Copyright
by
Gwennaël Beirnaert-Chartrel
2011

**The Thesis Committee for Gwennaël Beirnaert-Chartrel
Certifies that this is the approved version of the following thesis:**

**CFD Predictions of Heat Transfer Coefficient Augmentation
on a Simulated Film Cooled Turbine Blade Leading Edge**

**APPROVED BY
SUPERVISING COMMITTEE:**

Supervisor:

David G. Bogard

Robert D. Moser

**CFD Predictions of Heat Transfer Coefficient Augmentation
on a Simulated Film Cooled Turbine Blade Leading Edge**

by

Gwennaël Beirnaert-Chartrel, B.S.

Thesis

Presented to the Faculty of the Graduate School of

The University of Texas at Austin

in Partial Fulfillment

of the Requirements

for the Degree of

Master of Science in Engineering

The University of Texas at Austin

May 2011

Dedication

To Isabelle and Bruno, my parents, to Bruno, my second father
and to Gaël and Camille, my brother and my sister,
for their continued love and support.

Acknowledgements

First of all, I would like to thank Dr. Bogard for his guidance and wisdom that have helped me to successfully complete this thesis and to have offered me the opportunity to work on this exciting area of research. He has been a great teacher and I am very grateful to him. I also would like to thank Dr. Moser for reviewing this thesis and providing numerous pertinent comments.

Furthermore I would like to thank all the TTRCL students. First I would like to thank Sibi Mathew who was very helpful in introducing me to film cooling and CFD. I would like also to thank Jason Albert and Tom Dyson who helped me in setting up the experiments for the new graduate lab class. And finally I would like to thank Todd Davidson, Marc Nathan, Emily Pfautsch and Randy Williams for having made this lab a nice place to work.

Moreover I would like to thank Angel Bolhouse, Tai Duong, Claire Florette, Sigfried Haering, Jude Osara and Neil Shah, all the TAs I had the chance to work with.

I also would like to thank the “French team”: Maria Benzekri, Erwan Chabert, Claire Davis, Mathieu Leduc and Philippe Marouby for their friendship and all the good times spent together over the past couple of years. Additionally, I would like to thank Elizabeth Kincaid for her continued support in the process of writing this thesis.

Finally, I am very grateful to my beloved family which has always been a great source of love and support. In particular I would like to thank my Mom who has taught me the value of work and to be ambitious.

May 6th, 2011

Abstract

CFD Predictions of Heat Transfer Coefficient Augmentation on a Simulated Film Cooled Turbine Blade Leading Edge

Gwennaël Beirnaert-Chartrel, M.S. E.

The University of Texas at Austin, 2011

Supervisor: David G. Bogard

Computations were run to study heat transfer coefficient augmentation with film cooling for a simulated gas turbine blade leading edge. The realizable k - ϵ turbulence model (RKE) and Shear Stress Transport k - ω turbulence model (SST) were used for the computational simulations. RKE computations completed at a unity density ratio were confirmed to be consistent with experimental measurements conducted by Yuki et al. (1998) and Johnston et al. (1999) whereas SST computations exhibited significant discrepancies. Moreover the effect of the density ratio on heat transfer coefficient augmentation was studied because experimental measurements of heat transfer coefficient augmentation with film cooling are generally constrained to unity density ratio tests. It was shown that heat transfer coefficient augmentation can be simulated using unity density ratio jets, but only when scaled with the momentum flux ratio of the coolant jets.

Table of Contents

List of Tables	x
List of Figures	xi
Nomenclature	xvii
Chapter 1: Introduction and literature review	1
1.1. Gas turbine theory	1
1.2. Airfoil cooling theory	4
1.3. Experimental method of heat transfer measurement	8
1.4. Literature review	9
1.4.1. Experimental study of the heat transfer coefficient augmentation	9
1.4.2. Experimental study of the effect of density ratio on the heat transfer coefficient	13
1.4.3. Computational turbulence models	14
1.4.4. Computational heat transfer coefficient without film cooling	18
1.4.5. Computational predictions of heat transfer coefficient augmentation due to film cooling	20
1.5. Objectives of current study	22
Chapter 2: Computational setup	24
2.1. Description of the model	24
2.2. Operating parameters and boundary conditions	27
2.3. Varying density ratio simulations	29
2.4. Turbulence model specifications	30
2.5. Convergence	31
2.6. Grid construction	31
2.7. Grid independence	32
2.7.1. RKE grid independence	32
2.7.2. SST grid independence	36

Chapter 3:	Results and Discussion	40
3.1.	Study of the heat transfer coefficient without film cooling	40
3.2.	Comparison of heat transfer coefficient augmentation predictions with experimental data	47
3.2.1.	Evaluation of the effects of the non-uniform heat flux foil of Yuki.....	47
3.2.2.	Evaluation of the effects of the non-uniform heat flux of Mouzon	52
3.2.3.	Comparison of predictions and experiments for low mainstream turbulence	55
	Impingement plate effect	56
	Comparisons of RKE and SST predictions with an impingement plate.....	58
3.2.4.	Comparison of predictions and experiments for high mainstream turbulence	67
	Comparisons of RKE and SST predictions with an impingement plate.....	69
3.3.	Study of mechanism causing enhancement of the heat transfer coefficient due to coolant jets	75
3.4.	Study of the effect of the heat flux rate.....	83
3.5.	Evaluation of the effects of holes in the heat flux plate.....	84
3.6.	Study of the effect of the mainstream temperature	87
3.7.	Study of the effect of the coolant proprieties at DR=1.5	89
3.7.1.	Air proprieties at 300K and at 200K.....	89
3.7.2.	Injection of CO ₂	93
3.8.	Study of the effect of the density ratio	95
3.8.1.	Momentum flux ratio	95
3.8.2.	Blowing ratio	98
3.8.3.	Velocity ratio	102
Chapter 4:	Conclusion	107
4.1.	Summary of the results and conclusion	107
4.2.	Recommendations for future work	111

Appendix.....	112
Models.....	112
Materials	112
Solution Methods.....	114
Solution Controls	114
References.....	115
Vita.....	118

List of Tables

Table 2.1: Boundary conditions for the CFD model.....	28
Table 2.2: Characteristics of the prism layers tested for the RKE turbulence model	34
Table 2.3: Characteristics of the prism layers tested for the SST turbulence model	37
Table 3.1: Comparison between fluid and electric differential equations	49
Table 3.2: Change in momentum flux ratio and blowing ratio through each row due to the use of an impingement plate	58
Table 3.3: Comparison of the air proprieties and dimensionless numbers	88
Table 3.4: Comparison of the coolant proprieties values for the two cases considered	90
Table 3.5: Comparison of the fluid proprieties between air and CO ₂	93

List of Figures

Figure 1.1: Diagram of a gas turbine jet engine (http://en.wikipedia.org/wiki/Gas_turbine)	1
Figure 1.2: Idealized Brayton cycle (http://en.wikipedia.org/wiki/Brayton_cycle)	2
Figure 2.1: Schematic of the leading edge model (from Mathew (2010)).....	25
Figure 2.2: Front view of the leading edge model (from Mathew (2010)).....	25
Figure 2.3: Schematic of the 3D computational domain (from Ravelli et al. (2010)).....	26
Figure 2.4: Dimensions of the 3D computational domain (from Mathew (2010)).....	27
Figure 2.5: Comparison of the heat transfer coefficient lateral distributions at $s/d = 5$ for different numbers of cells.....	33
Figure 2.6: Comparison of RKE heat transfer coefficient lateral distributions at $s/d = 5$ for four different heights of the prism layer.....	35
Figure 2.7: Thermal boundary layer at $s/d = 5$ and at $z/d = 5.5$	35
Figure 2.8: Comparison of the SST heat transfer coefficient lateral distributions at $s/d = 5$ for three different first cell sizes.....	36
Figure 2.9: Comparison of heat transfer coefficient lateral distributions at $s/d = 5$ for four different heights of the prism layer	38
Figure 2.10: Comparison of the heat transfer coefficient lateral distributions at $s/d = 5$ for different numbers of cells.....	39
Figure 2.11: Grid resolution at the stagnation plane for the SST grid.....	39
Figure 3.1: Comparison of laterally averaged Frossling number	41
Figure 3.2: Schematic of the leading edge facility used by Yuki et al. (1998).....	42
Figure 3.3: Spanwise variations in Fr with the RKE and SST turbulence model.....	43
Figure 3.4: Leading edge surface mesh for the RKE turbulence model simulations	44
Figure 3.5: Lateral distribution of h_f at $I = 2.2$ and h_0 at $s/d = 5$ with the RKE and SST turbulence model.....	46

Figure 3.6: Deviations in heat flux due to non-uniform electrical current flow around holes. Contours of $q_{\text{foil with holes}} / q_{\text{foil without holes}}$ in the experiments. (From Yuki et al., 1998)	48
Figure 3.7: Wall heat flux rate boundary condition mimicking the experimental condition	48
Figure 3.8: Effect of the non-uniform heat flux on the laterally averaged heat transfer coefficient augmentation values for $DR = 1$, $I = 2.2$ and for $Tu = 0.5\%$	50
Figure 3.9: Contour plots of h_f/h_0 for a) the non-uniform heat flux and b) for the uniform heat flux.....	51
Figure 3.10: Experimental wall heat flux rate boundary condition imposed by Mouzon (2005).....	52
Figure 3.11: Comparison of the laterally averaged heat transfer coefficient augmentation values between the uniform heat flux case and the Mouzon's experiment	53
Figure 3.12: Ratio of heat transfer coefficient augmentation between the uniform heat flux case and the simulation of Mouzon's experiment	54
Figure 3.13: Comparison of simulated RKE laterally averaged heat transfer coefficient augmentation with experimental data at $DR = 1.0$, $I = 2.2$ and $Tu = 0.5\%$	55
Figure 3.14: Effect of the impingement plate on the simulated laterally averaged heat transfer coefficient augmentation for $DR=1$, $I=2.2$ and $Tu=0.5\%$	57
Figure 3.15: Comparison of simulated laterally averaged heat transfer coefficient augmentation with experimental data at $DR = 1$, $I = 2.2$ and $Tu = 0.5\%$	58
Figure 3.16: Comparison of simulated laterally averaged heat transfer coefficient augmentation with experimental data at $DR=1$, $I=0.56$ and $Tu=0.5\%$	59

Figure 3.17: Comparison of the heat transfer coefficient augmentation distribution at $s/d = 5.6$ and at $I = 2.2$	60
Figure 3.18: Frossling number distribution at $M = 0.4$ and $M = 0.8$ from Rozati and Tafti (2007)	61
Figure 3.19: Comparison of the heat transfer coefficient augmentation distribution at $s/d = 5.6$ and at $I = 2.2$	62
Figure 3.20: Overlay of h_f/h_0 ($M = 1.9$, $DR = 1.0$) and η ($M = 2.0$, $DR = 1.1$) from Yuki et al. with the RKE h_f/h_0 ($M = 2.0$, $DR = 1.0$) from the present study at $s/d = 5.6$	63
Figure 3.21: Experimental contour plots of h_f/h_0 at $DR = 1.0$, $I = 2.2$ and $Tu < 0.5\%$ from Yuki et al. (1998) and Johnston et al. (1999).....	65
Figure 3.22: Computational contour plots of h_f/h_0 at $DR = 1.0$, $I = 2.2$ and $Tu = 0.5\%$ for RKE and SST turbulence models.....	66
Figure 3.23: Comparison of RKE (no impingement plate) laterally averaged heat transfer coefficient augmentation at $I = 2.2$ for low and high mainstream turbulence with experimental data.....	67
Figure 3.24: Comparison of RKE (no impingement plate) laterally averaged h_f/h_0 at $Tu = 0.5\%$ with $h_{f,Tu=17\%}/h_{0,Tu<0.5\%}$ at $I = 2.2$	68
Figure 3.25: Comparison of simulated laterally averaged heat transfer coefficient augmentation with experimental data at $DR = 1$, $I = 2.22$ and $Tu = 17\%$	69
Figure 3.26: Comparison of simulated laterally averaged heat transfer coefficient augmentation with experimental data at $DR = 1$, $I = 0.56$ and $Tu = 17\%$	70
Figure 3.27: Comparison of the heat transfer coefficient augmentation distribution at $s/d = 5.1$ and at $I = 2.2$ for high mainstream turbulence	71

