UNIVERSITY OF CINCINNATI

Date: _____

I, _____,
hereby submit this work as part of the requirements for the degree of:

in:

It is entitled:

This work and its defense approved by:

Chair: _____

Single-Hole Film Cooling on a Turbine-Blade Leading-Edge Model

A thesis submitted to the
Graduate School of the University of Cincinnati
In partial fulfillment of the requirement for the degree of
Master of Science
In the Department of Mechanical Engineering
of the College of Engineering

by

SUMAN MISHRA

B. Tech., Mechanical Engineering
Regional Engineering College, Warangal, India

March 10, 2008

Committee Chair. Dr. Urmila Ghia

Abstract

The present study numerically investigates the effectiveness of film cooling on a turbine blade leading-edge model through a single-hole coolant exit. The model used in this study has the same dimensions as that of an earlier experimental investigation by Ekkad et al., 2006. The cylindrical model, a half cylinder with a flat after-body attached to it, provides a good representation of the leading edge of a turbine blade. The coolant hole is situated approximately at the center of the cylindrical model along the spanwise direction and makes an angle of 21.5 degrees to the true leading edge and 20 degrees to the spanwise direction. A multi-block grid is generated using GridGen, and the flow is simulated using the flow solver FLUENT. A highly clustered unstructured grid is developed around the leading edge of the model with an initial Δy corresponding to a y^+ of 1. The unstructured domain imitates the wind-tunnel as used in the experimental study of Ekkad et al., 2006, and the leading-edge model is placed at the center of the domain. Blowing ratio, M , is defined as the mass flux ratio of the coolant and the mainstream. Simulations are carried out for different blowing ratios M , ranging from 0.25 to 1.0, with the $k-\epsilon$ realizable turbulence model and the $k-\omega$ SST model. The flow is assumed to have a free-stream turbulence intensity of 0.75%. Additionally, the enhanced wall function approach is used as the near-wall treatment in the computational model for the simulation with $k-\epsilon$ realizable turbulence model. Results obtained indicate an increased film-effectiveness for low blowing ratio in the far downstream region. Also, in the vicinity of the coolant jet exit, it is observed that increase in blowing ratio increases the film cooling effectiveness. It is also concluded that steady RANS-based turbulence models under-predict the interaction between the jet and the mainstream at the jet exit, and the spreading of coolant downstream.

Acknowledgements

I am deeply indebted to my advisor Dr. Urmila Ghia for her support and guidance. This work would not have been possible without her continued help and inspiration. I would like to thank my co-advisor Dr. Kirti N. Ghia for his guidance throughout this research work. I would also like thank Dr. Milind Jog for reviewing my thesis. I am thankful to Dr. Hugh Thornburg of Major Shared Resource Center, Dayton, Ohio, for his help and guidance with some computational aspect of my work. Thanks to my colleagues at Computational Fluid Dynamics Research Laboratory, University of Cincinnati for their help at various times.

My sincere thanks to Dr. Kumar Vemaganti, for providing me with his computational resources. I would like to thank Larry Schartman and Rob Ogden for all their technical support with computational issues. I would also pay my gratitude to Ohio Supercomputing Center for providing computational resources.

Special thanks to my parents, for their continued love and support. Finally, I would like to thank my wife, Sarbani, who has been a great source of strength and motivation throughout this work.

Table of Contents

	<i>Page</i>
<i>Abstract</i>	ii
<i>Acknowledgement</i>	iv
<i>Table of Contents</i>	v
<i>List of Tables</i>	vi
<i>List of Figures</i>	vii
<i>Nomenclature</i>	ix
 Chapter 1 - Introduction	
<i>1.1 Motivation</i>	1
<i>1.2 Scope of Thesis</i>	4
 Chapter 2 - Literature Review	
<i>2.1 Heat Transfer Analysis on a Cylindrical-Leading-Edge Model</i>	6
<i>2.2 Leading-Edge Film Cooling</i>	7
 Chapter 3 - Study of Heat Transfer on the Leading-Edge Model - 2D	
<i>3.1 Simulation Geometry and Grid Generation</i>	10
<i>3.2 Turbulence Modeling</i>	16
<i>3.3 Boundary Conditions and Solver Configurations for Heat Transfer Study without the cooling Jet</i>	13
<i>3.4 Boundary Conditions and Solver Configurations for Heat Transfer Study with the cooling Jet</i>	13
 Chapter 4 - Leading-Edge Film Cooling - 3D	
<i>4.1 Simulation Geometry and Grid Generation</i>	21
<i>4.2 Boundary Conditions</i>	23
<i>4.3 Solver Configurations</i>	24
<i>4.4 Turbulence Modeling</i>	25
 Chapter 5 - Results and Discussion	
<i>5.1 Heat Transfer Study without the Cooling Jet - 2D</i>	29
<i>5.2. Leading-Edge Film Cooling – 2D</i>	38

5.3 <i>Leading-Edge Film Cooling - 3D</i>	39
Chapter 6 – Conclusion and Future Work	
6.1 <i>Conclusion</i>	59
6.1 <i>Future Work</i>	61
<i>Rereferences</i>	62

List of Tables

<i>Table 1. List of cases for study of 2D heat transfe on the cylindrical-leading-edge model– Laminar Flow.....</i>	<i>29</i>
<i>Table 2. List of cases for study of 2D heat transfer on the cylindrical-leading-edge model – Turbulent Flow.....</i>	<i>36</i>
<i>Table 3. List of cases for study of 3D film cooling on the cylindrical-leading-edge model</i>	<i>40</i>

List of Figures

<i>Fig 1. Cut-out view of the gas turbine engine.....</i>	<i>1</i>
<i>Fig 2. Film cooling holes on a turbine blade.....</i>	<i>2</i>
<i>Fig 3. The cylindrical leading-edge model.....</i>	<i>2</i>
<i>Fig 4. The cylindrical leading-edge model inside the wind tunnel.....</i>	<i>5</i>
<i>Fig 5. The 2D computational domain.....</i>	<i>5</i>
<i>Fig 6. Close-up view of the 2D computational grid.....</i>	<i>13</i>
<i>Fig 7. The boundary conditions on the 2D computational grid</i>	<i>15</i>
<i>Fig 8. The boundary conditions on the 2D computational grid – Constant-cooling.....</i>	<i>19</i>
<i>Fig 9. Dimensions of the cylindrical leading-edge model – top view.....</i>	<i>20</i>
<i>Fig 10. Dimensions of the cylindrical leading-edge model – right-hand side view</i>	<i>21</i>
<i>Fig 11. The 3D computational grid</i>	<i>22</i>
<i>Fig 12. The 3D computational grid – zoomed-in view of the jet exit region</i>	<i>22</i>
<i>Fig 13. The boundary conditions on the 3D computational grid</i>	<i>23</i>
<i>Fig 14. Opened out view of the leading edge(After Ekkad et al, 2006).....</i>	<i>27</i>
<i>Fig 15. Comparison of Frossling number distribution. Numerical and Experimental Results, Re = 40,000.....</i>	<i>30</i>
<i>Fig 16. Comparison of Frossling number distribution. Numerical and Experimental Results, Re = 60,000.....</i>	<i>31</i>
<i>Fig 17. Comparison of Frossling number distribution. Numerical and Experimental Results, Re = 70,800.....</i>	<i>32</i>
<i>Fig 18. Comparison of Frossling number distribution. Numerical and Experimental Results, Re = 100,000.....</i>	<i>32</i>
<i>Fig 19. Frossling number distribution. FLUENT Results for all Reynolds numbers.....</i>	<i>33</i>

