Investigation of film cooling effectiveness and enhancement of cooling performance

Sangkwon Na
Iowa State University

Follow this and additional works at: https://lib.dr.iastate.edu/rtd
Part of the Aerospace Engineering Commons

Recommended Citation
https://lib.dr.iastate.edu/rtd/1925

This Dissertation is brought to you for free and open access by the Iowa State University Capstones, Theses and Dissertations at Iowa State University Digital Repository. It has been accepted for inclusion in Retrospective Theses and Dissertations by an authorized administrator of Iowa State University Digital Repository. For more information, please contact digirep@iastate.edu.
Investigation of film cooling effectiveness
and enhancement of cooling performance

by

Sangkwon Na

A dissertation submitted to the graduate faculty
in partial fulfillment of the requirements for the degree of

DOCTOR OF PHILOSOPHY

Major: Aerospace Engineering

Program of Study Committee:
Tom I-P. Shih, Major Professor
Francine Battaglia
Richard H. Pletcher
Alric Rothmayer
Z. J. Wang

Iowa State University
Ames, Iowa
2006
INFORMATION TO USERS

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleed-through, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

UMI Microform 3328090
Copyright 2008 by ProQuest LLC.
All rights reserved. This microform edition is protected against unauthorized copying under Title 17, United States Code.

ProQuest LLC
789 E. Eisenhower Parkway
PO Box 1346
Ann Arbor, MI 48106-1346
Graduate College
Iowa State University

This is to certify that the doctoral dissertation of

Sangkwon Na

has met the dissertation requirements of Iowa State University

Signature was redacted for privacy.

Major Professor

Signature was redacted for privacy.

For the Major Program
# TABLE OF CONTENTS

**NOMENCLATURE** vi

**ABSTRACT** ix

**CHAPTER 1. INTRODUCTION** 1
  1.1 Motivation and Objectives 1
  1.2 Literature Survey 2
    1.2.1 Effects of roughness 3
    1.2.2 Injection from slots 5
    1.2.3 Injection from circular holes 6
      1.2.3.1 Effects of blowing ratio 6
      1.2.3.2 Effects of density ratio 9
      1.2.3.3 Effects of cooling hole diameter-to-length ratio 11
      1.2.3.4 Effects of freestream turbulence 13
      1.2.3.5 Effects of flow on film-cooling effectiveness 15
    1.2.4 Injection from shaped holes 19
    1.2.5 Computational Studies on Film Cooling 21
  1.3 Dissertation Organization 26

**CHAPTER 2. PROBLEM DESCRIPTION** 27
  2.1 Film Cooling of a Flat Plate 27
  2.2 Film Cooling with Upstream and Downstream Flow Modifiers 28
  2.3 Film Cooling by Using a New Shaped Hole 29

**CHAPTER 3. FORMULATION AND NUMERICAL METHOD OF SOLUTION** 34
  3.1 Conservation/Balance Equations 34
  3.2 Turbulence Modeling 36
  3.3 Boundary Conditions 43
  3.4 Numerical Methods of Solution 44

**CHAPTER 4. CFD ANALYSIS OF FILM COOLING** 48
  4.1 Abstract 48
  4.2 Introduction 49
  5.3 Problem Description 51
  4.4 Formulation and Numerical Method of Solution 52
  4.5 Grid System Used 53
  4.6 Results 54
4.7 Summary 55

CHAPTER 5. EFFECTS OF COATING BLOCKAGE AND ROUGHNESS ON FILM COOLING EFFECTIVENESS AND SURFACE HEAT TRANSFER 66
5.1 Abstract 66
5.2 Introduction 66
5.3 Problem Description 69
5.3.1 3-D Problem: with and without TBC 69
5.3.2 2-D Problem: with and without TBC and TBC roughness 71
5.4 Formulation and Numerical Method of Solution 71
5.5 Grid System Used 72
5.5.1 3-D Problem: with and without TBC 72
5.5.2 2-D Problem: with and without TBC and TBC roughness 73
5.6 Results 74
5.6.1 3-D Problem: with and without TBC 74
5.6.2 2-D Problem: with and without TBC and TBC roughness 75
5.7 Summary 76

CHAPTER 6. PREVENTING HOT-GAS INGESTION BY FILM-COOLING JET VIA FLOW-ALIGNED BLOCKERS 102
6.1 Abstract 102
6.2 Introduction 103
6.3 Problem Description 105
6.4 Formulation and Numerical Method of Solution 107
6.5 Grid-Sensitivity and Validation Study 107
6.6 Results 109
6.7 Summary 113

CHAPTER 7. INCREASING ADIABATIC FILM-COOLING EFFECTIVENESS BY USING AN UPSTREAM RAMP 126
7.1 Abstract 126
7.2 Introduction 127
7.3 Problem Description 129
7.4 Formulation and Numerical Method of Solution 131
7.5 Grid-Sensitivity and Validation Study 131
7.6 Results 133
7.6.1 Nature of the Flow Induced by an Upstream Ramp 133
7.6.2 Adiabatic Effectiveness 134
7.6.3 Heat Transfer 135
7.7 Summary

CHAPTER 8. INCREASING ADIABATIC EFFECTIVENESS BY USING RAMP AND BLOCKERS

8.1 Grid-Sensitivity
8.2 Nature of the Flow
8.3 Adiabatic Effectiveness
8.4 Summary

CHAPTER 9. A NEW SHAPED HOLE

9.1 Grid-Sensitivity
9.2 Nature of the Flow
9.3 Adiabatic Effectiveness
9.4 Summary

CHAPTER 10. CONCLUSION

10.1 Four New Film Cooling Design Paradigms
10.2 Effects of TBC Blockage and Roughness on the Film Cooling

BIBLIOGRAPHY

ACKNOWLEDGEMENTS
NOMENCLATURE

Roman Symbols

A \quad \text{surface area}

C_p \quad \text{ratio of specific heat}

D \quad \text{film-cooling hole diameter}

e \quad \text{specific internal energy}

\epsilon_0 \quad \text{total energy}

H \quad \text{heat transfer coefficient} = \frac{q''}{(T_w - T_{aw})} \text{ or distance between the ridge and valleys of a shaped hole}

Ku \quad \text{kurtosis of roughness}

k \quad \text{turbulence kinetic energy or thermal conductivity}

L \quad \text{length of hole}

M \quad \text{blowing ratio} = \frac{(\rho U)_c}{(\rho U)_{\infty}}

P \quad \text{static pressure}

Pr \quad \text{Prandtl Number} = \frac{\mu C_p}{k}

q'' \quad \text{surface heat flux}

Re \quad \text{Reynolds number}

Rq \quad \text{root mean square}

S \quad \text{source of property per unit volume or mean rate of strain}

s \quad \text{spacing between holes}

Sk \quad \text{skewness of roughness}

T, T_c, T_{\infty} \quad \text{temperature, coolant inlet temp., hot-gas temp.}

T_w, T_{aw} \quad \text{wall temperature, adiabatic wall temperature}
\( u, v, w \) Cartesian velocity components in \( x, y, z \) directions

\( U_c \) average speed of coolant flow in hole

\( U_\infty \) speed of hot gas at freestream

\( u_r \) friction velocity: \((\tau_w/\rho)^{0.5}\)

\( V \) cell volume

\( x, y, z \) Cartesian coordinates

\( y^+ \) distance to wall in wall coordinate: \( \rho u y/\mu \) (\( y \) is normal distance from the wall)

**Greek Symbols**

\( \alpha \) injection angle of cooling jets or angle of upstream ramp

\( \beta \) distance between upstream ramp end and the leading edge of exit of hole

\( \Gamma \) diffusion coefficient

\( \gamma \) specific heat ratio

\( \delta \) boundary layer thickness

\( \varepsilon \) dissipation rate

\( \eta \) adiabatic effectiveness/normalized temperature: \( (T_\infty-T)/(T_\infty-T_c)\)

\( \bar{\eta} \) laterally averaged adiabatic effectiveness (i.e., averaged along the spanwise direction)

\( \mu \) molecular dynamic viscosity

\( \mu_r \) eddy viscosity

\( \nu \) molecular kinematic viscosity

\( \rho \) density

\( \tau_w \) wall shear stress
Subscripts
aw  adiabatic wall value

c   cooling flow value

\(i, j, k\) indices for Cartesian coordinate

t  turbulence

w  wall value

0  total value

\(\infty\) mainstream

Abbreviations
CFD  computational fluid dynamics
CRV  counter rotating vortex
CVP  counter rotating vortex pair
DNS  direct numerical simulation
DR   density ratio
EDM  electrical discharge machining
LES  large eddy simulation
MBC  metallic-bond coating
N-S  Navier-Stokes
RANS Reynolds-averaged Navier-Stokes
TBCs thermal barrier coatings
TGO  thermally grown oxide
VR   velocity ratio
ABSTRACT

Advanced gas turbines are designed to operate at increasingly higher inlet temperatures to increase efficiency and specific power output. This increase in the operating temperature is enabled by advances in high-temperature resistant materials such as super alloys and thermal-barrier coatings (TBCs) and by the development of effective cooling methods that lower the temperature of all surfaces that come in contact with the hot gases. Since the lower-temperature air used for cooling is extracted from the compressor, efficiency considerations demand effective cooling with minimum cooling flow.

This study focuses on film cooling with a twofold objective. The first is to examine the effects of TBC blockage and surface roughness on film-cooling effectiveness. The second objective is to explore, develop, and evaluate more efficient film-cooling methods. This study is accomplished by using second-order accurate computational fluid dynamics (CFD) analyses of the "compressible" Navier-Stokes equations in which the details of the film-cooling geometry and relevant flow features are resolved. To ensure that the solutions generated are meaningful, a grid-sensitivity study was performed for each configuration examined. Also, a validation study was performed to assess the turbulence models used.

On TBC blockage, results obtained show that when the mass-flux ratio is fixed at 0.5, blockage reduces adiabatic effectiveness by decreasing the amount of coolant flow through the cooling hole. However, the amount of decrease in adiabatic effectiveness is less than expected in that the coolant flow is reduced considerably but adiabatic effectiveness is reduced only slightly. This indicates the TBC blockage configuration studied resulted in a more efficient film-cooling hole.
On more effective film-cooling methods, this study focused on ways to reduce or prevent the hot-gas entrainment, which has reduced the usefulness of this important and widely used method of cooling. Four promising design paradigms were developed and evaluated: flow-aligned blockers, upstream ramp, ramp and blocker, and momentum-preserving shaped holes. The usefulness of all paradigms was evaluated by examining how they improve or degrade the film cooling of a flat plate in which the coolant is injected from a plenum through one row of inclined circular holes. The flow-aligned blockers were developed to minimize the entrainment of hot gases underneath film-cooling jets by the counter-rotating vortices within the jets. Computations were performed to assess the usefulness of rectangular prisms as blockers in increasing film-cooling adiabatic effectiveness without unduly increasing surface heat transfer and pressure loss. Results obtained show that with these blockers, the laterally averaged adiabatic effectiveness at 15D downstream of the film-cooling hole is as high as that at 1D downstream. The upstream ramp with a backward-facing step was developed to modify the interaction between the approaching boundary-layer flow and the film-cooling jet to increase film-cooling effectiveness. Results obtained show the laterally averaged adiabatic effectiveness can be two or more times higher than the case without the ramp. Also, the ramp was found to increase the surface coverage by each film-cooling jet in protecting the exposed hot surfaces. The flow-aligned blockers and the ramp were combined to utilize the advantages of both design concepts. Results obtained show the combined blocker and ramp to perform better than the blocker by itself and the ramp by itself. Lastly, momentum-preserving shaped holes were developed to minimize hot-gas entrainment and to enhance coverage with improved penetration when compared to existing shaped holes. The new shape-hole
concept is actually similar to the blocker concept except instead of protruding from the surface, it cuts into the surface. With a W-shaped, momentum-preserving shaped hole, the laterally averaged effectiveness from the hole exit up to 15D downstream of the hole increased from 50% to 100% when compared to unshaped hole. One of the major advantages of the newly proposed shaped hole is greatly reduced drag in addition to greatly improved film-cooling effectiveness.
1.1 Motivation and Objectives

Gas turbine engines are widely used for electric generation and propulsion. One effective way to increase efficiency is to increase the temperature that enters the turbine stage. The temperatures sought today are so high – up to 2650 °F for electric power generation and 3000 °F for aircraft – that no materials can maintain structural integrity with reasonable service life. Therefore, it is critical to improve not only the materials used to make turbines, but also the effectiveness and efficiency with which turbines are cooled, referred to as thermal management.

The first-stage stator of the gas turbine experiences the highest gas temperatures, and so poses the greatest challenge on both the materials and the cooling. On materials, recent developments in thermal-barrier coatings (TBCs) have shown considerable promise, but TBCs become rough and can disappear with service. Also, TBCs can adversely affect cooling. On thermal management, film cooling in addition to internal and impingement cooling are needed. Film cooling cools by injecting lower temperature gas extracted from the compressor through rows of holes with the goal of forming an insulating layer next to the material surface so that the hot gases never touches the surface (Fig. 1.1). Though, film cooling is indispensable, especially for the first-stage stator, the cooling jets always lift off the surface that they are intended to protect and entrain hot gas to the surface, and this remains an unsolved problem.

The main objective of this study is to use computational fluid dynamics (CFD) simulations to explore, develop, and evaluate film-cooling strategies that reduce cooling-jet
lift off and hot-gas entrainment so that film cooling can be made more effective in protecting surfaces. The second objective is to examine the effects of TBC blockage and surface roughness on film cooling. Grid sensitivity and validation studies will be performed to ensure that the CFD results generated are meaningful.

![Figure 1.1. Schematic of film cooling of a flat plate.](image)

1.2 Literature Survey

In this section, we will review the literature on film cooling with focus on cooling-jet lift off, hot-gas entrainment, and computational methods. But, before presenting this review, some technical terms are defined and explained.

The usefulness of film cooling is defined by a parameter known as film-cooling adiabatic effectiveness or simply as effectiveness, and it is defined by

$$\eta = \frac{T_{aw} - T_{on}}{T_{aw} - T_c}$$

where \(\eta\) is the film cooling effectiveness; \(T_{aw}\) is the adiabatic wall temperature; \(T_{on}\) is the temperature of the hot gas; and \(T_c\) is the temperature of the injected coolant. The often used integrated value of \(h\), referred to as laterally averaged film cooling effectiveness, is defined by
\[
\eta = \frac{T_{\infty} - T_{\text{wall}}}{T_{\text{in}} - T_{c}}
\]

Other parameters of importance in film cooling are density ratio \((\rho_c / \rho_a)\), mass flux or blowing ratio \((\rho_c U_c / \rho_a U_a)\), lateral spacing between the centers of film-cooling holes \((s)\), inclination of the film-cooling hole with respect the surface to be cooled \((\alpha)\), film-cooling hole length to diameter ratio \((L/D)\), hot-gas boundary layer thickness to film-cooling hole diameter ratio \((\delta/D)\), cooling jet Reynolds number \((U_c D / \nu_c)\), and hot gas Reynolds number \((U_x x / \nu_x)\). The velocity ratio \((U_c / U_a)\) and the momentum flux ratio \((\rho_c U_c^2 / \rho_a U_a^2)\) are used to describe interactions between the jet and mainstream.

1.2.1 Effects of roughness

All surfaces in the stators and rotors such as blades, vanes, endwalls, and hubs that come in contact with the combustor’s hot gases must be cooled in which the cooling must account for the surface roughness that can develop. The degree and the nature of the roughness and material degradation due to mechanisms such as erosion, fuel deposition, corrosion, and spallation of thermal-barrier coatings depend on the environment from which the air is ingested, the effectiveness of cooling management such as film cooling in maintaining material temperatures within acceptable limits, the operating conditions, and the duration of service.

Though computational fluid dynamics (CFD) has provided some meaningful results for film-cooling (Lin and Shih, 1999; Shih et al., 1999; Walters and Leylek, 2000; Lin and Shih, 2001; Shih and Sultanian, 2001; Liu and Pletcher, 2005), challenges remain in turbulence modeling (Durbin and Shih, 2005) and grid resolution requirements (Shih, 2006). This challenge is exacerbated by manufacturing issues related to thermal-barrier coatings.
(TBCs) and by surface roughness that develop during service.

On TBCs, the manufacturing process creates non-uniformities in the coating about the film-cooling holes. The extent of non-uniformity can be significant. To illustrate, it is noted that typical film-cooling holes are 0.8 mm in diameter, whereas the thickness of the TBC varies from 0.3 to 0.5 mm and the thickness of the metallic-bond coat that connects the TBC to the turbine metal surface varies from 0.1 to 0.2 mm. Figure 1.2 illustrates the non-uniformity that can result. From Fig. 1.2, it can be seen that the non-uniformity can have a significant effect of how the film-cooling jet emerges from the film-cooling hole. There are two ways to overcome this non-uniformity. One way is to remove the TBC and metallic-bond coating (MBC) in the film-cooling hole by a combination of electrical discharge machining (EDM) and laser drilling. But, this can be expensive and lasers cannot be used to fashion shaped holes. The other is to understand and assess the effects of the non-uniformity on the effectiveness of the film-cooling jet.

On roughness, turbine surfaces invariably become rough with service (Taylor, 1990; Tarada and Suzuki, 1993; Bons et al., 2001). This material degradation in the form of surface roughness is known to increase surface heat transfer in a significant way (Tarada and Suzuki, 1993; Blair, 1994; Hoff's et al., 1996; Abuaf et al., 1998; Bogard et al., 1998; Hodge et al., 2001; Bons, 2002). For a given cooling management, increase in surface heat transfer increases material temperature, which hastens further material degradation. The importance of surface roughness on surface heat transfer has lead many investigators to study this problem. For film cooling, the main challenge is to understand the effects of roughness on film-cooling effectiveness and surface heat transfer. To date, no CFD studies have been reported on film cooling with TBC blockage or roughness.
1.2.2 Injection from slots

Early work focused on use of slots because it provides a uniform film that flows along the downstream surface and does not bring in lateral thermal differences, which can cause thermal stress in turbine engine components and shorten the life of the components. Wieghardt first measured film temperature from a two-dimensional slot on a flat plate with various density ratios of 0.81 to 0.91 and blowing rate from 0.2 to 1.14 in 1943 and a great number of studies are accomplished by Goldstein (1968), Pederson (1977), Wieghardt (1946), Eckert (1964), Seban and Back (1962), Metzger (1968), Whitelaw (1967) and etc. for 30
Goldstein and Haji-Sheikh (1967) proposed several models that account for the effects of blowing on the entrainment and distribution of temperature within the boundary layer. Metzger et al. (1968) investigated slot jet with various injection angles ranging from 20 to 60 deg, blowing ratios, temperature ratios, and slot-lengths to hole. Goldstein (1971) gives excellent review of the investigation through 1971. Recently, computational studies are carried on the slot jet by Irmisch (1995), Garg and Gauger (1995). Irmisch (1995) examined the a turbine blade airfoil with leading edge slots using unstructured grid to capture the complex airfoil shape and used a realistic film cooling geometry. Garg and Gauger (1995) studied the effect of exit velocity and temperature distribution on the film cooling performance. Despite good performance of two-dimensional slots, they are not often used because of mechanical design considerations (Fig. 1.4). As a result, the concerns of investigators move to the film cooling using the discrete holes.

![Schematics of slot and a row of inclined holes.](image)

Figure 1.4. Schematics of slot and a row of inclined holes.

### 1.2.3 Injection from circular holes

#### 1.2.3.1 Effects of blowing ratio

The features of the film cooling using the discrete hole is based on jet in a cross flow. In the study of the jet in a cross flow, in general, a single jet ejection from a relatively long
delivery pipe was considered, and the hole was smooth and oriented perpendicular to the mainstream. A great deal of film cooling via a row of holes was investigated by Eckert and his co-workers. Goldstein (1968) mentioned the deformation of the jet is strongly affected by the blowing ratio, and, in general, the injected fluid penetrates farther into the stream when the blowing ratio is larger. He showed effect of blowing rate on centerline film cooling effectiveness in Fig. 1.5, which is most typical feature of film cooling from rows of holes. As the coolant-to-mainstream blowing ratio, $M$, increases, the effectiveness first increases, and reaches a maximum at a blowing ratio of 0.5, then decreases. This maximum is attributed to the penetration of the jet into the mainstream as opposed to its attaching on the surface. The effectiveness was strongest near the row of holes and very weak for downstream. From the investigation on the blowing ratio on the effectiveness by Brown and Saluja (1979), the optimal blowing ratio was 0.5 for injection into favorable and adverse free stream pressure gradient regions. They reported also the optimum blowing ratio for the three rows of holes is also about 0.4 to 0.5.

Crabb et al. (1981) studied relatively high velocity ratios of 1.15 and 2.3 with single-component Laser Doppler Velocimetry (LDV) and X-sensor hot-wire anemometry for an inclination of 90 deg. They showed a change from anisotropic turbulence in the region near the jet exit to a more isotropic flow further downstream by means of measuring the turbulence normal stresses. The most detailed study of the fluid mechanics associated with a jet issuing into a mainstream was made by Andreopoulos and Rodi (1984). They studied a single normal jet issuing from a pipe on three velocity ratios; 0.5, 1.0, and 2.0 using triple-sensor hot wire to measure all three velocity components simultaneously. The mainstream created a pressure field which influenced the pipe flow upstream of the exit, causing a highly
non-uniform exit profile skewed towards the downstream edge of the hole. Once emerged, the jet was bent over abruptly by the cross stream and two longitudinal counter-rotating vortices formed, causing the jet cross-section to appear kidney shaped. The highest turbulence levels and shear stress levels occurred at locations from two to four diameters downstream of the hole, and were coincident with the maximum mean velocity gradients. The large velocity gradients in this region were due to a shear layer that existed above the wake region, downstream of the hole exit. Far downstream of the hole the flow develops towards a standard boundary layer. An important conclusion from their work was that turbulent processes were significant at lower velocity ratios.

Bergeles et al. (1977) studied the region near the exit of an inclined jet. They used a single jet, issuing from a 50D long pipe with an inclination of 30 deg over a range of velocity ratio from 0.1 to 1.5. They reported that the strong disturbances are caused by injection, but the jet remains attached to the surface for velocity ratio of 0.3 and less. For velocity ratios of 0.5 and above, the jet lifted off the surface, allowing penetration of the mainstream fluid beneath it. Maximum values of the wall effectiveness occurred for a velocity ratio of 0.5.

![Figure 1.5. Effect of blowing rate on centerline film cooling effectiveness from Goldstein et al. (1968).](image-url)
1.2.3.2 Effects of density ratio

For the flowfield of jet issuing into the mainflow, a number of experiments were performed at density ratios near unity. Fewer studies have matched the large jet-to-mainstream density ratio typical of film-cooled turbine blades, where the density ratio is near two. These studies have shown that the density ratio is a significant film-cooling parameter; i.e., the film-cooling wall effectiveness improves as the density ratio increases, with either the mass-flux ratio or the velocity ratio held constant.