Figure 3.28: Experimental contour plot of h_f/h_0 at $DR = 1.0$, $I = 2.2$ and $Tu = 17\%$ from Johnston et al. (1999)	72
Figure 3.29: Computational contour plots of h_f/h_0 at $DR = 1.0$, $I = 2.2$ and $Tu = 17\%$ for RKE and SST turbulence models	73
Figure 3.30: Comparison of RKE laterally averaged h_f/h_0 from this study and from York and Leylek (2002) study with experimental data at $DR = 1$, $I =$ 2.25 and $Tu = 0.5\%$	74
Figure 3.31: Contours of heat transfer coefficient augmentation for $DR = 1.0$ and $I =$ 4.0	76
Figure 3.32: Contours of heat transfer coefficient augmentation for $DR = 1.0$ and $I =$ 4.0 in the stagnation holes region	76
Figure 3.33: Velocity vectors on a plane normal to the leading edge and located right at the exit of the stagnation hole ($z/d = 0.63$)	77
Figure 3.34: Overlay of heat transfer coefficient augmentation h_f/h_0 and adiabatic effectiveness at $s/d = 5$ for $I = 2.67$	78
Figure 3.35: Contour plots of the velocity swirling strength at a) $s/d = 5$, b) $s/d = 7$ and c) $s/d = 9$	79
Figure 3.36: a) Pressure and b) normalized density contours downstream of the off- stagnation hole at $s/d = 5$	80
Figure 3.37: Coolant and mainstream streamlines interaction in the off-stagnation hole region	81
Figure 3.38: Predicted normalized temperature contours along $s/d = 5.1$ plane according to SST model, RKE model compared with experimental data (Mathew (2010))	82

Figure 3.39: Comparison of simulated laterally averaged heat transfer coefficient values at $DR = 1.0$ and $M = 2.0$ for three different heat flux rates	84
Figure 3.40: Effect of holes in the heat flux foil on the heat transfer coefficient without film cooling.....	85
Figure 3.41: Comparison of the laterally averaged heat transfer coefficient without film cooling for the two configurations	86
Figure 3.42: Spanwise correction factor	87
Figure 3.43: Comparison of the laterally averaged heat transfer coefficient augmentation for a fluid temperature at 300K and 200K	88
Figure 3.44: Comparison of laterally averaged heat transfer coefficient values at $I = 2.67$	90
Figure 3.45: Comparison of heat transfer coefficient distribution at $s/d = 5$ and at $I = 2.67$	91
Figure 3.46: Effect of the CO ₂ proprieties on the heat transfer coefficient distribution at $s/d = 5$	94
Figure 3.47: Effect of the CO ₂ proprieties on the laterally averaged heat transfer coefficient	95
Figure 3.48: Comparison of the laterally averaged h_f/h_0 at constant momentum flux ratio of $I = 4$	96
Figure 3.49: Comparison contour plots of h_f/h_0 for two density ratios at constant momentum flux ratio of $I = 4.0$	97
Figure 3.50: Comparison of h_f/h_0 distribution at $s/d = 5$ for two density ratios at constant momentum flux ratio of $I = 4.0$	98
Figure 3.51: Comparison of the laterally averaged h_f/h_0 at constant blowing ratio of $M = 2.0$	99

Figure 3.52: Comparison contour plots of h_f/h_0 for two density ratios at constant blowing ratio of $M = 2.0$	100
Figure 3.53: Comparison of h_f/h_0 distribution at $s/d = 5$ for two density ratios at constant blowing ratio of $M = 2.0$	101
Figure 3.54: Comparison of the laterally averaged h_f/h_0 at constant velocity ratio of $VR = 2.0$	103
Figure 3.55: Comparison contour plots of h_f/h_0 for two density ratios at constant velocity ratio of $M = 2.0$	104
Figure 3.56: Comparison of h_f/h_0 distribution at $s/d = 5$ for two density ratios at constant velocity ratio of $VR = 2.0$	105
Figure 3.57: Comparison of SST h_f distribution at $s/d = 5$ for two density ratios at constant momentum flux ratio of $I = 2.67$	106
Figure 3.58: Comparison of SST h_f distribution at $s/d = 5$ for two density ratios at constant blowing ratio of $M = 2.0$	106

Nomenclature

Symbols

c	Specific heat capacity
DR	Density ratio = ρ_c/ρ_∞
D	Leading edge diameter
d	Hole diameter
Fr	Frossling number = $Nu_D/Re_D^{1/2}$
h_f	Heat transfer coefficient with film cooling
h_0	Heat transfer coefficient without film cooling
h_f/h_0	Heat transfer coefficient augmentation
I	Momentum flux ratio = $\rho_c U_c^2 / \rho_\infty U_\infty^2$
k	Thermal conductivity
M	Blowing ratio = $\rho_c U_c / \rho_\infty U_\infty$
$\dot{m}$	Mass flow rate
Nu_D	Nusselt number = hD/k
p	Hole-to-hole pitch
q''	Heat flux
q	Heat transfer
Re_D	Reynolds number = $\rho U D / \mu$
s	Streamwise coordinate
T	Temperature
Tu	Turbulent intensity level
U	Flow velocity
VR	Velocity ratio = U_c / U_∞
y^+	Non-dimensional wall distance = $\rho u_\tau y / \mu$
z	Spanwise coordinate

Greek symbols

Δq_r	Net heat flux reduction
η	Adiabatic effectiveness
ϕ	Non-dimensional surface temperature
Λ	Integral length scale
ρ	Density
μ	Dynamic viscosity

Subscripts and superscripts

aw	Adiabatic wall
c	Coolant
f	With film cooling
in	At the entry of the hole
out	At the exit of the hole
w	At the wall
0	Without film cooling
∞	Mainstream

Chapter 1: Introduction and literature review

The intention of this first chapter is to provide an overall understanding of the fundamental principles behind gas turbine engines. The use of cooling techniques, such as film cooling, will be discussed. Finally a review of pertinent film cooling experimental and computational studies that have been conducted in the past will be carried out.

1.1. GAS TURBINE THEORY

Gas turbine engines have transformed the world of power generation and are used in a wide range of applications from transport industry (aircraft, ship, and tank) to the energy world (gas turbine power plant). As illustrated in Figure 1.1, a gas turbine engine is composed of a compressor, a combustor, a turbine and an exhaust nozzle.

Figure 1.1: Diagram of a gas turbine jet engine (http://en.wikipedia.org/wiki/Gas_turbine)

Most modern gas turbine engines operate on a cycle theorized first by George Brayton during the 1870's. The Brayton cycle, shown in Figure 1.2, is composed of four stages: an isentropic compression in the compressor, a constant pressure heat addition in

the combustion chamber, an isentropic expansion in the turbine and finally a constant pressure heat rejection in the exhaust nozzle.

Figure 1.2: Idealized Brayton cycle (http://en.wikipedia.org/wiki/Brayton_cycle)

The thermal efficiency of the Brayton cycle is defined as:

$$\eta_{th,Brayton} = \frac{Net\ work}{Heat\ in} = \frac{W_{net}}{Q_{in}} \quad (1.1)$$

In an ideal cycle, the net work output is the subtraction of the work extracted from the isentropic expansion in the turbine by the work used by the compressor to compress the gas whereas the net work input is the heat added by the burning fuel in the combustion chamber. Assuming constant heat capacity, the efficiency of the Brayton cycle can be now expressed by the following equation:

$$\eta_{th,Brayton} = \frac{(h_3 - h_4) - (h_2 - h_1)}{(h_3 - h_2)} = \frac{c_p(T_3 - T_4) - c_p(T_2 - T_1)}{c_p(T_3 - T_2)} = 1 - \frac{T_4 - T_1}{T_3 - T_2} \quad (1.2)$$

Besides processes between 2 and 3 and between 4 and 1 are constants pressure processes:

$$\frac{P_1}{P_2} = \frac{P_4}{P_3} \quad (1.3)$$

Moreover the two others processes are isentropic, therefore:

$$\left(\frac{T_1}{T_2}\right)^{\gamma/(\gamma-1)} = \left(\frac{T_4}{T_3}\right)^{\gamma/(\gamma-1)} \quad (1.4)$$

Finally the thermal efficiency of the Brayton cycle can be expressed as:

$$\eta_{th,Brayton} = 1 - \frac{T_4}{T_3} = 1 - \frac{T_{\text{exhaust gasses}}}{T_{\text{turbine inlet}}} \quad (1.5)$$

By this definition, in order to increase the efficiency of the gas turbine engine, the turbine inlet temperature T_3 has to be increased. As a consequence, to face the increasing demand on energy performance, temperatures of gases exiting the combustor have been increased to temperatures which are now in excess of the maximum sustainable temperatures of the metals in turbine airfoils. These high temperature gases exiting the combustor represent a significant threat to the components of the turbine: first stage vane and blade, especially on the stagnation region where the heat transfer is high. For this reason, engine turbine components must be cooled.

In addition to advances in material science which has been creating new alloys with higher melting temperature, various cooling strategies have been engineered to allow an increase of the turbine inlet temperature and to increase durability of the airfoils.

One method which has been employed is called Thermal Barrier Coating (TBC) and involves covering the external surfaces of vanes and blades with a thin layer of a low conductivity material, such as ceramic, in order to insulate it from the hot mainstream gases. Essentially, this low conductivity material diminishes the amount of heat entering the solid.

The second method of providing thermal protection is internal cooling. Cooled air is extracted from the compressor and is ducted to cool the internal passages of the vanes and blades. As this technique aims at increasing the internal heat transfer coefficient, a perforated plate referred to as an impingement plate is commonly used in the leading edge of the vane to increase the internal heat transfer coefficient through high velocity jets.

Finally, the last method, and the one of interest for this study, is film cooling which consists of discharging cooled air, called coolant, extracted from the compressor, through discrete holes drilled in the walls of the vane. The coolant exiting from the holes is considerably colder than the hot mainstream gas and provides a thin and insulating layer of air along the external surface of the turbine vane, offering thus a thermal barrier to reduce the amount of heat transfer to the wall.

Overall, film cooling as well as internal cooling and TBC allow gas turbines to operate at much higher temperatures than the melting temperature of the vane material.

The use of film cooling has to be optimized. Indeed film cooling requires air to be extracted from the compressor and thus leads to a decrease of the working fluid that passes through the combustor and turbine. As a consequence this results in a loss in the efficiency and power output of the engine. Since the amount of coolant used has to be limited, only the most exposed area is protected. As a result most of the holes are located on the leading edge.

1.2. AIRFOIL COOLING THEORY

In the analysis of film cooling, the main equation to consider is that of convective heat transfer:

$$q'' = h \Delta T \quad (1.6)$$

where q'' is the heat flux per area, h is the heat transfer coefficient and ΔT is the temperature difference providing the potential for heat transfer.

Likewise the heat transfer coefficient with film cooling, h_f , is defined by the following equation:

$$h_f = \frac{q_f''}{(T_{aw} - T_w)} \quad (1.7)$$

where T_{aw} is the temperature of an adiabatic wall with film cooling and T_w is the temperature of the conductive wall. Therefore the adiabatic wall temperature T_{aw} is assumed to be the driving temperature for heat transfer in a conducting wall. This assumption was found by Harrison (2008) to be reasonable on a flat plate except for some regions, generally along the centerline, where a 300% difference between the heat transfer predicted using T_{aw} and actual heat fluxes was observed. This assumption was also validated by Dobrowolski (2010) on a leading edge. Overall the heat transfer was well predicted except between the holes where disparity up to 30% was noticed.

The heat transfer coefficient h_f is often normalized by the heat transfer coefficient with no film cooling h_0 . The ratio obtained h_f/h_0 is named the heat transfer coefficient augmentation.

Moreover the adiabatic wall temperature is usually reported under a non-dimensional parameter, the adiabatic effectiveness, η , defined as:

$$\eta = \frac{T_{aw} - T_{\infty}}{T_c - T_{\infty}} \quad (1.8)$$

where T_c is the coolant jet temperature at the exit of the film cooling hole and T_{∞} is the mainstream flow temperature. A perfect cooling condition ($T_{aw}=T_c$) gives an adiabatic effectiveness η of 1 whereas a no cooling configuration ($T_{aw}=T_{\infty}$) leads to an adiabatic effectiveness η of 0.