Fig. 20. Comparison of Nusselt number distribution. Numerical and Experimental Results, $Re = 70,800$	34
Fig. 21. Comparison of Nusselt number distribution. Numerical and Experimental Results, $Re = 100,000$	35
Fig. 22. Comparison of Frossling number distribution. Numerical and Experimental Results, $Re = 40,000$, Turbulent flow solution.....	36
Fig. 23. Comparison of Frossling number distribution. Numerical and Experimental Results, $Re = 60,000$, Turbulent flow solution.....	37
Fig. 24. Velocity vectors at the jet exit region – Film cooling – 2D.....	38
Fig. 25. Comparison of film cooling effectiveness – 2D. Numerical and Experimental results.....	39
Fig. 26. Temperature contours, turbulent flow solution ($k-\epsilon$ realizable), $M = 0.25$	42
Fig. 27. Temperature contours, turbulent flow solution ($k-\omega$ SST), $M = 0.25$	43
Fig. 28. Comparison of film cooling effectiveness, $M=0.25$. Numerical and Experimental results.....	43
Fig. 29. Temperature contours, turbulent flow solution ($k-\epsilon$ realizable), $M = 0.50$	44
Fig. 30. Temperature contours, turbulent flow solution ($k-\omega$ SST), $M = 0.50$	45
Fig. 31. Comparison of film cooling effectiveness, $M=0.50$. Numerical and Experimental results.....	45
Fig. 32. Temperature contours, turbulent flow solution ($k-\epsilon$ realizable), $M = 0.75$	46
Fig. 33. Temperature contours, turbulent flow solution ($k-\omega$ SST), $M = 0.75$	47
Fig. 34. Comparison of film cooling effectiveness, $M=0.75$. Numerical and Experimental results.....	47
Fig. 35. Temperature contours, turbulent flow solution ($k-\epsilon$ realizable), $M = 1.0$	49
Fig. 36. Temperature contours, turbulent flow solution ($k-\omega$ SST), $M = 1.0$	49
Fig. 37. Comparison of film cooling effectiveness, $M=1.0$. Numerical and Experimental results	50
Fig. 38. Film cooling effectiveness for all blowing ratios. Experimental.....	51
Fig. 39. Film cooling effectiveness for all blowing ratios. Numerical - $k-\epsilon$ Realizable	52

<i>Fig. 40. Film cooling effectiveness for all blowing ratios. Numerical – $k-\omega$ SST</i>	52
<i>Fig. 41. Comparison of film cooling effectiveness with York et al. 2003, $M=1.0$</i>	53
<i>Fig. 42. Comparison of film cooling effectiveness for different turbulence intensities, $k-\omega$ SST</i>	
<i>turbulence model</i>	54
<i>Fig. 43. Comparison of film cooling effectiveness for different turbulence intensities, $k-\varepsilon$ Realizable</i>	
<i>turbulence model</i>	55

Nomenclature

Latin letters

D	<i>Cylinder diameter (m)</i>
d	<i>Coolant hole diameter (m)</i>
H	<i>Local heat transfer coefficient (W/m²-K)</i>
K	<i>Thermal conductivity (W/m-K)</i>
L	<i>Length of coolant hole (m)</i>
M	<i>Blowing ratio ($\rho_c V_c / \rho_\infty V_\infty$)</i>
Re	<i>Reynolds number</i>
Pr	<i>Prandtl number</i>
X	<i>Stream-wise distance from center of the hole (m)</i>
t	<i>Time</i>
k	<i>Turbulent kinetic energy</i>

Greek Letters

η	<i>Film cooling effectiveness</i>
ω	<i>Specific dissipation</i>
ε	<i>Turbulent dissipation</i>
μ	<i>Dynamic viscosity</i>
ν	<i>Kinematic viscosity</i>

Subscripts

N_u	<i>Nusselt Number</i>
F_r	<i>Frossling number ($\frac{N_u}{\sqrt{Re}}$)</i>

C_p	<i>Specific heat (kJ/kg-K)</i>
T_{aw}	<i>Adiabatic wall temperature (K)</i>
T_f	<i>Film temperature</i>
T_c	<i>Coolant temperature (K)</i>
T_∞	<i>Mainstream temperature (K)</i>
Re_D	<i>Reynolds number based on leading edge diameter</i>
ρ_c	<i>Coolant density</i>
ρ_∞	<i>Mainstream density</i>
μ_t	<i>Eddy viscosity</i>

Chapter 1

Introduction

1.1. Motivation

Gas turbine blades with their prolonged exposure to high temperature gases become vulnerable to failure. Because of high thermal load particularly at the leading-edge, investigation of flow and heat transfer becomes very critical in that region. The present study investigates the effectiveness of film cooling to protect the leading-edge surface. Film cooling is achieved by creating a buffer zone between the surface and the high temperature flow by introduction of a secondary fluid. This secondary fluid zone reduces the heat transfer between the surface and the high temperature mainstream flow and protects the turbine. It also allows a higher inlet temperature to the turbines by protecting the blade surfaces. Gas turbines require a high inlet temperature to achieve increased efficiency. Thus a higher thermal efficiency is achieved through film cooling. The present study concentrates on the leading edge of the turbine blade with a single hole angled at 20 degrees to the spanwise direction and 21.5 degrees to the true leading edge. The spanwise angle increases the lateral momentum of the jet and results in better spreading. The lower stream-wise angle of the coolant hole results in lower lift-off and contributes towards increased contact with the surface.

Fig 1. Cut-out view of the gas turbine engine

Fig 2. Film-cooling holes on a turbine blade

Fig. 3. The cylindrical leading-edge model

The convectional heat transfer from the film to the surface determines the surface temperature. The film effectiveness is defined as

$$\eta = \frac{T_f - T_\infty}{T_c - T_\infty} \dots\dots\dots(1)$$

Where

T_f = film temperature

T_c = Coolant temperature

T_∞ = Mainstream temperature

For an adiabatic surface boundary condition film temperature T_f is equal to the adiabatic surface temperature T_{aw} . The model used in this study is the same as that of an experimental investigation done earlier by Ekkad et al (2006). The cylindrical model, a half cylinder and a flat after-body attached to it, well represents the leading edge of a turbine blade. The half cylinder is 88.9 mm in diameter with a length of 364 mm. The flat after-body is attached to it to constitute the model of a turbine blade leading edge. The coolant hole is situated approximately at the center of the model along the spanwise direction, and makes an angle of 21.5 degrees to the true leading edge and 20 degrees to the spanwise direction. The spanwise angle increases the lateral momentum of the jet and results in better spreading. The stream-wise angle of the coolant hole results in lower lift-off and contributes towards increased contact with the surface. A multi-block grid is generated using GridGen, and the flow is simulated using the flow solver FLUENT. A highly clustered structured grid is developed around the leading edge of the model with an initial Δy corresponding to a y^+ value of 1 and a growth ratio of 1.1 till 15d from the surface. The outer unstructured domain imitates the wind-tunnel used in the

experimental study of Ekkad et al (2006), and the leading-edge model is placed at the center of the domain.

This work is part of a larger study that investigates film cooling of a turbine blade leading-edge model with pulsed blowing. In a typical gas turbine, 20-25% of the mainstream air is consumed for film cooling purposes [Ekkad et al. 2006]. This reduces the thrust of the gas turbine and thereby reduces the efficiency. Pulsed blowing works on an on-and-off cycle of coolant flow where the effective blowing ratio, i.e the time averaged blowing ratio, is less than for the corresponding constant blowing. Thus, pulsed film cooling protects the blade surface which consumes less amount of the mainstream flow and hence results in a higher thrust and efficiency.

1.2 Scope of the Thesis

The overall scope of this work involves investigation of flow and heat transfer on a turbine-blade leading-edge model considering

- ▶ 2D laminar and turbulent flow without coolant injection
- ▶ 2D turbulent flow with coolant injection,
- ▶ 3D turbulent flow with coolant injection

The 2D studies were performed with Reynolds numbers; 40000, 60000, 70800 and 100000. Also, $k-\omega$ SST turbulence model was used to simulate the flow and heat transfer. The blowing ratio, M is defined as the ratio of mass flux of the mainstream and the coolant. It is expressed as

$$M = \frac{\rho_c V_c}{\rho_\infty V_\infty} \quad (2)$$

The 3D film cooling studies were performed with four different blowing ratios ranging from 0.25 to 1.0, with the $k-\varepsilon$ realizable turbulence model and $k-\omega$ SST turbulence model. Additionally, the enhanced wall function approach is used as the near-wall treatment for the the $k-\varepsilon$ Realizable turbulence model. Film cooling effectiveness is computed for various blowing ratios and compared to the experimental results of Ekkad et al. 2006. Also, the work studies the impact of using a turbulence model in heat transfer calculations. It is observed that three-dimensional simulations represent the experimental conditions better and predict more accurate results for film cooling.

Chapter 2

Literature Review

2.1 Heat Transfer Analysis on a Cylindrical-Leading-Edge Model

A review of the literature brings to light a significant number of references for flows with geometries of either a flat plate or a cylinder in cross flow. Research conducted with these geometries includes both experimental and computational approaches to understand the flow and thermal physics of the flow.

