The difference in the adiabatic effectiveness of the side holes in the top figure are due to the top jet passing into the more refined region. A conclusive grid independency study would

require an additional round of grid refinement, however, this requires computational resources that are not available at the present time. Thus, the study proceeded with the bottom grid of Figure 11, to get a preliminary sense of the unsteady nature of the flow.

Figure 11. Top: Coarse side holes, Bottom: Refined side holes

4.2 Validation

The usefulness of the turbulence models in predicting the film-cooling performance is assessed by comparing computed and experimentally measured adiabatic effectiveness. The experimental data were provided by The University of Texas at Austin [21]. The computational setup, including geometry and flow parameters, is based

on the experimental setup as described in the problem description. The velocity profile imposed at the inlet of the computational domain was provided by the experimentalists, however, the turbulence profiles were not provided. Figure 12 shows laterally-averaged adiabatic effectiveness values for all the cases.

Figure 12. Computed and experimentally measured laterally-averaged adiabatic effectiveness

These plots show one undeniable source of error, which is the turbulence models. The SST and $k-\epsilon$ models show over a 60% difference at some point in every case

and this discrepancy will be explored more in a later section. The other source of error is in the conduction in the experiment. However, a conduction correction has already been applied to the experimental data provided, but it does not completely account for the near-hole region ($x/d < 4$). There is some agreement between the CFD and the experimental data, but it is inconsistent. The $k-\epsilon$ model performs well at low BR , but consistently under predicts the cooling at moderate and high BR . The SST model is closest to the experiment for the in-line case at $BR = 1.5$, but otherwise over predicts or under predicts cooling by as much as 100%. The LES case shows good agreement with the data except for small differences in the near-hole region. Figure 13 through Figure 15 below shows a more qualitative sense of where the adiabatic effectiveness discrepancies arise. The experimental contours in this figure also show the readily apparent effect of conduction in the near-hole region. SST shows a profile that overestimates lateral spreading of the jet, but manages to correctly predict coolant jet separation and reattachment for all blowing ratios. The $k-\epsilon$ model is closer to getting the width of the cooling footprints correct, but fails to see coolant jet separation as BR increases. LES does best at capturing the experimental cooling footprint.

Figure 13. Computed and experimentally measured adiabatic effectiveness contours for $BR = 0.5$

Figure 14. Computed and experimentally measured adiabatic effectiveness contours for $BR = 1.0$

Figure 15. Computed and experimentally measured adiabatic effectiveness contours for $BR = 1.5$

4.3 Effects of Operating Parameters on Adiabatic Effectiveness

This section will detail the trend in cooling performance. The trend is critical because if design decisions are to be made based on CFD, the solutions must accurately account for changing operating conditions. Laterally-averaged adiabatic effectiveness for the in-line crossflow direction for both SST and $k-\epsilon$ is seen in Figure 16 along with the contours. The models agree in the trend that performance drops as BR increases for $x/d < 20$. SST shows large dips followed by a spike in cooling just downstream of the hole due to separation and reattachment of the cooling jet for all BR s. These large separation regions are not seen in $k-\epsilon$, which shows no jet separation at all for $BR = 0.5$ and very little separation for higher BR s. The higher BR also means the jet has more momentum and is able to penetrate further in the lateral direction, which is accounted for in both models. The increasing laterally-averaged adiabatic effectiveness for $x/d > 10$ for the $k-\epsilon$ case suggests that coolant spreads out as it

mixes with the hot-gas enlarging its footprint on the flat plate. For all BR s studied, SST shows a much wider coolant footprint, as seen by the contours, than $k-\epsilon$. This accounts for the higher lateral averages, and the physics behind this will be shown in a later section.

Figure 16. Adiabatic effectiveness for in-line crossflow direction

Figure 17, shows how BR affects the cooling performance when the coolant is flowing in the counter crossflow direction. In this case, the trends between the turbulence models do not agree. SST shows $BR = 0.5$ to be the superior case over $BR = 1.0$ by a large margin, but $k-\epsilon$ shows $BR = 1.0$ to be superior everywhere except closest to the hole. SST again shows large dips and spikes in cooling near the hole due to separation and reattachment, notably on the $z/d < 0$ side of the hole. Again, SST shows a wider coolant footprint, but here it has two distinct peaks as

seen in the contour. This is a strange feature for a round film-cooling hole and is not seen in the k - ϵ solution or the experimental results. This will receive more attention when the flow field is analyzed more closely.