In order to quantify the performance of a particular film cooling configuration, these two nondimensional parameters, heat transfer coefficient augmentation h_f/h_0 and adiabatic effectiveness η , are usually analyzed together. Indeed it is important to recognize that the injection of coolant through cooling holes generates turbulence by mixing and vortices by interaction with the mainstream and thus increases the heat transfer coefficients (i.e. $h_f/h_0 > 1$). If the heat transfer coefficient augmentation is too

high, the beneficial effects of film cooling are lost. Therefore, the adiabatic effectiveness levels must be large enough to compensate for this increased heat transfer coefficient. That is why there is a parameter evaluating the total benefit of film cooling, named the net heat flux reduction and defined by Sen et al. (1996) as:

$$\Delta q_r = \frac{q_0 - q_f}{q_0} = 1 - \frac{q_f}{q_0} = 1 - \frac{h_f}{h_0} \left(\frac{T_{aw} - T_w}{T_\infty - T_w} \right) \quad (1.9)$$

where q_f is the heat flux with film cooling and q_0 is the heat flux without film cooling.

For convenience, a non-dimensional surface temperature ϕ is defined as:

$$\phi = \frac{T_\infty - T_w}{T_\infty - T_{c,in}} \quad (1.10)$$

where $T_{c,in}$ is the temperature of the coolant at the entry of the coolant hole. As a consequence, the net heat flux reduction can be simply expressed as:

$$\Delta q_r = 1 - \frac{h_f}{h_0} \left(1 - \frac{\eta}{\phi} \right) \quad (1.11)$$

If the heat transfer into the solid increases because of the presence of film cooling, the net heat flux reduction is negative whereas if the heat transfer decreases, the net heat flux reduction is positive and the metal temperature of the airfoil decreases. Finally evaluating the heat transfer coefficient augmentation is clearly critical to estimate the net heat flux reduction and is the main purpose of this thesis.

In order to make sure that the testing conditions can be compared with real engine operation and to characterize the film cooling performance, several nondimensional parameters are used in film cooling theory. The density ratio of the coolant to the mainstream flow is the first nondimensional parameter of interest and is defined by the following equation:

$$DR = \frac{\rho_c}{\rho_\infty} \quad (1.12)$$

Because of temperature difference between the coolant and the mainstream, there is an inherent density difference. Even though a characteristic real engines density ratio is

around 2, experimental measurements of heat transfer coefficient augmentation with film cooling are generally constrained to unity density ratio tests, and these results have been assumed to be applicable to high density ratio jets. A major objective of this study was to evaluate this assumption. As a consequence, density ratios of 1 and 1.5 were employed to evaluate the effect of the density ratios on the heat transfer coefficient augmentation.

The amount of coolant used to cool the surface is critical in the design process. To determine the flow rate of the coolant, three parameters are commonly used. The first parameter is the blowing ratio, M , relating the mass flux of the coolant to that the mainstream flow. Then the blowing ratio is defined such that:

$$M = \frac{\rho_c U_c}{\rho_\infty U_\infty} \quad (1.13)$$

where U_c is the average velocity at the exit of the coolant holes. This parameter is usually used when the coolants jets stay attached to the surface of the vane.

The second parameter commonly used to describe a cooling configuration is the momentum flux ratio, I , providing a ratio of the momentum fluxes of the coolant to the mainstream and defined as:

$$I = \frac{\rho_c U_c^2}{\rho_\infty U_\infty^2} \quad (1.14)$$

This parameter is often used if the separation of the coolants jets from the surface of the vane is likely. Finally the last parameter that is used to characterize the coolant flow rate is the velocity ratio, VR , defined as the velocity of the coolant jet divided by the velocity of the mainstream flow:

$$VR = \frac{U_c}{U_\infty} \quad (1.15)$$

1.3. EXPERIMENTAL METHOD OF HEAT TRANSFER MEASUREMENT

Many studies have been conducted in the past to measure the heat transfer coefficient. Three different methods exist: a transient liquid crystal technique; a heat-mass transfer analogy technique; a heat flux foil technique. The most commonly used method is the last one which consists in applying a heat flux on an external surface thanks to a thin foil. This same method was used in the simulations presented in this study. Heat transfer coefficients are determined at a unit density ratio in order to reduce the uncertainty on the driving temperature measurement. Indeed if the mainstream and the coolant temperature is equal (i.e. $DR = 1$), the driving temperature of the heat transfer simply is the mainstream temperature. However at higher density ratio, the adiabatic wall temperature is the driving temperature. The uncertainty in the measurement of the adiabatic wall temperature is greater than the uncertainty in the measurement of the mainstream temperature. At a density ratio of $DR = 1$, the equation to calculate the heat transfer coefficient becomes:

$$h_f = \frac{q_f''}{(T_w - T_\infty)} \quad (1.16)$$

Heat flux foil studies have been mainly conducted at a unit density ratio. However some studies were able to achieve higher density ratios by using a foreign gas as the coolant. For example, carbon dioxide was often used in order to obtain a density of $DR = 1.5$. However the effect of using a coolant with different fluid properties was not studied in these studies. For example, the thermal conductivity of carbon dioxide is half that of air at room temperature. This effect was studied in the present study.

1.4. LITERATURE REVIEW

The present study investigated CFD predictions of heat transfer coefficient on a cylindrical leading edge. As a consequence, this literature review is focused on experimental studies and computational predictions of heat transfer coefficient without film cooling as well as heat transfer coefficient augmentation.

1.4.1. Experimental study of the heat transfer coefficient augmentation

Many experimental studies have been carried out in the past to study the heat transfer coefficient augmentation. Several flow condition parameters such as the turbulence intensity and the shape of the holes have been tested.

Mick and Mayle (1988) conducted heat transfer measurements, using a constant heat flux surface, for secondary air injection through three row of holes (± 15 degrees and $+ 44$ degrees from the stagnation line) into the stagnation region of a circular leading edge and a flat afterbody. The holes were angled 30 degrees to surface in the spanwise direction and 90 degrees in the streamwise direction. For the three different blowing ratio ($M = 0.38$, $M = 0.64$ and $M = 0.97$), heat transfer coefficients near the holes with injection were found to be as much as two times those without. Far downstream, the heat transfer coefficients remained about 10 percent above that without injection.

Karni and Goldstein (1990) used a naphthalene sublimation technique to study the effect of surface injection on the mass transfer (i.e. heat transfer by analogy) from a circular cylinder with a single row of cooling jets in a cross-flow. Downstream of the cooling jets, the spanwise average of mass transfer was found to be increased by a factor of two over that without film cooling at a blowing ratio of $M = 2$, and the size of the elevated mass transfer region was observed to increase with the blowing ratio.

Mehendale and Han (1992) studied the influence of high mainstream turbulence on leading edge heat transfer coefficient. High mainstream turbulence was produced by a

passive grid and a jet grid ($Tu = 9.67$ and 12.9% respectively). These high turbulence results were compared with low turbulence ($Tu = 0.4\%$) one. Experiments were performed using a blunt body with a semi cylinder leading edge with a flat afterbody. Spanwise and streamwise distributions of heat transfer coefficient were obtained for three different blowing ratios ($M = 0.4, 0.8$ and 1.2) through four rows of holes located at ± 15 and ± 40 deg from the stagnation line. The results indicated that the heat transfer coefficient with film cooling increased with increasing blowing ratio. Also the heat transfer coefficient for a blowing ratio of $M = 0.8$ increased with mainstream turbulence: this increase was up to 30% between the $Tu = 0.4\%$ case and the $Tu = 12.9\%$ case. However for the two other blowing ratios ($M = 0.4$ and $M = 1.2$) the effect was not as systematic as for $M = 0.8$. Moreover the heat transfer coefficient in the leading edge region with no film cooling was found to increase by 50% with increasing mainstream turbulence level when compared to the low turbulence case. The effect diminished at the end of the flat sidewall due to the decay of mainstream turbulence.

Several studies have been conducted at the Turbulence and Turbine Cooling Research Laboratory (TTCRL) at the University of Texas at Austin on the measurement of heat transfer on the leading edge of a turbine airfoil.

Yuki (1998) examined heat transfer to a model turbine blade leading edge. The leading edge was a quarter-cylinder with a flat afterbody. The heat transfer tests were run at a density ratio of $DR = 1.0$. A thin stainless steel sheet was attached to the leading edge to provide a heat flux surface. Curved copper bars were attached on either end of the plate such that electrical current flowed spanwise.

On the stagnation line, the heat transfer coefficient was found to be increased by 80% for a blowing ratio $M = 1.9$. Downstream of the off-stagnation holes, the heat transfer coefficient was more than doubled. The maximum values of heat transfer

coefficient were observed at two different regions, one on the coolant jet path and the other on the side of the coolant jet. Thermal field measurements revealed this was probably due to a streamwise vortex formed because of the interaction of coolant jet with the mainstream flow. When these data were combined with adiabatic surface effectiveness, η , for a constant momentum flux ratio, to produce net heat flux reduction results, it was found that film cooling on this geometry would reduce heat transfer to the exit of the coolant jet while increase heat transfer to the side of the coolant jet. Indeed, at this last location, even if the coolant jet cools the surface, the heat transfer coefficient was enhanced by the vortical motion described previously, causing then additional heat transfer. Moreover Yuki (1998) found that larger blowing ratios increased the heat transfer coefficient augmentation, h_f/h_0 .

Johnston (1999) used the same facility as Yuki and studied high mainstream turbulence effects on film cooling of a leading edge, especially on the heat transfer. Comparisons of high turbulence (turbulence intensity equal to 17 percent) results with those under low turbulence ($Tu < 0.5\%$) displayed a more rapidly diminishing effect of coolant injection in the downstream direction. Additionally laterally averaged heat transfer coefficient augmentation values displayed decreases of 10 to 20 percent for all blowing ratios with the addition of high mainstream turbulence. The decreases were shown to be due to the increased heat transfer coefficient without film cooling, h_0 , even if slightly higher heat transfer rates due to film cooling, h_f , were also noticed. Indeed h_0 at high turbulence was 40 percent greater than at low turbulence.

Unlike Yuki (1998) and Johnston (1999), Mouzon (2005) used a semi-cylindrical model to study the heat transfer coefficient on a leading edge with forward diffused coolant holes. These experiments were carried out at a density ratio of $DR = 1.0$ and a “two-foil” method comprising two strips of stainless steel was used to generate the heat

flux on the surface. This method consisted of a small foil that ran between the stagnation and off-stagnation row of holes. The larger foil started just beyond the off-stagnation row of holes. It was found that shaped coolant holes increased heat transfer coefficient augmentation with increasing blowing ratios over the range $M = 0.5$ to $M = 2.0$ at low and high turbulence intensity. For example this increase was around 25% between $M = 1.0$ and $M = 2.0$. Also, like Johnston (1999), Mouzon (2005) found that the heat transfer coefficient augmentation was decreased by 15 to 25% for the high mainstream turbulence intensity compared to the low mainstream turbulence intensity case. Moreover shaped hole heat transfer coefficient augmentation were found to be surprisingly increased by 10% for a blowing ratio of $M = 1.5$ in comparison to that measured by previous studies using round holes. Indeed it was presumed that the shaped holes offered the advantage of reducing the disturbance to the boundary layer, thereby reducing the heat transfer coefficient augmentation.

A second type of heat flux foil was manufactured by Mouzon (2005). In this “full-foil” technique, holes were cut out of the sheet to correspond to the location of the coolant holes. The bus bars were attached at opposite ends in the streamwise direction (i.e. downstream of the two rows of off-stagnation coolant holes). The current flow restrictions created laterally in between the coolant resulted in small regions of very high heat flux. In the end, it was decided by the author that the results had too high of an uncertainty to be used.

In conclusion, even though the studies presented previously used different geometry of leading edge and of coolant holes, there is a general agreement on the values of the laterally averaged heat transfer coefficient values and on the effect of the mainstream turbulence. The heat transfer coefficient was commonly found to be augmented by as much as a factor of 2. However some studies exhibited discrepancies in

terms of peak of heat transfer coefficient values. Lastly, any of these previous studies showed the heat transfer coefficient augmentation scaled with the momentum flux ratio.