![Figure 1.6. Effect of coolant density on film effectiveness from Pedersen et al. (1972).](image)

Pedersen (1977) studied the effect of the coolant density in the same geometry of the experiments by Goldstein (1974). He used the density ratios of jet to mainstream flow between 0.75 and 4.17 and found the density ratio has a strong effect on the film cooling effectiveness for injection through holes. The results for lateral averaged effectiveness,
which were measured at a downstream location of 10 diameters, are shown in Fig. 1.6. Figure 1.6 shows the general results are very close to those of the investigation by Goldstein (1969). However, the maximum effectiveness increases as the coolant-to-mainstream density ratio increases. From their studies, the results gave the maximum effectiveness at a value of blowing ratio of about 0.4. This result may be interpreted as the jet remains attached to the wall at values of coolant-to-mainstream below 0.45 and penetrate the mainstream. Le Brocq et al. (1973), Launder and York (1974), Foster and Lampard (1980), Yoshida and Goldstein (1984), Pietrzik et al. (1989), and others studied the effect of density ratio on the film cooling effectiveness with various blowing ratio for a row of inclined jet at a number of injection angle. Le Brocq et al. (1973) injected Freon into an air mainstream to obtain a density ratio of 4.23. The flat plate was covered with a staggered array holes, inclined at 45 deg. The holes were spaced eight diameters apart in the lateral and streamwise directions. A mean velocity profile, taken two diameters behind a hole for a mass-flux ratio of 0.53, was compared with a unit-density profile having equal mass-flux ratio and a unit density ratio jet of equal momentum-flux ratio. These profiles suggested that the Freon jet remained attached to the wall and exhibited the same general shape as the unit density ratio jet of equal momentum-flux. The unit density ratio jet of equal mass-flux, which had 4.23 times the momentum-flux of the Freon jet, separated from the wall. For a small mass-flux ratio, \( M=0.2 \), the velocity profile for the Freon jet was nearly coincident with the velocity profile of a unit density ratio jet at the same mass-flux. Thus, this result is consistent with the finding that for small values of the mass-flux ratio, the wall effectiveness is primarily a function of the mass-flux ratio and is less dependent on the density ratio. Goldstein et al. (1974) reported that the use of a relatively dense secondary fluid, as might be encountered in many
applications, requires a significantly higher blowing rate to cause jet separation from the surface than when the densities of the freestream and secondary stream are the same. Launder and York (1974) used the same film-cooling geometry as Le Brocq (1973), but injected CO₂ into air to get a more realistic density ratio of 1.5. Velocity profiles, taken four diameters downstream of the first row of holes, for three different mass-flux ratios, 0.2, 0.54, and 0.65, were compared. From their studies, the jet remained attached to the plate with the mass-flux ratio of 0.2, while the two jets at higher mass-flux ratios had separated.

1.2.3.3 Effects of cooling hole diameter-to-length ratio

Early film cooling studies included long cooling hole length to diameter while using shorter cooling hole length to diameter ratios which are more representative of turbines in recent years. Goldstein et al. (1974) focused on hole length to diameter with $L/D = 5.2$, Pederson et al. (1977) with $L/D = 40$, and Sinha et al. (1991) with $L/D = 1.75$. In their study, density ratio was slightly larger or less than unity. Kadotani and Goldstein (1979)) and Yoshida and Goldstein (1984) studied jets issuing from pipes longer than $50D$ with an inclination of 35 deg, for velocity ratios of 0.35 and 0.50, respectively. Crabb et al (1981) studied the hydrodynamics of a normal jet of $L/D = 30$ in crossflow using hot-wire in the far field and LDV in the near field. Andreopoulos and Rodi (1985) investigated the turbulence field for a normal jet in crossflow with $L/D = 12$, $\rho_c/\rho_\infty = 1.0$, and $U_c/U_\infty = 0.5$. Lutum and Johnson (1999) investigated also it in the $1.75$ to $18$ range and presented little changes in film cooling effectiveness and concluded that coolant flow characteristics remained unchanged for $L/D > 7$.

Otherwise, Pietrzyk et al. ($L/D = 3.5$), Sinha et al. ($L/D = 3.5$), Schmidt et al. ($L/D = 4.0$), Bons et al. ($L/D = 3.5$) (1996), and Kohli and Bogard ($L/D = 2.8$ and 3.5) (1995)
documented short hole length for $L/D<4$. They commonly considered plenum as a high pressure vessel, which was located normal to the film cooling hole. With a very short $L/D$ of 1.75, Sinha et al. (1991) studied film cooling effectiveness downstream of holes with various density ratio, 35 deg streamwise injection, and low freestream turbulence intensity. Schmidt et al. (1996) investigated film cooling performance with a cooling hole length to diameter of 4.0, and noted the differences in adiabatic effectiveness that existed between round streamwise injection and compound angle injection with round and shaped holes. Their studies were performed at a density ratio of 1.6 and mass flux ratio of 0.5-2.5. Kohli and Bogard (1995) expanded cooling hole length to diameter to 2.8 and 3.5 to investigate 35 deg and 55 deg streamwise injection with a density ratio of 1.6. Similarly, a hole length to diameter of 3.5 was used by Bons et al. (1996) and Pietrzyk et al. (1989; 1990) for studies of and hole inclination angle of 35 deg in the streamwise direction. Differences between short and long hole injection have been numerically investigated as well. Walters and Leylek (1997; 2000), Berhe and Patankar (1996), and Ferguson et al. ($L/D = 1.75$ and 4.0) (1998) accomplished several numerical studies for short hole. Walters and Leylek (2000) reported the two relatively short hole lengths to diameter, 1.75 and 3.5, related with the flow characteristics at the hole exit and the interaction of the exiting jets with crossflow. They showed that the effects of the separation originated by the configuration of hole and plenum at the hole entrance have less time to attenuate as length to diameter decreases, and therefore exert more influence on the jet exit conditions. Leylek and Zerkle (1993) performed three dimensional computation and compared their results to the experiments of Pietrzyk et al. (1989; 1990) and Sinha et al. (1991). The operating configurations included film hole to diameter of 1.75 and 3.5, inclination angle of 35 deg, blowing ratio from 0.5 up to 2, and
density ratio of 2. They found that film cooling experiments with long length to diameter may be misleading for engine applications. In a numerical study, Berhe and Patankar (1996) also computed the influence of hole length to diameter of 2.8 and 4.9 and reported that the shorter hole results in higher laterally averaged effectiveness than the longer one, for $M = 0.5$ and 1.0, thus, the increase in near-field centerline effectiveness due to the longer hole is more than compensated by a better lateral spreading with the shorter hole. Burd et al. (1996) reported hydrodynamic measurements comparing 35 deg streamwise injections for two short and long injection hole length-to-diameter ratios of 2.3 and 7. They found hole length to diameter significantly influences the hole-exit velocity profiles and the manner by which the coolant and freestream flows interact.

1.2.3.4 Effects of freestream turbulence

Freestream turbulence level is an important parameter, and increasing turbulence intensity generally results in a decrease of centerline effectiveness at all downstream locations. The laterally averaged effectiveness values, however, increases with higher turbulence intensity at higher blowing ratio. Most early film cooling studies in the literature were carried out with freestream turbulence intensity in the range of 0.4 to 2 % from Goldstein et al. (1968). Kadotani and Goldstein (1979) used turbulence generating grids and found varying degrees of turbulence influence. They concluded the turbulence intensity is one of most important parameters, which are of greatest importance in changing the effectiveness. Launder and York (1974) found no influence at a freestream turbulence intensity of 4%. They also reported a drop in film cooling effectiveness due to increased freestream turbulence intensity in the presence of a favorable pressure gradient. Brown and Minty (1975) and Brown and Saluja (1979) studied film cooling from a single hole and a row
of holes exiting into accelerating and decelerating flows, and found losses in cooling
effectiveness for freestream turbulence intensity ranging from 1.7 to 8%. They reported
higher freestream turbulence intensity lowers the centerline effectiveness for locations within
15 diameter downstream exit hole for blowing rates less than 1.25. Jumper et al. (1991)
studied the influence of freestream turbulence in comparison between 0.5% and 14-17% on
film cooling effectiveness with an inclination angle of 30 deg, and found a faster decay in
film cooling effectiveness when higher freestream turbulence intensity was introduced.
Bons et al. (1996) investigated variations of film cooling effectiveness with several velocity
ratios and L/D=3.5 when freestream turbulence intensities of 0.9%, 6.5%, 11.5%, and 17.5%
were provided. They reported high freestream turbulence intensity drops film cooling
effectiveness by up to 70% in the region directly downstream of the injection hole due to
enhanced mixing. At the same time, high freestream turbulence produces a 50 to 100%
increase in film cooling effectiveness in the near hole regions between holes due to
accelerated spanwise diffusion of the cooling fluid, which produces an earlier merge of the
coolant jets from adjacent holes. Schmidt et al. (1996), however, found film cooling
effectiveness depends on the coolant to freestream momentum flux ratio when freestream
turbulence intensity increases. Freestream turbulence intensity reduced film cooling
effectiveness downstream from the hole for film cooling with low momentum flux ratios
when high freestream turbulence intensity increased, but it increased values of film cooling
effectiveness when momentum flux ratios were large. MacMullin et al. (1989) measured
freestream turbulence intensity in the range of 7 to 18%. Gogineni et al. (1996) measured
freestream turbulence intensity values of 1 to 17% and used two color particle image
velocimetry to investigate velocity and vorticity field with 35 deg inclined, single row
injection. Wang et al. (1996) reported the flowfield just downstream of injection for two freestream turbulence intensity levels, 0.5% and 12%, by means of using three-wire anemometry. They computed the eddy viscosity values from the data in the lateral direction and direction normal to wall, and the ratio of the two. This ratio explains the anisotropy of turbulent momentum transport.

1.2.3.5 Effects of flow on film-cooling effectiveness

One of the most important physical phenomena associated with the jet in crossflow is the formation of vortical structures strongly affecting jet behavior. Kamotani and Greber (1972), and Fearn and Weston (1974), found the configuration of a counter-rotating vortex pair (CRV) flow structure dominating the cross-section of the jet. Moussa (1977), Andreopoulos and Rodi (1984), Fric (1989), Kelso et al. (1995; 1996), and Camussi et al. (2002) provided more detailed experimental results in the nearfield, and they suggested that the CRV is formed by the vortex sheet issuing from the hole.

Andreopoulos and Rodi (1984) explained these flow characteristics in the fields as follows:

The most obvious feature of the jet in a crossflow is the mutual deflection of jet and crossflow. The jet is bent over by the cross-stream, while the latter is deflected as if it were blocked by a rigid obstacle, the difference being that the jet interacts with the deflected flow and entrains fluid from it. In the case of the small velocity ratio ($R = 0.5$), the flow behaves as if a partial, inclined ‘cover’ were put over the front part of the exit hole, causing the jet streamlines to start bending while still in the discharge tube and the jet to bend over completely right above the exit and also to lift up the oncoming flow over the bent-over jet. In the case of the higher velocity ratio ($R = 2$) the jet is only weakly affected near the exit and penetrates in the crossstream before it is bent over. In both cases, wake regions with very complex three-dimensional flow patterns flow patterns form in the lee of the jet. (Andreopoulos and Rodi, 1984, pp. 94)

Acharya et al. (1991) summarized film cooling flow as follows:

The majority of the studies reported in the 1970’s and 1980’s were motivated by
VSTOL-related applications and several flow visualization and experimental studies were conducted to understand the characteristics of the jet-crossflow interactions. Figure 1.7 shows a cartoon from Fric and Roshko illustrating the various structures generated when a jet is injected normally into an unbounded crossflow. Unlike a rigid cylinder in crossflow, the boundaries of the jet are compliant and entraining, causing the jet to bend over. Periodic shedding of wake vortices has been observed particularly when the jet blowing rate \( U_c/U_\infty \) is greater than 1.

The jet structure itself is dominated by a pair of kidney shaped counter-rotating vortex pair (CVP), and both the shearing between the jet and the crossflow and the vorticity issuing from the jet exit has been attributed to be the source of the CVP. There are however different mechanisms proposed on the reorientation of the jet-hole vorticity into the CVP structure. Upstream of the jet, due to the adverse pressure gradients, a horse-shoe vortex system is formed, which wraps around the base of the jet traveling downstream with vorticity counter to the CVP. (Acharya et al., 2001, pp. 94)

Figure 1.7. Schematic of the flow field of a jet in crossflow by Fric and Roshko. (Fric and Roshko, 1994, pp 2)
From the experiments of Kelso et al. (1996) in both water and air, it is suggested that periodic vortex ring rollup from the hole occurs for the jet in crossflow, and jet superposed on this process is a re-orientation of this shear layer vorticity imposed by the crossflow, which leads to a folding of the cylindrical vortex sheet. The superposition of these two mechanisms results in the interpretation of the evolution of the jet shear layer vortex rings, where there is a tilting of the upstream portion of the ring oriented with the mean curvature of the jet, and a tilting and folding of the downstream portion of the ring aligned with the direction of the jet.

The re-orientation of the shear layer is seen to lead to this tilting. Moussa et al. (1977), Andreopoulos and Rodi (1984), and Kelso et al. (1996) thought these tilting and folding contribute to the circulation of the CRV. The experiments of Kelso et al. (1996) suggested that the shear layer of the jet folds and rolls up very near to the hole exit, leading to or contributing to the formation of the CRV. Kuzo (1995) and Smith and Mungal (1998) suggested that the CRV can instantaneously be either symmetric or asymmetric in shape under specific circumstances and that end views of the jet in the farfield can reveal axisymmetric as well as sinusoidal motion of the CRV. The jet in crossflow is influenced by the presence of other vortex systems in the flow field. Krothapalli (1990), Kelso and Smits (1995) mentioned that horseshoe vortices form upstream of the jet and it influences and are coupled with periodic vortices which form in the wake of the jet. Moussa et al. (1977), Fric and Roshko (1994), and Smith and Mungal (1998) investigated the structure of wake vortices which were wall vortices and upright vortices forming in the wake of the jet. Fric and Roshko (1994) suggested that the vorticity in the wake region originated from the injection wall boundary layer, where the boundary layer fluid wraps around the jet,
separating on its lee side and acting to form the upright vortices. Kelso & Lim (1996) and Haven and Kurosaka (1997) showed the important role played by vortices in the evolution of film-cooling jets.

Lastly, the counter-rotating vortices (CRVs) was found to lift the jet off the surface that it is intended to entrain hot gases underneath it while an anti-kidney pair appears to have a sense of rotation opposite to that of the CRVs, and so can counteract the undesirable tendencies of the CRVs. Thus, it is of interest to develop strategies to control the formation and strength of these vortices in a way that leads to more effective film cooling. There are many ways to alter the structure of these vortices. Since the vorticity in the cooling jet originate from the flow in the film-cooling hole, the boundary layer upstream of the film-cooling hole, and the boundary-layer/cooling jet interactions, most investigators have focused on the geometry of the film-cooling hole. Shaped-diffusion hole is one approach. Haven and Kurosaka (1997) and Hyams et al. (2000) investigated the effects of shape holes on the vorticity dynamics of mainflow/film-cooling jet interactions. Another approach to alter the vortical structures is via vortex generators. Haven and Kurosaka (1997) investigated the effects of placing vanes inside film-cooling holes that produce vortices in the same sense as the anti-kidney vortices. Zaman and Foss (2005) and Zaman (1998) investigated the effects of tabs placed at the film-cooling-hole exit. From their studies, it was shown vortex generators are quite effective in reducing jet penetration. However, both studies only investigated jets that lifted-off the surface once exiting the film-cooling holes. Shih, et al. (2001) proposed placing a strut or obstruction within each film-cooling hole that do not necessarily generate appreciable vortices but can cause vortices inside film-cooling holes to be stretched and tilted in a way that would re-distribute the magnitude of the vortices inside
the CRVs and the anti-kidney pair. Bunker (2002) proposed creating a trench about each film-cooling hole exit to modify the boundary-layer/cooling jet interactions.

1.2.4 Injection from shaped holes

The shaped hole provides generally much better lateral coverage of the test surface and also attenuates jet lift-off. Effectiveness data confirmed the observations that the shaped film hole afforded better lateral coverage and better centerline effectiveness. In addition, the shaped film holes provided significant improvements in cooling performance at any blowing ratio and density ratio. Goldstein et al. (1994) hypothesized that the increased exit area of the shaped holes was responsible for slowing the coolant flow such that less penetration through the oncoming boundary layer and into the mainstream occurred. Ekkad et al. (1997) and Ligrani et al. (1994) both found that laterally averaged effectiveness increases for compound-angle injection relative to streamwise injection. Thole et al. (1998) studied flow-field measurements for film cooling holes that expand laterally and/or forward near the exit of the hole. The purpose of the expansion of the exit of the hole was to increase the lateral spread of the coolant film downstream of the holes and to minimize the penetration of the coolant flow into the mainstream. While this shape improved the uniformity of the film over the surface compared with cylindrical holes, the hole expansion causes separation in the hole and inefficient diffusion. Gritsch et al. (2000) have presented heat transfer, and adiabatic effectiveness measurements, which was conducted over a range of blowing ratios M=0.25 to 75 at an external crossflow Mach number of 0.6 and a coolant-to-mainflow density ratio of 1.85. As compared to the cylindrical hole, expanded holes showed lower heat transfer coefficients downstream of the injection location, particularly at high blowing ratios. The laidback fanshaped hole provided a better lateral spreading of the
injected coolant than the fanshaped hole which leaded to lower laterally averaged heat transfer coefficients. Sargison et al. (2002) investigated the converging slot-hole or console to improve the heat transfer and aerodynamic loss performance of turbine vane and rotor blade cooling systems. Christian Saumweber et al. (2003) conducted experiments to investigate the effect of elevated free-stream turbulence on film cooling performance of shaped holes with expanded exits, a fan-shaped (expanded in lateral direction), and a laidback fan-shaped hole (expanded in lateral and streamwise direction). Their results indicated that shaped and cylindrical holes exhibited very different reactions to elevated free-stream turbulence levels. For cylindrical holes film cooling effectiveness is reduced with increased turbulence level at low blowing ratios whereas a small gain in effectiveness can be observed at high blowing ratios. For shaped holes, increased turbulence intensity is detrimental even for the largest blowing ratio (M = 2.5). In comparison to the impact of turbulence intensity the effect of varying the integral length scale is found to be of minor importance. Papell (1984) experimentally studied a hole with a cusp (similar in appearance to a kidney bean), which induced strong longitudinal vortex structures within the film hole. He explained that creating these vortical structures enabled the crossflow to use its energy to force the jet down to the surface, rather than itself creating the counter-rotating vortices. Further, the postulated that the placement of the cusp on the leeward side (TE) forced the film hole secondary flow to rotate in a direction opposite of that traditionally observed in cylindrical film holes. Papell (1984) offered evidence in the form of adiabatic effectiveness and film coverage data to support his findings that his cusp-shaped holes provided better film cooling performance.
1.2.5 Computational Studies on Film Cooling

A number of rigorous and state-of-the-art CFD studies on film cooling based on steady Reynolds-averaged Navier-Stokes (RANS) calculation procedures (Garg and Gaugler (1995); Berhe and Patankar (1996); Walters and Leylek (1997); Lakehal et al. (1998), Acharya et al. (2001)) have been carried out. Since most film-cooling flows are turbulent, CFD analysis of film cooling can be divided into four categories: (1) those based on direct numerical simulation (DNS) of turbulence, (2) those based on large-eddy simulations (LES) of turbulence, (3) those based on Reynolds or Favre averaged Navier-Stokes (RANS) equations, and (4) those based on a combination of the above such as LES away from walls and RANS next to walls, referred to as detached eddy simulation (DES). Of these methods, DNS, LES, and DES have the highest potential to provide the best results in terms of accuracy (Liu and Pletcher, 2003). But, they are expensive computationally for design purposes – at least with existing computing resources. With today’s computing capabilities, RANS based on eddy-diffusivity and Reynolds stress models are the ones used.

Integral models were first applied to predict the behavior of jets in cross flow. The integral equations were derived by taking into consideration of a balance of forces acting over an elementary control volume of the jet. In this calculation, a set of ordinary differential equations were obtained, and they were solved analytically or numerically. The effect of pressure drag, entrainment of crossflow fluid, and spreading rates were simulated by way of empirical relations. Abramovich (1963), Crowe and Riesebieter (1967), and Chien and Schetz (1975) developed integral models to predict the behaviors of the jet in a crossflow in this way. Their model predictions demonstrated fairly good agreement with experimental data, but these models had a severe drawback that is the lack of generality.
In the 1980's, finite difference methods moved to integral methods as the analysis tools with the rapid advances in computational resources and with the development of better and faster algorithms. A number of numerical studies of the jet in cross-flow involved the solution of the Reynolds Averaged Navier Stokes (RANS) equations and the energy equation with closure for turbulent quantities obtained through a turbulence model. The simplest form of the turbulence model employed is based on the Boussinesq eddy viscosity approximation where the turbulent stresses are represented as:

\[-\rho u_i u_j = -\frac{2}{3} \rho k \delta_{ij} + 2 \mu_\epsilon S_{ij}\]

where \(k\) is the turbulent kinetic energy, \(S_{ij}\) is the mean rate of strain, and \(\mu_\epsilon\) is the eddy diffusivity. In algebraic models, \(\mu_\epsilon\) is expressed simply by an algebraic expression. The Baldwin-Lomax model (1978) is the most commonly used algebraic model. The eddy diffusivity is expressed in terms of mixing length in both the inner and outer layers. Fougeres and Heider (1994) solved the unsteady three-dimensional Navier-Stokes equations using mixing length and obtained predictions for a film cooled plate and a nozzle guide vane with coarse grid. Garg and Gaugler (1995) used the Baldwin-Lomax model (1978) to obtain predictions of flow and heat transfer over a film-cooled C3X vane using data provided by Hylton et al (1988) and a VKI rotor measured by Camci and Arts (1985). The computational model consisted of a series of holes in the spanwise directions, and the calculations performed by nearly 0.5 million grid points were reasonably well resolved. Although the Baldwin-Lomax (1978) model had been used very widely, and with reasonable success, the applicability of algebraic models was also quite limited for film cooling flows with strong pressure gradients and separation in the immediate vicinity of injection.

The most RANS simulations for jet in a cross flow have employed various k-\(\epsilon\) models
originally proposed by Launder and Spalding (1974) to obtain the distribution of eddy viscosity. Three dimensional computation was studied by Patankar and Spalding (1972) and Patankar et al. (1973). Patankar et al. early used this model to perform a detailed study of the jet in a cross flow, and they obtained reasonable agreement with experimental data for the jet trajectory and streamwise velocity in spite of using a relatively coarse (15×15×10) grid. Demuren (1993) published a detailed analysis on modeling turbulent jets in cross flow, and systematically reviewed the various models reported till 1985.

Tafti and Yavuzkurt (1990) computed film cooling with one row of injection into a turbulent boundary layer including a two-dimensional, parabolic model with low-Reynolds-number, k-ε turbulence model. A systematic study of film cooling by Demuren et al. (1993) revealed that the very complex flow field established behind the jet was not properly resolved and the turbulent mixing process was crudely simulated with the eddy viscosity model. Demuren (1993) also carried out computations using a multigrid method and a second-moment closure model to approximate the Reynolds stresses. Although a fairly good prediction of mean flow trends was reported, there was considerable uncertainty regarding the accuracy of jet penetration height. Leylek and Zerkle (1993) performed three-dimensional Navier-Stokes computation and compared their results to the experiments of Pietrzyk et al. (1989; 1990) and Sinha et al. (1991) and found counter-rotating vortices and local jetting effects were included in the film cooling exit flow. Multigrid calculations by Claus and Vanka (1990) failed to predict the horseshoe vortex even with a highly refined grid. This was attributed partly to the inability of the k-ε model to resolve the complex turbulence field. Findlay et al. (1996) included the plenum in the computational domain for streamwise inclined jets. The computations underpredicted the streamwise injection of fluid from the
jet and the flow field was not in good agreement with experimental results for most of the domain. Ajersch et al. (1997) conducted an extensive experimental investigation and a companion numerical simulation using a low-Re $k$-$\varepsilon$ model along with a nonisotropic extension to the effective viscosity for near-wall turbulence. The streamwise velocity in the jet wake was overpredicted and the recirculation region behind the jet was found to be smaller and closer to the surface than that observed in the measurements. Noticeable overprediction of shear stresses was observed and the simulation could not capture the local minimum in kinetic energy, which was measured in the wake region of the jet. For a square jet injected vertically into a crossflow, comparisons of predictions and measurements (Hoda and Acharya, 2000) revealed that RANS procedures with an array of turbulence models (from two-equation models to Reynolds stress models) significantly underpredict the lateral shear stress (responsible for the lateral mixing and spreading).