Figure 17. Adiabatic effectiveness for counter crossflow direction

Taking a closer look at the effect of internal-crossflow direction on the cooling effectiveness, reveals additional discrepancies between the turbulence models. Figure 18 shows exactly how the cooling trend changes with crossflow direction. SST predicts better cooling near the hole, but less cooling further downstream for the counter crossflow direction for $BR = 1.0$ and $BR = 1.5$ and better or equal cooling for $BR = 0.5$. The k - ϵ model agrees that cooling improves closer to the hole for $BR = 1.0$ and $BR = 1.5$, but it also shows cooling improves further downstream as well. However, k - ϵ shows a decrease in performance for $BR = 0.5$. The experimental data

is plotted together here, which shows in general that cooling performance improves for the counter-crossflow direction, with the biggest impact occurring for $BR = 1.0$.

Figure 18. Comparison of in-line and counter crossflow directions for SST, k - ϵ , and experiment

4.4 Discharge Coefficient

The discharge coefficient is the ratio of the actual discharge from the hole to the theoretical discharge and is a measure of the efficiency of the film cooling. The discharge coefficient will be defined as follows:

$$\Delta P_i = P_i - 1 \text{ atm} \quad (4.1)$$

$$U_{actual,i} = \frac{\dot{m}_i}{\rho A_c} \quad (4.2)$$

$$U_{ideal,i} = \sqrt{\frac{2\Delta P_i}{\rho}} \quad (4.3)$$

$$C_{d,i} = \frac{U_{actual,i}}{U_{ideal,i}} \quad (4.4)$$

A linear pressure drop is assumed in the channel and is defined by the inlet and outlet pressures of the coolant channel, which is seen in Figure 19. The points in the figure signify the pressures at the inlet, outlet, and at each of the eight holes. The density is chosen to be the density of nitrogen at 1 atm and 202 K.

Figure 19 reveals that the counter crossflow case requires higher pressure in the channel to achieve the same BR as the in-line crossflow case. However, the overall pressure drop in the channel is equal. It is not surprising then that the discharge coefficients in Figure 20 shows that the counter crossflow cases have lower discharge coefficients. This means that there are more losses in the hole for these cases. Another interesting point is that the in-line cases are relatively insensitive to changes in BR ,

whereas the counter cases see a much larger effect. A physical explanation for these differences is examined in the following section.

Figure 19. Pressure drop in the internal-cooling channel for $BR = 0.5$

Figure 20. Computed discharge coefficients at each hole

4.5 Nature of Flow

4.5.1 Steady-State Behavior

How the flow enters the hole, how it moves through the hole, and how it exits the hole and interacts with the hot-gas are the important flow features that will be analyzed. Figure 21 shows how flow separates upon entering the holes for each crossflow direction. Since the flow has a component of velocity parallel to the channel a swirling motion develops. This differs from a plenum-fed hole which simply sees a separation bubble. This swirling effect is a critical feature, as it completely changes the flow structure; references [9]-[14] have already stressed the importance of coolant-channel crossflow on film-cooling performance. An important difference in this study, however, is the addition of a compound angle which means the flow direction matters. For the in-line crossflow case, flow needs to turn less to enter the hole, which means there is a weaker vortical motion induced by the separation, seen by lower velocity magnitude in Figure 21. The counter crossflow case has a much stronger vortical motion induced by the separation due to the flow having to turn much more to enter the hole. Both turbulence models agree with this behavior, however, SST predicts a stronger vortex than $k-\epsilon$ in every case. The increased turning seen in the counter crossflow cases also leads to more losses at the entrance of the coolant hole, which is one reason why the counter crossflow cases have a lower discharge coefficient. As the blowing ratio increases, more coolant is being sucked into the hole and thus the strength of the vortex induced by the separation increases; again, this trend is seen in both models.