1.4.2. Experimental study of the effect of density ratio on the heat transfer coefficient

Few studies measuring the impact of density on the heat transfer have been conducted. Ammari (1990) studied the effect of density ratio on the heat transfer coefficient from a film-cooled flat plate using a heat-mass transfer analogy. In addition to a density ratio of $DR = 1$, density ratios of $DR = 1.38$ and $DR = 1.52$ were achieved by using foreign gases. For normal injection, the heat transfer coefficient at a given blowing parameter was found to be insensitive to the variation of density ratio, whereas for 35 degrees injection, a dependence was observed. Downstream of the holes, an increase in density ratio was seen to result in a decrease in the heat transfer coefficient augmentation at all blowing ratios: difference in laterally averaged heat transfer coefficient augmentation as high as 8% at a blowing ratio of $M = 1.0$ and 18% at a blowing ratio of $M = 2.0$ were observed. Indeed as the blowing ratio increases, the difference in momentum between the heavier and lighter gas increased, and this was reflected in a corresponding increase in the difference in heat transfer coefficient augmentation between the two sets of results. However at a low blowing ratio of $M = 0.5$, which usually gives the maximum effectiveness on a flat plate configuration, the influence of density ratio on heat transfer coefficient was small. It should be noted that if the momentum flux ratio, I , and not the blowing ratio, M , would have been matched, no change in h_f/h_0 would have been noticed. Indeed, even though a perfect match of the momentum flux ratio is not possible afterwards because of the blowing ratios selected experimentally, the results at $I = 2.25$ ($DR = 1$ and $M = 1.5$) and at $I = 2.67$ ($DR = 1.5$ and $M = 2$) were very similar.

Ekkad et al. (1998) used a transient liquid crystal technique to measure the heat transfer coefficient and carbon dioxide (CO_2) gas for coolant to study the effect of coolant density on the heat transfer coefficient on a cylindrical leading edge. The injection of CO_2 was performed through two rows of cylindrical holes located at $\pm 15^\circ$ from the stagnation line. Two different density ratios ($DR = 1$ and 1.5) were tested at three different blowing ratios ($M = 0.4, 0.8$ and 1.2). The heat transfer was found to be reduced by 10% at all blowing ratios when the density ratio was increased from $DR = 1.0$ to $DR = 1.5$. Indeed it was justified that since higher density coolant is heavier than the mainstream fluid, this reduced coolant jet to mainstream interaction, thus reducing turbulent mixing, which in turn produced lower heat transfer coefficients compared to a coolant with the same density as mainstream. From a momentum flux ratio point of view, CO_2 as coolant has lower momentum flux ratio compared to air at the same blowing ratio. The momentum flux ratio and the velocity ratio were not matched when comparing the two density ratios. However heat transfer coefficients with film cooling were found to increase with increasing momentum flux ratios whatever the density ratio was.

1.4.3. Computational turbulence models

Computational Fluid Dynamics (CFD) has become a critical part of engineering design. The choice of the turbulence model in CFD is important to obtain results as accurate as possible. There are four major turbulence simulation techniques: Steady Reynolds-Averaged Navier Stokes (RANS), Unsteady Reynolds-Averaged Navier Stokes (URANS), Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS).

RANS is the most widely used in CFD. It requires models for turbulence and often other transport equations. Because the averaged Navier Stokes equation is not closed, quantities such the Reynolds stress have to be modeled. Unlike RANS models,

URANS models are able to capture fluctuations or unsteady behavior in the mean quantities.

DNS is the most precise turbulence simulation approach because the Navier-Stokes equations are numerically solved without any turbulence model. All the eddy length scales are resolved. As a consequence DNS computational cost is very high and is thus impractical to complex engineering problems. Unlike DNS, LES resolves only the large scales of the solution, while the small eddies are modeled, in order to reduce the computational cost of the simulation. Because they are computationally expensive, both LES and DNS are rarely employed nowadays in the industry.

Whereas many computational studies of film cooling conducted in the past focused on prediction of the adiabatic effectiveness, only a few are available on the prediction of heat transfer coefficient. While there is no comparison of CFD codes on a leading edge, there exist some studies for flat plate and full vane geometries.

Three turbulence models (realizable k- ϵ model (RKE), standard k- ω (SKW) and Reynolds's stress model (RSM)) were compared by Harrison and Bogard (2008) to seek the most accurate turbulence model to predict adiabatic effectiveness and heat transfer coefficients on a flat plate with film cooling. These three turbulence models gave reasonable results. The laterally averaged h_f/h_0 distributions were reasonably predicted by the three models: the difference with experimental data was less than 10%. However the centerline h_f/h_0 was under-predicted and the lateral variations in h_f/h_0 were over-predicted. For example, with the RKE model at a blowing ratio of $M=0.5$, the centerline and spanwise h_f/h_0 predictions were found to be respectively 15% lower and 10% higher in comparison with experimental values. Overall the RSM model was found to result in the least under-predicted centerline h_f/h_0 values with a difference of 5 %.

Zhang and Hassan (2006) compared CFD centerline heat transfer coefficient augmentation predictions on a flat plate with experimental data from Eriksen (1971) at a blowing ratio of $M = 1$. The standard $k-\omega$ (SKW), the Reynolds-Stress (RSM), the Spalart-Allamaras (SA) and the standard $k-\varepsilon$ (RKE) turbulence models were tested. The RSM model gave the best results in the near hole region (a few percent difference with experimental data) but resulted in 10% lower heat transfer coefficient augmentations values far from the hole. The SKW and SA gave similar results. They both resulted in slightly underpredicted values of 5-10% close to the hole and in overpredicted values of 5% away from the hole. The RKE model was found to result in the closest agreement with experiments. It gave underpredicted values of 5% in the near hole region but then beyond this region gave excellent result. Additionally the performance of different wall treatments (standard wall functions, non-equilibrium wall functions and enhanced wall treatment) with the RKE model was compared. An insignificant difference in centerline h_f/h_0 was found.

Luo and Razinsky (2007) compared the performance of the V2F turbulence model developed by Medic and Durbin (2002) with a low-Reynolds-number $k-\varepsilon$ model and a nonlinear quadratic $k-\varepsilon$ model on a C3X cooled turbine vane. The V2F model is similar to the standard $k-\varepsilon$ model but also models near-wall turbulence by using an elliptic relaxation methodology. Predictions of heat transfer coefficients on the pressure and suction sides of the vane for the low-Reynolds-number $k-\varepsilon$ model and the nonlinear quadratic $k-\varepsilon$ model were found to be higher up to 50-75% compared to experimental data because of over production of turbulent kinetic energy near the stagnation point. However V2F model gave closer and more reasonable results with 10% discrepancies on the pressure side and 25% on the suction sides compared to measured values. Additionally the V2F model over-predicted by 40% the heat transfer coefficient at the

stagnation line whereas it was over-predicted by 100% by the $k-\epsilon$ model. Nevertheless all the models over-predicted the heat transfer coefficients at the end of transition and in the turbulent flow region. The ability of a CFD model to correctly predict heat transfer coefficient is strongly dependent on its capacity in predicting the boundary layer transition and turbulent heat transfer.

Dees et al. (2010) performed an experimental and computational conjugate heat transfer study of an internally cooled C3X simulated turbine vane, i.e. on the same vane as in Luo and Razinsky's study. At a turbulent intensity of $Tu = 20\%$, the heat transfer coefficient at the stagnation line predicted by the standard $k-\omega$ model was 20% higher when compared to the experiment and the prediction agreed well with measured heat coefficient values on the suction side of the airfoil after the boundary layer transition.

To conclude the review of Luo and Razinsky (2007) and Dees et al. (2010) studies, the V2F model was found to give satisfactory predictions of the heat transfer coefficient. However the improvement at the stagnation line was not significant when compared to the standard $k-\omega$ model. Additionally only one study with the V2F turbulence model were conducted on a film cooling configuration. As a consequence this model was not considered in the present study.

Finally modified versions of $k-\epsilon$ and $k-\omega$ models called the realizable $k-\epsilon$ (RKE) model and Shear Stress Transport (SST) $k-\omega$ model were used in the present study because, as underlined in the above background, they are found to be the most accurate turbulence models tested for film cooling simulations.

The RKE model developed by Shih et al. (1995) is a modified version of the standard $k-\epsilon$ model. It includes two modifications: a new formulation for the turbulent viscosity and a new transport equation for the dissipation rate, ϵ , based on the dynamic equation of the mean-square vorticity fluctuation. The RKE model ensures positivity of

normal stresses and Schwarz inequality for shear stresses. This model has been shown to provide a better prediction of separated flows and flows with complex secondary flow features. Walters and Leylek (2000) showed the superior performance of the RKE model over the standard k- ϵ turbulence model by comparing loss coefficient prediction in a simulation of a turbine airfoil cascade.

The SST model developed by Menter (1994) combines the k- ϵ and k- ω models in order to overcome the weaknesses of both. Indeed this model blends the robustness of the standard k- ω model in the inner parts of the boundary layer with the free-stream independence of the standard k- ϵ model. A blending function is incorporated in this model and allows switching between the k- ω model in the viscous sublayer and log layer parts of the boundary layer and the k- ϵ model in the outer region of the boundary layer and in freestream flow. This model was shown to better predict flow in adverse pressure gradients and separating flow in the Menter's study.

1.4.4. Computational heat transfer coefficient without film cooling

Few simulations have been conducted to study the heat transfer coefficient around a cylinder or a cylindrical leading edge. Usually the heat transfer coefficient without film cooling is presented in terms of a non-dimensional quantity: the Frossling number defined as:

$$Fr = \frac{Nu_D}{Re_D^{1/2}} \quad (1.17)$$

York and Leylek (2002) conducted realizable k- ϵ heat transfer coefficient simulations on a leading edge model that is similar to the one used in this study. Frossling values were compared to experimental values obtained by Geidt (1949) which showed experimentally that the Frossling number is equal to unity at the stagnation line of a cylinder-in-crossflow. The computational Frossling number value at the stagnation line

was found to be over predicted by 5% compared the experimental one. Moreover, downstream, from an angle of 45 degrees, the numerical prediction exhibited increasing larger value of Frossling number compared to experimental data. York and Leylek (2002) stated that the downstream increase of the Frossling number is due to the large blockage in the computations owing to passage geometry and causes much stronger acceleration of the mainstream than in the study of Geidt (1949). Indeed acceleration due the blockage thins the boundary layers and as a consequence increases heat transfer.

Szcepanik et al. (2004) performed a numerical study of heat transfer from a cylinder in cross flow using two dimensional steady state RANS (Reynolds-Averaged Navier Stokes) simulations. The standard $k-\omega$ turbulence model as well as a $k-\omega$ model for which the calculation of some turbulent quantities had been modified, were compared to experimental data. The modified $k-\omega$ turbulence model was born from the tendency of two equation turbulence models to over-predict levels of turbulent kinetic energy close to a stagnation point. The modified $k-\omega$ turbulence model used in the study from Szcepanik et al. (2004) was the one proposed by Medic and Durbin (2002). The modifications aim at reducing the calculated levels of turbulence by either increasing the production of dissipation ω or decreasing the production of turbulent kinetic energy k . Basically Durbin and Medic's modification developed a limiting criterion for this turbulent time scale because they had observed high values of the turbulent time scale T_t (defined as $T_t = 1/(C_\mu \omega)$ in an unmodified $k-\omega$ turbulence model) as well as k close to the stagnation line. The purpose of this limiting criterion on T_t is to limit the turbulent viscosity which will result in a change of the production and dissipation of k and ω . Szcepanik et al. (2004) observed improved heat transfer results by using this T-limited model. At a Reynolds number of $Re = 21,580$ the stagnation point heat transfer coefficient was found to be reduced by 40% compared to the unmodified $k-\omega$ turbulence model and to be 10%

lower than the experimental value. This reduction in stagnation point heat transfer was caused by reduced levels of k at the stagnation point.

1.4.5. Computational predictions of heat transfer coefficient augmentation due to film cooling

Whereas the heat transfer coefficient augmentation on a flat plate is close to unity, the leading edge experiences larger heat transfer coefficient augmentation values up to 2 at certain locations downstream of the coolant holes. As a consequence predicting them accurately is more critical for this geometry.

Of numerical studies of leading edge film cooling, only three predictions of the heat transfer coefficient are available in the open literature. York and Leylek (2002) conducted realizable k - ϵ heat transfer coefficient simulations on a leading edge model identical to the one used by Yuki et al. (1998) and Johnston et al. (1999), i.e. a quarter-cylinder with a flat afterbody. A two-layer near wall treatment was used allowing the resolution of the viscous sublayer. The heat transfer coefficient with film cooling was obtained by specifying a constant heat flux on the leading edge surface. It should be noted that, in this study, a two-dimensional simulation was run to calculate the heat transfer coefficient without film cooling and then to report the heat transfer coefficient augmentation. Two separate regions of high heat transfer coefficient were predicted computationally downstream of the off-stagnation holes whereas only one large region of elevated heat transfer coefficients is observed experimentally. Moreover the numerical prediction of the laterally averaged heat transfer coefficient augmentation exhibited good agreement for low blowing ratios with experimental data from Yuki et al. (1998). However, regardless of the blowing ratio, up to 10 % higher heat transfer coefficient augmentation values were predicted directly after the off-stagnation hole. Additionally, computational values of heat transfer coefficient augmentation in the area between two

stagnation holes were appreciably lower than the values experimentally measured at the same location: for example, at $M = 1.5$, h_f/h_0 is around 1.25 for the computational prediction whereas h_f/h_0 is about 1.7 for the experiment.