Several sources can be cited which describe the boundary layer theory and empirical heat transfer correlations for the flat plate and the cylinder. In the book by Schlichting (2000), the fundamental laminar boundary layer theory is outlined. There are detailed examples and discussion for the two basic geometries, as well as the presentation of data collected for coefficient of pressure, velocity field and Nusselt number. Another source with profound insight and a concise summary of these flows is the book by White (2003). Although this second book focuses on viscous flow more than on heat transfer, it is still a valuable resource for understanding the physics involved in the flow over these geometries. A third fundamental book by Bejan (1972) completes the discussion with regard to the forced-convection heat transfer. This book gives many details on external flow thermal physics. An overview of the correlations between Reynolds number and heat transfer is presented by Whitaker (1972) who refined the correlations to achieve better fit to the reported experimental results.

Considering that the leading edge of the model is a half-cylinder, heat transfer studies on a cylinder-in cross-flow are assumed relevant to compare with as the first step. The most widely referenced results for the cylinder-in-cross-flow are presented by Geidt (1949), and are included in most of the textbooks on the subject. Experimental results of heat transfer are also collected from the dissertation of Ou (1991) and from work by Mick and Mayle (1977) for geometries similar to the geometry used in the present study. Experimental work by Ou (1991) presents results for three different Reynolds numbers; 25,000, 40,000 and 100,000, with a free-stream turbulent intensity of 0.75%. The experimental results obtained from Mick & Mayle (1977), are at a Reynolds number of 100,000, and a free stream turbulence intensity of 0.4%.

2.2 Leading-Edge Film Cooling

Film cooling has been extensively studied over the past decades. A majority of the studies have been done on a flat surface. However, only a handful of studies have worked on film cooling over surface shapes as the one used in this study. Ekkad et al. (2006), investigated film cooling for blowing ratio ranging from 0.25 to 2.0 on the same geometry as used in the present study. For a gas turbine, typically 20-25% of the mainstream air is used for cooling. Ekkad et al. (2006) presented a novel concept of using pulsed cooling that consumes less coolant air, and contributes towards higher thrust for gas turbines. They reported that the duty cycle, i.e., the ratio of coolant valve opening time to its closing time, inversely affects the film cooling effectiveness up to a certain limit. The results obtained show that lowering the effective blowing ratio by reducing the

duty cycle results in higher film effectiveness and reduced heat transfer. Another study by Ekkad et al. (1998) presented the effect of coolant density and free-stream turbulence on the film effectiveness using a transient liquid crystal method. The study reported that a higher-density coolant provides higher film effectiveness for all blowing ratios. Zheng et al. (2000) numerically predicted the film-cooling effectiveness on a quarter-cylindrical model. As in the present study, Zheng et al. (2000) used the flow solver FLUENT to arrive at the heat transfer results with two different turbulence models, the standard two-equation model and RNG two-equation realizable model. Ou et al. (1991) investigated film effectiveness using three rows of holes on a cylindrical model with four different blowing ratios, two different turbulent intensities, 1% and 20%, and two different Reynolds numbers 30,000 and 60,000. They reported increase in film effectiveness and heat transfer coefficient for high turbulence intensities with increase in blowing ratio from 1.0 to 2.0 and decrease thereafter. Mick et al. (1977) performed tests on a turbine blade leading edge model, and investigated the film effectiveness and heat transfer coefficients on the surface for blowing ratios, 0.38, 0.64 and 0.97. The results from this study indicated a lower heat load on the leading edge blade surface for all blowing ratios except 0.97.

A previous paper by Sinha et al. (1991) studied the effectiveness of using cryogenically cooled air on a flat, adiabatic, test surface. They investigated the effect of blowing ratio (M) and momentum ratio on the film cooling effectiveness. This study reported film-cooling effectiveness to be proportional to density ratio and inversely varying with momentum ratio. Both Sinha et al. (1991) and Ekkad et al. (1998) worked on two different test models, and showed that increase in coolant density increases the

film-cooling effectiveness. Another study by Walters et al. (2001) presented a comprehensive numerical investigation of film cooling on a flat surface. The study showed good agreement of film-cooling effectiveness results with the experimental study of Sinha et al. (1991) and utilized the $k-\varepsilon$ turbulence model.

The literatures discussed so far concentrate on heat transfer on a leading-edge model and on study of film cooling effectiveness on a flat plate. With that understanding, the present study investigates film cooling and heat transfer on a cylindrical model. The cylindrical model is a half-cylinder with a flat after-body that imitates the leading edge of the turbine blade. Initially, a 2D heat transfer analysis on a bluff body is performed in a systematic approach to investigate film cooling on a 3D leading-edge model. The 2D simulations were performed with a laminar flow assumption and also, with a turbulent flow assumption using $k-\omega$ SST turbulence model. The 3D film cooling simulations were performed using a turbulent flow assumption using $k-\varepsilon$ Realizable turbulence model and $k-\omega$ SST turbulence model. The results were compared with the study of Ekkad et al. 2006. The next chapter concentrates on 2D heat transfer study on the cylindrical model.

Chapter 3

Study of Heat Transfer on the Leading Edge Model - 2D

This chapter focuses on heat transfer study for the turbine-blade leading-edge model without any film cooling holes. The purpose this study is to predict the local heat transfer on the surface in the form of Frossling number. It also emphasizes on the effectiveness of a turbulence model in predicting heat transfer more accurately than attainable with the laminar flow assumptions. Film cooling effectiveness is also investigated on the two dimensional leading-edge model. Four values of Re , namely, 40000, 60000, 70800 and 100000 are considered and correspond to the values used in the experiments with which the results of the present study are compared.

3.1 Simulation geometry and grid generation

The physical geometry in this study has the same dimensions as in the experimental work by Ou (1991). The model consists of a half-cylinder with a flat plate attached to it. As described earlier, the diameter of the half-cylinder is 88.9 mm, and the flat after-body is 360 mm in length. The mainstream inlet for the simulation is located 720 mm upstream of the center of the half cylinder. The computational domain extends to 400 mm in the $\pm y$ direction from the surface of the leading-edge model to imitate the exact wind tunnel wall locations.

Fig. 4. The cylindrical-leading-edge model inside the wind tunnel.

Fig. 5. The 2D computational domain.

The hybrid grid topology used for the simulations is developed using GridGen. A highly clustered, structured C-grid (Fig. 6) is developed around the leading edge with an initial grid spacing of 0.02 mm and a growth ratio of 1.05 until 100 mm from the surface, i.e., till the end of the C-grid domain. An unstructured grid domain is created between the structured C-grid and the outer rectangular domain boundary. The thermal boundary layer thickness is assumed to be of the same order of magnitude as the viscous boundary layer thickness for air, for which the Prandtl number is close to unity. Thus, the near-surface grid spacing is calculated considering the viscous boundary layer thickness to ensure that a sufficient number of grid points lie inside the boundary layer. The final grid consists of 206,203 grid points.

Fig. 6. Close-up view of the 2D computational grid

3.2 Turbulence Model

For the present simulation $k-\omega$ shear-stress transport (SST) model was employed to predict the heat transfer to the surface. The $k-\omega$ shear-stress transport (SST) model was developed by Menter (1994) to blend the $k-\omega$ model in the near-

wall region with the k - ε model away from the surface [FLUENT documentation]. The k - ω SST model leads to more accurate and reliable in flows involving separation. Flow over present body involves flow separation and k - ω SST turbulence model is therefore employed to predict the flow features and heat transfer with good accuracy. The k - ω SST turbulence model uses the standard k - ω turbulence model in the near-wall region and k - ε model in the freestream. This is achieved by multiplying the standard k - ω model and the k - ε model with a blending function such that the function activates k - ω model at the boundary and the k - ε model away from the surface [FLUENT documentation]. So, it benefits from the accuracy of k - ω turbulence model in the near-wall region and the free-stream-independence of k - ε model away from the surface

As proposed by Menter (1993), the k - ω SST turbulence model uses the following transport equations to simulate the flow field.