References (Kercher, 2003; Kercher, 2005) list all experimental and CFD studies of film cooling in the open literature up to 2004. Though a number of rigorous and state-of-the-art CFD studies on film cooling based on RANS have been carried out, Walter & Leylek (1997) noted that a lot of the earlier work employed simplified mathematical models, used highly dissipative numerical schemes (e.g., first-order upwind), used wall functions (i.e., did not integrate to the wall), had poor quality grids (e.g., highly skewed and warped cell shapes especially in the region about the film-cooling holes), and/or could not afford to generate grid-independent solutions because of limited computing resources. In addition, there are concerns on lack of convergence to steady state and uncertainties from variability in the different CFD codes. Finally, there is considerable uncertainty on the adequacy of the eddy-diffusivity and Reynolds stress models used to capture the most important flow physics
governing surface heat transfer.

Seen from the literature reviewed, a great deal of experimental and computational studies have been done to increase the surface adiabatic effectiveness. It can be summarized by the key factors of coverage of cooling flow on the surface exposed to the hot gases as follows: roughness, 2-D slot, CRVs, density ratio, turbulence level, and shaped hole.

Roughness is one of the main components in increasing heat transfer. 2-D slot is very useful in the coverage of the cooling jet. The adiabatic effectiveness increases up to 0.5 of a blowing ratio when a density ratio is 2.0, but it decreases when a blowing ratio is above 0.5. The main factor is viewed as lift off, because the flow injected from the hole does not bend due to strong momentum and it detaches from the surface. Moreover, entraining hot gases to the surface, CRVs raise the effect of lift off and make film cooling worse. High density ratio, even higher than $M = 0.5$, increases effectiveness. The cause of increase of effectiveness by density ratio is also found in lift off because coolant flow attaches to the surface at the high density ratio. A high level of turbulence decreases centerline effectiveness. However, it increases the laterally averaged effectiveness, because the mainflows with higher level turbulence increase the diffusion when they are mixed with coolant and accordingly causes the coolant to spread laterally. The challenge of shaped hole is the increase of the diffusion with modified exit hole (i.e., expanded exit hole).

It is found that lift off can be attenuated by means of reducing momentum of cooling jet. The main cause of decreasing of wall effectiveness is lift off of the cooling jet. CRVs accelerate and result in the increase of effects of lift off. As a result, the coverage of the coolant jet is limited to the small region attached by the coolant on the surface.

Endeavors to increase this coverage have focused on the configurations of the hole
which is thought to be directly connected with cooling jet. However, lift off and entrainment of hot gases still remains as a problem, because they always appear in the film cooling using rows of jet in a crossflow. In this study, we turn our angle of view to the modification of the surface. There has been little effort made to modify the surface to reduce these problems. We propose four paradigms of the film cooling in this study.

1.3 Dissertation Organization

Organization of this dissertation is follows: The film-cooling problems studied are outlined in Chapter 2. The details of the problem formulation and numerical methods of solution are given in Chapter 3. In chapter 4, the validation and grid sensitivity studies are described. In chapter 5, the effects of TBC blockage and roughness on film cooling are presented. Chapters 6 to 9 provide details on the performance of the four new design paradigms developed. Finally, Chapter 10 gives a summary of the key findings of this dissertation.
CHAPTER 2. PROBLEM DESCRIPTION

In this study, the following eight problems were studied:

1. 2-D baseline. Film cooling of a flat plate in which the coolant is injected from one row of inclined circular holes.
2. 2-D baseline with TBC and with and without roughness.
3. 3-D baseline.
4. 3-D baseline with TBC.
5. 3-D baseline with holes replaced by shaped holes.
6. 3-D baseline with blockers.
7. 3-D baseline with ramp.
8. 3-D baseline with ramp and blockers.

In this chapter, only problems 3, 7, and 8 are described in detail. The details of the other problems are given in Chapters 5, 6, and 7.

2.1 Film Cooling of a Flat Plate

This problem is taken from the experimental study of Kohli & Bogard (1995). A schematic diagram of this problem is shown in Fig. 2.1. For this problem, the cooling jets emerge from a plenum through one row of circular holes. Each hole has a diameter $D$ of 12.7 mm, a length of $2.8D$, and an inclination of $35^\circ$ relative to a plane. The operating conditions are as follows. The fluid for the main flow (hot gas) and coolant is air. The main flow has a temperature of $T_{\infty} = 298$ K upstream of the film-cooling holes, and the coolant has a temperature of $T_c = 188$ K, while in the plenum. This gives a density ratio of $DR = 1.6$. The average velocity in all film-cooling holes is $U_c = 6.25$ m/s. The freestream
velocity of the main flow is $U_{\infty} = 20$ m/s. This gives a mass flux or blowing ratio of $M = 0.5$. The velocity profile is uniform at the inflow boundary, and the flow in the boundary-layer is assumed to be turbulent from the leading edge. All walls are adiabatic, and the back pressure at the outflow boundary is maintained at 1 atm.

For this problem, the computational domain is taken to be the region bounded by the dashed lines in Fig. 2.1 (b) and the solid lines in Fig 2.1 (c). As can be seen, periodicity is assumed in the spanwise direction so that only one film-cooling hole needs to be examined. In addition, the “upper channel wall” (i.e., the wall without film-cooling holes) was moved closer to the wall with the film-cooling holes. This was done to reduce the size of the computational domain and hence computational cost. The errors incurred by this are minimized by making the “upper channel wall” sufficiently far away and by making it inviscid (i.e., the velocity there can slip despite the viscous nature of the flow) so that boundary layers will not form there.

2.2 Film Cooling with Upstream and Downstream Flow Modifiers

Schematic diagrams of the film-cooling problem studied with the blockers and the ramp are shown in Figs. 2.1 and 2.2. For this problem, the cooling jets emerge from a plenum through one row of circular holes. Like the 3-D baseline problem, each hole has a diameter $D$ of 12.7 mm, a length of 3.5$D$ mm, and an inclination of 35° to the flat plate. The spacing between the centers of the film-cooling holes in the spanwise direction is 3$D$. Since the film cooling is for a flat plate in which the cooling jet emerges from a row of inclined holes, the blockers are taken to be pairs of parallel ribs or fence-like protrusions from the flat plate with rectangular cross sections as shown in Figs. 2.1 and 2.2. These “rectangular prism” blockers are located 1$D$ downstream of the film-cooling hole. Each
blocker has height of 0.6D and thickness of D/5, and it is separated by a distance of D. The upstream ramp has a length of 2D and an angle of 14° to the flat plate, and is located 0.5D upstream of the film-cooling hole. Other dimensions of the geometry of the ramp and blockers are given in Figs. 2.1 and 2.2. The operating conditions are the same as the 3-D baseline problem.

2.3 Film Cooling by Using a New Shaped Hole

Schematic diagrams of the momentum-preserving shaped holes studied are shown in Figs. 2.3 and 2.4. The shaped hole has a W-shape with a ridge in the middle and a pair of symmetric valleys. As shown Figs. 2.3 and 2.4, each valley has a depth of 0.5D, radius of R₁ = 0.3 mm and R₂ = 0.3 D in curvature, and H is either 0 or 0.4D (for the cross section at 20.4 mm downstream of the hole). The width of the W-shaped hole is expanded (1.1D) symmetrically in the spanwise direction (A-A) from the center of the hole exit to 20.4 mm downstream of the hole exit. From Fig. 2.3(c), W₁, W₂, and W₃ are 10.4 mm, 10.0 mm, and 2D along the centerline (B-B), respectively. L₁ and L₂ are 7.2 mm and 5.08 mm, respectively. Three types of configurations are examined. A short shaped hole has a height of H = 0.4D between the ridge and the surface of the flat plate, and its shape disappears at 25.8 mm downstream of the hole exit. Two long shaped holes have two heights of H = 0 and H = 0.4, respectively. They have constant cross section extended to the end of the surface. Other dimensions that describe the geometry are given in Figs. 2.2 and 2.3.

In order to compare to the effects of film cooling with the W-shaped hole, the operating conditions are the same as flat plate problem.
Figure 2.1. Schematic of film cooling of a flat plate from a row of inclined circular holes. (Kohli and Borgard, 1995) (not drawn to scale). (a) 3-D view. (b) Top view (X-Z). (c) Side view (X-Y).
Figure 2.2. Schematics of the ramp and blockers of a film-cooling hole. (a) 3D diagram. (b) the configurations of blockers. (c) the configurations of ramp
Figure 2.3. Schematic of shaped hole placed on the film-cooling holes and under the flat plate for the 3-D problem. (a) 3-D computational domain studied. (b) 3-D view of a short shaped hole. (c) dimensions of cross sections of A-A and B-B of a short shaped hole.
Figure 2.4. Schematic of shaped hole placed on the film-cooling holes and under the flat plate for the 3-D problem. (a) Short shaped hole. (b) Long shaped hole.
CHAPTER 3. FORMULATION AND NUMERICAL METHOD OF SOLUTION

The eight film-cooling problems summarized in Chapter 2 were modeled by the ensemble-averaged continuity, momentum (full compressible Navier-Stokes), and energy equations for a thermally and calorically perfect gas. The effects of turbulence were modeled by using the two-equation realizable k-ε model. In all cases, the integration of all equations is to the wall (i.e., wall functions are not used).

Solutions to the aforementioned governing equations were obtained by using Version 6.1.18 of the Fluent-UNS code. The following algorithms in Fluent were invoked. Since only steady-state solutions were of interest, the SIMPLE algorithm was used. The fluxes at the cell faces representing advection were interpolated by using second-order upwind differences. The fluxes at the cell faces representing diffusion were interpolated by using second-order central differences. For all computations, iterations were continued until all residuals for all equations plateau to ensure convergence to steady-state has been reached. At convergence, the scaled residuals were always less than $10^{-6}$ for the continuity equation, less than $10^{-6}$ for the three components of the velocity, less than $10^{-8}$ for the energy equation and, and less than $10^{-5}$ for the turbulence quantities.

3.1 Conservation/Balance Equations

The equations governing non-reacting compressible flows are the continuity, momentum, and energy equations, given by

Conservation of Mass (Continuity)
\[ \frac{\partial}{\partial x_j} \left[ \rho u_j \right] = 0 \quad (3.1) \]

Balance of Linear Momentum

\[ \frac{\partial}{\partial x_j} \left[ \rho u_i u_j + p \delta_{ij} - \tau_{ij} \right] = 0, \quad i = 1, 2, 3 \quad (3.2) \]

Conservation of Energy

\[ \frac{\partial}{\partial x_j} \left[ \rho u_j \rho e_0 + u_j p + q_j - u_i \tau_{ij} \right] = 0 \quad (3.3) \]

For a Newtonian fluid and assuming Stokes hypothesis, the viscous stress is given by:

\[ \tau_{ij} = 2 \mu S_{ij}^* \quad (3.4) \]

where the strain rate is defined by

\[ S_{ij}^* \equiv \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) \quad (3.5) \]

The heat-flux, \( q_j \), is given by Fourier’s law

\[ q_j = -\lambda \frac{\partial T}{\partial x_j} = -C_p \frac{\mu}{Pr} \frac{\partial T}{\partial x_j} \quad (3.6) \]

where the Prandtl number, \( Pr \), is defined by

\[ Pr \equiv \frac{C_p \mu}{\lambda} \quad (3.7) \]

To close these equations, it is also necessary to specify an equation of state. Assuming a calorically perfect gas, the following relations are valid:

\[ \gamma \equiv \frac{C_p}{C_v}, \quad p = \rho RT, \quad e = C_v T, \quad C_p - C_v = R \quad (3.8) \]

where \( \gamma, C_p, C_v \), and \( R \) can be treated as constant at low-speed non-reacting flows.

The total energy \( e_0 \) is defined by

\[ e_0 = e + \frac{u_i u_i}{2} \quad (3.9) \]
3.2 Ensemble-Averaging and Turbulence Modeling

Since the flows of interest are turbulent and it is not feasible to perform direct or large-eddy simulation with our existing computing capabilities, the Reynolds-Averaged Navier-Stokes (RANS) approach was adopted. With RANS, the variables in Eqs. (3.1) to (3.9) are first decomposed into the mean and the fluctuating components. For example, velocity is decomposed as follows:

\[ u_i = \bar{u}_i + u'_i \]  \hspace{1cm} (3.10)

where \( \bar{u}_i \) and \( u'_i \) are the mean and fluctuating components, respectively \((i=1,2,3)\).

Similarly, pressure and other scalar quantities are decomposed as

\[ \phi_i = \bar{\phi}_i + \phi'_i \]  \hspace{1cm} (3.11)

where \( \phi \) denotes pressure, energy, and other scalar quantities. After substituting Eqs. (3.10) and (3.11) into Eqs. (3.1) to (3.9), Eqs. (3.1 to (3.3) are ensemble averaged. Since only the steady-state solutions are of interest, we could also have done time averaging instead of ensemble averaging. By dropping the time derivatives and the overbar on the mean quantities, the ensemble-averaged equations become

\[ \frac{\partial}{\partial x_i} (\rho u_i) = 0 \]  \hspace{1cm} (3.12)

\[ \frac{\partial}{\partial x_i} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_i}{\partial x_i} \right) \right] + \frac{\partial}{\partial x_i} (\rho u_i u_j) \]  \hspace{1cm} (3.13)

\[ \frac{\partial}{\partial x_i} (u_i (\rho E + p)) = \frac{\partial}{\partial x_i} \left( k_{\text{eff}} \frac{\partial T}{\partial x_i} - \sum_j h_{ij} J_{ij} + u_i (\tau_{ij})_{\text{eff}} \right) + S_i \]  \hspace{1cm} (3.14)
Equations (3.12) to (3.14) are often referred to as the Reynolds-averaged Navier-Stokes equations. These equations have additional terms that represent the effects of turbulence, known as Reynolds stresses, $\overline{u_iu_j}$, which must be modeled.

**Modeling of the Reynolds Stresses**

As noted, ensemble-averaging produces the Reynolds stresses terms that must be modeled. Many approaches have been developed for modeling the Reynolds stresses, and they are classified into the following categories (Shih & Sultanian (2001)): Differential Reynolds stress models (DSMs), algebraic Reynolds stress models (ASMs), nonlinear constitutive relations (NCRs), and eddy-viscosity models (EVMs).

Of the RANS models, DSMs represent the highest level of closure. The major advantage of DSMs is that they are founded on a more rigorous foundation, and naturally account for more physics of turbulent flows such as streamline curvature, rotation, and stress anisotropy, which are prevalent in internal and film cooling flows. DSMs also can account for the effects associated with the time-lagged response of turbulence to changes in the mean flow. With DSMs, one needs to solve six transport equations for the six Reynolds stresses plus one equation for the length scale (typically, the dissipation rate of turbulent kinetic energy $\varepsilon$ is used). Depending on how velocity-temperature fluctuations are modeled, three additional transport equations may be needed. In these transport equations, modeling is needed for terms interpreted as molecular diffusion, dissipation of Reynolds stresses by the smaller scales, pressure-strain redistribution of Reynolds stresses, and pressure-temperature redistribution of thermal energy as well as the diffusion, production, and destruction of $\varepsilon$. Despite their sophistication, DSM is limited by the data from which they are calibrated as well as by assumptions that are often invoked such as isotropic dissipation, isotropic
diffusion, and one-point correlation in modeling pressure-strain correlations. Of particular concern is the modeling in the near-wall region. This is because DSMs are largely developed and calibrated by data from homogeneous flows, but wall-bounded flows can be highly nonhomogeneous. Also, DSMs may not be able to fully account for near-wall effects due to pressure reflection and eddy flattening and squeezing.

To reduce the number of transport equations that must be solved in DSMs, ASMs neglect the convection and diffusion terms in the Reynolds stress transport equations. This converts all Reynolds stress transport equations from partial differential equations (PDEs) to algebraic equations. The convection and diffusion terms can be dropped if turbulent convection and diffusion are small or convection is approximately equal to diffusion so that they cancel. Since these assumptions are rarely valid for engineering flows, Rodi proposed an alternative derivation of ASMs. His approach is based on the following two assumptions. First, convection minus diffusion of Reynolds stresses is proportional to convection minus diffusion of turbulent kinetic energy. Second, the Reynolds stress tensor normalized by the turbulent kinetic energy is constant along a pathline. With this closure, some effects of convection and diffusion are accounted for, and only two PDEs are needed to compute the turbulence quantities, either $k$ and $\varepsilon$ or $k$ and some other parameter for the length scale such as $\omega = \varepsilon/k$.

Instead of deriving a transport equation for each Reynolds stress, NCRs represent Reynolds stresses by a finite series expansion. First, the most general expansion for a second-order closure is derived. In this expansion, the leading two term represent an eddy-diffusivity model. The remaining terms are functions of the mean strain-rate and rotation
tensors. Afterwards, mathematical and physical constraints along with experimental data are used to determine the closure constants to ensure proper behavior in limiting cases.

EVMs tackle the Reynolds-stress closure problem by assuming that the Reynolds stresses are aligned with the mean rate of strain (i.e., the molecular analogy) and by introducing the concept of eddy viscosity as a proportionality factor. Analogous to molecular viscosity from kinetic theory of gases, the eddy viscosity is a function of a length and a velocity scale. These scales can be represented by algebraic equations or by one or more PDEs. In algebraic models such as those due to Prandtl, Cebeci-Smith, and Baldwin-Lomax, the production of turbulent kinetic energy, \( k \), is assumed to be everywhere equal to its rate of dissipation, \( \varepsilon \). With this local equilibrium assumption, the eddy viscosity becomes a function of the local mean flow field. Differential-equation models do not make this assumption, and as a result, require one or more PDEs to compute the evolution of the scales. The velocity scale is invariably computed from the PDE for \( k \) because that equation requires minimal modeling (e.g., the pressure-strain term vanishes). The PDE for the length scale is more difficult to derive. The first successful one was based on \( \varepsilon \), resulting in the \( k-\varepsilon \) model. Another popular variation, the \( k-\omega \) model, uses a PDE for \( \omega \) (\( \omega = \varepsilon/k \)) because \( \omega \) behaves better than \( \varepsilon \) in the near-wall region. But, the \( k-\omega \) model gives solutions that depend on the freestream \( k \). Overcoming this deficiency produced the shear stress transport (SST) model, which blends \( k-\omega \) in the near-wall region with \( k-\varepsilon \) further away from the wall.

The Realizable \( k-\varepsilon \) Model

In this study, we employ a two-equation eddy-diffusivity model known as the realizable \( k-\varepsilon \) model. In this model, as with all eddy-diffusivity models, the Reynolds stresses are modeled as
where $\mu_t$ is the turbulent viscosity, and is modeled as

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

where $C_{\mu}$ is typically modeled as a constant far away from walls ($C_{\mu} = 0.09$), but is variable in the near-wall region (e.g., in the viscous sublayer and the buffer region).

The realizable $k$-$\epsilon$ model (Shih et al., 1995) differs from the standard $k$-$\epsilon$ model in two important ways: (1) The realizable $k$-$\epsilon$ model contains a new formulation for the turbulent viscosity. (2) A new transport equation for the dissipation rate, $\epsilon$, has been derived from an exact equation for the transport of the mean-square vorticity fluctuation. The realizable $k$-$\epsilon$ model has shown to more accurately predict the spreading rate of both planar and round jets.

Also, it provides superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients, separation, and recirculation. The term "realizable" means that the model satisfies certain mathematical constraints on the normal stresses, consistent with the physics of turbulent flows. To understand this, consider Eq. (3.15a) for the normal Reynolds stress in an incompressible strained mean flow:

$$\overline{u^2} = \frac{2}{3} k - 2\nu_t \frac{\partial U}{\partial x}$$

Using Eq. (3.15b) for $\nu_t = \mu_t / \rho$, one obtains the result that the normal stress, $\overline{u^2}$, which by definition is a positive quantity, becomes negative (i.e., "non-realizable") when the strain is large enough to satisfy

$$\frac{k}{\epsilon} \frac{\partial U}{\partial x} > \frac{1}{3C_{\mu}}$$

This condition is not satisfied in the near-wall region, where the strain is typically large enough to keep $\overline{u^2}$ positive. However, in certain regions such as the viscous sublayer and the buffer region, the strain can be large enough to violate this condition, leading to non-realizable behavior. The model is designed to ensure that the normal stresses remain positive and consistent with the physics of turbulent flows.
The most straightforward way to ensure the realizability (positivity of normal stresses and Schwarz inequality for shear stresses) is to make $C_{\mu}$ variable by sensitizing it to the mean flow (mean deformation) and the turbulence $(k, \varepsilon)$. The notion of variable $C_{\mu}$ is suggested by many modelers including Reynolds (1987), and is well substantiated by experimental evidence. For example, $C_{\mu}$ is found to be around 0.09 in the inertial sublayer of equilibrium boundary layers, and 0.05 in a strong homogeneous shear flow.

Another weakness of the standard $k$-$\varepsilon$ model or other traditional $k$-$\varepsilon$ models lies with the modeled equation for the dissipation rate ($\varepsilon$). The well-known round-jet anomaly (named based on the finding that the spreading rate in planar jets is predicted reasonably well, but prediction of the spreading rate for axisymmetric jets is unexpectedly poor) is considered to be mainly due to the modeled dissipation equation.

The realizable $k$-$\varepsilon$ model proposed by Shih et al. (1995) was intended to address these deficiencies of traditional $k$-$\varepsilon$ models by adopting the following: (1) a new eddy-viscosity formula involving a variable $C_{\mu}$ originally proposed by Reynolds (1987), and (2) a new model equation for dissipation ($\varepsilon$) based on the dynamic equation of the mean-square vorticity fluctuation.

The modeled transport equations for $k$ and $\varepsilon$ in the realizable $k$-$\varepsilon$ model are as follows:

\[
\frac{\partial}{\partial x_i} (\rho ku_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_s + S_k 
\]

\[
\frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} - C_{1\varepsilon} \frac{\varepsilon}{k} C_{2\varepsilon} G_\varepsilon + S_\varepsilon
\]

where
In these equations, $C_2$ and $C_{1e}$ are constants. This model has been extensively validated for a wide range of flows (Shih et al., 1995; Kim et al., 1997), including rotating homogeneous shear flows, free flows including jets and mixing layers, channel and boundary layer flows, and separated flows. For all these cases, the performance of the model has been found to be substantially better than that of the standard $k$-$\varepsilon$ model. Especially noteworthy is the fact that the realizable $k$-$\varepsilon$ model resolves the round-jet anomaly; i.e., it predicts the spreading rate for axisymmetric jets as well as that for planar jets.