Rozati and Tafti (2007) conducted a numerical investigation of leading edge film cooling with Large Eddy Simulation (LES). The leading edge was the one Ekkad et al. (1998) used experimentally (i.e. two rows of coolant holes located at ± 15 deg from the stagnation line). Downstream of the coolant holes, prediction of laterally averaged Frossling number were found to be within the experimental studies of Ekkad's study for blowing ratios of $M = 0.4$ and $M = 0.8$. However, at a blowing ratio of $M = 1.2$, a larger difference with experimental data was noticed, up to 10% about 1.5 d downstream of the coolant holes.

Like the York and Leylek's realizable k- ϵ study, LES computations showed two regions of high heat transfer coefficient augmentation values downstream of the coolant holes for all blowing ratios.

In addition they were able to identify the influential coherent structures in the flow. The following definitions were used to describe the location of flow and structures: in the lateral direction, the side that the jet blows from is specified with prefix "aft" and the side the jet blows to, specified with prefix "fore". In the streamwise direction along the blade surface, "leeward" is used to denote the downstream side of the jet and "windward" the upstream side of the jet.

At a low blowing ratio of $M = 0.4$, three types of coherent structures were recognized: a primary entrainment vortex at the leeward aft-side of the coolant hole; vortex tubes at the windward side of the coolant hole; and hairpin vortices typical of turbulent boundary layers produced by the turbulent interaction of the coolant and mainstream downstream of injection. The primary vortex dynamics were described

further. At the leeward edge of the coolant hole, a low pressure region formed, which resulted in a primary vortex that draws the flow from the aft-side. This primary vortex was acknowledged to be the key element in the mainstream entrainment underneath the coolant and to be responsible for the high values of Frossling number observed at the aft-side of the coolant hole.

The effect of blowing ratio on the mechanism of jet-mainstream interaction was also studied. It was shown that the combined effect of a larger turbulent shear layer and a stronger mainstream entrainment, as the blowing ratio increases, caused higher heat transfer coefficient value.

1.5. OBJECTIVES OF CURRENT STUDY

This thesis has two main objectives. The first is to compare the heat transfer coefficient augmentation prediction for a simulated film cooled turbine blade leading edge obtained with the realizable k- ϵ turbulence model as well as the k- ω SST turbulence model with experimental data available in the literature. This is accomplished notably by comparing the contour plots of h_f/h_0 , the laterally averaged h_f/h_0 values and the h_f/h_0 distributions. Moreover an investigation into the interaction flow physics between the mainstream flow and the coolant jets was conducted in order to determine mechanisms causing enhancement of the heat transfer.

This thesis also explores the validity of some experimental techniques to determine the heat transfer coefficient augmentation, specifically the use of a heat flux foil that generates a non-uniform heat flux and the use of a unit density ratio. Indeed there exists an assumption in the literature that the heat transfer coefficient values obtained experimentally at a unit density ratio are valid also for higher density ratios. Velocity

ratio, blowing ratio and momentum flux ratio were used as points of reference and two density ratios ($DR = 1.0$ and $DR = 1.5$) were compared to study the coolant density effect.

Chapter 2: Computational setup

An objective of this thesis was to compare the CFD prediction of the heat transfer coefficient augmentation with experimental results. Therefore, the computational leading edge model had similar geometry and flow conditions to the experimental leading edge model. This chapter will describe the computational domain used and the operating parameters and boundary conditions specified and finally will describe how the grid independence of the solution was verified. The simulations were run in FLUENT, version 12.1, a CFD software package based on finite volume method.

2.1. DESCRIPTION OF THE MODEL

Since one of the purposes of the present study was to compare the computational prediction of heat transfer coefficients with experimental data, the geometry of the leading edge used in these simulations was identical to the computational study carried out by Dobrowolski et al. (2009) and as close as possible to the physical model employed by Yuki et al. (1998) and Johnston et al. (1999). However it should be pointed out that these studies had a quarter circle model with a suction slot on the bottom whereas the present study used a half cylinder model.

As indicated in the schematic of the geometry shown in Figure 2.1, the outer diameter and the inner diameter of the leading edge were respectively 50.8 mm and 25.4 mm. A 10.9 mm long flat portion was located downstream of the half-cylindrical part. In most of the simulations conducted in the present study, an impingement plate was employed and had holes 5 mm in diameter and directed the coolant from the plenum towards the internal surface of the model.

Figure 2.1: Schematic of the leading edge model (from Mathew (2010))

The film cooling configuration included three rows of holes positioned along the stagnation line and at 25 degrees from the stagnation line. The coolant hole diameter was $d = 3.18$ mm. The holes were angled at 20 degrees to the surface and were oriented normally to the flow direction (i.e. with a 90 degrees compound angle). They were located with a hole-to-hole pitch of $p = 24.2$ mm i.e. $p/d = 7.6$ as illustrated in Figure 2.2.

Figure 2.2: Front view of the leading edge model (from Mathew (2010))

However it should be recognized that the model presented in Figure 2.1 and Figure 2.2 is not the computational domain. Figure 2.3 shows the 3D computational domain used in the present study. Periodic and symmetric conditions were applied to save computational time. Periodic boundaries spaced by a hole-to-hole pitch distance, was applied on the side walls. A symmetry boundary condition was applied at the stagnation line so that only half of the leading edge was simulated. The top of the wind tunnel was modeled with a no-slip boundary condition.

Figure 2.3: Schematic of the 3D computational domain (from Ravelli et al. (2010))

The dimensions of the computational domain are presented in Figure 2.4. Mathew (2010) showed that using an extended domain, that had the freestream inlet boundary

condition located at 163 mm from the leading edge instead of 63.7 mm, gave similar results to those using the present domain.

Figure 2.4: Dimensions of the 3D computational domain (from Mathew (2010))

2.2. OPERATING PARAMETERS AND BOUNDARY CONDITIONS

The operating parameters and boundary conditions were chosen in order to match those in the wind tunnel from previous recent experiments conducted at the TTCRL. The mainstream inlet velocity was set up as a constant velocity profile at 15 m/s. A turbulence intensity of $Tu = 5.9\%$ and a turbulence length scale of $\Lambda = 19$ mm were set at the inlet. However, in some simulations, these turbulence parameters were adjusted to match the experimental values from the Yuki et al. and Johnston studies. Even though the coolant turbulence intensity was not measured experimentally, it was presumed to be 1% because the coolant comes from a large reservoir. The coolant turbulent length scale was also assumed to be half of the diameter of the impingement holes. A mass flow inlet condition

was specified through the coolant hole of the impingement plate. The coolant mass flow was varied to match the average blowing ratio M desired. Table 2.1 summarizes the boundary conditions used in these simulations.

Table 2.1: Boundary conditions for the CFD model

Inlet mainstream velocity (m/s)	15
Inlet mainstream temperature (K)	300
Inlet mainstream turbulence intensity (%)	5.9
Inlet mainstream turbulent length scale (mm)	19
Coolant temperature (K)	300
Coolant mass flow rate (kg/s)	4.1×10^{-4} for $M=2$
Coolant turbulence intensity (%)	1
Coolant turbulent length scale (mm)	2.5

Finally the Reynolds number in these simulations ($Re = 48,500$) was somewhat different from the Reynolds number at which Yuki et al. (1998) and Johnston et al. (1999) ran their experiments ($Re = 65,000$) because the simulations were initially run to match the Reynolds number from previous recent experiments at the TTCRL and not from Yuki et al. and Johnston et al.'s studies. As a consequence, the Reynolds number was not matched mistakenly. However it was justified later in the present study that the Reynolds number does not have a significant impact on the heat transfer coefficient augmentation.

A constant heat flux of $q'' = 6,000 \text{ W}/(\text{m}^2 \cdot \text{K})$ was applied on the external surface of the leading edge. The local heat transfer coefficient was then computed using its definition:

$$h = \frac{q''}{(T_w - T_\infty)} = \frac{6,000 \text{ W}/(\text{m}^2 \cdot \text{K})}{(T_w - 300\text{K})} \quad (2.1)$$

The internal surfaces of the leading edge as well as the impingement plate and the glue layer were set up as adiabatic walls.

Since the air and coolant temperature ranged from 300K to 370K because of the heating on the external surface, fluid properties such as density, thermal conductivity, viscosity and specific heat were made temperature dependent.

2.3. VARYING DENSITY RATIO SIMULATIONS

Simulations were run at two different density ratios: $DR = 1.0$ and $DR = 1.5$. For the density ratio of $DR = 1.0$ cases, the coolant had the exact same properties of the mainstream flow that is to say of air. However, to achieve a density ratio of 1.5, the coolant density had to be 1.5 times larger but at the same time the coolant temperature had to still be at 300K to eliminate ambiguity on the driving temperature. As a consequence, a different density equation was used for the coolant. However other coolant proprieties such as viscosity, thermal conductivity and specific heat were still those of air.

The density equation used for the coolant was the incompressible ideal gas equation. The molecular weight was set up at 43.449 kg/kmol so that the density was 1.5 times larger at 300K. Actually, different density ratios were achievable as long as the molecular weight of the coolant gas is selected appropriately.

Since the $DR = 1.5$ simulations implied the mixing and the transport of two different gases, the “species transport without reactions” modeling option was activated on FLUENT. This option enables the calculation of multi-species transport equations predicting the local mass fraction of each species. The mixture properties were defined as ideal gas mixing law properties that is to say the local mass fraction of coolant was considered to calculate the local properties of the mixture.

The uniqueness of these simulations at a non unit density ratio lies in the fact that the conditions imposed computationally are not achievable experimentally. As a consequence these simulations were very valuable to isolate the effects of density ratio on the heat transfer coefficient.

2.4. TURBULENCE MODEL SPECIFICATIONS

As justified in the first chapter, two turbulence models from FLUENT were used in the present study: the realizable k - ϵ (RKE) model and the Shear Stress Transport (SST) k - ω model because of their superior performance shown in the literature review. Both models were run with Green-Gauss node based, segregated, steady state and implicit solvers. Also, a pressure based solver with a SIMPLE pressure-velocity coupling method was used.

Usually simulations were started as laminar flow and then after 500 iterations they were switched to turbulent flow (RKE or SST) with second order accuracy. This method allowed the residuals to be low enough when switching to the turbulence model.

For RKE, the enhanced wall treatment option was used in order to resolve the turbulent boundary layer all the down to the viscous sub-layer. Pressure gradient effects and thermal effects options were also selected.

The low-Reynolds-number correction option was activated for SST. That feature, also named transitional flows option, allows the correction of the turbulent viscosity by a damping coefficient. Indeed two equation turbulence models, such as k - ω , have the tendency to over-predict levels of turbulence kinetic energy close to a stagnation point that causes an over-prediction of the heat transfer coefficient. The low-Reynolds-number

correction option has an effect on the production and dissipation of k and ω and thus on the heat transfer coefficient by reducing the level the turbulent viscosity.

2.5. CONVERGENCE

In order to estimate the convergence of the simulations, the residuals are commonly kept below 10^{-9} for energy, below 10^{-4} for ω and below 10^{-6} for all other residuals. However the use of these criteria to estimate the convergence is insufficient. Indeed residuals are the ratio of the difference between two successive iterations to the first iteration and could be artificially low since the converged solution differed significantly from the initial guess which is the freestream inlet conditions. As a consequence a second convergence criterion was used. At convergence, the spanwisely average of the heat transfer coefficient at three different locations ($s/d = 1$, $s/d = 5$ and $s/d = 9$) changed by no more than 0.01% for at least 200 iterations. The value of 0.01% was chosen low enough so that it eliminated the possible slow fluctuations.