Figure 21. Streamlines colored by velocity magnitude for SST and $BR = 1.0$
(Left: in-line. Right: counter)

As flow moves up the hole it is divided into high and low velocity regions. Low velocity in the vortex core causes the flow around it to get squeezed, which increases the velocity of the fluid. This effect is very similar to how a separation bubble in a plenum-fed hole creates a high velocity region. A swirling motion in the hole is another consequence of the vortex that is created at the hole entrance. This swirling rotates the high and low velocity regions around each other. For the in-line crossflow, flow in the hole rotates counter-clockwise, and for the counter crossflow, flow rotates clockwise in the hole. Figure 22 compares velocity contours in the hole between LES, SST, and $k-\epsilon$ for Case 2b.

Comparing the performance of the turbulence models it is seen that LES and $k-\epsilon$ are most similar in terms of the shapes of the high and low velocity regions and how the magnitudes dissipate towards the end of the hole. SST, on the other hand, maintains a clear separation between high and low velocity regions. The stronger vortex is seen for SST in cross-section 1 where the velocity magnitude and tangential-velocity component (seen by the vectors) is much higher than $k-\epsilon$. Cross-section 5 also reveals that SST and $k-\epsilon$ agree with the swirling rate as both models show the velocity

regions rotating approximately the same degree by the end of the hole; however, LES predicts less rotation.

Figure 22. Velocity magnitude in the hole for $BR = 1.0$ and counter crossflow

Figure 23 shows q -criterion for SST and k - ϵ and λ_2 -criterion for LES which help to visualize the vortex core through the hole. This again shows that SST and k - ϵ may both be over predicting the rotation in the hole. This is a serious concern because where the high and low velocity regions exit the hole will determine how the coolant is spread over the flat plate, this directly influences the adiabatic effectiveness.

Figure 23. Iso-surfaces of the vortex core in the hole

It is important to remember that the LES results here are time-averaged and SST and $k-\epsilon$ are just steady RANS so while it is not apparent, this vortex in the hole is quite unsteady. Its radius grows and shrinks as it moves around, which constantly changes the velocity distribution. This will play a significant role when it is seen how the flow exits the hole, and is analyzed in more detail later.

Figure 24 shows how the velocity in the hole compares for the in-line and counter crossflow directions for both models. As was mentioned earlier, the swirling in the hole for the in-line case is opposite that of the counter crossflow case. This puts the high and low velocity regions on opposite sides as flow moves up the hole. The maximum velocity in the hole is less for the in-line case, especially for SST. This is due to the larger separation region that forms for the counter crossflow case, which squeezes the fluid around it to a greater degree. Both models, again, agree in terms of how much the flow rotates in the hole for the in-line case seen by the velocity distribution in cross-section 5. However, SST shows a higher tangential velocity component and a stronger vortex upon entering the hole.

The effect of BR is seen in Figure 25 which shows cross-section 3 (see Figure 24 for cross-section details) for the in-line and counter crossflow cases. Velocity increases in the hole as BR increases, this is expected since mass flow increases, by definition of BR . These images reveal that for the in-line case the vortex core is relatively insensitive to BR by remaining at the five o'clock position in the contour. However, in the counter crossflow case this vortex core does not stay in the same location and as BR increases there is less swirling in the hole. This means there are less losses in the hole and the flow can make the large turn more efficiently at higher BR . This was accounted for in Figure 20 where the discharge coefficient in the counter crossflow cases increased by a larger degree for increases in BR than the in-line crossflow cases.

Figure 24. Velocity magnitude in the hole for $BR = 1.0$

Figure 25. Comparison of velocity magnitude for cross-section 3

Taking a general look at the consequence of swirling flow in the hole it is possible to see that there will exist a position where these high and low velocity regions exit that provide optimal cooling performance. This means the length of the hole is a critical parameter when internal crossflow is involved, since short holes will allow for less rotation and long holes will allow for more rotation.

Figure 26 shows streamlines, colored by velocity magnitude, exiting the hole for Case 2b. These streamlines correspond to the velocity distributions that were seen in Figure 22. A common feature in all models is the strong counter-clockwise rotating vortex that forms on the downstream edge of the hole and becomes aligned with the freestream direction when it encounters the higher momentum hot gas. The classic counter-rotating-vortex-pair that occurs for streamwise injected holes is not seen since the vortex that should form on the right side of a streamwise injected hole is stifled due to the stagnation of the hot gas on the upstream side of the compound-angled coolant jet. This has been a well-documented feature of compound angle injection.