As in other $k$-$\varepsilon$ models, the eddy viscosity is computed from

$$
\mu_t = \rho C_{1e} \frac{k^2}{\varepsilon} 
$$

The difference between the realizable $k$-$\varepsilon$ model and the standard $k$-$\varepsilon$ models is that $C_\mu$ is no longer constant. It is computed from

$$
C_\mu = \frac{1}{A_o + A_s \left( kU^*/\varepsilon \right)} 
$$

where

$$
U^* = \sqrt{S_y S_y + \tilde{\Omega}_y \tilde{\Omega}_y} 
$$

and

$$
\tilde{\Omega}_y = \Omega_y - 2\varepsilon_{yk} \omega_k 
$$

$$
\Omega_y = \overline{\Omega}_y - \varepsilon_{yk} \omega_k 
$$

where $\overline{\Omega}_y$ is the mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity $\omega_k$. The model constants $A_o$ and $A_s$ are given by

$$
A_o = 4.04, \quad A_s = \sqrt{6} \cos \phi 
$$
where

$$
\phi = \frac{1}{3} \cos^{-1}\left(\sqrt{6}W\right), \quad W = \frac{S_y S_z S_{ij}}{S^3}, \quad \tilde{S} = \sqrt{S_y S_z}, \quad S_j = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
$$

(3.27)

It can be seen that $C_\mu$ is a function of the mean strain and rotation rates, the angular velocity of the system rotation, and the turbulence fields ($k$ and $\epsilon$). $C_\mu$ in Equation (3.12) can be shown to recover the standard value of 0.09 for an inertial sublayer in an equilibrium boundary layer.

In FLUENT, the term $-2\epsilon_{ij} \omega_k$ is, by default, not included in the calculation of $\tilde{\Omega}_y$. This is an extra rotation term that is not compatible with cases involving sliding meshes or multiple reference frames. The model constants $C_2, \sigma_k,$ and $\sigma_\epsilon$ have been established to ensure that the model performs well for certain canonical flows. The model constants are

$$
C_{1e} = 1.44, \quad C_2 = 1.9, \quad \sigma_k = 1.0, \quad \sigma_\epsilon = 1.2
$$

(FLUENT user manual 6.1)

3.3 Boundary Conditions

For this problem, the computational domain is taken to be the region bounded by the solid lines shown in Fig. 2.1. As can be seen, periodicity is assumed in the spanwise direction so that only one film-cooling hole and one pair of blockers need to be examined. In addition, the “upper channel wall” (i.e., the wall without film-cooling holes) was moved closer to the wall with the film-cooling holes. This was done to reduce the size of the computational domain and hence computational cost. The errors incurred by this are minimized by making the “upper channel wall” sufficiently far away and by making it inviscid (i.e., the velocity there can slip despite the viscous nature of the flow) so that boundary layers will not form there.
The fluid for the main flow (hot gas) and coolant is air. The main flow above the flat plate has a uniform freestream temperature $T_\infty$ of 298 K and a uniform freestream velocity $U_\infty$ of 20 m/s along the x-direction at the inflow boundary. At the outflow boundary, the static pressure is maintained at 1 atm. The flow in the boundary-layer is assumed to be turbulent from the leading edge of the flat plate. The coolant has a temperature $T_c$ of 188 K in the plenum. This gives a density ratio DR of 1.6. When the average velocity in the film-cooling holes $U_c$ is 6.25 m/s, the mass flux or blowing ratio $M$ is 0.5. Other blowing ratios were also studied, by varying the velocity at the inflow of the plenum that feeds the film cooling holes.

Two types of boundary conditions were applied on the flat plate for the heat transfer study. When the film-cooling adiabatic effectiveness is sought, the flat plate is made adiabatic. When the surface heat transfer coefficient is sought, the flat plate is maintained at a constant wall temperature $T_w$ of 243K. All other walls, including the walls of the film-cooling holes and the plenum, are made adiabatic. The back pressure at the outflow boundary above the flat plate is maintained at the standard atmospheric pressure.

3.4 Numerical Methods of Solution

3.4.1 Discretization

A control-volume-based technique is used to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control-volume basis. Discretization of the governing equations is illustrated by considering the steady-state conservation equation for transport of
a scalar quantity, $\phi$. This is demonstrated by the following equation written in integral form for an arbitrary control volume $V$ as follows:

$$
\int \rho \vec{v} \cdot d\vec{A} = \int \Gamma_\phi \nabla \phi \cdot d\vec{A} + \int S_\phi dV
$$

(3.28)

where

- $\rho$ density
- $\vec{v}$ velocity vector
- $\vec{A}$ surface area vector
- $\Gamma_\phi$ diffusion coefficient for $\phi$
- $\nabla \phi$ gradient of $\phi$
- $S_\phi$ source of $\phi$ per unit volume

Equation (3.27) is applied to each control volume, or cell, in the computational domain. The two-dimensional, tetragonal cell shown in Fig. 3.1 is an example of such a control volume. Discretization of Eq. (3.27) on a given cell yields

$$
\sum_{f}^{N_{faces}} N_{faces} \rho_f \vec{v}_f \phi_f \cdot \vec{A}_f = \sum_{f}^{N_{faces}} \Gamma_\phi (\nabla \phi)_n \cdot \vec{A}_f + S_\phi V
$$

(3.29)

where

- $N_{faces}$ number of faces enclosing cell
- $\phi_f$ value of convected through face
- $\rho_f \vec{v}_f \phi_f \cdot \vec{A}_f$ mass flux through the face
- $\vec{A}_f$ area of face $f$
Discrete values of the scalar $\phi$ are stored at the cell centers ($c_0$ and $c_1$ in Fig. 3.1). However, face values $\phi_f$ are required for the convection terms in Eq. 3.28 and must be interpolated from the cell center values. This is accomplished using an upwind scheme.

Figure 3.1  Control volume used to illustrate discretization of a scalar transport equation

Upwinding means that the face value $\phi_f$ is derived from quantities in the cell upstream, or "upwind," relative to the direction of the normal velocity $v_n$ in Eq. 3.29. The diffusion terms in Eq. 3.29 are central-differenced and are always second-order accurate.

3.4.2 Second-order upwind scheme

When second-order accuracy is desired, quantities at cell faces are computed using a multidimensional linear reconstruction approach. In this approach, higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about
the cell centroid. Thus when second-order upwinding is selected, the face value $\phi_f$ is computed using the following expression:

$$\phi_f = \phi + \nabla \phi \cdot \Delta \vec{s}$$

where $\phi$ and $\nabla \phi$ are the cell-centered value and its gradient in the upstream cell, and $\Delta \vec{s}$ is the displacement vector from the upstream cell centroid to the face centroid. This formulation requires the determination of the gradient $\nabla \phi$ in each cell. This gradient is computed using the divergence theorem, which in discrete form is written as

$$\nabla \phi = \frac{1}{V} \sum_{f}^{N_{\text{faces}}} \phi_f \vec{A}_f$$

Here the face values $\phi_f$ are computed by averaging $\phi$ from the two cells adjacent to the face. Finally, the gradient $\nabla \phi$ is limited so that no new maxima or minima are introduced.
CHAPTER 4. CFD ANALYSIS OF FILM COOLING

A paper presented at 44th Aerospace Sciences Meeting and Exhibition, January 9-12, 2006, Reno, Nevada

S. Na, B. Zhu, M. Bryden, F. Cunha, and T.I-P. Shih

4.1 Abstract

Computational fluid dynamics studies of film cooling have met with varying degrees of success. This paper describes an effort that uses the Fluent-UNS code, Version 6.1.22, with second-order upwind differencing and three eddy-diffusivity turbulence models – realizable k-ε, shear-stress transport (SST), and Spalart-Allmaras (S-A) – to compute two film-cooling configurations for which experimental data are available to assess the accuracy of the computed results. One configuration is film cooling of a flat plate in which the coolant is injected from a plenum through one row of inclined circular holes.

For the flat-plate configuration, results obtained show the SST model to provide better predictions of the laterally-averaged adiabatic effectiveness than the realizable k-ε and the S-A models. However, all three models significantly over predict the centerline adiabatic effectiveness. For this more complex flow, the laterally averaged adiabatic effectiveness was predicted surprisingly well by the realizable k-ε model. The local adiabatic effectiveness and the local heat transfer coefficient were predicted less accurately, but the qualitative trends were predicted correctly. Also, for both configurations, the computed results reveal considerable insight on the nature of the flow, how they are affected by the geometry, and how the flow affects adiabatic effectiveness and surface heat transfer.
4.2 Introduction

As computational fluid dynamics (CFD) becomes more widely used and more accepted as a design and analysis tool, it becomes increasingly more important to be able to assess its accuracy and its range of applicability. This paper addresses the CFD analysis of film cooling. Film cooling is selected for examination for three reasons (Goldstein, 1971 and 2001; Sue, 1985; Han et al, 2000; Sundén and Faghri, 2001). First, it is an important and widely used method for insulating and cooling materials whose surfaces are exposed to high temperature gases. Second, efficient and effective film cooling requires a good understanding of the detailed fluid mechanics and their effects on surface heat transfer – information that could be provided by CFD. Third, CFD has not been successful in predicting film cooling and its predictions are often treated with suspect.

Since most film-cooling flows are turbulent, CFD analysis of film cooling can be divided into four categories (Shih and Sultanian, 2001; Durbin and Shih, 2005): (1) those based on direct numerical simulation (DNS) of turbulence, (2) those based on large-eddy simulations (LES) of turbulence, (3) those based on Reynolds or Favre averaged Navier-Stokes (RANS) equations, and (4) those based on a combination of the above such as LES away from walls and RANS next to walls, referred to as detached eddy simulation (DES). Of these methods, DNS, LES, and DES have the highest potential to provide the best results in terms of accuracy. But, they are expensive computationally and have unacceptably slow turn-around time for design purposes – at least with existing computing resources. With today’s computing capabilities, RANS based on eddy-diffusivity and Reynolds stress models are the ones used.

References, Kercher (2002; 2005), all experimental and CFD studies of film cooling
in the open literature up to 2004. Though a number of rigorous and state-of-the-art CFD studies on film cooling based on RANS have been carried out, Walter & Leylek (2000) noted that a lot of the earlier work employed simplified mathematical models, used highly dissipative numerical schemes (e.g., first-order upwind), used wall functions (i.e., did not integrate to the wall), had poor quality grids (e.g., highly skewed and warped cell shapes especially in the region about the film-cooling holes), and/or could not afford to generate grid-independent solutions because of limited computing resources. In addition, there are concerns on lack of convergence to steady state and uncertainties from variability in the different CFD codes. Finally, there is considerable uncertainty on the adequacy of the eddy-diffusivity and Reynolds stress models used to capture the most important flow physics governing surface heat transfer.

With the advent of robust, versatile, and well maintained, state-of-the-art CFD commercial codes that everyone can have access to, some of the uncertainties in the variability in models, algorithms, grids, and codes of previous studies can be removed since all can repeat the work of previous investigators if grid and other input files are provided. The objective of this work is to setup a database of computed solutions on film cooling for all to access with the goal of creating an open environment, where users can examine and openly critique efforts on verification and validation of CFD codes and solutions for film cooling.

To get things started, this paper describes an effort that uses the Fluent-UNS code, Version 6.1.22, with second-order upwind differencing and several turbulence models to compute a flat plate film-cooling configuration for which experimental data are available to assess the accuracy of the computed results. All input and output files generated for these two configurations, including the grid systems, the solutions generated, and the convergence
history can be downloaded from www.public.iastate.edu/~tomshih/CFD/film-cooling/. The goal is to continue the buildup and the critique of the database to provide guidelines and best practices on models, algorithms, grids, and codes.

The remainder of this paper is organized as follows. First, the flat plate film-cooling configurations studied are described. This is followed by the formulation, the numerical method of solution, the grid, the grid sensitivity study, and the computed results. Everything needed to repeat our study are given either here or at the aforementioned URL.

5.3 Problem Description

In this study, flat plate film-cooling configuration is examined. For film cooling of a flat plate, the coolant is injected from a plenum through one row of inclined circular holes. This configuration was selected because experimental data are available to assess the accuracy of the computed results. Details of the geometry and operating conditions are given below.

This problem is taken from the experimental study of Kohli & Bogard (1995). A schematic diagram of this problem is shown in Fig. 4.1. For this problem, the cooling jets emerge from a plenum through one row of circular holes. Each hole has a diameter $D$ of 12.7 mm, a length of $2.8D$, and an inclination of $35^\circ$ relative to a plane tangent to the flat plate. The operating conditions are as follows. The fluid for the main flow (hot gas) and coolant is air. The main flow has a temperature of $T_\alpha = 298$ K upstream of the film-cooling holes, and the coolant has a temperature of $T_c = 188$ K, while in the plenum. This gives a density ratio of $DR = 1.6$. The average velocity in all film-cooling holes is $U_c = 6.25$ m/s. The freestream velocity of the main flow is $U_\alpha = 20$ m/s. This gives a mass flux or blowing ratio of $M = 0.5$. The velocity profile is uniform at the inflow boundary, and the flow in the boundary-layer is
assumed to be turbulent from the leading edge. All walls are adiabatic, and the back pressure at the outflow boundary is maintained at 1 atm.

For this problem, the computational domain is taken to be the region bounded by the solid lines shown in Fig. 4.1. As can be seen, periodicity is assumed in the spanwise direction so that only one film-cooling hole needs to be examined. In addition, the “upper channel wall” (i.e., the wall without film-cooling holes) was moved closer to the wall with the film-cooling holes. This was done to reduce the size of the computational domain and hence computational cost. The errors incurred by this are minimized by making the “upper channel wall” sufficiently far away and by making it inviscid (i.e., the velocity there can slip despite the viscous nature of the flow) so that boundary layers will not form there.

4.4 Formulation and Numerical Method of Solution

The two film-cooling problems just described are modeled by the ensemble-averaged conservation equations of mass (continuity), momentum (full compressible Navier-Stokes), and energy for a thermally and calorically perfect gas. Three different turbulence models were employed for the purpose of validation, and they are the one-equation Spalart-Allmaras (S-A) model (Spalart and Allmaras, 1992), the two-equation realizable k-ε model (Shih et al. 1995), and the two-equation shear-stress-transport (SST) model (Menter, 1991 and 1993). In all cases, the integration is to the wall (i.e., wall functions are not used).

Solutions to the conservation equations and the three turbulence models were obtained by using Version 6.1.22 of the Fluent-UNS code. Fluent-UNS generates solutions by using the SIMPLE and the SIMPLEC algorithms for problems with steady states. Since SIMPLE is more stable for problems with complicated flow features, SIMPLE was used. All equations (conservation and turbulent transport) are integrated over each cell of the grid
system. The fluxes at the cell faces are interpolated by using second-order upwind differencing. In all cases, computations were carried out until the residual plateau to ensure convergence to steady-state has been reached. At convergence, the scaled residuals were always less than $10^{-6}$ for $u$ and $w$, less than $10^{-7}$ for $v$, less than $10^{-8}$ for energy, and less than $10^{-4}$ for the turbulence quantities.

4.5 Grid System Used

Accuracy of CFD solutions is strongly dependent upon the grid system, which must be constructed to minimize grid-induced errors and to resolve the relevant flow physics. When using Fluent-UNS, cells in the grid system can be made up of hexagons or tetrahedrons. Though grid systems based on tetrahedrons are easier to generate, especially for problems with complicated geometries, hexahedrons are used in this study because they are known to provide more accurate solutions for flows with boundary layers. The grid systems used for the two film-cooling configurations studied are described below.

Because of the presence of the film-cooling hole and the need to resolve the sharp gradients of the flow variables next to all walls as well as the cooling-jet and the approaching flow interaction region, the generation of a high quality grid with hexagons can be a challenge. Figure 4.3 shows the nine grid systems generated for this study. The numbers of grid points in these nine grid systems are as follows: Grid 1: 1,190,432, Grid 2: 1,018,238, Grid 3: 1,018,238, Grid 4: 492,670, Grid 5: 1,151,136, Grid 6: 715,594, Grid 7: 1,223,162, Grid 8: 2,546,286, and Grid 9: 2,811,028. For all grid systems, $y^+$ of the first cell away from walls was less than unity.

The rationale for the generation of these grids is as follows: Grid 1 in Fig. 4.3 is essentially an H-H grid that has grid lines clustered next to the flat plate and the wall of the
film-cooling hole. As a result, this grid system has streaks of highly clustered grid lines that propagate from the film-cooling hole and from the flat plate into the flow domain, where clustering is not needed. Since these streaks may adversely affect the accuracy of the solutions, other grids were constructed by wrapping grid lines around solid walls to prevent such streaks from forming (Chi et al., 2002). Grid 2 was the first of such grids generated. Grid 3 is Grid 2 with elliptic smoothing. Grid 4 is a coarser version of Grid 3 with even smoother grids (goal is trying to find an optimal grid). Grid 5 is a refined version of Grid 4 via h-refinement in a region about the film-cooling hole and jet. Grid 6 is an r-refined version of Grid 4. As can be seen in Fig. 4.3, mesh refinement was accomplished by using either r- or h-refinement.

4.6 Results

In this section, the results obtained for the two film-cooling configurations are described and compared with experimental data. Only steady-state solutions are of interest, and each solution is iterated until the residuals plateau. In all cases, the scaled residual is less than $10^{-5}$ for the continuity equation, less than $10^{-4}$ for X-, Y-, and Z-component velocity equations, less than $10^{-7}$ for energy equation, and less than $10^{-3}$ for the turbulent equations.

Figures 4.4 to 4.7 show the results obtained by all grids for the centerline adiabatic effectiveness and the laterally averaged adiabatic effectiveness, respectively. From this figure, it can be seen that all grids employed provided similar results except just downstream of the film-cooling hole, even though the number of cells ranged from 492,670 to 2,811,028. The reason that all grids provided reasonably and similarly good results is that guidelines from previous studies (e.g., Refs. Cruse, 1997 and Lin and Shih, 1999) were employed to generate them. One goal of this grid sensitivity study is to see how coarse can the grid be and still...
provide reasonable results. From Figures 4.4 to 4.7, Grid 6 with 715,594 is most optimal.

Figure 4.8 shows how well the three turbulence models predict laterally averaged and centerline adiabatic effectiveness when compared to measured values from Kohli & Bogard (1995). From this figure, it can be seen that all models over predict centerline adiabatic effectiveness. The SST and S-A models were able to predict the laterally averaged adiabatic effectiveness reasonably well. At the leading edge of the film-cooling hole, the SST model predicted a boundary-layer thickness of 0.13D, a shape factor of 1.68, and Reynolds number based on the freestream velocity and momentum thickness of 1,297. The corresponding measured values are 1.2D, 1.51, and 1,100.

Figures 4.9 to 4.11 show the predicted flow characteristics. Figures 4.9 and 4.10 show the normalized temperature and pressure distributions about the film-cooling hole. These figures show the high stagnation pressure caused by the hot gas impinging on the cooling jet. It also shows the mixing layer between the hot gas and coolant near the film cooling hole and the evolution of the coolant jet along the plate, including the entrainment of hot gas underneath the coolant jet at X/D = 1 and 5.

4.7 Summary

The purpose of this paper is to set up a database for the verification and validation of CFD analysis of film cooling. This paper describes results obtained for flat plate film-cooling configurations. All input files needed to generate the results described in this paper, including all grid systems, and the solutions generated can be downloaded from www.public.iastate.edu/~tomshih/CFD/film-cooling/. We invite the continued buildup of this database by the community and open the critique of the database to provide guidelines and best practices on models, algorithms, grids, and codes.
Figure 4.1. Schematic of film cooling of a flat plate from a row of inclined circular holes (not drawn to scale).

Figure 4.2. Grid system for flat-plate film-cooling problem.
(a) Grid 1 to 9 in X-Y plane at $Z = 0$.

(b) Grid 1 to 9 in Y-Z plane at $X = 0$. 
Figure 4.3. Grid systems generated for the flat-plate film-cooling problem. Grid 1 to 9 ordered from left to right and top to bottom.
Figure 4.4. Adiabatic effectiveness along $Y = 0$. (a) from $X/D = 0$ to $X/D = 30$. (b) from $X/D = 4$ to $X/D = 8$
Figure 4.5. Laterally averaged adiabatic effectiveness along Y = 0. (a) from X/D = 0 to X/D = 30. (b) from X/D = 1 to X/D = 3.
Figure 4.6. Adiabatic effectiveness at \( X/D = 1 \). (a) from \( Y/D = 0 \) to \( X/D = 1.5 \). (b) from \( Y/D = 0 \) to \( X/D = 0.3 \).
Figure 4.7. Adiabatic effectiveness at X/D = 5. (a) from Y/D = 0 to X/D = 1.5. (b) from Y/D = 0 to X/D = 0.2
Figure 4.8. Laterally averaged and centerline adiabatic effectiveness predicted by realizable k-ε, SST, and Spalart Allmaras models and comparison with measured values.
Figure 4.9.  Predicted normalized temperature (left) and pressure (right) for the flat plate film-cooling problem.
Figure 4.10.  Predicted normalized temperature (left) and pressure (right) for the flat-plate film-cooling problem at X/D = 1 (top) and 5 (bottom).

Figure 4.11.  Streamlines colored by temperature for the flat-plate film-cooling problem.
CHAPTER 5. EFFECTS OF COATING BLOCKAGE AND ROUGHNESS ON FILM COOLING EFFECTIVENESS AND SURFACE HEAT TRANSFER

A paper presented at 44th Aerospace Sciences Meeting and Exhibition, January 9-12, 2006, Reno, Nevada

S. Na, F. Cunha, M. Chyu, and T.I-P. Shih

5.1 Abstract

Computations were performed to study the effects of blockage by thermal-barrier coatings (TBC) on film-cooling of a flat plate from a row of inclined holes or an inclined slot. For film-cooling through an inclined slot, computations were also performed to study the effects of TBC roughness. Results obtained show that if the mass-flux ratio is kept the same at 0.5, then TBC blockages reduces adiabatic effectiveness because the amount of coolant flow through each hole or across the slot is less. However, the amount of decrease in adiabatic effectiveness was less than expected based on the amount of reduced coolant flow. One reason for the better performance from the TBC blocked film-cooling hole is the step created by the TBC blockage at the upstream side of the film-cooling hole, which affected how the approaching boundary layer flow interacts with the film-cooling jet. Surface roughness was found to increase displacement thickness and hence is an additional source of blockage within the film-cooling hole.

5.2 Introduction

Stators and rotors in gas-turbine engines operate in very harsh environments, involving high temperatures and debris that can cause material degradation. All surfaces in the stators and rotors such as blades, vanes, endwalls, and hubs that come in contact with the
combustor's hot gases must be cooled in which the cooling must account for the surface roughness that can develop. The degree and the nature of the roughness and material degradation due to mechanisms such as erosion, fuel deposition, corrosion, and spallation of thermal-barrier coatings depend on the environment from which the air is ingested, the effectiveness of cooling management such as film cooling in maintaining material temperatures within acceptable limits, the operating conditions, and the duration of service.

For the first-stage stator, where the environment is the harshest, film cooling is needed in addition to internal cooling. The goal of film cooling is to form an insulating layer of cooler air between the stator surface and the hot gas from the combustor (Han et al., 2000; Goldstein, 2001; Sundén and Faghri, 2001). Though computational fluid dynamics (CFD) has provided some meaningful results for film-cooling (see, e.g., Lin and Shih, 1999; Shih et al., 1999; Walter and Leylek, 2000; Lin and Shih, 2001; Shih and Sultanian, 2001), challenges remain in turbulence modeling (Durbin and Shih, 2005) and grid resolution requirements (Shih, 2006). This challenge is exacerbated by manufacturing issues related to thermal-barrier coatings (TBCs) and by surface roughness that develop during service.