2.6. GRID CONSTRUCTION

The GAMBIT grid generator was used to construct an unstructured tetrahedral mesh. Because of the complexity of the geometry (sharp angles, strong curvature), it was only possible to mesh the domain with tetrahedral cells. Even if a hexahedral meshing is recognized to give faster converging results, it resulted in highly skewed cells. However, high grid resolution was required in order to better capture near wall effects. Capturing the turbulent boundary layer all the way down to the viscous sub-layer was necessary for predicting the growth of the coolant separation and the heat transfer from the wall. As a consequence, a prism layer was attached to the external surface of the leading edge. The

first cell was located at a distance close enough to the surface so that the first cell node was inside the viscous sub-layer: a y^+ value of 1 was desired. Indeed the enhanced wall treatment employed in the RKE turbulence model as well as the low-Reynolds-number correction option for SST required a y^+ value at the wall-adjacent cell on the order of 1. The y^+ , called, the non dimensional wall distance, is a parameter that defines the structure of the structure of the prism layer and is defined by the following equation:

$$y^+ = \frac{\rho u_\tau y}{\mu} \quad (2.2)$$

where y is the dimensional wall distance, μ the dynamic viscosity and u_τ the frictional velocity ($u_\tau = \tau_w/\rho$ where τ_w is the wall shear stress and ρ is the density).

2.7. GRID INDEPENDENCE

Grid independence was established in two steps that needed to be carried out in parallel. The first step was to prove the grid independence outside of the prism layer and the second one was to determine the prism layer independence. Moreover the overall grid independence was shown for two grids corresponding to the two turbulence models used in this study.

2.7.1. RKE grid independence

In order to establish the grid independence outside of the prism layer for the grid used for the realizable k- ϵ turbulence model, three different grids were developed. They had different total number of cells (from 3.3 to 10.8 million cells) but had the same prism layer (the one determined by the second step).

The spanwise heat transfer coefficient distribution downstream of the off-stagnation holes (i.e. at $s/d = 5$) was used to establish the grid independence. Indeed

looking at the spanwise average of the heat transfer coefficient was found to be insufficient to evaluate the independence of the grid.

Figure 2.5: Comparison of the heat transfer coefficient lateral distributions at $s/d = 5$ for different numbers of cells

It should be noted, based on Figure 2.5, that the first mesh of 3.3 million cells had first peak values in heat transfer coefficient that were too low compared to the other grids. However the second and third meshes were in good agreement. For these two meshes, the values of the first peak were within 0.4% and the values of the second peak were within 0.6%. Finally the 7.0 million cells grid was used for production simulations.

Then the prism layer independence was evaluated. Three parameters are characteristics of a prism layer: the first row size, the growth rate and the total height. The first row size was set at 0.01mm so that the y^+ value was about 1. The maximum y^+ value was found to be 1.2.

Then, as shown in Table 2.2, different heights of prism layer were tested on 7.0 million cells grid which had been shown as grid independent outside the prism layer by the first step.

Table 2.2: Characteristics of the prism layers tested for the RKE turbulence model

	Mesh#1	Mesh#2	Mesh#3	Mesh#4	Mesh#5
First cell size (mm)	0.01	0.01	0.01	0.01	0.01
Growth factor	1.2	1.2	1.2	1.2	1.1
Number of rows	12	16	20	22	31
Prism layer height (mm)	0.4	0.9	1.9	2.7	1.8

As shown in Figure 2.6, the total height of the prism layer had a strong effect on the value of the second peak of the heat transfer coefficient distribution. The prism layers with a total height of 0.4 mm and 0.9 mm were apparently not thick enough to capture the thermal boundary layer. However a prism layer with a total height of 1.9 mm was enough for this purpose since the prism layer with a total height of 2.7 mm gave results in good agreement with the smaller prism layer: the values of the second peak are within 1.3% difference for these two prism layers. Indeed the thermal boundary layer at the second peak position was found to be 4 mm thick as shown in Figure 2.7.

Lastly the growth rate was set at 1.2 as it was found that a value of 1.1 or 1.2 had no impact of the heat transfer coefficient values as long as the total height of the prism layer is constant. Indeed results from mesh#3 and mesh#5 were identical.

Figure 2.6: Comparison of RKE heat transfer coefficient lateral distributions at $s/d = 5$ for four different heights of the prism layer

Figure 2.7: Thermal boundary layer at $s/d = 5$ and at $z/d = 5.5$

2.7.2. SST grid independence

The same methodology was used to evaluate the grid independence for the $k-\omega$ turbulence model. First the prism layer independence was asserted. According to the FLUENT manual (2009), the SST $k-\omega$ turbulence model with the Low-Reynolds correction option activated requires the same mesh guidelines as for the enhanced wall treatment i.e. a y^+ value around 1 at the wall-adjacent cell. The impact of the first cell value was studied specifically for the SST turbulence model as shown in Figure 2.8. Whereas the 0.05 mm first cell grid resulted in y^+ value around 5, the 0.01 mm and 0.005 mm first cell grids led to y^+ value less than 1. This necessity for a y^+ value around 1 is clearly demonstrated in Figure 2.8. Finally the first cell size was fixed at 0.01 mm and the maximum y^+ value was found to be 1.3.

Figure 2.8: Comparison of the SST heat transfer coefficient lateral distributions at $s/d = 5$ for three different first cell sizes

A grid with a growth rate of 1.2 when compared to a grid with a growth rate of 1.1 having the same prism layer height was found to give slightly different results. Moreover the convergence with this grid required many more iterations to be obtained. As a consequence, a growth rate of 1.1 was used.

Then different heights of the prism layer were tested to evaluate the right value to use. Table 2.3 compares the characteristics of the four different prism layers examined.

Table 2.3: Characteristics of the prism layers tested for the SST turbulence model

	Mesh#1	Mesh#2	Mesh#3	Mesh#4
First cell size (mm)	0.01	0.01	0.01	0.01
Growth rate	1.1	1.1	1.1	1.1
Number of rows	24	28	31	33
Prism layer height (mm)	0.9	1.4	1.8	2.2

The lateral distributions of the heat transfer coefficient are compared in Figure 2.9. Once again a total height of 1.8 mm was found to be thick enough to capture the thermal and hydrodynamic boundary layers using the SST $k-\omega$ turbulence model. The averaged difference in heat transfer coefficient values between the 1.8 mm and 2.2 mm high prism layers was determined to be as low as 1%.

Figure 2.9: Comparison of heat transfer coefficient lateral distributions at $s/d = 5$ for four different heights of the prism layer

Regarding the grid independence outside of the prism layer, two different grids were tested as shown in Figure 2.10. There was a large discrepancy in the intermediate peak in heat transfer coefficient values between the 7.9 million cell case and the 12 million cell case. This difference is probably due to the inherent singularities in the SST model. As a consequence, the grid independence of the solution was not achieved. Nevertheless the 7.9 million cells grid was used because the anomaly was localized and because the difference between the laterally averaged values was only 1% at this location. So it should be kept in mind, when the SST results will be presented in this study, that they are suspect due to resolution problems.

Figure 2.10: Comparison of the heat transfer coefficient lateral distributions at $s/d = 5$ for different numbers of cells

In conclusion, the grid independence was shown for each turbulence model grid. These two grids were not very different from each other. Their prism layer had the same first size cell (0.01 mm) and almost the same height (1.8-1.9 mm).

Finally the prism layer and the grid resolution close to wall for the SST grid are illustrated in Figure 2.11.

Figure 2.11: Grid resolution at the stagnation plane for the SST grid

Chapter 3: Results and Discussion

The main focus of this study was to evaluate conventional experimental practice of using a density ratio of $DR = 1$ tests to determine high density ratios heat transfer coefficient augmentation.

First a study of the heat transfer coefficient without film cooling (h_0) is conducted. Then this chapter presents the computational predictions of the RKE and SST turbulence models and compares them with experimental data at unit density ratio. After that, mechanism causing enhancement of the heat transfer coefficient due the coolant jets is studied. Finally the effect of the density ratio on h_f/h_0 is presented with considerations to the momentum flux ratio, the blowing ratio and the velocity ratio effects.

3.1. STUDY OF THE HEAT TRANSFER COEFFICIENT WITHOUT FILM COOLING

The heat transfer coefficient without film cooling (h_0) was determined using a three dimensional leading edge with a uniform heat flux specified on the surface. Even though the physical problem is only two dimensional, the three dimensions simulation was done to verify the spanwise uniformity of the predicted h_0 . However, as the following results show, spanwise variations were observed.

A comparison of computation predictions and experimental measurements of the heat transfer coefficient without film cooling for a mainstream turbulence intensity of $Tu = 0.5\%$ is presented in Figure 3.1 in terms of a non-dimensional quantity: the Frossling number, Fr , defined by the equation (1.17).

The results presented in Figure 3.1 were obtained from studies conducted at different Reynolds number. However the Frossling is expected to be independent of the Reynolds number because of its definition.

Figure 3.1: Comparison of laterally averaged Frossling number

The stagnation line was located at $s/d = 0$ in each of these studies. A Frossling number of $Fr = 1.0$ at the stagnation line of a cylinder in cross-flow was established experimentally by Geidt (1949). A lower value of Frossling number was measured by Yuki et al. (1998) at the stagnation line in comparison with the unity value experimentally measured by Mouzon (2005). The difference was attributed to the use of suction slot at stagnation as shown in Figure 3.2. Indeed, in Yuki et al. study, instead of using a full leading edge centered in a test section, a half-section leading edge was used with a suction slot immediately in front of the leading edge to adjust the location of the stagnation line. As a consequence, this suction slot actually increased the effective diameter of the cylinder. Mouzon (2005) solved this issue by using a half cylindrical leading edge and obtained a value close to 1.0 at the stagnation line.

Figure 3.2: Schematic of the leading edge facility used by Yuki et al. (1998)

Computational Frossling values at the stagnation line were slightly larger than 1.0, and there was an increase in Fr for the three computational models immediately downstream of the stagnation line. Dobrowolski (2009) showed that this slope could be attributed to the symmetry condition. Indeed Dobrowolski ran an additional simulation with a complete leading edge and this simulation gave the zero slope characteristic of the Frossling number downstream of the stagnation line.

Unlike with the realizable $k-\epsilon$ turbulence model, the computational predictions of h_0 with the SST $k-\omega$ turbulence model and with the laminar case were similar to the experimental data of Mouzon (2005).

Spanwise variations in the heat transfer coefficient value up to 5% were observed with the RKE and SST turbulence models as shown in Figure 3.3. In the laminar case, the variations in h_0 were significantly larger as much as 15%.

Figure 3.3: Spanwise variations in Fr with the RKE and SST turbulence model

Efforts were made to try to resolve this issue. First a complete half cylindrical leading edge was modeled to eliminate issues related to the stagnation plane symmetry. Additionally, in order to study grid independence issues, different grids were used: the prism layer first cell size was varied by a factor up to 5 and the number of cells was increased by a factor up to 3. Overall all the different configurations tested did not change the laterally averaged values but also did not change significantly the magnitude of the spanwise variations. However it was found that the locations of these variations were different from one grid to another. This may indicate a slight grid dependence of the heat transfer coefficient without film cooling. Finally this spanwise nonuniformity is probably due grid inhomogeneities around the holes as shown in Figure 3.4. However these random spanwise variations were not noticed in h_f simulations.

Figure 3.4: Leading edge surface mesh for the RKE turbulence model simulations

No studies were found in the literature on spanwise variations of heat transfer coefficient without film cooling. Because this is a two dimensional problem, previous studies only used a two dimensional computational domain to obtain h_0 .

However it should be acknowledged that these spanwise variations looked significant in Figure 3.3 because of the scale which is focused on a small range of Frossling number. Indeed when the lateral distribution of h_0 is compared with the one of h_f , as shown in Figure 3.5, the spanwise variations in h_0 appeared to be insignificant and negligible.

Finally, when reporting the heat transfer coefficient augmentation in the rest of this study, the h_f and h_0 values were obtained both with the same turbulence model and with the same three dimension computational domain.

Figure 3.5: Lateral distribution of h_f at $I = 2.2$ and h_0 at $s/d = 5$ with the RKE and SST turbulence model

3.2. COMPARISON OF HEAT TRANSFER COEFFICIENT AUGMENTATION PREDICTIONS WITH EXPERIMENTAL DATA

In this section, computational predictions of the heat transfer coefficient augmentation were compared with experimental data from Yuki et al. (1998) and Johnston et al. (1999) because these studies are the only experimental data that had the same commonly used round hole configuration. However it should be acknowledged that their model was not a full leading edge thus this was not a perfect comparison.