Figure 26. Streamlines for $BR = 1.0$ and counter crossflow from two viewpoints

The $k-\epsilon$ result is closest to LES based on the streamlines which would be expected based on the similarity of cross-section 5 in Figure 22. However, there is one significant issue with the $k-\epsilon$ solution and that is how well it seems to stay attached to the surface on the right side of the hole (seen by high adiabatic effectiveness) despite slightly higher momentum than LES and streamlines exiting at approximately the same angle.

SST has quite a different streamline pattern and seems to come out in almost two distinct jets. This is a direct consequence of the large velocity gradient in the hole

which maintains two distinct regions of high and low velocity throughout, as seen in previous figures. This difference in momentum causes the jet to split in two as it hits the hot gas. This effect is more pronounced for the counter crossflow cases because the low velocity region is on the left of the hole and does not have the high momentum region of the jet to “protect” it from the hot-gas as is the case for the in-line crossflow. This is why the adiabatic effectiveness exhibited two peaks in Figure 17.

The effect of BR and crossflow direction is shown in Figure 27 for SST and Figure 28 for $k-\epsilon$. These figures show that the in-line case has streamlines on the downstream end of the hole moving left, while streamlines on the upstream side are moving towards the right; this is just the opposite for the counter crossflow cases. The simplified streamlines sketched in Figure 29 illustrates this point. This is one reason why the adiabatic effectiveness seen in Figure 13 through Figure 15 is pushed towards the left side of the hole. The other factor is the lower momentum coolant on the left side of the jet which is less likely to lift off. However, all models still see some separation on the left side of the hole despite this low momentum fluid.

Figure 27. Streamlines exiting the cooling hole for SST

Figure 28. Streamlines exiting the cooling hole for $k-\epsilon$

Figure 29. Sketch of coolant streamlines exiting the hole

To see how the coolant is distributed in the streamlines a non-dimensional temperature will be defined:

$$\theta = \frac{T - T_{\infty}}{T_c - T_{\infty}} \quad (4.5)$$

Non-dimensional temperature contours are shown in Figure 30 at two locations downstream of the hole. Based on the differences in the streamlines from Figure 26 it is no surprise that SST shows a much different temperature distribution than LES and $k-\epsilon$. Notably there is a large inflection point in the contour as a result of the coolant jet splitting into two distinct parts. The entrainment of hot gas by the strong vortex is seen clearly in SST at $x/d = 5$. The red spot in the SST contour at $x/d = 2$ shows that most the coolant has lifted off and also that this model predicts less mixing with the hot-gas.

The excessive attachment of the coolant jet for $k-\epsilon$ that was mentioned earlier is seen again here with the coolest part of the jet touching the surface. Neither SST nor $k-\epsilon$ predict the same non-dimensional temperature distribution as LES which sees the coolest region of the jet more towards $z/d = 0$. Assuming LES is closest to the

experimental temperature distribution, it is seen that $k-\epsilon$ over predicts attachment and under predicts the inflection in the profile whereas SST shows just the opposite.

Figure 30. Non-dimensional temperature contours for $BR = 1.0$ and counter crossflow

The effect of BR and crossflow direction is shown in Figure 31 for SST and Figure 32 for $k-\epsilon$. The inflection in the contour continues to be a major difference between the models, except for $BR = 1.5$ for the in-line case which have a similar structure except for the amount of separation. For $BR = 1.0$ and $BR = 1.5$ the coolest region of the jet has lifted off from the flat plate, which is seen in both models. The exception is $BR = 1.0$ and counter crossflow for $k-\epsilon$, which still shows the coolest region attached. This explains why adiabatic effectiveness improved for this case in Figure 18.

Figure 31. Non-dimensional temperature contours for SST

Figure 32. Non-dimensional temperature contours for $k-\epsilon$

Some important conclusions can be drawn from the steady-state data presented thus far. It has been seen that $k-\epsilon$ is most similar to the LES in terms of flow structure, but there are still some small differences in velocity magnitudes and amount of swirling in the hole. The distinct deficiency in $k-\epsilon$, however, is its inability to predict the separation of the coolant jet; this was seen on the flat plate in Figure 13 through Figure 15. For $BR = 0.5$, when the coolant jet does not separate,

$k-\epsilon$ does pretty well and is within 10% of the experiment, but continues to do worse as BR increases. SST, on the other hand, does much better than $k-\epsilon$ at predicting the amount of separation at each BR and correctly predicts less separation for the counter crossflow cases. However, the insufficient mixing of the flow, on average, in the hole, creates a coolant jet that does not correctly spread the coolant over the flat plate leading to errors in adiabatic effectiveness.