On TBCs, the manufacturing process creates non-uniformities in the coating about the film-cooling holes. The extent of non-uniformity can be significant. To illustrate, it is noted that typical film-cooling holes are 0.8 mm in diameter, whereas the thickness of the TBC varies from 0.3 to 0.5 mm and the thickness of the metallic-bond coat that connects the TBC to the turbine metal surface varies from 0.1 to 0.2 mm. Figure 5.1 illustrates the non-uniformity that can result. From Fig. 5.1, it can be seen that the non-uniformity can have a significant effect of how the film-cooling jet emerges from the film-cooling hole. There are two ways to overcome this non-uniformity. One way is to remove the TBC and MBC in the
film-cooling hole by a combination of EDM and laser drilling. But, this can be expensive and laser cannot be used to fashion shaped holes. The other is to understand and assess the effects of the non-uniformity on the effectiveness of the film-cooling jet.

On roughness, turbine surfaces invariably become rough with service (Taylor, 1990; Tarada and Suzuki, 1993; Bons et al., 2001). This material degradation in the form of surface roughness is known to increase surface heat transfer in a significant way (Tarada and Suzuki, 1993; Blair, 1994; Hoff's et al., 1996; Bogard et al. 1998; Abuaf et al., 1998; Hodge et al., 2001; Bons, 2002). For a given cooling management, increase in surface heat transfer increases material temperature, which hastens further material degradation. The importance of surface roughness on surface heat transfer has lead many investigators to study this problem. For film cooling, the main challenge is to understand the effects of roughness on film-cooling effectiveness and surface heat transfer.

To date, no CFD studies have been reported on film cooling with TBC blockage or roughness. The objective of this study is as follows:

1. Perform three-dimensional (3-D) CFD simulations of film cooling on a flat plate from a row of circular holes (Kohli & Bogard's experiment, 1995) with and without TBC blockage.

2. Perform two-dimensional (2-D) CFD simulations of film cooling on a flat plate with and without TBC blockages and with and without roughness.

Roughness is simulated in 2-D instead of 3-D because the multiple length scales of the roughness geometry requires a number of cells to resolve.

The remainder of this paper is organized as follows. First, the film-cooling problems studied are described. This is followed by the formulation, the numerical method of solution,
the grid, the grid sensitivity study, and the computed results.

5.3 Problem Description

As noted in the Introduction, two film-cooling configurations are examined. One configuration is film cooling of a flat plate in which the coolant is injected from a plenum through one row of inclined circular holes with and without TBC – henceforth referred to as the 3-D problem. The other configuration is film cooling of the same flat plate in which the coolant is injected from a plenum through one inclined slot with and without TBC and roughness in the TBC – henceforth referred to as the 2-D problem. The 3-D problem was selected because experimental data are available to assess the accuracy of the computed results for the case without TBC. Details of the geometry and operating conditions are given below.

5.3.1 3-D Problem: with and without TBC

Schematic diagrams of the 3-D problem studied are shown in Figs. 5.3 and 5.4. Figure 5.3 shows the geometry of the flat plate with film-cooling holes and the plenum, and Fig. 5.4 shows the TBC placed on top of the flat plate and a part of the film-cooling hole near the plate surface that can be coated.

For this problem, the cooling jets emerge from a plenum through one row of circular holes. Each hole has a diameter D of 12.7 mm, a length of 3.5D, and an inclination of 35° relative to a plane tangent to the flat plate. When there is TBC, its thickness is 0.5D. Other details of the geometry are given in Figs. 5.3 and 5.4.

The operating conditions are as follows. The fluid for the main flow (hot gas) and coolant is air. The main flow has a temperature $T_\infty$ of 298 K upstream of the film-cooling holes, and the coolant has a temperature $T_c$ of 188 K, while in the plenum. This gives a
density ratio DR of 1.6. The freestream velocity $U_{\infty}$ of the main flow is 20 m/s. The velocity profile is uniform at the inflow boundary, and the flow in the boundary-layer is assumed to be turbulent from the leading edge of the flat plate. All walls are adiabatic, and the back pressure at the outflow boundary is maintained at 1 atm.

At the plenum inflow, in addition to specifying the coolant temperature, either the velocity or the pressure must be specified. In this study, the following three different conditions were examined: (1) Adjust the velocity at the plenum inflow so that the average velocity $U_c$ in the film-cooling hole is 6.25 m/s to yield a mass flux or blowing ratio $M$ of 0.5 for cases with and without TBCs. (2) Fix the velocity at the plenum inflow to be 0.28 m/s (vertically upwards towards the film-cooling hole). This velocity yields a mass flux ratio of 0.5 when there are no TBCs. (3) Fix the static pressure at the plenum inflow to be 128.4 Pa gage. This static pressure gives a mass flux ratio of 0.5 when there are no TBCs. Conditions 2 and 3 are essentially the same, but were examined for consistency. The purpose is to see how the mass flux ratio change with TBC since TBC increases loss and changes

For this problem, the computational domain is taken to be the region bounded by the solid and dashed lines in Fig. 5.3. Note that periodicity is assumed in the spanwise direction so that only one film-cooling hole needs to be examined (see Fig. 5.3(a)). In addition, the "upper channel wall" (i.e., the wall without film-cooling holes) was moved closer to the wall with the film-cooling holes (Fig. 5.3(b)). This was done to reduce the size of the computational domain and hence computational cost. The errors incurred by this are minimized by making the "upper channel wall" sufficiently far away and by making it inviscid (i.e., the velocity there can slip despite the viscous nature of the flow) so that boundary layers will not form there.
5.3.2 2-D Problem: with and without TBC and TBC roughness

The 2-D problem is identical to the 3-D problem just described except for the following differences. The first difference is having a slot instead of a row of holes. For the 2-D problem, the inclined film-cooling hole is replaced by a slot of width $D$ and length $3.5D$ inclined at $35^\circ$ relative to a plane tangent to the flat plate. The second difference is that two TBC thicknesses were examined instead of one: $0.5D$ and $0.75D$. Also, the geometry of this TBC differs slightly from those for the 3-D problem, and is shown in Fig. 5.5. The fourth difference is that three different roughness geometries were examined for the TBC with $0.5D$ thickness. The geometry of the roughness is shown in Fig. 5.6. It is a 2-D slice taken from a 3-D rough surface measured by Bons et al. (2001). The characteristics of this 2-D roughness are summarized in Table 5.1. The fifth difference is the boundary conditions imposed at the plenum. Like the 3-D problem, the coolant temperature is still specified to be $188$ K to give a density ratio $DR$ of 1.6, but the values differ for the imposed velocity and pressure. These are the corresponding second condition imposed at the plenum inflow: (1) Adjust the velocity at the plenum inflow so that the average velocity $U_c$ in the film-cooling hole is $6.25$ m/s to yield a mass flux or blowing ratio of $M = 0.5$ for cases with and without TBCs. (2) Fix the velocity at the plenum inflow to be $1.07$ m/s instead of $0.28$ m/s. This velocity yields a mass flux ratio of 0.5 when there are no TBCs. (3) Fix the static pressure at the plenum inflow to be $151.6$ Pa gage instead of $128.4$ Pa gage. Like the 3-D problem, this static pressure gives a mass flux ratio of 0.5 when there are no TBCs.

5.4 Formulation and Numerical Method of Solution

The 3-D and the 2-D film-cooling problems just described are modeled by the ensemble-averaged conservation equations of mass (continuity), momentum (full
compressible Navier-Stokes), and energy for a thermally and calorically perfect gas. The realizable k-ε model was used to model the increased mixing due to turbulence. In all cases, the integration is to the wall (i.e., wall functions are not used).

Solutions to the conservation equations and the three turbulence models were obtained by using Version 6.1.22 of the Fluent-UNS code. Fluent-UNS generates solutions by using the SIMPLE and the SIMPLEC algorithms for problems with steady states. Since SIMPLE is more stable for problems with complicated flow features, SIMPLE was used. All equations (conservation and turbulent transport) are integrated over each cell of the grid system. The fluxes at the cell faces are interpolated by using second-order upwind differencing. In all cases, computations were carried out until the residual plateau to ensure convergence to steady-state has been reached. At convergence, the scaled residuals were always less than $10^{-6}$ for $u$ and $w$, less than $10^{-7}$ for $v$, less than $10^{-8}$ for energy, and less than $10^{-4}$ for the turbulence quantities.

5.5 Grid System Used

Accuracy of CFD solutions is strongly dependent upon the grid system, which must be constructed to minimize grid-induced errors and to resolve the relevant flow physics. When using Fluent-UNS, cells in the grid system can be made up of hexagons or tetrahedrons. Though grid systems based on tetrahedrons are easier to generate, especially for problems with complicated geometries, hexahedrons are used in this study because they are known to provide more accurate solutions for flows with boundary layers. The grid systems used for the two film-cooling configurations studied are described below.

5.5.1 3-D Problem: with and without TBC

Because of the presence of the film-cooling hole and the need to resolve the sharp
gradients of the flow variables next to all walls as well as the cooling-jet and the approaching flow interaction region, the generation of a high quality grid with hexagons can be a challenge. Figures 5.7 to 5.9 show the type of grid systems generated for this study. The numbers of cells used for the 3-D problem without TBC is 2,118,132, and the number of cells used for the 3-D problem with TBC is 2,267,484. For both grid systems, Fig. 5.11 shows equiangle skewnesses are almost 0.1 and less than 0.8 in the histograms for without and with TBC. Figure 5.10 shows y+ of the first cell away from walls is less than unity and at least five cells within a y+ of five. These two grids were arrived at after a grid sensitivity study that included the following grids: 2,118,132 cells, 2,559,930, and 3,210,692 for the 3-D problem without TBC and 2,267,484 cells and 3,017,940 cells for the 3-D problem with TBC.

5.5.2 2-D Problem: with and without TBC and TBC roughness

Figures 5.12 and 5.13 show the grid system employed for the 2-D problem with and without TBC and TBC roughness. Table 5.2 gives a summary of the grids used. Like the 3-D problem, the y+ of the first cell away from walls is less than unity in Fig. 5.26. From Fig. 5.27, equiangle skewnesses are almost 0.1 and less than 0.8 in the histograms for all 2-D problems. All grids were arrived at after a grid sensitivity study, and Table 5.3 gives a summary of the sensitivity study. The used grid cells are 43,669 for the 2-D problem without TBC, 44,489 and 44,911 for the 2-D problem with 0.5D mm and 0.75D mm TBCs, and 602,989, 390,069, 620,357, and 205,250 for the 2-D problem with 0.5D mm TBC and four roughnesses, Rq1, Rq2, Rq3, and Rq4, respectively. The shear stress and surface effectiveness with h-refined grid are shown in Figs. 5.28 and 5.29, respectively. The calculations on different grids show remarkably good agreement.
5.6 Results

In this section, the results obtained for the two film-cooling problems are described. Only steady-state solutions are of interest, and each solution is iterated until the residuals plateau. In all cases, the scaled residual is less than $10^{-5}$ for the continuity equation, less than $10^{-4}$ for X-, Y-, and Z-component velocity equations, less than $10^{-7}$ for energy equation, and less than $10^{-3}$ for the turbulent equations.

For the film-cooling problem studied, experimental data exist for the case without TBC and TBC roughness. Details of this validation study, reported in Ref. (Na, 2006), are summarized in Fig. 5.15. From this figure, it can be seen that grid-independent solutions were generated by a sequence of grids. The solution over predicted the centerline adiabatic effectiveness and under predicted the laterally averaged adiabatic effectiveness. The trends, however, are predicted correctly.

5.6.1 3-D Problem: with and without TBC

Figure 5.16 shows the centerline and the laterally averaged adiabatic effectiveness to decrease significantly when there is TBC if the mass flux ratio is maintained at 0.5. Figures 5.17 to 5.21 show why this reduction takes place. These figures show that with TBC, there is considerable blockage in the film-cooling hole, which increases the coolant exit velocity. Thus, to keep the mass-flux ratio at 0.5, the net flow of coolant from the film-cooling hole had to decrease. The amount of decrease is about 50% since about 50% of the film-cooling-hole, cross-sectional area is blocked. From this perspective, the reduction in adiabatic effectiveness is not so bad. Note that at 5D downstream of the film-cooling hole, the reduction is about 15% for the center adiabatic effectiveness and 35% for the laterally
averaged adiabatic effectiveness. This indicates that TBC blockage is producing a more effective film-cooling-hole design.

To better understand why this is so, Fig. 5.19 shows the step at the upstream side of the film-cooling hole created by the TBC is modifying how the approaching boundary-layer flow interacts with the exiting film-cooling jet. Figure 5.18 and 5.19 show that this interaction increases the magnitude of the counter-rotating vortices, which increases lateral spreading of the cooling jet. Figure 5.20 shows the location of the maximum pressure about the film-cooling jet on the surface of the flat plate shifted from just upstream of the film-cooling jet to two sides about the film-cooling jet. Figure 5.21 shows the magnitude of the shear stress on the plate surface to be modified in a similar manner.

Figures 5.22 to 5.24 show the evolution of the film-cooling jet with and without TBC in which the mass-flux ratio is maintained at 0.5. From these figures, it is interesting to note that the coverage with and without TBC is comparable though the cooling flow rate differs by as much as 50%.

Figure 5.25 shows the situation in which the flow rate through the film-cooling hole with and without TBC are kept the same. From this figure, it can be seen that the centerline and the laterally averaged adiabatic effectiveness again decrease significantly when there is TBC. Though this seems contradictory based on the earlier discussion, it is not. The reason the film-cooling is less effective this time is because the mass-flux ratio is greatly increased by the TBC blockage. As a result, the film-cooling jet lifted off.

5.6.2 2-D Problem: with and without TBC and TBC roughness

Figure 5.30 show the adiabatic effectiveness of the 2-D problem with and without TBC and TBC roughness. From this figure, the following observations can be made. First,
with TBC blockage, the adiabatic effectiveness decreases as the blockage increases. This is because the net flow rate of coolant decreases with blockage if the mass-flux ratio is maintained at 0.5. Second, for a given TBC blockage, the adiabatic effectiveness decreases as roughness (Rq) increases. This is because roughness induces complex motion about the peaks and valleys. These complex fluid motion increases the displacement thickness, which increases the effective blockage in the hole and surface heat transfer on the surface.

5.7 Summary

A CFD study was performed to examine the effects of TBC blockage on the film-cooling of a flat plate from a row of inclined holes or an inclined slot. For film-cooling through an inclined slot, the effects of TBC roughness were also examined. Results obtained show that if the mass-flux ratio is kept the same at 0.5, then TBC blockages reduces adiabatic effectiveness because the amount of coolant flow through each hole or across the slot is less. However, the amount of decrease in adiabatic effectiveness was found to be much less than expected since the coolant flow was reduced significantly more to keep the mass-flux ratio the same. One reason for the better performance from the TBC blocked film-cooling hole is the step created by the TBC blockage at the upstream side of the film-cooling hole, which affected how the approaching boundary layer flow interacts with the film-cooling jet. On surface roughness, they were found to increase displacement thickness and hence is an additional source of blockage within the film-cooling hole and increases mixing on the film-cooled surface.

Acknowledgement

This research was supported by the Department of Energy's University Turbine
System Research Program with Dr. R. Wenglarz and Dr. L.P. Golan as the Technical Monitors (SCIES Project 04-01-SR115). The authors are grateful for this support. The authors are also grateful to Fluent for providing the Fluent-UNS code.
Table 5.1. Statistical Properties of the 2-D Surface Roughness Shown in Fig. 5.6*

<table>
<thead>
<tr>
<th></th>
<th>Rq1</th>
<th>Rq2</th>
<th>Rq3</th>
<th>Rq4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Min.</td>
<td>-0.8773</td>
<td>-1.7546</td>
<td>-3.5093</td>
<td>-6.4122</td>
</tr>
<tr>
<td>Max.</td>
<td>1.0157</td>
<td>2.0314</td>
<td>4.0629</td>
<td>7.4238</td>
</tr>
<tr>
<td>Rq</td>
<td>0.43</td>
<td>0.87</td>
<td>1.74</td>
<td>3.18</td>
</tr>
<tr>
<td>Ku</td>
<td>2.4090</td>
<td>2.4090</td>
<td>2.4090</td>
<td>2.4090</td>
</tr>
<tr>
<td>Sk</td>
<td>0.0007</td>
<td>0.0059</td>
<td>0.0475</td>
<td>0.2899</td>
</tr>
</tbody>
</table>

*Rq1, Rq2, Rq3, and Rq4 denote the names of the 4 rough surfaces. All numbers are in mm. Min and Max refers to the lowest valley and the highest peak, respectively, where the mean roughness is set equal to zero. Rq, Ku, and Sk are the rms, the kurtosis, and the skewness of the roughness, respectively.

Table 5.2 Grids used for the 2-D problems

<table>
<thead>
<tr>
<th>case</th>
<th>Cell #</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>43,669</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 0 mm roughness</td>
<td>44,489</td>
</tr>
<tr>
<td>0.75D TBC, Rq = 0 mm roughness</td>
<td>44,911</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 0.43 mm roughness</td>
<td>602,989</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 0.87 mm roughness</td>
<td>390,069</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 1.74 mm roughness</td>
<td>620,357</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 3.18 mm roughness</td>
<td>205,250</td>
</tr>
</tbody>
</table>
Table 5.3  The results from the 2-D problems for grid independency study

<table>
<thead>
<tr>
<th>case</th>
<th>Cell #</th>
<th>Grid #</th>
<th>$\tau$ (pa)</th>
<th>T (k)</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>43,669</td>
<td>44,282</td>
<td>0.656</td>
<td>205.99</td>
</tr>
<tr>
<td></td>
<td>138,721</td>
<td>139,868</td>
<td>0.672</td>
<td>206.05</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 0 mm</td>
<td>44,489</td>
<td>45,119</td>
<td>0.752</td>
<td>212.88</td>
</tr>
<tr>
<td></td>
<td>142,931</td>
<td>144,110</td>
<td>0.755</td>
<td>213.21</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 0.43 mm</td>
<td>602,989</td>
<td>680,415</td>
<td>0.696</td>
<td>213.08</td>
</tr>
<tr>
<td></td>
<td>1,077,349</td>
<td>1,150,210</td>
<td>0.698</td>
<td>213.14</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 0.87 mm</td>
<td>390,069</td>
<td>424,994</td>
<td>0.683</td>
<td>220.74</td>
</tr>
<tr>
<td></td>
<td>923,919</td>
<td>954,495</td>
<td>0.689</td>
<td>220.14</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 1.74 mm</td>
<td>620,357</td>
<td>687,751</td>
<td>0.543</td>
<td>222.31</td>
</tr>
<tr>
<td></td>
<td>1,579,367</td>
<td>1,642,831</td>
<td>0.598</td>
<td>222.63</td>
</tr>
<tr>
<td>0.5D TBC, Rq = 3.18 mm</td>
<td>205,250</td>
<td>208,310</td>
<td>0.506</td>
<td>215.25</td>
</tr>
<tr>
<td></td>
<td>726,764</td>
<td>735,751</td>
<td>0.495</td>
<td>215.03</td>
</tr>
<tr>
<td>0.75D TBC, Rq = 0 mm</td>
<td>44,911</td>
<td>45,547</td>
<td>0.841</td>
<td>229.65</td>
</tr>
<tr>
<td></td>
<td>144,157</td>
<td>145,347</td>
<td>0.858</td>
<td>229.11</td>
</tr>
</tbody>
</table>
Figure 5.1. Non-uniform coating of TBCs about a film-cooling hole.

Figure 5.2. A patch of rough surface, where colors show valleys and peaks of the roughness (compliments of Jeffrey Bons, 2001).
Figure 5.3. Schematic of film cooling of a flat plate from a row of inclined circular holes (no TBCs and not drawn to scale) (Kohli and Bogard, 1995). (a) 3-D view. (b) Top view (X-Z). (c) Side view (X-Y).
Figure 5.4. Schematic of TBC placed on top of the flat plate and the film-cooling holes for the 3-D problem.

Figure 5.5. Schematic of TBC placed on top of the flat plate and the film-cooling slot for the 2-D problem.
Figure 5.6. Geometry of the three roughnesses imposed on the TBC (see Table 1 for the statistics).
Figure 5.7. Grid system for the 3-D problem without TBC. Left is overall grid and right is grid in hole.

Figure 5.8. Grid system for the 3-D problem with TBC.
Figure 5.9. 2-D views of grid for the 3-D problem with TBC.
Figure 5.10. $y^+$ values on the surface without and with TBC for the 3-D problems.

Figure 5.11. Histograms of equiangle skewness values for without and with TBC for the 3-D problems.
Figure 5.12. Grid System for the 2-D problem without TBC.

Figure 5.13. Grid System for the 2-D Problem with BC and TBC roughness.
Figure 5.14. Grid-independent study for 0.5D TBC: (a) adiabatic effectivenesses at X/D = 3 in the spanwise direction. (b) Centerline adiabatic effectiveness.
Figure 5.15. Centerline and laterally averaged adiabatic effectiveness for the 3-D problem without TBC with hole length of 2.8 (Na et al. 2006).
Figure 5.16. Centerline and laterally averaged adiabatic effectiveness for the 3-D problem with and without TBC and $M = 0.5$. 
Figure 5.17. Velocity field in the Y-Z plane at X/D = 1 downstream of the film-cooling hole with M= 0.5. Left: no TBC. Right: 0.5D thick TBC.
Figure 5.18. Normalized temperature in the Y-Z plane at X/D = 1 downstream of the film-cooling hole with M= 0.5. Left: no TBC. Right: 0.5D thick TBC.

Figure 5.19. Normalized temperature in the middle X-Z plane with M= 0.5. Left: no TBC. Right: 0.5D thick TBC.
Figure 5.20. Gage pressure (Pa) on the flat plate surface with $M = 0.5$. Left: no TBC. Right: 0.5D thick TBC.

Figure 5.21. Magnitude of shear stress (Pa) on the flat plate surface with $M = 0.5$. Left: no TBC. Right: 0.5D thick TBC.
Figure 5.22. Surface adiabatic effectiveness with $M = 0.5$. Left: no TBC. Right: 0.5D thick TBC.
Figure 5.23. Normalized temperature in the middle X-Z plane with M = 0.5. Left: no TBC. Right: 0.5D thick TBC.