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_k \nu_T) \frac{\partial k}{\partial x_j} \right] \quad (2)$$

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha S^2 - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[(\nu + \sigma_\omega \nu_T) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \quad (3)$$

where F_1 is calculated from the following relationship

$$F_1 = \tanh \left\{ \left[\min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500 \nu}{y^2 \omega} \right), \frac{4 \sigma_{\omega^2} k}{CD_{kw} y^2} \right] \right]^4 \right\} \quad (4)$$

and CD_{kw} is

$$CD_{k\omega} = \max\left(2\rho\sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10}\right) \quad (5)$$

The expression for P_k

$$P_k = \min\left(\tau_{ij} \frac{\partial U_i}{\partial x_j}, 20\beta^* k\omega\right) \quad (6)$$

The eddy viscosity is calculated from the following equation

$$\nu_T = \frac{a_1 k}{\max(a_1 \omega, \Omega F_2)} \quad (7)$$

Where $\alpha_1=5/9$, $\alpha_2=0.44$, $\beta_1=3/40$, $\beta_2= 0.0828$, $\beta^*=0.09$, $\sigma_{k1}=0.85$, $\sigma_{k2}=1.0$,
 $\sigma_{\omega 1}=0.5$, $\sigma_{\omega 2}=0.856$

3.3 Boundary Conditions and Solver Formulations for Heat Transfer Study without the Cooling Jet

The computational domain extends to 400 mm on either side, which is about 5 times the diameter of the half-cylinder. These boundary locations are sufficiently far away from the body, so that slip-wall boundary conditions can be imposed on these boundaries. To replicate the experimental set-up, the mainstream inlet flow is considered to have a uniform velocity profile with a temperature of 332 K. The initial pressure at the exit boundary is kept uniform at one atmosphere, and is allowed to develop as the flow progresses. The surface of the bluff-body is isothermal at 295K.

The body is assumed to be of Plexiglas and has a density of 1162 kg/m^3 , conductivity of $0.21 \text{ W/m}^2\text{K}$ and C_p value of 1465 kJ/kg K .

The study uses FLUENT solver to simulate the flow and heat transfer. As described earlier, a two dimensional study of flow is performed as the initial step towards investigation to film cooling effectiveness on a 3D cylindrical model. Once the solver is validated for heat transfer calculations on a cylindrical model, 3D film cooling simulations are performed. The highest Mach number for the simulations is 0.037; hence, the flow can be considered incompressible. The momentum and energy equations are second-order upwinded. An unsteady segregated solver with a predictor-corrector type adaptive time step simulates the flow for Reynolds numbers of 40,000, 60,000, 70,800, and 100,000.

The Semi Implicit Pressure Linked Equation (SIMPLE) algorithm is used for coupling the velocity and pressure equations. The SIMPLE method mainly consists of the following steps comprising an iteration.

- 1) An initial guess for pressure or the preceding-iteration value of pressure is taken as the starting value and the governing equations are solved for velocity.
- 2) Once the velocity is obtained, the pressure equation is solved in order to obtain the new pressure distribution.
- 3) The now pressure distribution is used to calculate the corrected velocity.
- 4) Steps 2 and 3 are repeated until the convergence criterion is satisfied.

Twenty iterations are performed for each time-step to ensure convergence at each time-step. The coefficients of drag and lift are monitored during the simulations

simultaneously to ensure convergence of aerodynamic properties. In addition, the Frossling Number distribution is plotted over the surface at various time intervals until no variation is observed in the distribution. On an average, 2.5 seconds of flow-time ensures convergence for all the Reynolds numbers examined in this study. The simulations are run on 6 AMD 2200+ processors. It took approximately 72 hours to arrive at convergence for each

case.

Fig. 7. Boundary conditions for the 2D computational grid

3.4 Boundary Conditions and Solver Configurations for Heat Transfer Study with the Cooling Jet

The present simulation uses the same leading-edge geometry and grid, as used in the no-cooling simulations. The coolant is supplied through a slot in the leading-edge region that makes an angle of 21.5 degree to the leading edge in the +y direction. The diameter of the coolant hole is 4.76 mm.

The flow and heat transfer are simulated over the 2D bluff-body with a constant blowing ratio of 1, and a Reynolds number of 60,000. The flow is assumed turbulent with a turbulence intensity of 0.75%. The $k-\omega$ SST model is used to simulate the flow and heat transfer. The boundary conditions used for this simulation are same those as in no-cooling cases, except at the wall of the cylindrical model. Unlike the iso-thermal boundary condition used in the no-cooling case, the cooling case uses an adiabatic surface boundary condition [Fig.8]. Also, this simulation uses a coolant slot to inject coolant onto the surface. The coolant jet inlet has a uniform velocity profile and at a temperature of 297.4K. It has a density of 1.19 kg/m³. The mainstream is at a temperature of 332 K and has a density of 1.0657 kg/m³. A single blowing ratio, $M = 1.0$ is used to simulate the flow. The results are discussed in chapter 5.2.

Fig. 8. Boundary conditions for the 2D computational grid – Constant-cooling

Chapter 4

Leading-Edge Film Cooling - 3D

4.1. 3D Simulation Geometry

The physical 3D geometry [Fig. 3] in this 3D study uses the same 2D geometry as used in preceding 2D simulations except that the model has a span-wise length of 364 mm. Also, the coolant hole is angled 20 degrees to the surface in the span-wise direction. The coolant hole is situated approximately midway on the cylindrical model in the span-wise direction and has a diameter of 4.76 mm with a length to diameter (L/d) ratio of 11.69. The cylindrical leading-edge model is placed at the center of a wind tunnel. The mainstream inlet for the wind tunnel is located 720 mm upstream of the half cylinder.

Fig. 9. Dimensions of the cylindrical-leading-edge model – top view

Fig. 10. Dimensions of the cylindrical-leading-edge model – side view

4.2 Grid Generation

The hybrid grid topology used for the simulations is developed in two steps. First, the geometric model is developed using solid-modeling software, SolidEdge, and subsequently, the grid is developed using the grid generation software GridGen. First, a structured C-grid is developed around the bluff-body with highest level of cell concentration at the jet exit region. Then, a multi-block grid is developed around the body by extruding the surface grid with a growth ratio of 1.1 upto $10d$ from the surface where d is the cooling pipe diameter. It is also ensured that the initial grid spacing at the surface corresponds to a y^+ value 1. An unstructured grid block is created between the structured C-grid developed around the bluff body, and the outer domain boundary. The final grid consists of 2.4 million cells.

Structured C
grid around
the leading-
edge

Fig. 11. The 3D computational grid

Geometric
extrusion of
surface grid
with a growth
ratio of 1.1

The coolant
pipe interface

Fig. 12. The 3D computational grid – close-up view of the jet exit region

4.3 Boundary Conditions

In the computational domain, a uniform velocity profile is assumed at the inlet of the wind tunnel, situated approximately at a distance of 720mm from the leading-edge. As in the experimental set up by Ekkad et al (2006), the inlet velocity is 12.63 m/s i.e. equivalent to a

Fig. 13. The boundary conditions on the 3D computational grid

Reynolds number of 60,000 with the leading-edge diameter, D , as the characteristic length. Also, the inlet flow is at a constant temperature of T_{∞} , 332 K and with a density of 1.0657 kg/m^3 . Considering the fact that the outer boundary domain is sufficiently far away from the surface, a slip-wall boundary condition is applied at the far-field. Also, a slip-wall boundary condition does not require clustering at the boundary that helps save computational time. The downstream boundary is located at 360 mm from the leading edge. The pressure at the exit is kept constant at one atmospheric pressure and is allowed to develop as the flow develops. At the inlet of the hole, the coolant temperature is 297.4K and the coolant velocity is assumed uniform with the value determined from the specified blowing ratio, $M (= \rho_c V_c / \rho_{\infty} V_{\infty})$. An adiabatic boundary condition is applied on the test surface. Four different blowing ratios; 0.25, 0.5, 0.75, 1.0, are considered for simulations with a Reynolds number 60,000.

4.4 Numerical methodology and Solver Configurations

The steady 3D Navier-Stokes equations are solved to simulate the flow and heat transfer. The flow is simulated with two turbulence models, namely, $k-\omega$ SST and $k-\varepsilon$ realizable turbulence models are used to simulate the flow and heat transfer. For the simulation with $k-\varepsilon$ Realizable turbulence model, the near-wall quantities are calculated by enhanced wall functions available in FLUENT software. The highest Mach number for the simulations is 0.037; hence the flow is considered incompressible. Momentum

and energy equations were second-order upwinded. The solution is considered converged when the residuals of continuity reached below 10^{-5} .