3.2.1. Evaluation of the effects of the non-uniform heat flux foil of Yuki

To perform heat transfer tests, Yuki et al. (1998) and Johnston et al. (1999) attached a thin stainless steel sheet to the leading edge to provide a heat flux surface. Curved copper bars were attached on either end of the plate such that electrical current flowed spanwise. The heat flux plate did not provide a uniform heat flux because of the presence of the holes which caused an uneven distribution of electrical current around the holes. As the electrical current flowed around the cross-sectional area of the holes, the current increased, while the current decreased between holes in the heat flux above and below the holes. These increased and decreased current areas caused increased and decreased heat flux areas as illustrated in red and blue in Figure 3.6. The non-uniformity was quantified by Yuki et al. using experimental measurements. The surface temperatures on this heat flux foil were measured with no flow. Therefore the heat transfer coefficient h was assumed constant. As h is constant, the local measured surface temperatures were proportional to the local heat flux.

Figure 3.6: Deviations in heat flux due to non-uniform electrical current flow around holes. Contours of $q_{\text{foil with holes}} / q_{\text{foil without holes}}$ in the experiments. (From Yuki et al., 1998)

Figure 3.7: Wall heat flux rate boundary condition mimicking the experimental condition

To obtain a similar boundary condition, a two-dimensional simulation was run to simulate the spanwise flow of current in the heat flux foil using an incompressible

inviscid flow model. Indeed the electronic-hydraulic analogy is a widely used analogy for electrical current flow in a metal conductor. Because the electrical and hydraulic differential equations have the same form as shown in Table 3.1, the response will be similar. In this analogy, the volumetric flow rate Q of a fluid is similar to the electrical current I . Since the heat flux rate is proportional to the current I , the local velocity of the flow was used to determine the local heat flux.

Table 3.1: Comparison between fluid and electric differential equations

Fluid	Electric
$\Delta\phi = 0$ (Laplace equation)	$\Delta V = 0$
$div(\vec{V}) = 0$	$div(\vec{j}) = 0$
$\vec{V} = \overrightarrow{grad}(\phi)$	$\vec{j} = -\sigma \overrightarrow{grad}(V)$
$\vec{V}$ velocity vector ϕ velocity potential	$\vec{j}$ current density vector V velocity potential σ electrical conductivity $\sigma = l/(R.A)$

However it was found that a mistake was made in the boundary conditions of this simulation. As a consequence, the wall heat flux boundary condition presented in Figure 3.7 is erroneous. Nevertheless the computational boundary condition simulated similarly the high heat flux regions above and below the holes and the low heat flux regions on the left and right of the holes noticed experimentally. However the computational maximum and minimum heat fluxes were respectively higher and lower than the experimental values. Finally the computational boundary condition represents a more extreme case and is still of interest.

Yuki et al. (1998) and Johnston et al. (1999) assumed that the unevenness of the heat flux does not affect the value of h_f/h_0 since the local heat flux is dependent only on electrical current flow and the voltage across the foil was same for all tests. The equation for heat transfer with film cooling is

$$h_f = \frac{q_f''}{(T_{w,f} - T_\infty)} \quad (3.1)$$

The equation for heat transfer without film cooling is

$$h_0 = \frac{q_0''}{(T_{w,0} - T_\infty)} \quad (3.2)$$

Since q_f is equal to q_0 at every local position, the local value of h_f/h_0 becomes

$$\frac{h_f}{h_0} = \frac{(T_{w,f} - T_\infty)}{(T_{w,0} - T_\infty)} \quad (3.3)$$

So that the calculation of h_f/h_0 is supposed to be independent of the uneven heat flux

In order to validate this assumption, Figure 3.8 compares laterally averaged heat transfer coefficient augmentation values for a uniform heat flux foil and the foil used in the experiments.

Figure 3.8: Effect of the non-uniform heat flux on the laterally averaged heat transfer coefficient augmentation values for $DR = 1$, $I = 2.2$ and for $Tu = 0.5\%$

The values for these two cases were very similar and matched within an averaged 1.0% difference. The contour plots of h_f/h_0 for these two cases are presented in Figure 3.9 and did not exhibit any significant difference. As a consequence, it can be concluded that the assumption affirming that the unevenness of the heat flux does not affect the value of h_f/h_0 is justified.

Figure 3.9: Contour plots of h_f/h_0 for a) the non-uniform heat flux and b) for the uniform heat flux

3.2.2. Evaluation of the effects of the non-uniform heat flux of Mouzon

Mouzon (2005) used a “two-foil” method which consisted of two strips of stainless steel: a small foil that ran between the stagnation and off-stagnation row of holes and a larger foil that started just beyond the off-stagnation row of holes. The experimental wall heat flux rate boundary condition imposed by Mouzon is shown in Figure 3.10.

Figure 3.10: Experimental wall heat flux rate boundary condition imposed by Mouzon (2005)

This advantage of this method is that it provides a constant heat flux rate on the foils. However the drawback is that the leading edge is not fully covered by the foils. The assumption behind this design is that the regions not heated will not impact of the heat transfer coefficient calculation.

In order to study this configuration, heat transfer coefficient simulations with cooling and without cooling, for the experimental boundary condition imposed by Mouzon, were run. Figure 3.11 compares laterally averaged heat transfer coefficient augmentation values for the simulation of Mouzon’s experiment with the uniform heat flux case.

Figure 3.11: Comparison of the laterally averaged heat transfer coefficient augmentation values between the uniform heat flux case and the Mouzon's experiment

It is evident that, from Figure 3.11, the laterally averaged heat transfer coefficient augmentation values did not match. Immediately downstream of the coolant holes, there were discrepancies up to 20%. It is clear that using this two-foil method largely under-predicted the heat transfer coefficient augmentation.

In order to investigate this large difference, Figure 3.12 is a contour plot representing the ratio of heat transfer coefficient augmentation of the uniform heat flux case over the Mouzon experiment.

Figure 3.12: Ratio of heat transfer coefficient augmentation between the uniform heat flux case and the simulation of Mouzon's experiment

The large differences in heat transfer coefficient augmentation appeared to be located right after the exit of the coolant holes. At these particular locations, the discrepancies were found to be up to 80%. For the non-uniform heat flux, at these positions, a new thermal boundary grows and then allows an enhancement of the heat transfer coefficient whereas, for the uniform heat flux, the thermal boundary layer starts at the exit the coolant holes resulting then in lower heat transfer coefficient values at the edge of the second heat flux foil.

Because the present study did not use a shaped hole configuration as Mouzon did, computational predictions were not compared to the experiment data from Mouzon's study.

3.2.3. Comparison of predictions and experiments for low mainstream turbulence

A RKE computational simulation was run with the same flow conditions as in the experiments of Yuki et al. (1998) and Johnston et al. (1999): unit density ratio, a momentum flux ratio of $I = 2.2$ (i.e. a blowing ratio of $M = 1.5$) and at a mainstream turbulence intensity of $Tu = 0.5\%$. In order to ensure the similarities of boundary conditions, heat transfer coefficient simulations with cooling and without cooling were obtained with the heat flux boundary condition that mimicked the one imposed in Yuki and Johnston's studies as shown in Figure 3.7. The laterally averaged heat transfer coefficient augmentation values are compared in Figure 3.13.

Figure 3.13: Comparison of simulated RKE laterally averaged heat transfer coefficient augmentation with experimental data at $DR = 1.0$, $I = 2.2$ and $Tu = 0.5\%$

Downstream of the off-stagnation holes (i.e. $s/d > 4.0$), the RKE data were in good agreement with experimental data. Yuki reported an uncertainty in h_f/h_0 of ± 0.044

over the entire leading edge whereas Johnston stated an overall uncertainty of ± 0.05 . However, between the stagnation and off-stagnation holes (i.e. $0.6 < s/d < 3.0$), the discrepancy was large and as much as 7.5%. This difference could be explained by the change in the geometry between computations and experiments. Indeed Yuki et al. (1998) and Johnston et al. (1999) used a quarter-cylinder leading edge instead of using a full leading edge.

Impingement plate effect

An impingement plate was used in most of the simulations because it better simulated real engine conditions, where an impingement plate is commonly used. Only one simulation at low mainstream turbulence was run without an impingement plate. As a consequence, it was important to evaluate the impact of the impingement plate on the heat transfer coefficient. To do so, a simulation with an impingement plate was run and compared to the one without an impingement plate presented previously with the exact same flow conditions.

Figure 3.14: Effect of the impingement plate on the simulated laterally averaged heat transfer coefficient augmentation for $DR=1$, $I=2.2$ and $Tu=0.5\%$

Laterally averaged heat transfer coefficient augmentation values decreased significantly in the region between the holes, as shown in Figure 3.14. Downstream of the off-stagnation holes the values were not appreciably changed.

The difference observed in simulations between a case with an impingement plate and a case without was attributed to a change in the split of flow exiting the two rows, as shown in Table 3.2, even though the overall coolant mass flow rate stayed constant. With an impingement plate, less coolant flows through the stagnation holes resulting in lower heat transfer coefficient values just downstream of the stagnation line. As a consequence, a lower momentum flux ratio causes a decrease in h_f/h_0 . A 35% decrease in the local momentum flux ratio at the stagnation line was noticed with the addition of the

impingement plate. This change was previously noticed by Ravelli et al. (2010) who evaluated the effect of internal impingement cooling.

Table 3.2: Change in momentum flux ratio and blowing ratio through each row due to the use of an impingement plate

Averaged values $I = 2.2$ and $M = 1.5$	Local stagnation values		Local off-stagnation values	
With an impingement plate	$I = 1.35$	$M = 1.16$	$I = 2.79$	$M = 1.67$
Without an impingement plate	$I = 2.10$	$M = 1.45$	$I = 2.31$	$M = 1.52$

Comparisons of RKE and SST predictions with an impingement plate

RKE and SST computational simulations with an impingement plate were run with the same experimental flow conditions as described previously. The predictions of laterally averaged heat transfer coefficient augmentation values are compared in Figure 3.15.

Figure 3.15: Comparison of simulated laterally averaged heat transfer coefficient augmentation with experimental data at $DR = 1$, $I = 2.2$ and $Tu = 0.5\%$

In the region between the stagnation and off-stagnation lines, SST prediction was found to give closer results to experimental than RKE prediction. However, downstream of the off-stagnation holes, while RKE data were within experimental uncertainties, SST gave predictions of 20% larger laterally averaged h_f/h_0 values. Once again it should be recalled that the SST results are suspect because of resolution problems presented in Chapter 2.

The same comparison was conducted at a momentum flux ratio of $I = 0.56$ (i.e. a blowing ratio of $M = 0.75$) and is presented in Figure 3.16. Downstream of the off-stagnation holes, the RKE prediction was within the experimental uncertainties of Johnston et al.'s data while the SST prediction was within the experimental uncertainties of Yuki et al.'s data. Between the stagnation and off-stagnations holes, SST predicted values were in good agreement with experimental data whereas RKE values were under-predicted and slightly outside of the uncertainties range.

Figure 3.16: Comparison of simulated laterally averaged heat transfer coefficient augmentation with experimental data at $DR=1$, $I=0.56$ and $Tu=0.5\%$

The simulated heat transfer coefficient augmentation distributions at $I = 2.2$ are compared in Figure 3.17 at a plane downstream of the off-stagnation holes $s/d = 5.6$ with the experimental distribution provided by Yuki et al. (1998).

Figure 3.17: Comparison of the heat transfer coefficient augmentation distribution at $s/d = 5.6$ and at $I = 2.2$

It is evident from Figure 3.17 that the RKE and SST distributions did not have the same general trend when compared to the experimental one. Large difference between the RKE and the SST distribution were noticeable. The distribution predicted by the SST model exhibited very high values. There was a 60% difference between RKE and SST in the peak values of h_f/h_0 whereas a 20% difference was observed in the laterally averaged values in Figure 3.16.

However although two clearly separate peaks can be differentiated for the simulated RKE distribution, a plateau of high values is observed for the experimental distribution. York and Leylek (2002) conducted a computational study on the heat transfer coefficient augmentation using the RKE turbulence model. In this study, two very similar regions of high h_f/h_0 were also observed. Moreover Rozati and Tafti (2007) conducted a LES study and observed, downstream of the off-stagnation holes, two regions of high h_f at blowing ratio of $M = 0.4$ but only one larger region at $M = 0.8$ as shown in Figure 3.18. In conclusion, the results given by the RANS turbulence models were less close to the experimental data than the results given by the LES simulation.