Figure 5.24. Normalized temperature in the Y-Z plane at X/D = 3 (left) and X/D = 7 (right) with M = 0.5 Top: no TBC. Bottom: 0.5D thick TBC.
Figure 5.25. Centerline and laterally averaged adiabatic effectiveness for the 3-D problem with and without TBC and the same velocity at the plenum inflow.
Figure 5.26. $y^+$ for the 2-D problems

(a) w/o TBC, w/o roughness

(b) 0.5D TBC, w/o roughness

(c) 0.5D TBC, $R_q = D/4$
Figure 5.27. Equiangle skewness for the 2-D problems.
Figure 5.28. Shear stresses for the 2-D problems.
Figure 5.29. Surface effectivenesses for the 2-D problems.
Figure 5.30. Adiabatic effectiveness for the 2-D problem with and without TBC and TBC roughness for $M = 0.5$. 
CHAPTER 6. PREVENTING HOT-GAS INGESTION BY FILM-COOLING JET VIA FLOW-ALIGNED BLOCKERS

A paper to be submitted to The Journal of Heat Transfer

T. I-P. Shih and S. Na

6.1 Abstract

Flow aligned blockers are proposed to minimize the entrainment of hot gases underneath film-cooling jets by the counter-rotating vortices within the jets. Computations, based on the ensemble-averaged Navier-Stokes equations closed by the realizable $k$-$\varepsilon$ turbulence model, were used to assess the usefulness of rectangular prisms as blockers in increasing film-cooling adiabatic effectiveness without unduly increasing surface heat transfer and pressure loss. The Taguchi’s design of experiment method was used to investigate the effects of the height of the blocker (0.2D, 0.4D, 0.8D), the thickness of the blocker (D/20, D/10, D/5), and the spacing between the pair of blockers (0.8D, 1.0D, 1.2D), where D is the diameter of the film-cooling hole. The effects of blowing ratio (0.37, 0.5, 0.65, 1.0) were also studied. Results obtained show that blockers can greatly increase film-cooling effectiveness. By using rectangular prisms as blockers, the laterally averaged adiabatic effectiveness at 15D downstream of the film-cooling hole is as high as that at 1D downstream. The surface heat transfer was found to increase slightly near the leading edge of the prisms, but reduced elsewhere from reduced temperature gradients that resulted from reduced hot gas entrainment. However, pressure loss was found to increase somewhat because of the flat rectangular leading edge, which can be made more streamlined.
6.2 Introduction

To increase thermal efficiency and specific thrust, advanced gas turbine stages are designed to operate at increasingly higher inlet temperatures (Suo, 1985). This increase is made possible by advances in materials such as super alloys and thermal-barrier coatings and by advances in cooling technology such as internal, film, impingement, and other techniques (Suo, 1985; Metzger, 1985; Moffat, 1987). With cooling, inlet temperatures can far exceed allowable material temperatures. Though cooling is an effective way to enable higher inlet temperatures, efficiency considerations demand effective cooling to be accomplished with minimum amount of cooling air since it takes energy to pump the cooling air through the turbine system, which operates at high pressures.

For advanced gas turbines, the first-stage stator and rotor typically require film cooling, which strives to form a blanket of cooler air next to the material surface to insulate the material from the hot gas (Golstein, 1971). Many investigators have studied the effects of design and operating parameters on film cooling. These include film-cooling hole inclinations and length-to-diameter ratios, spacing between holes, geometry of holes including shaped holes, surface curvatures, mainflow turbulence, embedded vortices in the mainflow, and unsteadiness from rotor-stator interactions (see, e.g., reviews by Han et al. (2000), Goldstein (2001), Sundén & Faghri (2001), and Shih & Sultanian (2001); in addition see the comprehensive bibliography provided by Kercher (2003 and 2005)).

Of the previous studies, Kelso & Lim (1996) and Haven et al. (1997) showed the important role played by vortices in the evolution of film-cooling jets. One pair, referred to as the counter-rotating vortices (CRVs), was found to lift the jet off the surface that it is intended to protect and to entrain hot gases underneath it. The other pair, referred to an anti-
kidney pair, was shown to have a sense of rotation opposite to that of the CRVs, and so can counteract the undesirable tendencies of the CRVs. Thus, it is of interest to develop strategies to control the formation and strength of these vortices in a way that leads to more effective film cooling.

There are several ways to address this problem. One way that has been proposed by several investigators is to alter the structure of these vortices. These include alterations by using shaped-diffusion holes and slots (e.g., Haven et al. (1997), Hyams et al. (1997), and Thole et al. (1998)), by judicious placement of vortex generators (Haven & Kurosaka (1996)), by constructing tabs at hole exit (Zaman & Foss (2005) and Zaman (1998)), by inserting struts inside film-cooling holes (Shih, et al. (1999)), and by creating a trench about the exit of each film-cooling hole (Bunker (2002)). An alternative way is to prevent the CRVs from entraining hot gases by downstream treatment, and this has not been reported.

In this paper, flow-aligned blockers (Fig. 6.1) are proposed to minimize the entrainment of hot gases by the CRVs so that film-cooling effectiveness improves without unduly increasing surface heat transfer and pressure loss. Since extended surfaces can increase surface heat transfer and this is undesirable on the hot-gas side, it is noted that the blockers can be constructed in the thermal-barrier coating (TBC) system by using the ceramic top coat, which has very low thermal conductivity (private communication with Bunker (2005)). The objective of this study is twofold. The first is to assess the usefulness of the "blocker" concept in improving the adiabatic effectiveness of film-cooling jets, to examine the nature of the flow induced by the blockers, and to show how they minimize hot-gas entrainment. The second objective is to perform a parametric study to examine the effects of design parameters for a generic blocker. This study will be accomplished by using
computational fluid dynamics (CFD) analysis that accounts for the three-dimensional nature of the flow and resolves the hot gas and film-cooling jet interactions above the plate as well as the flow in the plenum and in the film-cooling holes.

6.3 Problem Description

To demonstrate the usefulness of flow-aligned blockers to improve film-cooling effectiveness, the problem of film-cooling of a flat plate from a row of inclined circular holes is studied. The problem selected is similar to the experimental study of Kohli & Bogard (1995) so that the meaningfulness of this computational study can be assessed by comparing the CFD predictions with the measurements.

Schematic diagrams of the film-cooling problem studied with and without blockers are shown in Figs. 6.1 to 6.3. For this problem, the cooling jets emerge from a plenum through one row of circular holes. Each hole has a diameter \( D \) of 12.7 mm, a length of 3.5\( D \), and an inclination of 35° relative to the flat plate. The spacing between the centers of the film-cooling holes in the spanwise direction is 3\( D \). Since the film cooling is for a flat plate in which the cooling jet emerges from a row of inclined holes, the flow-aligned blockers are taken to be pairs of parallel ribs or fence-like protrusions from the flat plate with rectangular cross sections as shown in Figs. 6.1 and 6.3. These “rectangular prism” blockers are located 1\( D \) downstream of the film-cooling hole. Each blocker has height \( b \) and thickness \( c \), and separated by a distance \( a \). Three values of \( a \), \( b \), and \( c \) were examined, and they are as follows: 0.8\( D \), \( D \), and 1.2\( D \) for the spacing \( a \); 0.2\( D \), 0.4\( D \), and 0.6\( D \) for the height \( b \); and \( D/20 \), \( D/10 \), and \( D/5 \) for the thickness \( c \). Other dimensions that describe the geometry are given in Figs. 6.1 to 6.3.

The operating conditions are as follows. The fluid for the main flow (hot gas) and
coolant is air. The main flow above the flat plate has a freestream temperature $T_{\infty}$ of 298 K and a freestream velocity $U_{\infty}$ of 20 m/s along the $x$-direction. The flow in the boundary-layer is assumed to be turbulent from the leading edge of the flat plate. The coolant has a temperature $T_c$ of 188 K in the plenum. This gives a density ratio $DR$ of 1.6. When the average velocity in the film-cooling holes $U_c$ is 6.25 m/s, the mass flux or blowing ratio $M$ is 0.5. Two other blowing ratios were also studied, 0.37 and 0.65, by varying the velocity at the inflow of the plenum that feeds the film cooling holes.

Two types of boundary conditions were applied on the flat plate for the heat transfer study. When the film-cooling adiabatic effectiveness is sought, the flat plate is made adiabatic. When the surface heat transfer coefficient is sought, the flat plate is maintained at a constant wall temperature $T_w$ of 243 K. All other walls, including the walls of the film-cooling holes and the plenum, are made adiabatic. The back pressure at the outflow boundary above the flat plate is maintained at the standard atmospheric pressure.

For this problem, the computational domain is taken to be the region bounded by the solid lines shown in Figs. 6.2. As can be seen, periodicity is assumed in the spanwise direction so that only one film-cooling hole and one pair of blockers need to be examined. In addition, the “upper channel wall” (i.e., the wall without film-cooling holes) was moved closer to the wall with the film-cooling holes. This was done to reduce the size of the computational domain and hence computational cost. The errors incurred by this are minimized by making the “upper channel wall” sufficiently far away and by making it inviscid (i.e., the velocity there can slip despite the viscous nature of the flow) so that boundary layers will not form there.
6.4 Formulation and Numerical Method of Solution

The problem just described was modeled by the ensemble-averaged continuity, momentum (full compressible Navier-Stokes), and energy equations for a thermally and calorically perfect gas. The effects of turbulence were modeled by using the two-equation realizable k-ε model (Shih et al., 1995). In all cases, the integration of all equations is to the wall (i.e., wall functions are not used).

Solutions to the aforementioned governing equations were obtained by using Version 6.1.18 of the Fluent-UNS code. The following algorithms in Fluent were invoked. Since only steady-state solutions were of interest, the SIMPLE algorithm was used. The fluxes at the cell faces representing advection were interpolated by using second-order upwind differences. The fluxes at the cell faces representing diffusion were interpolated by using second-order central differences. For all computations, iterations were continued until all residuals for all equations plateau to ensure convergence to steady-state has been reached. At convergence, the scaled residuals were always less than $10^{-6}$ for the continuity equation, less than $10^{-6}$ for the three components of the velocity, less than $10^{-8}$ for the energy equation and, and less than $10^{-5}$ for the turbulence quantities.

6.5 Grid-Sensitivity and Validation Study

Accuracy of solutions is strongly dependent upon the quality of the grid system in minimizing grid-induced errors and in resolving the relevant flow physics. In this study, a grid sensitivity study was carried out to determine the appropriate grid. Figure 6.4 illustrates this study for the case without blockers, which involved three grids – the baseline grid with 2.291 million cells, a finer grid with 2.716 million cells (adaptation 1), and a still finer grid with 5.252 million cells (adaptation 2). For the two finer grids, the additional cells were all
concentrated about the film-cooling hole and the hot gas/coolant jet interaction region, where the flow physics is most complicated. From this grid sensitivity study, the baseline grid was found to give essentially the same result for the centerline adiabatic effectiveness as those from adaptation 1 and 2 grids. The relative error in the “average” centerline adiabatic effectiveness is 0.4% when comparing results from the baseline grid with those from the adaptation 2 grid.

The grid systems used for this problem with and without blockers are shown in Fig. 6.5. When there are no blockers, the grid system used employ 2.291 million cells. When there are blockers, the grid systems used has cells that varied from 2.412 million to 2.478 million depending upon the height and thickness of each blocker. Figure 4.24 shows the equiangle skewness is almost closed to 0.1 and less than 0.6 in the histogram. For all grids used, the first grid point away from all viscous walls has a $y^+$ less than unity. Figure 6.10 shows $y^+$ values are less than unity at the first grid point from the wall. Also, the first 5 grid points have $y^+$ values within five. The surface effectivenesses with h-refined grid are shown in Fig. 6.11. Figure 6.11 shows the computed surface effectiveness at 3D downstream of the exit hole and along the centerline, and the calculations for different grids match well.

To assess the meaningfulness of this computational study, the grid-independent solutions generated for the problem of film-cooling over a flat plate were compared with the experimental data provided by Kohli & Bogard (1995) for $L/D = 2.8$. At the leading edge of the film-cooling hole, the computations predicted a boundary-layer thickness of 0.14D, a shape factor of 1.49, and a Reynolds number based on the freestream velocity and momentum thickness of 1,492. The corresponding measured values are 0.12D, 1.48, and 1,100, respectively. This comparison shows that the flow upstream of the film-cooling hole is
predicted reasonably well. Results for the predicted adiabatic effectiveness are shown in Fig. 6.6 along with experimentally measured ones. From this figure, it can be seen that the centerline adiabatic effectiveness is over predicted and that the laterally averaged adiabatic effectiveness is under predicted. This indicates that the realizable k-ε model over predicts normal spreading and under predicts lateral spreading of the cooling jet. Despite this, the trends are predicted correctly. Also, the qualitative features of the flow are captured by the computations. Thus, though the predictions are not accurate quantitatively, they are good enough to discern differences in film-cooling designs.

6.6 Results

As will be shown in this section, the proposed flow-aligned blockers do indeed greatly improve film-cooling adiabatic effectiveness without unduly increasing surface heat transfer or pressure rise. Instead of showing this for one configuration, the results will be presented in the following order. First, a parametric study that uses the Taguchi’s design of experiments (Taguchi, 1978) is described from which an “optimal” blocker design is identified. Then, the nature of the flow field induced by blockers is given for this optimal design. Here, optimal is used loosely since the blocker design considered is confined to be a rectangular prism.

6.6.1 Adiabatic Effectiveness

A Parametric Study via Taguchi’s Design of Experiments For the rectangular-prism blockers shown in Fig. 6.3, the effects of the following three design parameters are sought: 0.8D, D, and 1.2D for the spacing between blockers a; 0.2D, 0.4D, and 0.6D for the height of the blockers b; and D/20, D/10/, and D/5 for the thickness of the blockers c. If a full factorial study is to be performed (i.e., one parameter is varied at a time) to assess the effects of the
three parameters at the three levels, then a total of $3^3$ or 27 simulations will be needed. To reduce the number of simulations needed, the Taguchi fractional factorial (Taguchi, 1978; Dehnad, 1990) is employed, where the number of simulations can be reduced to six. These six simulations are summarized in Table 6.1.

The results of the simulations summarized in Table 6.1 for the adiabatic effectiveness are given in Figs. 6.7 to 6.8. Figure 6.7 gives the average adiabatic effectiveness. From this figure, it can be seen that the averaged adiabatic effectiveness is highest when $a = a_2 = D$, $b = b_3 = 0.6D$, and $c = c_3 = D/5$, which corresponds to run number 3 in Table 6.1. Thus, for the range of the parameters studied, the optimal design has the pair of rectangular prism blockers to be spaced $D$ apart and that each blocker should have a height of $0.6D$ and a thickness of $D/5$. Figure 6.7 also shows that $D$ may indeed be near optimum for the spacing between the blockers. However, optimum values for the height and thickness of the blocker remain unclear since the effects of these two parameters remained monotonic in the range studied. It is anticipated that the optimal height is related to the blowing ratio, and the optimal thickness of each blocker is related to the spacing between film-cooling holes since there is a region between film-cooling holes that are unprotected by film cooling. Thus, a true optimal design even for the simple configuration considered here requires further study.

Figures 6.8 and 6.9 show the computed surface-adiabatic effectiveness. Figure 6.8 shows the centerline and the laterally averaged adiabatic effectiveness for all six runs in Table 6.1 as a function of $X/D$. From this figure, it can readily be seen that all “blockers” investigated greatly improve laterally averaged adiabatic effectiveness. For the “optimal” case studied (run 3), Fig. 6.8(c) shows the blockers to maintain the laterally averaged adiabatic effectiveness at nearly the highest levels from $D$ to $15D$ downstream of the film-
cooling hole. At 15D downstream of the film-cooling hole, blockers improved laterally averaged adiabatic effectiveness by about a factor of two, which is quite significant. Figure 6.9 shows the surface adiabatic effectiveness as a function of Y/D at X/D = 3. From this figure, it can be seen that though the blockers may cause parts of the flat plate from being inadequately cooled, this is not the case. In fact, with the blockers, the adiabatic effectiveness is improved in all regions. One reason is that a part of the film-cooling jet is split by the blocker. Thus, Figs. 6.8 and 6.12 show flow-aligned blockers to be useful in improving film cooling effectiveness.

Figure 6.13 shows the predicted surface heat transfer coefficient on the flat plate without blockers and with the optimal blocker (run 3 configuration). The heat transfer coefficients were computed in three steps. First, simulations were performed with adiabatic walls to obtain the adiabatic surface temperature on the flat plate, $T_{aw}$. Next, computations were performed for the same configuration and operating conditions except the flat plate is maintained at a constant wall temperature $T_w$ of 243K to predict surface heat transfer per unit area, $q_w$. Then, the heat transfer coefficient $h$ is computed by $q_w / (T_w - T_{aw})$. From Fig. 6.13, it can be seen that the blockers increase surface heat transfer slightly near its leading-edge, but reduces surface heat transfer downstream of the blockers. The slight increase in surface heat transfer at the leading edge of the blocker may not be significant since the adiabatic effectiveness is high there. The reduced surface heat transfer downstream of the blockers resulted from reduced temperature gradients that arose from less hot gas entrainment. The average heat transfer rate per unit area for the entire flat plate with and without rectangular-prism blockers is -781.34 W/m², and -1094.70 W/m², respectively. The average heat transfer coefficient with and without rectangular-prism blockers is 24.40 W/m²·K and 25.02 W/m²·K,
respectively. Thus, in general the blockers studies were found to reduce surface heat transfer instead of increasing them. This also means that the extended surface due to blockers may not be a concern.

Though surface heat transfer was not increased by the blockers, computed results show that there is non-negligible pressure rise. When there are no blockers, the average pressure drop from the inflow to the outflow boundary above the flat plate is 10.66 Pa. When there are rectangular blockers, it increases to 16.07 Pa. This represents an increase of 5.41 Pa or 51%, which is considerable. The magnitude of the average shear stress for the flat plate without blockers is 1.14 Pa. The magnitude of the average shear stress for the case with blockers that include the shear stress on the flat plate and on the blockers is 0.95 Pa. This indicates the rise in pressure loss from the blockers is due to pressure of the leading and trailing edges instead of from shear. Thus, one way to reduce this pressure rise is to streamline the leading and the trailing edges. For example, instead of the flat leading and trailing faces as shown in Figs. 6.1 and 6.3, they can be rounded at the leading edge and pointed at the trailing edge, similar to that of an airfoil.

### 6.6.2 Nature of the Flow

With an "optimal" blocker design identified from the range of the design parameters investigated, this section examines how this blocker \((a = D, b = 0.6D, \text{ and } c = D/5)\) minimizes hot gas entrainment and thereby increase film-cooling adiabatic effectiveness. Figure 11 shows normalized temperature \((T_o-T)/(T_o-T_c)\) at two Y-Z planes, one located at \(X/D = 3\) and one at \(X/D = 7\) in which the blowing ratio is \(M = 0.5\) with and without blockers. From this figure, it can be seen that the two blockers confine the cooling flow within it and prevents the entrainment of hot gases. By \(X/D = 7\), the coolant is fairly well mixed along the
spanwise Y direction so that the temperature variation is mostly along Z. Since the blockers are placed D downstream of the film-cooling-hole exit, the cooling flow also wraps around the "outer" sides of the blockers. Thus, cooling extends beyond the blockers by as much as 0.2D beyond the blockers. This, of course, improved the film-cooling effectiveness outside of the blockers as shown in Fig. 6.12.

To further examine the usefulness of this blocker, simulations were done with slightly lower and slightly higher blowing ratios for the same blocker geometry (run 3). Results of these simulations are shown in Figs. 6.15 and 6.16. In these two figures, it can be seen that even with a blowing ratio of $M = 0.37$, laterally averaged adiabatic effectiveness is still quite respectable when there are blockers. With a lower blowing ratio, the cooling flow rate is less and so the wrap-around about the blockers is reduced. When the blowing ratio increases to $M = 0.65$, the laterally averaged adiabatic effectiveness improves further. With higher blowing ratio, more of the coolant spills over and around the blockers.

6.7 Summary

This study proposes a new design concept, referred to as "flow-aligned blockers" to increase the adiabatic effectiveness of film-cooling jets by minimizing hot-gas entrainment without unduly increasing surface heat transfer and pressure loss. Numerical simulations based on the compressible Navier-Stokes equations were performed to investigate the usefulness of a blocker geometry that has a rectangular cross section. A parametric study based on the Taguchi's method was used to examine the effects of three parameters: spacing between blockers, height of blockers, and the thickness of each blocker. A limited study on the effects of blowing ratio was also carried out. Results obtained show that the blockers studied are highly effective in preventing hot-gas entrainment and can increase adiabatic
effectiveness significantly by confining the coolant flow between the blockers. For the blockers studied, the laterally averaged adiabatic effectiveness at 15D downstream of the film-cooling hole can be as high as that at 1D downstream. The blockers studied were found to increase surface heat transfer only slightly in the region about the leading edge of the blockers. Downstream of the blockers, surface heat transfer was reduced. There is, however, some rise in pressure loss because of the flat leading and trailing edges, indicating a need for streamlining there.

Acknowledgement

This research was supported by the Department of Energy’s University Turbine System Research Program with Dr. R. Wenglarz and Dr. L.P. Golan as the Technical Monitors (SCIES Project 04-01-SR115). The authors are grateful for this support. The authors are also grateful to Fluent for providing the Fluent-UNS code.

Table 6.1. Summary of Runs of the Taguchi’s Study

<table>
<thead>
<tr>
<th>Run No.</th>
<th>a</th>
<th>b</th>
<th>c</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>a1 = 0.8D</td>
<td>b1 = 0.2D</td>
<td>c1 = D/20</td>
</tr>
<tr>
<td>2</td>
<td>a1</td>
<td>b2 = 0.4D</td>
<td>c2 = D/10</td>
</tr>
<tr>
<td>3</td>
<td>a2 = D</td>
<td>b3 = 0.6D</td>
<td>c3 = D/5</td>
</tr>
<tr>
<td>4</td>
<td>a2</td>
<td>b1</td>
<td>c2</td>
</tr>
<tr>
<td>5</td>
<td>a3 = 1.2D</td>
<td>b2</td>
<td>c3</td>
</tr>
<tr>
<td>6</td>
<td>a3</td>
<td>b3</td>
<td>c1</td>
</tr>
</tbody>
</table>
Figure 6.1. Schematic of blockers downstream of a film-cooling hole.

Figure 6.2. Schematic of film cooling of a flat plate from a row of inclined circular holes (not drawn to scale).
Figure 6.3. Schematic of "rectangular prism" blockers.

Figure 6.4. Grid-independent study: centerline adiabatic effectiveness for three grids.
Figure 6.5. Grid systems used. (a) No blockers. (b) With blockers. (c) Grid around film-cooling hole (top view). (d) Grid around film-cooling hole through center of hole.
Figure 6.6. Validation study: CFD predictions and comparison with experimental data of Kohli & Bogard (1995) ($L/D = 2.8$ to match experiment). (a) Laterally averaged. (b) Centerline.
Figure 6.7. Effects of a, b, and c on average adiabatic effectiveness.
Figure 6.8. Surface adiabatic effectiveness with and without blockers ($M = 0.5$). (a) Centerline adiabatic effectiveness for all 6 runs in Table 1. (b) Laterally averaged adiabatic effectiveness for all six runs in Table 1. (c) Laterally averaged adiabatic effectiveness for the optimal blocker, run 3 in Table 1.
Figure 6.9. Histogram of equiangle skewness values for the case 3 in the blockers.

Figure 6.10. The distributions of the y+ values for the first cell away from the flat plate with the ramp.
Figure 6.11. Grid-independent study for blockers: (a) adiabatic effectivenesses at $X/D = 3$ in the spanwise direction. (b) Centerline adiabatic effectiveness.
Figure 6.12. Surface adiabatic effectiveness with and without blockers with $M = 0.5$ at $X/D = 3$. (a) Region outside of blockers. (b) On top of blockers.
Figure 6.13. Predicted surface heat transfer coefficient (W/m$^2$-K). (a) No blockers. (b) With rectangular-prism blockers.