4.5 Turbulence Modeling

4.5.1 k - ε Realizable Turbulence Model

The k - ε Realizable turbulence model proposed by Shih et al. 1993 uses a new formula for eddy viscosity. The model satisfies a specific mathematical constraint on the normal Reynolds stress such that it is always positive. Unlike the standard k - ε model, k - ε Realizable turbulence model predicts the spreading rate of jets and separation of flow accurately. The k - ε Realizable turbulence model uses the following transport equations for turbulence kinetic energy and dissipation rate.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + P_b - \rho \varepsilon - Y_M + S_k \quad (8)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}(\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} P_b + S_\varepsilon \quad (9)$$

where P_k and P_b are generations of turbulent kinetic energy due to mean velocity gradient and buoyancy respectively. The expression for eddy viscosity is

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (10)$$

where C_μ is

$$C_\mu = \frac{1}{A_0} + A_s \frac{k U^*}{\varepsilon} \quad (11)$$

Where ε is turbulent dissipation and $A_0 = 4.04$, $C_{1\varepsilon} = 1.44$, $C_2 = 1.9$, $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.2$

4.5.2 The k - ω SST Model

The k - ω SST turbulence model has earlier been described in the previous no-cooling simulations. The k - ω SST model uses highly accurate and robust k - ω turbulence model near the wall and free-stream independence of k - ε model in the far field. As observed in the simulations without cooling jets, the SST model predicts the local Frossling number distribution accurately. In addition, k - ω SST turbulence model predicts the jet spreading with greater accuracy. Therefore the k - ω SST turbulence model is employed to simulate the flow.

The flow is simulated for four different blowing ratios, $M = 0.25, 0.5, 0.75$ and 1.0 for a Reynolds number of 60000 . To ensure convergence for all the simulations, the residuals of continuity were monitored. The solution is considered converged when the residuals reached below 10^{-5} . The simulations were run on a 6-processor P4 cluster. Each simulation took around 24 hours to converge.

For 3D simulations, the film temperature, and hence, the film-cooling effectiveness, is a function of the two surface coordinates, namely, the streamwise and the spanwise coordinates. Therefore, T_f and η can be presented as contour plots on the surface. In addition, spanwise-averaged values of η can be presented as a function of the streamwise distance X .

The film cooling effectiveness in this study has been span-wise averaged, and compared with the experimental results. The adiabatic wall temperature distribution profile is exported and the film-cooling effectiveness is span-wise averaged using Microsoft Excel.

Chapter 5

Results and Discussion

5.1 Heat Transfer Study without Cooling Jet - 2D -Laminar

Simulations are performed using FLUENT for four different Reynolds numbers, 40,000, 60,000, 70,800, and 100,000[Table 1]. Results are compared to the experimental results available from Ou (1991), and Mick and Mayle (1977) and the numerical results of Litzler et al. (2007) for a similar model. As the simulation geometry consists of a half-cylinder, the cylinder-in-a-crossflow results of Geidt (1949) are also considered relevant to compare with. So the Nusselt number distributions were compared to the results obtained from the experimental results of Geidt (1949).

Table 1. List of cases for study of 2D heat transfer on the cylindrical-leading-edge model
– Laminar Flow

	<i>Type</i>	<i>Reynolds Numbers</i>	<i>Comparison study</i>
2-D	Simulations without the cooling jet	40000	Ou (1991), Litzler (2007)
		60000	Litzler (2007)
		70800	Geidt (1949), Litzler (2007)
		100000	Ou (1991), Geidt (1949), Mick and Mayle (1977),Litzler (2007)

The results [Fig 15] from FLUENT agree with the experimental results as Ou (1991) in the stagnation region. FLUENT under-predicts the Frossling number at the

stagnation point from experimental results by 4%. As the flow develops, an earlier separation of the flow is observed in the simulation results as compared to the experiment. The flow separates at about 1.6 radians for the experimental data and, for the simulated cases in FLUENT. It is further observed that the Frossling number value for the simulated data differs from the experimental value of Ou (1991) by approximately 25% for the same location. The main reason for this is the turbulence intensities associated with the experimental results. The turbulence intensity considered by Ou (1991) for his experimental results is 0.75%, whereas a laminar flow condition was considered for the simulations in FLUENT.. The high kinetic energy associated with the turbulent flow in the experimental case delays the separation. Turbulence associated with experiments also results in increased heat transfer, thereby increasing the Frossling number value in the separation zone for the experimental cases. Also, for the simulations, the higher Reynolds number flows are observed to separate earlier than the low Reynolds number flows.

Fig. 15. Comparison of Frossling number distribution. Numerical and Experimental Results, $Re = 40,000$.

The Frossling number results of FLUENT, for Reynolds numbers 60,000, and 70,800 are shown, respectively, in Figures 16, and 17. Figure 18 shows the present Frossling number distribution along with the experimental results of Ou (1991), and Mick and Mayle (1977) for Reynolds number 100,000. At the stagnation point, FLUENT result is in agreement with the experimental result of Mick and Mayle (1977). Except for slight differences in the stagnation region, FLUENT predicts approximately the same value of Frossling number at the separation point as results obtained from Litzler et al. 2007. As the flow progresses, a shift in the Frossling number distribution obtained from FLUENT is observed from the experimental results.

As explained earlier, the shift from experimental results is likely because of turbulence intensity.

Fig. 16. Comparison of Frossling number distribution. Numerical and Experimental Results, Re = 60,000.

Figure 19 shows the Frossling number results of the simulation using FLUENT for all the Reynolds numbers.

Fig. 17. Comparison of Frossling number distribution. Numerical and Experimental Results, $Re = 70,800$.

Fig. 18. Comparison of Frossling number distribution. Numerical and Experimental Results, $Re = 100,000$.

Fig. 19. Frossling number distribution. FLUENT Results for all Reynolds numbers.

Next, present results are compared with Nusselt number distribution on a cylinder in cross-flow available from Giedt (1949). Considering that the leading edge of the model is a half-cylinder; the results of Giedt (1949) are assumed relevant to compare with. Figure 21 shows comparison of Nusselt number distribution obtained from FLUENT, to results of Giedt (1949) for Reynolds number 70,800. FLUENT seems to over-predict the experimental results of Giedt (1949) in the stagnation region. In Figure 21, the FLUENT results for Nusselt number are compared for Reynolds number of 100,000 with the experimental results of Giedt (1949). At the stagnation point, the Nusselt number values of Giedt (1949) match with the Nusselt number values

of the FLUENT simulations. As the flow develops, the same shift in separation point is observed as seen earlier for all Reynolds numbers.

Fig. 20. Comparison of Nusselt number distribution. Numerical and Experimental Results, $Re = 70,800$.

Fig. 21. Comparison of Nusselt number distribution. Numerical and Experimental Results, $Re = 100,000$.

5.2 Heat Transfer Study without the Cooling Jet - 2D –Turbulent

As seen from the results from the preceding section, the flow and heat transfer simulations on a two-dimensional leading-edge model with the laminar flow assumption predict the Frossling number consistent with the experimental results at the stagnation region. However, at the separation point, the numerical results do not predict the Frossling number quite accurately. Also, it is observed that the flow separates earlier by approximately 0.2 radians than the experimental for all Reynolds numbers. These differences may be due to the laminar flow assumptions in the numerical simulations. Hence, results are next obtained using turbulence flow simulations.

As laminar flow assumption does not predict the results with reasonable accuracy, study of heat transfer with a turbulent flow assumption is performed. Using the $k-\omega$ SST turbulence model, the flow is simulated for two different Reynolds numbers, 40000 and

60000[Table 2]. The results were compared with the experimental data of Ou (1991), and Mick and Mayle (1977).

Table 2. List of cases for study of 2D heat transfer on the cylindrical-leading-edge model
– Turbulent Flow

	<i>Type</i>	<i>Reynolds Numbers</i>	<i>Comparison study</i>
2-D	Simulations without the cooling jet	40000	Ou (1991), Mick and Mayle (1977)
		60000	Ou (1991), Mick and Mayle (1977)

Fig. 22. Comparison of Frossling number distribution. Numerical and Experimental Results, $Re = 40,000$, Turbulent flow solution

Figure 22, shows the Frossling number distribution on the leading-edge model with the turbulent flow assumption for a Reynolds number of 40,000. It shows a remarkable consistency with the experimental data. At the stagnation point, the numerical simulations predict approximately the same value of Frossling number as the

experimental results of Mick and Mayle (1977). As the flow progresses, the numerical results under-predict the experimental results. However, unlike the early separation prediction by the laminar flow assumption, turbulent flow, predicts the separation location consistent with the experiments. Also, as anticipated, the heat transfer in turbulent flow is higher and thus, the Frossling number value increased at the point of separation.

Fig. 23. Comparison of Frossling number distribution. Numerical and Experimental Results, $Re = 60,000$, Turbulent flow solution

Figure 23 shows the distribution of Frossling number on the leading-edge surface for a Reynolds number of 60,000. The result is compared to the data of Ou (1991) and Mick and Mayle (1977). At the stagnation point, the result obtained from present simulations is in agreement with the the data of Mick and Mayle (1977). As observed in the results of Reynolds 40,000, a noticeable delay in separation is observed in the turbulent flow as compared to laminar flow. This delayed separation-point predicted is consistent with both the experimental results.