4.5.2 Unsteady Jet Behavior

To understand how the jet behaves in time, Figure 33 shows instantaneous cooling data over several time steps with $\Delta t = 0.00045$ seconds. This figure shows there is a side-to-side shedding behavior of the jet, and the steady-state SST results have actually helped to understand the mechanism. The non-dimensional temperature contours exhibit a large inflection in the transient, a characteristic of SST (see Figure 31). This inflection was a direct consequence of the distinctive high and low momentum regions of the coolant jet. Therefore, the vortex in the hole grows in size, increasing the velocity difference in the hole, until it cannot get any larger. The high velocity is then ejected from the hole and the vortex shrinks, starting the process over again.

Figure 33. Instantaneous flat plate adiabatic effectiveness contours and non-dimensional temperature contours at $x/d = 2$ for Case 2b

Based on the complex flow features and inherent unsteadiness of the flow in this film cooling geometry it is not surprising that steady RANS methods do so poorly to predict the correct cooling performance. Also, SST and $k-\epsilon$ will predict different turbulence production, dissipation, and eddy viscosity in general, so the separation points and vortices present here will exaggerate their difference which has been clearly seen in the steady-state results.

5 CONCLUSIONS

A CFD study based on steady RANS with the SST and realizable k - ϵ turbulence models was performed to study a compound-angle film-cooling hole fed by an internal crossflow operating under laboratory conditions. One case was also performed with LES to get a sense of the unsteady nature of the flow. The simulations provided insight to the complex nature of the flow and the results on adiabatic effectiveness were compared with experimentally measured values.

When flow enters a cooling hole a separation region forms on the sharp edge, which induces a vortex and causes flow to swirl in the hole. This rotation is counter-clockwise for in-line crossflow and clockwise for counter crossflow. As BR increases the intensity of the swirling decreases. The clockwise swirling in the hole pushes the coolant jet to the left as it exits the hole. The vortex in the hole was found to be unsteady causing a side-to-side shedding of the jet. In general, SST and k - ϵ agreed on the flow features present in the flow.

In general, the adiabatic effectiveness predicted by steady RANS compared poorly with experimental data. The k - ϵ results compared best with experimental data at $BR = 0.5$ for both crossflow directions. This is because k - ϵ 's largest error is in predicting coolant jet separation and reattachment, but for $BR = 0.5$ the coolant jet does not separate, making this error negligible. SST performed best for $BR = 1.5$ for the in-line crossflow direction. This is because SST has difficulty predicting the swirling in the hole, but for $BR = 1.5$ and in-line crossflow the streamwise velocity component along the hole is dominant over the tangential component, making this

error negligible. LES was able to predict adiabatic effectiveness with reasonable accuracy for $BR = 1.0$ and counter crossflow.

REFERENCES

REFERENCES

- [1] Ligrani, P.M., Wigle, J.M., and Jackson, S.W., 1994, "Film-Cooling from Holes with Compound Angle Orientations: Part 2 Results Downstream of a Single Row of Holes with 6d Spanwise Spacing," *Journal of Heat Transfer*, Vol. 116, pp. 353-362.
- [2] Schmidt, D., Sen, B., and Bogard, D., 1996, "Film Cooling with Compound Angle Holes: Adiabatic Effectiveness," *Journal of Turbomachinery*, Vol. 118, pp. 807-813.
- [3] Goldstein, R.J. and Jin, P., 2001, "Film Cooling Downstream of a Row of Discrete Holes With Compound Angle," *Journal of Turbomachinery*, Vol. 123, pp. 222-230.
- [4] 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," *Journal of Turbomachinery*, Vol. 116, pp. 106-112.
- [5] Jung, I.S., and 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 Turbomachinery*, Vol. 122, pp. 153-163.
- [6] Aga, V., Rose, M., and Abhari, R.S., 2008, "Experimental Flow Structure Investigation of Compound Angled Film Cooling," *Journal of Turbomachinery*, Vol. 130, p. 031005.