Figure 6.14. Normalized temperature ($T_{o}-T$)/($T_{o}-T_{c}$) at Y-Z planes located at X/D = 3 and 7 for $M = 0.5$. (a) No blockers. (b) With blockers (Run 3).
Figure 6.15. Laterally averaged adiabatic effectiveness for several blowing ratios. Baseline has blowing ratio of 0.5

Figure 6.16. Normalized temperature \((T_o - T) / (T_o - T_c)\) at Y-Z planes located at X/D = 3 and 7. (a) M = 0.37. (b) M = 0.65.
CHAPTER 7. INCREASING ADIABATIC FILM-COOLING EFFECTIVENESS BY USING AN UPSTREAM RAMP

A paper to be submitted to The Journal of Heat Transfer

S. Na and T. I-P. Shih

7.1 Abstract

A new design concept is presented to increase the adiabatic effectiveness of film cooling jets without unduly increasing surface heat transfer and pressure loss. Instead of shaping the film-cooling hole at its downstream end as is done for shaped holes, this study proposes a geometry modification upstream of the film-cooling hole to modify the approaching boundary-layer flow and its interaction with the film-cooling jet. Computations, based on the ensemble-averaged Navier-Stokes equations closed by the realizable k-ε turbulence model, were used to examine the usefulness of making the surface just upstream of the film-cooling hole into a ramp with backward-facing step. The effects of the following parameters were investigated: angle of the ramp (8.5°, 10°, 14°), distance between the backward-facing step of the ramp and the film-cooling hole (0.5D, D), and blowing ratio (0.36, 0.49, 0.56, 0.98). Results obtained show that an upstream ramp with a backward-facing step can greatly increase film-cooling adiabatic effectiveness. The laterally averaged adiabatic effectiveness with ramp can be twice or more the case without the ramp. Also, the ramp increases the surface area that each film-cooling jet protects. However, using the ramp does increase drag. The increase in surface heat transfer was found to be minimal.
7.2 Introduction

To increase thermal efficiency and specific thrust, advanced gas turbine stages are designed to operate at increasingly higher inlet temperatures (Suo, 1985). This increase is made possible by advances in materials such as super alloys and thermal-barrier coatings and by advances in cooling technology such as internal, film, impingement, and other techniques (Suo, 1985; Metzger, 1985; Moffat, 1987). With cooling, inlet temperatures can far exceed allowable material temperatures. Though cooling is an effective way to enable higher inlet temperatures, efficiency considerations demand effective cooling to be accomplished with minimum amount of cooling air since it takes energy to pump the cooling air through the turbine system, which operates at high pressures.

For advanced gas turbines, the first-stage stator and rotor typically requires film cooling, which strives to form a blanket of cooler air next to the material surface to insulate the material from the hot gas (Goldstein, 1971). Many investigators have studied the effects of design and operating parameters on film cooling. These include film-cooling hole inclinations and length-to-diameter ratios, spacing between holes, geometry of holes including shaped holes, surface curvatures, mainflow turbulence, embedded vortices in the mainflow, and unsteadiness from rotator-stator interactions (see, e.g., reviews by Han, et al. (2000), Goldstein (2001), Sundén & Faghri (2001), and Shih & Sultanian (2001); in addition see the comprehensive bibliography provided by Kercher (2003, 2005).

Of the previous studies, Kelso & Lim (1996) and Haven et al. (1997) showed the important role played by vortices in the evolution of film-cooling jets. One pair, referred to as the counter-rotating vortices (CRVs), was found to lift the jet off the surface that it is intended to protect and to entrain hot gases underneath it. The other pair, referred to an anti-
kidney pair, was shown to have a sense of rotation opposite to that of the CRVs, and so can counteract the undesirable tendencies of the CRVs. Thus, it is of interest to develop strategies to control the formation and strength of these vortices in a way that leads to more effective film cooling.

There are many ways to alter the structure of these vorticies. Since the vorticity in the cooling jet originate from the flow in the film-cooling hole, the boundary layer upstream of the film-cooling hole, and the boundary-layer/cooling jet interactions, most investigators have focused on the geometry of the film-cooling hole. Shaped-diffusion hole is one approach. Haven et al. (1997) and Hyams et al. (1996) investigated the effects of shape holes on the vorticity dynamics of mainflow/film-cooling jet interactions. Another approach to alter the vortical structures is via vortex generators. Haven & Kurosaka (1996) investigated the effects of placing vanes inside film-cooling holes that produce vortices in the same sense as the anti-kidney vortices. Zaman & Foss (2005) and Zaman (1998) investigated the effects of tabs placed at the film-cooling-hole exit. These two studies showed vortex generators to be quite effective in reducing jet penetration. However, both studies only investigated jets that lifted-off the surface once exiting the film-cooling holes. Shih, et al. (1999) proposed placing a strut or obstruction within each film-cooling hole that do not necessarily generate appreciable vorticity but can cause vortices inside film-cooling holes to be stretched and tilted in a way that would re-distribute the magnitude of the vortices inside the CRVs and the anti-kidney pair. Bunker (2002) proposed creating a trench about each film-cooling hole exit to modify the boundary-layer/cooling jet interactions. So far, no one has studied modifying the geometry upstream of the film-cooling hole to improve film-cooling effectiveness without unduly increasing surface heat transfer and pressure loss.
In this paper, an "upstream ramp" (Fig. 7.1) is proposed to modify the boundary-layer/cooling jet interaction so that film-cooling effectiveness improves. Since extended surfaces such as a ramp could increase surface heat transfer and this is undesirable on the hot-gas side, it is noted that the ramp can be constructed in the thermal-barrier coating (TBC) system by using the ceramic top coat, which has low thermal conductivity (private communication with Bunker (2005)). The objective of this study is twofold. The first is to demonstrate the usefulness of the "upstream ramp" concept in improving the adiabatic effectiveness of film-cooling jets and examine the nature of flow induced by the upstream ramp without unduly increasing surface heat transfer and drag. The second is to perform a parametric study to examine the effects of design parameters for a generic ramp. This study will be accomplished by using computational fluid dynamics analysis that accounts for the three-dimensional nature of the flow and resolve the flow above the plate as well as the flows in the plenum and in the film-cooling hole.

7.3 Problem Description

To demonstrate the usefulness of an upstream ramp, the problem of film-cooling of a flat plate from a row of inclined circular holes is studied. The baseline problem without the upstream ramp is similar to the experimental study of Kohli & Bogard (1995) so that this computational study can be validated by comparing the CFD predictions with experimentally measured values.

Schematic diagrams of the film-cooling problem studied with and without an upstream ramp are shown in Figs. 7.1 to 7.3. For this problem, the cooling jets emerge from a plenum through one row of circular holes. Each hole has a diameter $D$ of 12.7 mm, a length of $3.5D$, and an inclination of $35^\circ$ relative to a plane tangent to the flat plate. The upstream
ramp studied has length 2D, makes an angle \( \alpha \) with respect to the flat plate, and is located \( \beta \) upstream of the film-cooling hole. The following values of \( \alpha \) and \( \beta \) were examined: 8.53°, 10°, and 14° for \( \alpha \) and 0.5D and D for \( \beta \). Other dimensions that describe the geometry are given in Figs. 7.1 to 7.3.

The operating conditions are as follows. The fluid for the main flow (hot gas) and coolant is air. The main flow above the flat plate has a freestream temperature \( T_\infty \) of 298 K and a freestream velocity \( U_\infty \) of 20 m/s along the x-direction. The flow in the boundary-layer is assumed to be turbulent from the leading edge of the flat plate. The coolant has a temperature \( T_c \) of 188 K in the plenum. This gives a density ratio \( D_R \) of 1.6. When the average velocity in the film-cooling holes \( U_c \) is 6.25 m/s, the mass flux or blowing ratio \( M \) is 0.5. Three other blowing ratios were also studied – 0.36, 0.56, and 0.98 – by varying the velocity at the inflow of the plenum that feeds the film cooling holes.

Two types of boundary conditions were applied on the flat plate for the heat transfer study. When the film-cooling adiabatic effectiveness is sought, the flat plate is made adiabatic. When the surface heat transfer coefficient is sought, the flat plate is maintained at a constant wall temperature \( T_w \) of 243K. All other walls, including the walls of the film-cooling holes and the plenum, are made adiabatic. The back pressure at the outflow boundary above the flat plate is maintained at the standard atmospheric pressure.

For this problem, the computational domain is taken to be the region bounded by the solid lines shown in Fig. 7.2. As can be seen, periodicity is assumed in the spanwise direction so that only one film-cooling hole need to be examined. In addition, the “upper channel wall” (i.e., the wall without film-cooling holes) was moved closer to the wall with the film-cooling holes. This was done to reduce the size of the computational domain and hence
computational cost. The errors incurred by this are minimized by making the "upper channel wall" sufficiently far away and by making it inviscid (i.e., the velocity there can slip despite the viscous nature of the flow) so that boundary layers will not form there.

7.4 Formulation and Numerical Method of Solution

In this study, the problem just described was modeled by the ensemble-averaged conservation equations of mass (continuity), momentum (full compressible Navier-Stokes), and energy for a thermally and calorically perfect gas. The effects of turbulence were modeled by using the two-equation realizable k-ε model (Shih et al. 1995). In all cases, the integration of all equations is to the wall (i.e., wall functions are not used).

Solutions to the aforementioned governing equations were obtained by using Version 6.1.18 of the Fluent-UNS code. The following algorithms in Fluent were invoked. Since only steady-state solutions were of interest, the SIMPLE algorithm was used. The fluxes at the cell faces representing advection were interpolated by using second-order upwind differences.

The fluxes at the cell faces representing diffusion were interpolated by using second-order central differences. For all computations, iterations were continued until all residuals for all equations plateau to ensure convergence to steady-state has been reached. At convergence, the scaled residuals were always less than $10^{-6}$ for the continuity equation, less than $10^{-6}$ for the three components of the velocity, less than $10^{-8}$ for the energy and, and less than $10^{-5}$ for the turbulence quantities.

7.5 Grid-Sensitivity and Validation Study

Accuracy of solutions is strongly dependent upon the quality of the grid system in minimizing grid-induced errors and in resolving the relevant flow physics. In this study, a grid sensitivity study was carried out to determine the appropriate grid. Figure 7.4 illustrates
this study for the case without blockers, which involved three grids – the baseline grid with 2.291 million cells, a finer grid with 2.716 million cells (adaptation 1), and a still finer grid with 5.252 million cells (adaptation 2). For the two finer grids, the additional cells were all concentrated about the film-cooling hole and the hot gas/coolant jet interaction region, where the flow physics is most complicated. From this grid sensitivity study, the baseline grid was found to give essentially the same result for the centerline adiabatic effectiveness as those from adaptation 1 and 2 grids. The relative error in the "average" centerline adiabatic effectiveness is 0.4% when comparing results from the baseline grid with those from the adaptation 2 grid.

The grid systems used for this problem with and without an upstream ramp are shown in Fig. 7.5. Without the upstream ramp, the grid system used employed 2.291 million cells. When there is an upstream ramp, the grid systems used had cells that varied from 2.282 million to 2.367 million, depending upon the angle of the ramp and distance from the film-cooling hole. For all grids used, the first grid point away from all viscous walls has a $y^+$ less than unity. Also, the first 5 grid points have $y^+$ values within five. Figure 7.6 shows the distributions of the $y^+$ values for the first cell away from the flap plate with the ramp for the case with $M = 0.98$, $\alpha = 14^\circ$, and $\beta = 0.5D$. From this figure, it can be seen the $y^+$ values are all less than unity. The surface effectivenesses with h-refined grid are shown in Figs. 7.7. Figure 7.7 shows the computed surface effectivenesses at 3D downstream of the exit hole and along the centerline, and the calculations for different grids match well.

To assess the meaningfulness of this computational study, the grid-independent solutions generated for the problem of film-cooling over a flat plate were compared with the experimental data provided by Kohli & Bogard (1995) for $L/D = 2.8$. At the leading edge of
the film-cooling hole, the computations predicted a boundary-layer thickness of 0.14D, a
shape factor of 1.49, and a Reynolds number based on the freestream velocity and
momentum thickness of 1,492. The corresponding measured values are 0.12D, 1.48, and
1,100, respectively. This comparison shows that the flow upstream of the film-cooling hole is
predicted reasonably well. Results for the predicted adiabatic effectiveness are shown in Fig.
7.8 along with experimentally measured ones. From this figure, it can be seen that the
centerline adiabatic effectiveness is over predicted and that the laterally averaged adiabatic
effectiveness is under predicted. This indicates that the realizable k-ε model over predicts
normal spreading and under predicts lateral spreading of the cooling jet. Despite this, the
trends are predicted correctly. Also, the qualitative features of the flow are captured by the
computations. Thus, though the predictions are not accurate quantitatively, they are good
enough to discern differences in film-cooling designs.

7.6 Results

Table 7.1 summarizes all cases simulated. Results for all simulations are given in Figs.
7.9 to 7.18. In this section, the nature of the flow induced by the upstream ramp is described
first. Afterwards, its effects on surface adiabatic effectiveness are given.

7.6.1 Nature of the Flow Induced by an Upstream Ramp

Figures 7.9 to 7.12 show the interaction of the approaching boundary-layer flow and
the film-cooling jet with and without the upstream ramp. In Fig. 7.9, it can be seen that when
there is no ramp, there is a significant pressure rise on the flat plate, just upstream of the film-
cooling hole because of approaching boundary-layer/cooling jet interaction. But, when there
is an upstream ramp, the static pressure upstream of the film-cooling hole is quite low. This
reduced pressure near the surface resulted because the boundary-layer flow, being diverted
upwards by the upstream ramp, now interacts with the film-cooling jet at a distance above the flat plate. The flow pattern created by the ramp is shown in Fig. 7.10. From this figure, it can be seen that there is a separated region that extends from the backward-facing step of the ramp to the upstream end of the film-cooling hole. The recirculating flow in this separated region entrains the cooler fluid from the film-cooling jet and cools the wall bounding the separated region. With the approaching boundary-layer deflected upwards, the cooling jet flow more easily through the cooling hole as shown in Fig. 7.11. Without an upstream ramp, the net pressure force on the cooling jet is high. This high pressure causes the jet to bend towards the flat plate and thereby reducing the effective cross-sectional area for film-cooling flow, which increases flow speed of the coolant and the effective blowing ratio. With an upstream ramp, the cooling jet is not deflected by the approaching boundary-layer flow until further above the surface so that the effective cross-sectional area for flow is higher and the effective blowing ratio is less. In Fig. 7.11, it can be seen that the large separated region in the film-cooling hole for the case without ramp essentially disappears for the case with a ramp. Since the approaching boundary layer is made up of hot gas, deflecting it above the film-cooling hole delays the entrainment of hot gases by the counter-rotating vortices in the film-cooling jets. This can be seen in Fig. 7.12. Not shown is that having the boundary-layer flow/cooling jet interactions taking place above the plate weakens the pressure rise and allows the cooling jet to spread out more laterally about the flat plate.

### 7.6.2 Adiabatic Effectiveness

Figures 7.13 to 7.17 show the results obtained for the film-cooling adiabatic effectiveness. From Fig. 7.13, it can be seen that with an upstream ramp, the surface adiabatic effectiveness is greatly improved and that a much greater surface about the film-
cooling hole is now protected by the cooling jet, including the region upstream of the film-cooling hole. This is consistent with the nature of the fluid flow described earlier (e.g., entrainment of coolant downstream of the ramp’s backward facing step by the recirculating flow in the separated region and the increased lateral spreading of the coolant upon exiting the film-cooling hole). Figures 7.14 to 7.16 show the effects of the ramp angle and the distance from the backward-facing step of the ramp to the film-cooling hole. From these figures, the following observations can be made. First, increasing the ramp angle (and hence the height of the backward-facing step), the higher is the laterally averaged adiabatic effectiveness, at least for the two angles studied. Second, placing the ramp 0.5D upstream gave higher laterally averaged adiabatic effectiveness than placing it D upstream. Bottomline is that placing a ramp upstream of a row of film-cooling holes increases laterally averaged adiabatic effectiveness significantly. The laterally averaged adiabatic effectiveness with ramp can be two or more times or higher than that without the ramp.

Figure 7.17 shows the effects of blowing ratio on adiabatic effectiveness. From this figure, it can be seen that laterally averaged adiabatic effectiveness can decrease if the blowing ratio is too high or too low. When it is too high, lift-off can take place. When it is too low, hot gas is entrained behind the backward-facing step. Thus, the optimal angle of the ramp or the height of the backward-facing step depends on the blowing ratio. For a ramp with $\alpha = 8.5^\circ$ that is located at $\beta = D$ upstream of the film-cooling hole, best results are obtained with blowing ratio around 0.5.

### 7.6.3 Heat Transfer

Figure 7.18 shows the predicted surface heat transfer coefficient on the flat plate with and without ramp and with the optimal blocker (Case 4 in Table 7.1). The heat transfer
coefficients were computed in three steps. First, simulations were performed with adiabatic walls to obtain the adiabatic surface temperature on the flat plate, $T_{aw}$. Next, computations were performed for the same configuration and operating conditions except the flat plate is maintained at a constant wall temperature $T_w$ of 243K to predict surface heat transfer per unit area, $q_w$. Then, the heat transfer coefficient $h$ is computed by $q_w / (T_w - T_{aw})$. From Fig. 7.18, it can be seen that the ramp changes substantially regions of high and low heat transfer regions. The average heat transfer rate per unit area for the entire flat plate with and without the ramp (Case 4 in Table 1) is -916.78 W/m$^2$ and -1094.70 W/m$^2$, respectively. The corresponding average heat transfer coefficient with and without the ramp is 27.04 W/m$^2$-K and 25.02 W/m$^2$-K, respectively. Thus, introducing a ramp can increase adiabatic effectiveness without unduly increasing surface heat transfer.

Though surface heat transfer was not increased by the ramp, computed results show that there is non-negligible pressure rise. When the ramp is not present, the average pressure drop from the inflow to the outflow boundary above the flat plate is 10.66 Pa. When there is a ramp (Case 4 in Table 7.1), it increases to 14.92 Pa. This represents an increase of 4.26 Pa or 40%, which is considerable. The magnitude of the average shear stress for the flat plate without ramp is 1.136 Pa. This indicates that the pressure loss is due to pressure drag instead of shear. This pressure drag can be reduced by reducing the height of the backward-facing step.

7.7 Summary

This study proposes placing a ramp upstream of a row of film-cooling holes to increase surface adiabatic effectiveness. By having a ramp upstream of the cooling hole, the approaching boundary-layer flow is deflected away from the surface and the film-cooling jet,
producing the following flow features: (1) the approaching boundary-layer flow/cooling jet interaction occurs further away from the surface and is weakened, (2) the diverted boundary-layer flow serves as a blocker that confines the cooling jet next to the surface and increases its lateral spreading, and (3) the recirculating flow downstream of the backward-facing step of the ramp entrains the coolant to cool the surface bounding it. These flow features were shown to greatly increase the adiabatic effectiveness of the film-cooling jets without unduly increasing surface heat transfer. The laterally averaged adiabatic effectiveness with ramp can be two or more times or higher than that without the ramp. The ramp geometry was found to depend on the blowing ratio. There is, however, some rise in pressure loss created by the pressure drag from the ramp.

ACKNOWLEDGMENT

This research was supported by the Department of Energy’s University Turbine System Research Program with Dr. R. Wenglarz and Dr. L.P. Golan as the Technical Monitors (SCIES Project 04-01-SR115). The authors are grateful for this support. The authors are also grateful to Fluent for providing the Fluent-UNS code.
Table 7.1. Summary of Simulations

<table>
<thead>
<tr>
<th>Case No.</th>
<th>α (deg)</th>
<th>β</th>
<th>M</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>8.53</td>
<td>0.5D</td>
<td>0.49</td>
</tr>
<tr>
<td>2</td>
<td>8.53</td>
<td>D</td>
<td>0.49</td>
</tr>
<tr>
<td>3</td>
<td>10.0</td>
<td>D</td>
<td>0.49</td>
</tr>
<tr>
<td>4</td>
<td>14.0</td>
<td>0.5D</td>
<td>0.49</td>
</tr>
<tr>
<td>5</td>
<td>14.0</td>
<td>D</td>
<td>0.49</td>
</tr>
<tr>
<td>6</td>
<td>8.53</td>
<td>D</td>
<td>0.36</td>
</tr>
<tr>
<td>7</td>
<td>8.53</td>
<td>D</td>
<td>0.56</td>
</tr>
<tr>
<td>8</td>
<td>8.53</td>
<td>D</td>
<td>0.98</td>
</tr>
</tbody>
</table>
Figure 7.1. Schematic of a upstream ramp.

Figure 7.2. Schematic of film cooling of a flat plate from a row of inclined circular holes (not drawn to scale and ramp not inserted).
Figure 7.3. Schematic of the upstream ramp design parameters.

Figure 7.4. Grid-independent study: centerline adiabatic effectiveness for three grids at $M = 0.5$. 
Figure 7.5. Grid systems used. (a) No ramp. (b) With ramp.
Figure 7.6. The distributions of the y+ values for the first cell away from the flat plate without and with the ramp.

Figure 7.7. Grid-independent study for upstream ramp: (a) adiabatic effectivenesses at X/D = 3 in the spanwise direction. (b) Centerline adiabatic effectiveness.
Figure 7.8. Validation study: CFD predictions and comparison with experimental data of Kohli & Bogard (1995) (L/D = 2.8 to match experiment). (a) Laterally averaged. (b) Centerline.
Figure 7.9. Static gage pressure (Pa) on the flat plate about the film-cooling hole ($\alpha = 8.5^\circ$, $\beta = 1D$, and $M = 0.49$). (a) No ramp. (b) With ramp.

Figure 7.10. Streamlines (colored by temperature with red high and blue low) induced by the ramp with $\alpha = 8.5^\circ$, $\beta = 1D$, and $M = 0.49$. 
Figure 7.11. Velocity vectors and streamlines in an X-Z plane that passes through the center of the film-cooling hole. Red denotes higher velocity region. Blue denotes low speed/separated region. (a) baseline, (b) with ramp
Figure 7.12. Normalized temperature \((T_{\infty} - T)/(T_{\infty} - T_c)\) at Y-Z plane located at \(X/D = 3\) (\(M = 0.49, \alpha = 14^\circ\) that is located at \(\beta = 0.5D\)). Left: No ramp. Right: With ramp.

Figure 7.13. Adiabatic effectiveness on the flat plate about the film-cooling hole (\(\alpha = 8.5^\circ, \beta = 1D, \text{ and } M = 0.49\)). (a) No ramp. (b) With ramp.
Figure 7.14. Adiabatic effectiveness with and without ramp. (a) Centerline adiabatic effectiveness. (b) Laterally averaged adiabatic effectiveness.
Figure 7.15. Adiabatic effectiveness for $\beta = D$ and $M = 0.49$. Top: $\alpha = 8.5^\circ$. Bottom: $\alpha = 14^\circ$.

Figure 7.16. Adiabatic effectiveness for $\alpha = 14^\circ$ and $M = 0.49$. Top: $\beta = D$. Bottom: $\beta = 0.5D$. 
Figure 7.17. Adiabatic effectiveness as a function of blowing ratio for a given ramp ($\alpha = 8.5^\circ$, $\beta = D$). (a) Centerline adiabatic effectiveness. (b) Laterally averaged adiabatic effectiveness.
Figure 7.18. Heat transfer coefficients ($M = 0.5$, $\alpha = 14^\circ$ and $\beta = 0.5D$). Top: No ramp. Bottom: With ramp.
CHAPTER 8. INCREASING ADIABATIC EFFECTIVENESS BY USING RAMP AND BLOCKERS

8.1 Grid-Sensitivity

Without the upstream ramp and blockers, the operating conditions and all configurations are exactly same as those of plat flat problem. Figure 8.1 shows the type of grid systems generated for this study. Figure 8.2 shows the distributions of the $y^+$ values for the first cell away from the flat plate with the ramp and blockers for the case of $M = 0.5$. Seen from this figure, $y^+$ values are all less than unity. Figure 8.3 shows the equiangle skewness is closed to 0.1 and less than 0.8 in the histogram. The surface effectivenesses with h-refined grid are shown in Fig. 8.4. Figure 8.4 shows the computed surface effectivenesses at 3D downstream of the exit hole and along the centerline, and the computation for different grids matched well.