5.3. Leading-Edge Film Cooling – 2D

Fig. 24. Velocity vectors at the jet exit region – Film cooling – 2D

Figure 24, shows the velocity vectors of the coolant and the mainstream flow. Clearly, a separation bubble is observed at the exit of the coolant jet. Also, it is observed that, less interaction occurs between coolant jet and mainstream in the jet exit region. That results in higher effectiveness at the jet exit. However, in the downstream, the film thickness becomes thinner and lowers the effectiveness. Figure 25, shows the film-cooling effectiveness results obtained from the present two-dimensional simulation for a Reynolds number 60000 and a blowing ratio 1. The film cooling effectiveness results obtained from this simulation is approximately 4 times the results of experimental results obtained by Ekkad et al. (2006). In the present simulations, two dimensional

simulations are unable to perform spanwise averaging. That is the primary reason of discrepancy of results between numerical and experimental results for 2D simulations. Also, coolant hole in the experimental work has a spanwise angle of 20 degrees which is absent in the 2D simulation. The spanwise angle of the coolant jet results in less streamwise momentum which reduces the lift-off and higher lateral momentum that increases the spreading. The experimental results are span-wise averaged. That cannot be implemented in a 2D simulation. That necessitates a 3D simulation that will employ the correct spanwise angle of the coolant hole and also, consider spanwise averaging to predict results. The next chapter concentrates on results obtained from 3D simulations.

Fig. 25. Comparison of film cooling effectiveness – 2D. Numerical and Experimental results

5.4 Leading-Edge Film Cooling - 3D- Turbulent

Three-dimensional simulations are performed with four blowing ratios, $M = 0.25$, 0.5, 0.75 and 1.0 with two different turbulence models, the $k-\varepsilon$ realizable and the $k-\omega$ SST turbulence model [Table 3]. The flow is simulated for a Reynolds number of 60000, with ρ_c and ρ_∞ of 1.19 kg/m^3 and 1.0657 kg/m^3 respectively.

Table 3. List of cases for study of 3D film cooling on the cylindrical-leading-edge model

Blowing Ratio	Turbulence Model	Comparison Study
0.25	k- ω SST	Ekkad et al. 2006, York and Leylek, 2003
0.5		Ekkad et al. 2006
0.75		Ekkad et al. 2006
1		Ekkad et al. 2006
0.25	k- ε realizable	Ekkad et al. 2006, York and Leylek, 2003
0.5		Ekkad et al. 2006
0.75		Ekkad et al. 2006
1		Ekkad et al. 2006

The results are compared to the experimental results of Ekkad et al. (2006). Also, the results, in the form of film cooling effectiveness for a blowing ratio 1, are compared with numerically obtained results of York and Leylek (2003) who worked on a similar leading-edge model. The blowing ratio, M is defined as the ratio of mass flux of the mainstream and the coolant. It is expressed as

$$M = \frac{\rho_c V_c}{\rho_\infty V_\infty} \quad (12)$$

The coolant jet inlet has a uniform velocity profile. Considering the temperature difference of the mainstream flow and the coolant, and the difference in densities between them, the velocity of coolant inlet is calculated for different blowing ratios.

5.3.1 Blowing Ratio = 0.25

Figures 26 and 27 show the temperature contours obtained using the two turbulence models, the $k-\epsilon$ realizable and the $k-\omega$ SST turbulence model, at the downstream of coolant jet exit. Results obtained show that the $k-\omega$ SST turbulence model, predicts a better attachment of the coolant flow to the surface. Figure 28 shows the results of laterally averaged film cooling effectiveness for both turbulence models. A better attachment is predicted by $k-\omega$ SST turbulence model and that results in a better film cooling effectiveness. Both turbulence models over-predict the experimental results in the jet-exit region. The results obtained from using $k-\epsilon$ realizable model predict a marginally better solution in the jet exit region than $k-\omega$ SST turbulence model. Further downstream, the $k-\omega$ SST turbulence model predicts a better solution than $k-\epsilon$ realizable model. The $k-\epsilon$ realizable model under-predicts the experimental results by 50% and the $k-\omega$ SST turbulence under-predicts by 15% in the downstream region. Nonetheless, in the downstream region, both the models under-predict the experimental; results and over-predict in the jet exit region. The primary reason for this discrepancy could be the grid density in the boundary layer. The initial grid spacing corresponds to y^+ of 1, whereas the growth ratio of grid Δy is 1.1. Typically a growth ratio of 1.02 to 1.05 along the boundary layer is desired to correctly resolve the thermal boundary layer. A growth ratio of 1.1 might be unable to resolve the thermal boundary layer adequately. Another reason could be the turbulence intensity associated with the flow. As observed in the previous 2D heat transfer calculations, turbulence affects the solution considerably. Higher turbulence intensity results in greater attachment of the flow to the surface. This results in lower heat transfer between the mainstream and the blade surface and will increase the

film-cooling effectiveness in the downstream area. Also, the steady approach, used here to simulate the mixing of the jet and mainstream might not resolve the mixing appropriately. An under-prediction of mixing results in higher attachment of coolant to the surface in the jet-exit region and over-prediction of film cooling effectiveness.

Fig. 26. Temperature contours, turbulent flow solution (k - ε realizable model), $M = 0.25$

Fig. 27. Temperature contours, turbulent flow solution ($k-\omega$ SST model), $M = 0.25$

Fig. 28. Comparison of film cooling effectiveness, $M=0.25$. Numerical and Experimental results

5.3.2 Blowing Ratio = 0.5

The temperature contours [Figs.29 and 30] obtained from the simulations with blowing ratio of 0.5 show a distinct increase in lateral momentum of the flow. Figure 31 shows the computed spanwise-averaged film-cooling effectiveness obtained from experimental study of Ekkad et al. (2006). It is observed that in the immediate vicinity of jet exit, the span-wise averaged film-cooling effectiveness is identical for both turbulence models. However, both sets of numerical results over-predict the experimental results in the jet exit region. As the flow progresses, in the downstream region the numerical results obtained under-predict the experimental results. The difference in film cooling effectiveness for the numerical results and the experimental results are approximately 25% at the far downstream region. In the jet exit region, the results obtained by both the turbulence models are approximately 1.5 times the results of Ekkad et al (2006).

Fig. 29. Temperature contours, turbulent flow solution (k - ϵ realizable model), $M = 0.50$

Fig. 30. Temperature contours, turbulent flow solution ($k-\omega$ SST model), $M = 0.50$

Fig. 31. Comparison of film-cooling effectiveness, $M=0.50$. Numerical and Experimental results

5.3.3 Blowing Ratio = 0.75

The result for blowing ratio 0.75 is similar to the result obtained for blowing ratio 0.5. A higher lateral momentum [Figs. 32 and 33] of the flow is observed for both the numerical results. The spreading is observed to be better in the $k-\epsilon$ realizable turbulence model results than in the $k-\omega$ SST turbulence model. In the immediate vicinity of jet exit the spanwise averaged film cooling effectiveness results of numerical simulations over-predict the experimental results. At x/d between 1.0 to 2.5 the numerical results differ by approximately 20% from the experimental results [Fig. 34]. Beyond x/d of 2.5 the numerical results drift from the experimental results by almost 100%. All the numerical results under-predict the experimental data in the downstream region.

Fig. 32. Temperature contours, turbulent flow solution ($k-\epsilon$ realizable model), $M = 0.75$

Fig. 33. Temperature contours, turbulent flow solution ($k-\omega$ SST), $M = 0.75$

Fig. 34. Comparison of film cooling effectiveness, $M = 0.75$. Numerical and Experimental results

5.3.4 Blowing Ratio =1.0.

A higher streamwise momentum associated with the flow due to higher blowing ratio results in lower attachment of the coolant to the surface. So spreading in the simulation for blowing ratio 1.0 [Figs. 35 and 36], is relatively less than the spreading observed for the low blowing ratio simulations. Figure 37 compares the spanwise-averaged film cooling effectiveness of numerical simulation to that of experimental. Similar to results obtained from all other blowing ratios, it is observed that numerical results for blowing ratio 1, over-predict the experimental results immediately after the jet exit and under-predict thereafter. This could be because the C-grid block around the leading-edge model is developed with normal extrusion from the surface grid using a growth factor of 1.1. A growth factor of 1.1 is rather high to be able to predict the heat transfer accurately.