- [7] Lee, S.W., Kim, Y.B., and Lee, J.S., 1997, "Flow Characteristics and Aerodynamic Losses of Film Cooling Jets With Compound Angle Orientations," *Journal of Turbomachinery*, Vol. 119, pp. 310-319.
- [8] McGovern, K.T., and Leylek, J.H., 2000, "A Detailed Analysis of Film-Cooling Physics: Part II Compound Angled Injection with Cylindrical Holes," *Journal of Turbomachinery*, Vol. 122, pp. 113-121.
- [9] Thole, K.A., Gritsch, M., Schulz, A., and Wittig, S., 1997, "Effect of a Crossflow at the Entrance to a Film-Cooling Hole," *Journal of Fluids Engineering*, Vol. 119, pp. 533-540.
- [10] Kohli, A., and Thole, K.A., 1997, "A CFD Investigation on the Effects of Entrance Crossflow Directions to Film-Cooling Holes," *Proc. Of the 32nd National Heat Transfer Conference, Baltimore, Maryland, USA*.
- [11] Wilfert, G., and Wolff, S., 2000, "Influence of Internal Flow on Film Cooling Effectiveness," *Journal of Turbomachinery*, Vol. 122, pp. 327-333.
- [12] Gritsch, M., Schulz, A., and Wittig, S., 2003, "Effect of Internal Coolant Crossflow on the Effectiveness of Shaped Film-Cooling Holes," *Journal of Turbomachinery*, Vol. 125, pp. 547-554.
- [13] Kissel H.P., Weigand, B., von Wolfersdorf, J., Neumann, S.O., 2007, "An Experimental and Numerical Investigation of the Effect of Cooling Channel Crossflow on Film-Cooling Performance," ASME Paper, GT2007-27102.
- [14] Saumweber, C., and Schulz, A., 2008, "Comparison of the Cooling Performance of Cylindrical and Fan-Shaped Cooling Holes with Special Emphasis on the Effect of Internal Coolant Crossflow," ASME paper, GT2008-51036.
- [15] Menter, F.R., 1993, "Zonal Two Equation $k-\omega$ Turbulence Models for Aerodynamic Flows," AIAA Paper 93-2906.

- [16] Shih, T.-H., Liou, W.W., Shabbir, A., Yang, Z., Zhu, J., 1994, “A New k - ϵ Eddy viscosity Model for High Reynolds Number Turbulent Flows Model Development and Validation,” *NASA Technical Memorandum*, 106721.
- [17] Chen, H.C., and Patel, V.C., 1988, “Near-Wall Turbulence Models for Complex Flows Including Separation,” *AIAA Paper*, Vol. 26, pp. 604-648.
- [18] Wolfshtein, M., 1969, “The Velocity and Temperature Distribution in One-Dimensional Flow with Turbulence Augmentation and Pressure Gradient,” *International J. Heat Mass Transfer*, Vol. 12, pp. 301-318.
- [19] *ANSYS®Fluent, Release 14.0*
- [20] Exa Corporation, 2013, “PowerFLOW® User’s Guide Release 5.0”
- [21] McClintic, J.W., Klavetter, S.R., Anderson, J.B., Winka, J.R., Bogard, D.G., Dees, J.E., Laskowski, G.M., Briggs, R., 2014, “The Effect of Internal Cross-Flow on the Adiabatic Effectiveness of Compound Angle Film-Cooling Holes,” *ASME Paper*, GT2014-25975.
- [22] Burd, S., Kaszeta, R., “What is Film Cooling,” University of Minnesota, <http://www.me.umn.edu/labs/tcht/measurements/what.html>
- [23] Aainsqatsi, K., 2008, “Turbofan Operation,” Wikipedia under GNU Free Documentation License, http://en.wikipedia.org/wiki/File:Turbofan_operation.svg
- [24] Han, J.C., Park, J.S., Lie, C.K., 1984, “Heat Transfer and Pressure Drop in Blade Cooling Channels with Turbulence Promoters,” Texas A&M University (prepared for NASA CR-3837)