The results of simulation are given in Fig. 8.5 to 8.13 and Table 8.1 and 8.2. In Section 8.2, the nature of flow induced by the upstream ramp and blockers is described. In Section 8.3, the effects of the ramp and blockers on surface adiabatic effectiveness are discussed.

8.2 Nature of the Flow

Figure 8.7 to 8.12 show the interaction of the approaching boundary-layer flow and the film-cooling jet with and without upstream ramp and blockers. The previous chapters, CHAPTER 6 and 7, show the upstream ramp and the blockers have the following effects. For the upstream ramp, when there is no ramp, pressure significantly rise just upstream of the film-cooling hole exit on the flat plate because of approaching boundary-layer/cooling jet interaction. However, when there is an upstream ramp, static pressure upstream of the film-
cooling hole exit is quite low. This reduces pressure near the surface because the boundary-layer flow, being diverted upwards by the upstream ramp, interacts with the film-cooling jet at a distance above the flat plate (Fig. 8.8 (c) and (d)). In Fig. 8.12, it is shown that the static pressure upstream of the film-cooling hole has the same trend as that of the upstream ramp and blockers. From Fig. 8.9, when there are no blockers, CRVs are placed in the middle at X = 3D. For the blockers, the two blockers confine the cooling flow within it and prevent the entrainment of hot gases. The upstream ramp and blocker use these two advantages. From Fig. 8.12, pathlines originated from the hole downstream of -0.5d and colored by temperature. In this figure, there is a separated region that extends from the backward-facing step of the ramp to the upstream end of the film-cooling hole exit when the blowing rate is less than 0.5. The recirculating flow in this separated region entrains the cooling fluid from the film-cooling jet and cools the wall bounding the separated region. With the approaching boundary-layer deflected upwards, the cooling jet flows through the cooling hole more easily. At the low mass flux ratio, coolant actively moves and fills the whole region upstream and around exit hole in Fig. 8.8. Thus, the effectiveness is much higher even at relatively small amount of the injected coolant through a hole. When mass flux is much higher ( >0.5), the appearance of “lift-off” is generous for the jet in a cross flow. Figure 8.10 shows that the detached coolant jets are attached onto the surface where the jets meet the blocker. In view of lift-off, introducing blockers solves the problem of lift-off. As a result, the effectiveness is re-increased at 1.5D distance of the exit hole in Fig. 8.6. When the ramp and blockers is not present, the average pressure drop from the inflow to the outflow boundary region is 10.66 Pa. When there are the ramp and blockers, it increases to 19.63 Pa. This represents an increase of almost 100%, which is considerable. The magnitude of the average shear stress
for the flat plate without the ramp and blockers is 1.136 Pa. This indicates that the pressure loss is due to drag instead of shear stress. This pressure drag can be reduced by reducing the height of the backward-facing step and using streamlined blockers.

8.3 Adiabatic Effectiveness

Figure 8.5 and 8.6 show the results obtained for the film-cooling adiabatic effectiveness. Figure 8.5 shows that, when the same inlet velocity condition is given as \( W_e = 0.28 \text{ m/s} \), the laterally averaged effectiveness is dramatically increased as much as 294%, 263%, 237%, and 216% at the location of 5d, 10d, 15d, and 20d downstream of the hole exit, respectively. Without the upstream ramp and blockers, baseline, cooling jet covers only about 1/3 of the computational surface and the values of the effectiveness decrease due to the entrainment of the hot gases and mixing in the near field and wake regions. However, from Fig. 8.7, 8.9, and 8.10, the cooling flows cover whole surface in the spanwise direction and much far downstream of the hole exit in the streamwise direction.

Figure 8.6 shows the laterally average effectiveness with different mass flux ratio. From the computations, the surface effectiveness increases as the mass flux ratio approaches 0.5. The effectiveness at \( M = 0.24 \) is also higher than baseline. That is, it is highly useful to raise the efficiency of the turbine because increasing mass flux of the coolant is one of factors reducing the performance of the highly efficient turbine. From a different point of view, this figure shows much higher effectiveness at mass flux ratio of 1.0. That is, even with strong lift-off, the local and whole surface effectiveness do not drop and is much higher due to attachment of cooling jets.

8.4 Summary

This study proposes flow-aligned blockers to minimize hot-gas entrainment and a
ramp upstream of a row of film-cooling holes to increase lateral spread of coolant. By presenting a ramp upstream of the cooling hole, the approaching boundary-layer flow is deflected away from the surface and the film-cooling jet. Recirculating flow downstream of the backward-facing step of the ramp entrains the coolant to cool the surface bounding it. And these flows laterally spread and fill whole surface upstream of the hole exit. These flow features resulted in great increase of the adiabatic effectiveness of the film-cooling without unduly increasing surface heat transfer.

By placing blockers, the detached coolant jets are attached onto the surface where the jets meet the blocker. Even when there is lift-off at the high blowing rate, $M = 1.0$, the laterally averaged adiabatic effectiveness at $15D$ downstream of the hole exit is as high as that at $1D$ downstream. The laterally averaged adiabatic effectiveness with ramp and blockers can be two or more times higher than that without the ramp and blockers.
Table 8.1. Surface average values for the different configurations at the same inlet velocity conditions.

<table>
<thead>
<tr>
<th></th>
<th>$M$</th>
<th>$P_{in}$ (pa)</th>
<th>$\tau_w$ (pa)</th>
<th>$T_{aw}$ (K)</th>
<th>$q_{w''}$ (W/m$^2$)</th>
<th>$h$ (W/m$^2$-k)</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>0.5</td>
<td>10.66</td>
<td>1.136</td>
<td>286.76</td>
<td>-1094.70</td>
<td>25.02</td>
</tr>
<tr>
<td>blockers</td>
<td>0.5</td>
<td>16.07</td>
<td>0.952</td>
<td>275.05</td>
<td>-781.34</td>
<td>24.10</td>
</tr>
<tr>
<td>ramp</td>
<td>0.49</td>
<td>14.92</td>
<td>0.878</td>
<td>276.90</td>
<td>-910.78</td>
<td>27.04</td>
</tr>
<tr>
<td>ramp &amp; blockers</td>
<td>0.46</td>
<td>19.63</td>
<td>0.705</td>
<td>271.28</td>
<td>-688.09</td>
<td>24.33</td>
</tr>
</tbody>
</table>

Table 8.2. Surface average values for the ramp & blockers at the different mass flux ratios.

<table>
<thead>
<tr>
<th></th>
<th>$M$</th>
<th>$P_{in}$ (pa)</th>
<th>$\tau_w$ (pa)</th>
<th>$T_{aw}$ (K)</th>
<th>$q_{w''}$ (W/m$^2$)</th>
<th>$h$ (W/m$^2$-k)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.24</td>
<td>16.92</td>
<td>0.736</td>
<td>281.73</td>
<td>-879.64</td>
<td>22.71</td>
</tr>
<tr>
<td>ramp &amp; blockers</td>
<td>0.46</td>
<td>19.63</td>
<td>0.705</td>
<td>271.28</td>
<td>-688.09</td>
<td>24.33</td>
</tr>
<tr>
<td></td>
<td>1.00</td>
<td>24.26</td>
<td>0.810</td>
<td>268.73</td>
<td>-634.74</td>
<td>24.67</td>
</tr>
</tbody>
</table>
Figure 8.1. Grid systems used for the ramp and blockers.
Figure 8.2. The distributions of the $y+$ values for the first cell away from the flat plate without and with the ramp and blockers.

Figure 8.3. Histogram of equiangle skewness values for the ramp and blockers.
Figure 8.4. Grid-independent study for the ramp and blockers: (a) adiabatic effectivenesses at $X/D = 3$ in the spanwise direction, (b) centerline adiabatic effectiveness
Figure 8.5. CFD comparisons of the cases without and with the ramp and blockers, blockers, and ramp with laterally adiabatic effectiveness at the same velocity inlet boundary condition, $W_c = 0.28\text{m/s}$.

Figure 8.6. CFD comparisons of the ramp and blockers with laterally adiabatic effectiveness at the different blowing ratios, $M = 0.24, 0.46, 1.0$. 
Figure 8.7. Comparison of surface adiabatic effectiveness at the same velocity inlet boundary condition, $W_c = 0.28\text{m/s}$. (a) without the ramp and blockers. (b) with blockers. (c) with ramp. (d) with the ramp and blockers.
Figure 8.8. Comparison of static gage pressure (Pa) at the same velocity inlet boundary condition, $W_c = 0.28\text{m/s}$. (a) without the ramp and blockers. (b) with blockers. (c) with ramp. (d) with the ramp and blockers.
Figure 8.9. Comparison of normalized adiabatic effectiveness and velocity vectors at the same velocity inlet boundary condition, $W_c = 0.28$ m/s, at $X/D = 3$. (a) left: without the ramp and blockers, right: with blockers. (b) left: with ramp, right : with the ramp and blockers. (c) left: the magnified view of the with blockers, right: : the magnified view of the with the ramp and blockers.
Figure 8.10. CFD comparisons of the ramp and blockers with surface adiabatic effectiveness at the different blowing ratios, (a) $M = 0.24$. (b) $M = 0.46$. (c) $M = 1.0$.

Figure 8.11. CFD comparisons of the ramp and blockers with static gage pressure (Pa) at the different blowing ratios, (a) $M = 0.24$. (b) $M = 0.46$. (c) $M = 1.0$. 
Figure 8.12. CFD comparisons of the ramp and blockers with pathlines at the different blowing ratios, (a) $M = 0.24$. (b) $M = 0.46$. (c) $M = 1.0$. 
CHAPTER 9. A NEW SHAPED HOLE

9.1 Grid-Sensitivity

The problem in the case without shaped hole is identical with that of flat plate. The operating conditions and all configurations without shaped hole are the same as flat plate problem. The sensitivity, quality, and validation without shaped hole are also the same as that of flat plate problem. Figure 9.1 shows the type of grid systems generated for this study. Figure 9.2 shows $y^+$ values for the first cell away from the flat plate with the ramp for the case of $M = 0.5$. All of the $y^+$ values are less than unity. In Fig. 9.3, the equiangle skewness is close to unity and less than 0.8. The surface effectivenesses with h-refined grid are shown in Fig. 9.4. Figure 9.4 shows the computed surface effectivenesses at 3D downstream of the hole exit and along the centerline, and the calculations with different grids match well.

The results of computation are shown in Figs. 9.5 to 9.11 and Table 9.1. In this section, effects on the nature of the flow induced by the shaped hole are described first. Then surface adiabatic effectiveness is discussed.

9.1 Nature of the Flow

The counter-rotating vortex pairs always appear in the film cooling using jet in a crossflow and entrain the hot gases to the surface, which is a fatal flaw of the method using jet in a cross flow. Previous works are mainly interested in increasing the hole exit size to increase the lateral spread of the cooling jets. From Thole et al.'s (1998) study with expand laterally and/or forward near the exit of the hole, the hole expansion causes separation in the hole and inefficient diffusion, while this shape improved the uniformity of the film over the surface compared with cylindrical holes. In the present work, the size of the exit of the hole is increased and placing the cooling flows under the flat surface decreases the entrainment of
the hot gases. The counter-rotating vortex pairs of the jet in a crossflow cause the hot gases to flow from both opposite cross-section ends to the centerline above the flat surface. However, placing the coolant under the flat surface reduces the value of the tangential velocity to the surface in the spanwise direction. The coolant flows facing downward to the surface change the direction to the centerline in the circulation, flowing parallel to the surface and forward to the centerline in the spanwise direction. Then there is momentum deficiency outside the circulation due to changing direction of the coolant flows. This is the mechanism how the hot gases are entrained to supplement the momentum deficiency. With a shaped hole, the cooling flows in the circulation at Y-Z plane also face downward to the flat surface at the region where the winding shaped hole meets the flat surface, but these flows do not change their directions. The momentum deficiency of the cooling flow is smaller than in the case without shaped hole and the amount of the entraining hot gases is also small due to the reduced tangential force of the cooling flows at Y-Z plane. Figures 9.9, 9.10, and 9.11 (b) show that the velocities without shaped hole are stronger right over the surface than that of the shaped hole at X=7D. From Fig. 9.7 and 9.8, the shaped hole of short length shows the mixing at the end of the shaped hole, X = 2D, and the adiabatic effectiveness slightly decreases in Figs. 9.8 (b) and 9.9 (b) because of the entrainment of the hot gases. When the distance (H) between the ridge of the shaped hole and the flat surface is zero, the high effectiveness results from the small mixing by the counter-rotating vortices. It is because the postulated ridge at the center acts as a block when the entrained hot gases flow from both outside to the center from Fig. 9.10.

When there is no shaped hole, the average pressure drop from the inflow to the outflow boundary is 10.66 Pa. When there is shaped hole, the drop is 4.72 Pa. This represents
a decrease of almost 56%, and this is different with other cases such as ramp, blockers. The magnitude of the average shear stress of the flat plate without shaped hole is 1.136 Pa. Pressure drag reduces when shaped hole is long and distance between the ridge and the plat surface increases as shown in Table 9.1.

9.2 Adiabatic Effectiveness

Figure 9.5 and 9.6 show the case of the long shaped hole with no distance between the flat surface and the ridge in the middle of the cross-section, $H = 0$, and baseline; the laterally averaged effectiveness increases from 50% to 100% up to 15D downstream of the hole exit when the shaped hole is introduced, but there are almost no differences in the values of the centerline effectiveness. The effects of film cooling follow the order of the case with shaped hole (i.e., short shaped hole), with long shaped hole with a distance (i.e., long shaped hole and 0.4D distance from the flat surface and a top between two symmetric valleys), and with long shaped hole with zero height of the distance (i.e., long shaped hole and zero distance from the flat surface and the top of the shaped hole between two symmetric valleys). Figure 9.9 (b) and Fig. 9.11 (a) show that the width of the cooling jet covering the concerned surface is over 2D when the short shaped hole is placed, while only 1D of width is covered when there is no shaped hole. As a result, the coverage of the cooling flow is enlarged, and thus the lateral averaged effectiveness increases even though the centerline effectiveness is constant. When the cross-section of this shaped hole stretches out along the downstream surface and to the end of the computational domain, the cooling jets cover the region of the shaped hole with small entrainment of the hot gases as shown in Figs. 9.8, 9.9, and 9.11 (b).

9.3 Summary

This study proposes flow-aligned shaped hole under the surface to minimize hot-gas
entrainments and to increase the exit size of the hole. The pressure drop from the inflow to the outflow boundary is only 4.72 Pa. This represents a decrease of almost 56% compared with the baseline, the case without shaped hole, and this is different from the results of the other cases such as ramp and blockers. By placing shaped hole, the laterally averaged effectiveness increases from 50% to 100% up to 15D downstream of the hole exit when shaped hole is introduced, but the values of the centerline effectiveness constantly maintain its values with shaped hole. The coverage of the cooling flows is close to 2D laterally when shaped hole is placed, but that is only 1D when there is no shaped hole. That is, the coverage of the cooling flow is expanded, and thus the lateral averaged effectiveness increases even when the centerline effectiveness is constant.

Table 9.1. Surface average values for the shaped hole at the different mass flux ratios

<table>
<thead>
<tr>
<th></th>
<th>M</th>
<th>$P_{in}$ (pa)</th>
<th>$t_{w}$ (pa)</th>
<th>$T_{aw}$ (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>baseline</td>
<td>0.49</td>
<td>10.66</td>
<td>1.136</td>
<td>286.76</td>
</tr>
<tr>
<td>Short 0.4D</td>
<td>0.49</td>
<td>11.10</td>
<td>1.103</td>
<td>283.57</td>
</tr>
<tr>
<td>Long 0.4D</td>
<td>0.49</td>
<td>4.72</td>
<td>1.008</td>
<td>281.26</td>
</tr>
<tr>
<td>Long 0D</td>
<td>0.49</td>
<td>6.43</td>
<td>0.968</td>
<td>278.73</td>
</tr>
</tbody>
</table>
Figure 9.1. Grid systems used for a new shaped hole: (a) top view, (b) side view.
Figure 9.2. The distributions of the y+ values for the first cell away from the flat plate without and with the shaped hole.

Figure 9.3. Histogram of equiangle skewness values for the shaped hole.
Figure 9.4. Grid-independent study for upstream shaped hole: (a) adiabatic effectivenesses at $X/D = 3$ in the spanwise direction. (b) Centerline adiabatic effectiveness.
Figure 9.5. CFD comparisons of laterally adiabatic effectiveness at the same velocity inlet boundary condition, $W_c = 0.28\text{m/s}$.

Figure 9.6. CFD comparisons of centerline effectiveness at the same velocity inlet boundary condition, $W_c = 0.28\text{m/s}$.
Figure 9.7. Comparisons of pressure distributions with different configurations at the same velocity inlet boundary condition, \( W_c = 0.28 \text{m/s} \). (a) without shaped hole. (b) short shaped hole with a length of 2D and a distance of 0.4D between the flat surface and a top. (c) long shaped hole and a distance of 0.4D between the flat surface and a top. (d) long shaped hole and zero distance between the flat surface and a top.
Figure 9.8. Comparisons of surface effectiveness distributions with different configurations at the same velocity inlet boundary condition, $W_c = 0.28$ m/s. (a) without shaped hole. (b) short shaped hole with a length of 2D and a distance of 0.4D between the flat surface and a top. (c) long shaped hole and a distance of 0.4D between the flat surface and a top. (d) long shaped hole and zero distance between the flat surface and a top.
Figure 9.9. Comparisons of normalized effectiveness and velocity vectors at $X/D = 3$ and $7$ at the same velocity inlet boundary condition, $W_c = 0.28$ m/s. (a) without shaped hole. (b) short shaped hole with a length of $2D$ and a distance of $0.4D$ between the flat surface and a top. (c) long shaped hole and a distance of $0.4D$ between the flat surface and a top.
Figure 9.10. Comparisons of normalized effectiveness and velocity vectors at $X/D = 3$ and $7$ at the same velocity inlet boundary condition, $W_c = 0.28 \text{m/s}$. (a) long shaped hole and a distance of $0.4D$ between the flat surface and a top. (b) long shaped hole and zero distance between the flat surface and a top.
Figure 9.11. Comparisons of normalized effectiveness and velocity vectors at $X/D = 3$ and 7 at the same velocity inlet boundary condition, $W_c = 0.28\text{m/s}$. (a) without and with a short shaped hole. (b) without and with a long shaped hole.
CHAPTER 10. CONCLUSION

Film cooling is important and a widely used method to protect from extremely harsh environment by creating an insulating layer. But film cooling often lifts-off over the surface and entrains the hot gases. In this study, we developed and evaluated four new design paradigms, which are blocker, ramp, shaped hole, and ramp and blocker. The effects of TBC blockage and TBC roughness on the film cooling are also studied because these are one of the main reason increasing heat transfer on the surface. Because this study is on the basis of computational studies, we assessed the computations with validation, grid sensitivity, grid quality, appropriateness of turbulence modeling.

10.1 Four New Film Cooling Design Paradigms

"Flow-aligned blockers” increase the adiabatic effectiveness of film-cooling jets by minimizing hot-gas entrainment without unduly increasing surface heat transfer and pressure loss. The results show that blockers have almost constant effectiveness of 0.3 at the region placed by them (i.e., 1D downstream of the exit hole to the end of the computation domain).

By introducing a ramp at 0.5D upstream of the cooling hole, the approaching boundary-layer flow is deflected away from the surface and the film-cooling jet, producing the following flow features: (1) the approaching boundary-layer flow/cooling jet interaction occurs further away from the surface and decreases, (2) the diverted boundary-layer flow serves as a blocker that confines the cooling jet next to the surface and increases its lateral spreading, and (3) the recirculating flow downstream of the backward-facing step of the ramp entrains the coolant to cool the surface bounding it. These flow features showed the increase of the adiabatic effectiveness of the film-cooling without unduly increasing surface heat transfer. The laterally averaged adiabatic effectiveness with ramp was twice or more the case
without the ramp.

The results from proposing the upstream ramp and the flow-aligned blockers showed features of upstream ramp and downstream blockers, increasing lateral spread and minimizing the entrainment of the hot gases, respectively. The laterally averaged effectiveness is dramatically increased as much as 294%, 263%, 237%, and 216% at the location of 5d, 10d, 15d, and 20d downstream of the exit hole, respectively. From the case without upstream ramp and blockers, baseline, cooling jet covers only about 1/3 of the computational surface but the cooling flows cover whole surface in the spanwise direction and much far downstream of the exit hole in the streamwise direction.

Momentum-preserved shaped hole is proposed to minimize hot-gas entrainments and to increase exit hole size. The result shows that it increases surface adiabatic effectiveness from 50% to 100% with increased coverage of the cooling flow. By presenting a shaped hole, the entrainment of the hot gases reduces and the heat transfer decreases by mixing of cooling flow and hot gases in the wake region. In this study, the pressure drop from the inflow to the outflow boundary above the whole surface is only 4.72 Pa. This represents a decrease of almost 56% compared to the baseline (i.e., the case without shaped hole) and this is different from the other cases such as ramp or blockers.

10.2 Effects of TBC Blockage and Roughness on the Film Cooling

The results show that: when mass-flux ratio was fixed at 0.5, TBC blockage reduced adiabatic effectiveness because of the decrease of the amount of coolant flow through the cooling hole. However, the amount of decrease in adiabatic effectiveness was less than the expected amount of decrease based on the amount of reduced coolant flow. The better performance in the TBC blocked film-cooling hole is, in part, interpreted as the function of
the step created by the TBC blockage at the upstream side of the film-cooling hole exit. The step affected how the approaching boundary layer flow interacts with the film-cooling jet. Surface roughness was found to increase displacement thickness and hence is an additional source of blockage within the film-cooling hole.

For future study, because the pressure drops are relatively large, designs reducing pressure drops need to be studied further. In order to predict more accurate computation, turbulence models using $v^2f$ or LES need to be considered. With the predicted results, the model needs to be tested with experimental studies.
BIBLIOGRAPHY


1787.


ACKNOWLEDGEMENTS

I would firstly like to express my gratitude to my advisor, Dr. Tom I-P. Shih for his guidance and support. This thesis is a much work better thanks to his supervision. Beside his academic guidance, I offer him my heartfelt thanks for his generosity, patience, encouragement, and good advice.

My thanks must also go to Dr. Richard H. Pletcher, Dr. Francine Battaglia, Dr. Alric Rothmayer, and Dr. Z. J. Wang for taking the time to serve my graduate committee.

This research was supported by the Department of Energy’s University Turbine System Research Program with Dr. R. Wenglarz and Dr. L.P. Golan as the Technical Monitors (SCIES Project 04-01-SR115). I am grateful for this support. I am also grateful to Fluent for providing the Fluent-UNS code.

I have to thanks to Dr. Bin Zhu and the wonderful supporters of my department, Delora Pfeiffer, Laurie Hoifeldt, Ann Laws, and Jim Wellman.

I also thanks to my friends, Yoo Jin Kwon, Seong-Wook Yoon, and Shourya Otta, for their companionship.

Finally, I like to thank to my parents and brothers and sisters for their love and support.