Fig. 35. Temperature contours, turbulent flow solution (k - ϵ realizable model), $M = 1.0$

Fig. 36. Temperature contours, turbulent flow solution (k - ω SST model), $M = 1.0$

Fig. 37. Comparison of film cooling effectiveness, $M=1.0$. Numerical and Experimental results

5.3.5 Effectiveness for All Blowing ratios.

Fig. 38. Film cooling effectiveness for all blowing ratios. Experimental, Ekkad et al. (2006)

Figure 38 shows the experimental spanwise-averaged film-cooling effectiveness results reported by Ekkad et al. (2006) for all the blowing ratios considered in the present study. It is observed that, at immediate jet exit region, the film-cooling effectiveness increases with increase in blowing ratio. Downstream of the jet exit, the blowing ratio of 0.25 has the highest effectiveness. This is also predicted by the numerical results obtained by using $k-\omega$ SST turbulence model [Fig 40]. It also predicts a higher film cooling effectiveness for a higher blowing ratio in the immediate vicinity of jet exit. It is also predicted by $k-\varepsilon$ realizable turbulence model [Fig. 39] in the jet exit region. It predicts a higher effectiveness for higher blowing ratio in the jet-exit region.

Fig. 39. Film-cooling effectiveness for all blowing ratios. Numerical - $k-\epsilon$ realizable model

Fig. 40. Film cooling effectiveness for all blowing ratios. Numerical - $k-\omega$ SST model

Figure 41 presents the numerical results for blowing ratio 1.0 to the numerical results of York and Leylek., (2003). The study by York and Leylek., (2003) has almost the same film cooling effectiveness as the present values in the jet exit region, with a difference of less than 5%. In the downstream region [Fig 41], the results of York and Leylek., (2003) show a higher film effectiveness than predicted in the present study, with a maximum difference of approximately 33% . The difference between the numerical results of York and Leylek., (2003) and the present study is largely due to the grid. It is important to not that the grid used by York and Leylek., (2003) had higher grid density around the surface with a total cell count of 4.8 million. Also, York and Leylek., (2003), used periodicity in their flow simulations to correspond to a series of coolant holes situated along the leading edge. This affects the numerical solution at the far downstream area. Although, the periodicity assumption in the simulations has only a small effect in the immediate vicinity of jet exit, it significantly affects the numerical results in the far downstream area. Hence, as anticipated, the present study exhibits a small difference from the results of York and Leylek (2003) in the jet exit region.

Fig. 41. Comparison of film cooling effectiveness with York et al. 2003, $M=1.0$

5.3.6 Film Cooling Effectiveness for Blowing Ratio = 1, Effect of Different Free-Stream Turbulence Intensities.

It is observed that for all the blowing ratios investigated in the present study, numerical results over-predict the experimental results at the jet exit, and under-predict in the downstream region. As described earlier, turbulence intensity in the free-stream is presumed to be one of the factors affecting the prediction of heat transfer significantly. To assess the impact of the free-stream turbulence intensity on the film cooling effectiveness, simulations are performed with two different turbulence intensities, 0.25% and 7.5%, for a single blowing ratio=1.0. Also, as in the earlier simulations, two turbulence models, the $k-\varepsilon$ realizable turbulence model and the $k-\omega$ SST model, are used to simulate the flow and heat transfer. The results are compared with the results obtained with lower turbulence intensities and also with the experimental results.

Fig. 42. Comparison of film cooling effectiveness for different turbulence intensities, $k-\omega$ SST

Turbulence Model

Fig. 43. Comparison of film cooling effectiveness for different turbulence intensities, $k-\varepsilon$

Realizable Turbulence Model

Figure 42 compares the film cooling effectiveness for different turbulence intensities for $k-\omega$ SST turbulence model and for a blowing ratio of 1. It is observed that a turbulence intensity of 0.25% has very little effect on the film cooling effectiveness whereas, a higher intensity of 7.5% results in lower film cooling effectiveness. In the jet exit region film cooling effectiveness obtained for 7.5% turbulence intensity is lower by almost 20% as compared to the numerical results obtained with turbulence intensity of 0.75%. In the downstream region, the turbulence intensity has negligible impact on the heat transfer and film cooling effectiveness. As anticipated, higher turbulence intensity results in higher mixing of the coolant and the mainstream in the jet exit, that results in higher heat transfer rate and lower film-cooling effectiveness. Figure 43 shows the corresponding results for the film-cooling effectiveness using the $k-\varepsilon$ realizable turbulence model. The results obtained show similar characteristics as the results obtained with the $k-\omega$ SST

Turbulence Model. In the jet exit region, we observe a higher mixing of the mainstream and coolant that predicts a lowered film cooling effectiveness. Further downstream, the computed results remain unaffected by the free-stream turbulence intensity.

5.3.4 Film Cooling Effectiveness for Blowing Ratio = 1.5, with a finer grid

Fig. 44. Comparison of film cooling effectiveness with a finer grid, $M=1.5$. Numerical and Experimental results

In addition to assessing the effect of turbulence intensity, a grid, with a higher grid density is also considered so as to assess the impact of grid density. The new grid consists of 3.9 million cells. As a test case, the flow is simulated without using any turbulence models. For a blowing ratio of 1.5, the results show remarkable agreement with experimental results of Ou and Rivir (2006).. The experimental study done by

Ekkad et al. 2006 and Ou and Rivir, 2006, use different spanwise distance for averaging the film cooling effectiveness. Therefore, for the present numerical simulation, post processing is performed for two different spanwise distance equivalent to spanwise distance of both the experimental studies. In the downstream region, numerically obtained film effectiveness (spanwise averaged according to experimental results obtained from Ou and Rivir, 2006)[Fig. 44] over-predicts experimental results of Ou and Rivir, 2006 by 15%. At the immediate vicinity of the jet exit region, numerical results under-predict the experimental results by 10%.

Also, the numerical results (spanwise averaged according to experimental results obtained from Ekkad et al., 2006) are compared with the experimental results obtained from Ekkad et al., 2006, for a blowing ratio of 1.0. As previously observed in experimental results of Ekkad et al. 2006, an increase in blowing ratio increases the effectiveness by 0.075 in the jet-exit region and it has negligible effect in the downstream region. Therefore, experimental results obtained from Ekkad et al. 2006 for a blowing ratio of 1 is considered relevant to compare with. Figure 44 shows the comparison of numerically obtained film cooling effectiveness to experimental results of Ekkad et al. (2006). In the jet exit region numerical prediction of film cooling effectiveness show a significant improvement in accuracy. Heat transfer simulation from the previous grid predicted a film cooling effectiveness 40% higher than the experimental. The numerical results obtained from the grid with higher grid resolution, show a remarkable improvement in results than the previous grid in the jet exit region. Also, in the

downstream region, between the film cooling effectiveness show a marginal difference of 15% from the experimental results.

These results strongly suggest that the heat transfer calculations are very sensitive to the grid refinement. The new grid consists about additional 1.5 million points than the grid used previously. Another change that has been incorporated in the new grid is the shape of the outer boundary. The circular outer domain is replaced by a rectangular boundary in the new refined grid. The circular boundary in the earlier grid was sufficiently far away from the cylindrical leading-edge model to affect the coolant flow or the heat transfer calculations. Therefore, it is presumed that change in shape of boundary has very little effect on the film cooling effectiveness calculations.

The next chapter focuses on the summary of the entire research and conclusions drawn. It also describes the factors that significantly affect the turbine blade heat transfer calculations that will be considered for future simulations.

Chapter 6

Conclusion and Future Work

A comprehensive analysis of flow and heat transfer over a leading-edge model is performed. The results are presented in three sections.

1. 2D laminar and turbulent simulations without cooling
2. 2D turbulent simulations with cooling, using the $k-\omega$ SST turbulence model and blowing ratio, $M = 1.0$
3. 3D turbulent simulations with cooling, including a cooling pipe, with four blowing ratios, 0.25, 0.5, 0.75, 1.0, and with two turbulence models: the $k-\epsilon$ realizable and the $k-\omega$ SST model.

It is observed that the laminar flow assumption in the two-dimensional heat transfer study predicts heat transfer at the stagnation point with reasonable accuracy. However, better accuracy in heat transfer prediction is achieved using the $k-\omega$ SST turbulence model. The low momentum flow near the solid surface gains kinetic energy from the flow near the edge of boundary layer through efficient mixing in a turbulent flow. This results in higher attachment of the flow onto the surface, and leads to higher heat transfer. Thus, turbulence in the flow increases the heat transfer than in laminar flow, and results in better agreement with the experimental results.

Spanwise angle of the coolant hole increases the lateral momentum of the flow and results in better spreading of the coolant on the blade surface. Also, the increased lateral momentum of the flow drives the flow in the spanwise direction. Two-

dimensional simulation of the flow to predict film cooling effectiveness is unable to incorporate the spanwise angle in its geometry. A three-dimensional simulation, more closely representing the actual experimental geometry and conditions, helps achieve better accuracy in predicting film cooling effectiveness.

- As observed in the numerical simulations and in the experimental work by Ekkad et al. (2006), a lower blowing ratio results in higher film-cooling effectiveness in the downstream region. However, film-cooling effectiveness increases with increase in blowing ratio in the immediate vicinity of cooling hole exit. Also, it is observed that, for all the blowing ratios considered, numerical results over-predict the experimental results at the jet exit, and under-predict downstream. A steady approach might under-predict the interaction between the mainstream and coolant. In the downstream region, prediction of lower interaction between the flows results in less attached coolant onto the surface and lower film-cooling effectiveness. In the immediate vicinity of the jet exit, the lower prediction of jet coolant mixing results in higher contact of coolant to the surface and that results in higher film-cooling effectiveness. An unsteady approach to the solution will better predict the mixing of coolant and mainstream, and might predict the numerical results better. The jet and the mainstream interaction occurs largely at the transition zone on the turbine-blade model. A RANS-based turbulence model might not be able to resolve this flow structure accurately. Finally, as we observed, heat transfer calculations are highly sensitive to the grid resolution. A refined grid needs to be used to predict the film cooling effectiveness accurately.

6.2 Future Work

Further improvement to the present study can be achieved by using an unsteady approach. An unsteady approach might predict jet and mainstream interaction more accurately. Also, an LES model can be used for future film-cooling simulations so as to predict the flow and heat transfer more accurately than possible with RANS-based models. A proper turbulence intensity in the numerical investigation should be used to represent the experimental conditions more closely and to predict results with better accuracy in the far downstream region.

This work is part of a larger study that focuses on pulsed-flow film cooling. The 3-D simulations should be continued to determine the optimum duty cycle and frequency that achieves higher film-cooling effectiveness on the turbine blade leading-edge and also, consumes less mainstream air to achieve increased engine efficiency.

References

- Azzi, A., Abidat, M., Jubran, B. A., and Theodoridis, G. S., (2001), "Film cooling predictions of simple and compound angle injection from one and two staggered rows", Numerical Heat Transfer, Part A, Vol. 40, pp. 273-294.
- Bejan A., (2004), Convection Heat Transfer, 3rd edn., John Wiley & Sons, New York,
- Bellows, W. J. and Mayle, R.E., (1986), "Heat Transfer Downstream of a Leading Edge Separation Bubble", Journal of Turbo machinery, Vol. 108, pp. 131-136.
- Bergeles, G., Gosman, A. D., and Launder, B. E., (1976), "Near-field character of a jet discharged through a wall at 30° to a mainstream", AIAA Journal, Vol. 15, No. 4., pp. 499-504.
- Bharti, R. P., Chhabra, R. P. and Eswaran, V., (2007), "A numerical study of the steady forced convection heat transfer from an unconfined circular cylinder", Journal of Heat and Mass Transfer, Vol 43, Issue 7, pp. 639-648.
- Chernobrovkin, A., and Lakshminarayana, B., (1999), "Numerical Simulation and Aerothermal Physics of Leading Edge Film Cooling", IMEchE, Proc Instn Mech Engrs, Vol. 213, Part A.
- Ekkad, S. V., Han, J. C., and Du, H., (1998), "Detailed film cooling measurements on a cylindrical leading edge model. effect of free-stream turbulence and coolant density", Journal of Turbomachinery, Vol. 120, pp. 799 – 807.
- Ekkad, S. V., Ou, S., and Rivir, R. B., (2006), "Effect of Jet pulsation and duty cycle on film cooling from a single jet on a leading edge model", Journal of Turbomachinery, Vol. 128., pp. 564-571.
- Geidt, W. H., (1949), "Investigation of Variation of Point Unit Heat Transfer Coefficient Around a Cylinder Normal to an Air Stream" ASME Trans., Vol. 71, pp. 375-371.

Hua, H. M., and Lykoudis, P. S., (1976), "Heat transfer from a constant temperature circular cylinder in cross-flow", International Journal of Heat and Mass Transfer, vol. 19, pp. 698-700

Kim, Y. J., and Kim, S.M., (2004), "Influence of shaped injection holes on turbine blade leading edge film cooling", International journal of heat and mass transfer, Vol. 47, pp. 245-256.

Lakehal, D., Theodoridis, G.S. and Rodi, W., (1997), "Computation of film cooling of a flat plate by lateral injection from a row of holes", International Journal of Heat and Fluid Flow, No. 19, pp 417-430

Lin, Y.L. and Shih, T.I.P., (2001), "Film cooling of a cylindrical leading edge with injection through rows of compound angle holes", Journal of Heat Transfer, Vol. 123, pp 645-654.

Litzler, J, Mishra, S., Ghia, U., Ghia, K. and Ou, S., (2007), "Two-dimensional low mach number heat transfer simulation of turbine blade leading-edge model", ASME paper FEDSM2007-37085, Presented in 5th Joint ASME/JSME Fluid Engineering Summer Conference, San Diego, CA.

Majumdar, S., and Rodi, W., (1985), "Numerical calculations of turbulent flow past circular cylinders", Symposium on Numerical and Physical Aspects of Aerodynamic Flows, pp. 3-13 to 3-25.

Menter, F.R., (1993), "Zonal Two Equation $k-\omega$ Turbulence Models for Aerodynamic Flows", AIAA Paper 93-2906

Menter, F.R., (1994), "Two-equation eddy-viscosity turbulence models for engineering applications", AIAA Journal, Volume 32, pp 269-289

Mick, W. J., and Mayle, R. E., (1977), "Stagnation Film Cooling and Heat Transfer, Including Its Effect Within the Hole Pattern", Journal of Turbo machinery, Volume 110, pp. 66-72.

Ou, S., (1991), "Leading edge film cooling heat transfer including the effect of mainstream turbulence" PhD. Dissertation, Texas A & M University.

Sarkar, S., Das, K. and Basu, D., (2001), "Film cooling on a turbine guide vane. a numerical analysis with a multigrid technique", Proceeding of institute of mechanical engineers, Vol. 215. Part A.

Schlichting, H., Gersten K., and Meyes K. (2000), Boundary-layer theory, 7th rev., Springer., Berlin, White, F.M., 1994. Fluid Mechanics, 3rd edn., McGraw-Hill, New York.

Sinha, A. K., Bogard, D. G., and Crawford, M. E., (1991), "Film-cooling effectiveness downstream of a single row of holes with variable density ratio", Journal of Turbomachinery, Vol. 113, pp. 442-449.

Szczepanik, K., Ooi, A, Aye, L. and Rosengarten, G., (2004), "A Numerical Study of Heat Transfer from a Cylinder in Cross Flow", 15th Australasian Fluid Mechanics Conference, 13-17 December.

Tyagi, M., and Acharya, S., (2003), "Large eddy simulation of film cooling from an inclined cylindrical jet", Journal of Turbomachinery, Vol. 125, pp. 734-742.

Walters, D. K., and Leylek, J. H., (2000), "A detailed analysis of film-cooling physics. Part I-Streamwise injection with cylindrical holes", Journal of Turbomachinery, Vol. 122, pp. 102-112.

Walters, D. K., and Leylek, J. H., (1997), "A systematic computational methodology applied to a three-dimensional film-cooling flowfield", Journal of Turbomachinery, Vol. 119, pp. 777-784.

Whitaker, S., (1972), "Forced convection heat transfer correlations for flow in pipes, past flat plates, single cylinders, single spheres, and for flow in packed beds and tube bundles", Journal AIChE, Vol. 17, pp 361-371.

York, W. D., and Leylek, J. H., (2003), "Leading Edge Film Cooling Physics – Part III. Diffused Hole Effectiveness", Journal of Turbomachinery, Vol. 125, pp 252-259.

Zheng, B., Lin. C. X., and Ebadian, M. A., (2000), "Numerical investigation of leading-edge film-cooling flow over experimental model", Proceedings of NHTC'00, Pittsburgh, PA, NHTC2000-12111.