Figure 3.18: Frossling number distribution at $M = 0.4$ and $M = 0.8$ from Rozati and Tafti (2007)

A reason for this difference between computations and experiments could have been an insufficient spatial resolution of the IR camera used in this study. Yuki et al. stated that the surface temperature measurements with the IR camera had a resolution of $0.5 d$. Indeed the discrete pixel levels were based on integrated averages over a $3 \text{ mm} \times 3 \text{ mm}$ area (the hole diameters in this study were 6.35 mm). Figure 3.22 presents the RKE and SST distribution that are averaged over $0.5 d$ in order to simulate the resolution of the IR camera. It is evident that if the two separate sharp peaks of h_f/h_0 values existed, they would have been captured by the IR camera because they were distant from a distance of $1 d$. Finally it can be concluded that the first peak in h_f/h_0 is an artifact from the RANS turbulence models.

Figure 3.19: Comparison of the heat transfer coefficient augmentation distribution at $s/d = 5.6$ and at $I = 2.2$

The peak of the h_f/h_0 levels does not correspond to the peak in the adiabatic effectiveness, η , as shown in Figure 3.20. Indeed Yuki et al. (1998) among other studies such as Johnston et al. (1999) and Mouzon et al. (2005) observed a shift in the position of these two peaks. In these studies, the peak of h_f/h_0 levels to the right of the peak in the η level was attributed to the generation of a streamwise vortex generated by the interaction of the mainstream with the laterally injected coolant jet. This interaction will be explored with more details later in the present study. Comparing the RKE h_f/h_0 levels with the h_f/h_0 levels and the η levels from Yuki et al.'s study emphasizes once again the artificial presence of the first peak in RKE h_f/h_0 levels.

Figure 3.20: Overlay of h_f/h_0 ($M = 1.9$, $DR = 1.0$) and η ($M = 2.0$, $DR = 1.1$) from Yuki et al. with the RKE h_f/h_0 ($M = 2.0$, $DR = 1.0$) from the present study at $s/d = 5.6$

Experimental and computational contour plots of h_f/h_0 at a momentum flux ratio of $I = 2.2$ presented in Figure 3.21 and Figure 3.22 respectively, highlight the differences described previously. On the computational plots, two regions of high h_f/h_0 values were noticeable downstream of the off-stagnation holes whereas, on the experimental plots, a single larger region of high h_f/h_0 values was visible. For the peak on the left side, the peak levels of heat transfer coefficient augmentation values presented in the RKE plot were sensibly similar to the experimental ones. However the levels in h_f/h_0 from the SST turbulence model were found to be over-predicted downstream of the off-stagnation holes with values as high as 2.8. The LES Frossling number distribution presented in Figure 3.18 exhibited, at a blowing ratio of $M = 0.8$, a maximum value of $Fr = 3.9$, that is to say a value of h_f/h_0 greater than 3.9 (the Frossling number of h_0 being less than 1.0 at this location). Although the LES simulation from Rozati and Tafti (2007) was conducted on a different leading edge geometry, it can be concluded that LES over-predicted the levels in h_f/h_0 when compared to experimental results.

Yuki et al. (1998)

Johnston et al. (1999)

Figure 3.21: Experimental contour plots of h_f/h_0 at $DR = 1.0$, $I = 2.2$ and $Tu < 0.5\%$ from Yuki et al. (1998) and Johnston et al. (1999)

Figure 3.22: Computational contour plots of h_f/h_0 at $DR = 1.0$, $I = 2.2$ and $Tu = 0.5\%$ for RKE and SST turbulence models

3.2.4. Comparison of predictions and experiments for high mainstream turbulence

Johnston et al. (1999) studied the influence of a high mainstream turbulence level ($Tu = 17\%$) on heat transfer coefficient augmentation. In Figure 3.23, the experimental data are compared with the RKE predictions without an impingement plate at momentum flux ratio of $I = 2.2$. Similar to the low turbulence case, the high turbulence laterally averaged h_f/h_0 values were found to be within the experimental uncertainties downstream of the off-stagnation line. Moreover, in the region between the stagnation and off-stagnation lines, the high turbulence predicted values were also in good agreement with experimental values while this was not the case for the low turbulence values in this region.

Figure 3.23: Comparison of RKE (no impingement plate) laterally averaged heat transfer coefficient augmentation at $I = 2.2$ for low and high mainstream turbulence with experimental data

Additionally the heat transfer coefficient augmentation was found to decrease with increasing mainstream turbulence. The values at mainstream high turbulence were found to be 10% lower than the one at low mainstream turbulence. This decrease in heat transfer coefficient augmentation with increasing mainstream turbulence was mainly due to a larger increase in h_0 than in h_f .

The laterally averaged heat transfer coefficient augmentation values for low and high turbulence are compared in Figure 3.24 by using a common h_0 to normalize the h_f values. It is clear that there was only a slight increase in the heat transfer coefficient for the high turbulence. h_f values were increased by only a few percents. This was in contrast to the experimental results of Johnston et al. (1999) who found an increase of at least 7%.

Figure 3.24: Comparison of RKE (no impingement plate) laterally averaged h_f/h_0 at $Tu = 0.5\%$ with $h_f, Tu=17\%/h_0, Tu<0.5\%$ at $I = 2.2$

Comparisons of RKE and SST predictions with an impingement plate

RKE and SST predictions of laterally averaged heat transfer coefficient augmentation values with an impingement plate and at high turbulence are compared with Johnston's data in Figure 3.25.

Figure 3.25: Comparison of simulated laterally averaged heat transfer coefficient augmentation with experimental data at $DR = 1$, $I = 2.22$ and $Tu = 17\%$

At high mainstream turbulence, both turbulence models studied gave predictions within the experimental uncertainties downstream of the off-stagnation holes. However in the region in between the holes, only the RKE predictions fell into that range. The SST turbulence model predicted lower values by 5%. If the impact of the impingement plate was taken into consideration, both turbulence models would probably give values in agreement with experimental data.

The same comparison was conducted at a different momentum flux ratio $I = 0.56$ and is presented in Figure 3.26. Almost all the RKE predicted values fell into the experimental uncertainties range. The SST turbulence model gave values far downstream of the off-stagnation holes in agreement with the experimental data. However, for $0 < s/d < 6.0$, the values were under-predicted.

Figure 3.26: Comparison of simulated laterally averaged heat transfer coefficient augmentation with experimental data at $DR = 1$, $I = 0.56$ and $Tu = 17\%$

The comparison of the computational prediction of heat transfer coefficient augmentation distribution, downstream of the off-stagnations holes, with the experimental data of Johnston et al. (1999), presented in Figure 3.27, showed that both turbulence models over-predicted the maximum values of h_f/h_0 . Similar to the low turbulence mainstream case, two peaks of high values were predicted computationally

whereas a plateau was measured experimentally. However it is unlikely that there would be two distinct peaks with high turbulence because the turbulence causes oscillation of the coolant jet.

Outside of the region of high values in h_f/h_0 , the RKE model was found to accurately predict the heat transfer coefficient augmentation. Finally the fact that the laterally averaged heat transfer coefficient augmentations values were accurately predicted downstream of the off-stagnation holes was misleading because large local errors were noticed.

Figure 3.27: Comparison of the heat transfer coefficient augmentation distribution at $s/d = 5.1$ and at $I = 2.2$ for high mainstream turbulence

Experimental and computational contour plots of h_f/h_0 at a momentum flux ratio of $I=2.2$ are presented in Figure 3.28 and Figure 3.29. Once again, only one peak in h_f/h_0

levels downstream of the second row of holes was noticeable experimentally whereas two peaks were visible in computations. The experimental maximum heat transfer coefficient augmentation value was $h_f/h_0 = 1.8$ while for the RKE model the maximum value was between $h_f/h_0 = 1.8$ and 2.0 and the SST maximum value was between $h_f/h_0 = 2.2$ and 2.4. This over-prediction of the h_f/h_0 levels by the RANS turbulence models was also underlined previously in the distribution presented in Figure 3.27.

Figure 3.28: Experimental contour plot of h_f/h_0 at $DR = 1.0$, $I = 2.2$ and $Tu = 17\%$ from Johnston et al. (1999)

Figure 3.29: Computational contour plots of h_f/h_0 at $DR = 1.0$, $I = 2.2$ and $Tu = 17\%$ for RKE and SST turbulence models

To conclude this section on the comparison with experimental data, the RKE turbulence model was found to provide overall better predictions than the SST turbulence model when compared to past experiments. As a consequence, the author decided to employ it in the rest of the present study.

The RKE simulations presented in the present study were found to better predict the heat transfer coefficient augmentation downstream of the off-stagnation holes than a previous study conducted by York and Leylek (2002) as shown in Figure 3.31. However their study better predict the h_f/h_0 values in the region between the stagnation and off-stagnation holes.

Figure 3.30: Comparison of RKE laterally averaged h_f/h_0 from this study and from York and Leylek (2002) study with experimental data at $DR = 1$, $I = 2.25$ and $Tu = 0.5\%$

The fact that the heat transfer coefficient without film cooling was obtained in York and Leylek's study with a two-dimensional simulation is a main explanation of the vastly different results. Indeed because the authors used a 2D simulation, they did not account for the presence of holes in the heat flux foil. However the present study used a 3D simulation with a foil with holes and thus better simulated the experimental conditions. The section 3.5 *Evaluation of the effects of holes in the heat flux plate* shows that there are significant differences immediately downstream of the coolant holes between a heat flux foil with holes and no holes. However this difference in h_0 does not account for the total difference in h_f/h_0 between the two RKE computations.

3.3. STUDY OF MECHANISM CAUSING ENHANCEMENT OF THE HEAT TRANSFER COEFFICIENT DUE TO COOLANT JETS

A CFD simulation to predict heat transfer coefficient augmentation was conducted for a density ratio of $DR = 1.0$, for a momentum flux ratio of $I = 4.0$ (i.e. a blowing ratio of $M = 2.0$) and at a mainstream turbulence intensity $Tu = 5.9\%$. Thermal and velocity fields from these simulations are analyzed to study the mechanism causing enhancement of the heat transfer coefficient.

Figure 3.31 shows a contour plot of the heat transfer coefficient augmentation. There is a region, between the off-stagnations holes, where the h_f/h_0 values were in the range 0.8-1.2. That means this region was not very affected by the coolant jets.

Figure 3.31: Contours of heat transfer coefficient augmentation for $DR = 1.0$ and $I = 4.0$

As shown in Figure 3.32, in between the holes on the stagnation line, the highest values of heat transfer coefficient augmentation are not immediately at the fore side of the jet but at a distance approximately two diameters from the exit of the holes.

Figure 3.32: Contours of heat transfer coefficient augmentation for $DR = 1.0$ and $I = 4.0$ in the stagnation holes region

Two regions of high values of h_f/h_0 around 1.5 were noticed in Figure 3.32. Indeed as the stagnation coolant jet exits the hole, it generates a pair of vortices that enhance the heat transfer coefficient. These vortices are clearly highlighted in Figure 3.33.

Figure 3.33: Velocity vectors on a plane normal to the leading edge and located right at the exit of the stagnation hole ($z/d = 0.63$)

Downstream of the off-stagnation holes, two regions of high heat transfer coefficient values are noticeable in Figure 3.31. The first peak was found to be due to the new thermal boundary layer introduced by the coolant jet. Figure 3.34 compares the laterally averaged h_f/h_0 with adiabatic effectiveness simulations run by Mathew (2010) at the same momentum flux ratio.

Figure 3.34: Overlay of heat transfer coefficient augmentation h_f/h_0 and adiabatic effectiveness at $s/d = 5$ for $I = 2.67$

This shows that, downstream of the off-stagnation holes, the first peak in the h_f/h_0 values corresponds to the highest effectiveness levels, i.e. where the coolant from the off-stagnation holes is. However, as described previously in Figure 3.20, this first peak in the h_f/h_0 levels was not noticed experimentally. As a consequence, the RKE turbulence model certainly over-predicted the heat transfer coefficient augmentation at this particular location.

A second peak in the h_f/h_0 values was noticed on the right of the first peak. It is due to a vortical flow pattern created from the interaction of the off-stagnation coolant jet and the mainstream flow, and is responsible for this increase in heat transfer. Contours of the velocity swirling strength are plotted in Figure 3.35 at three different positions downstream of the off-stagnation holes. It is evident that the strength of the vortical pattern on the aft side of the coolant decreased as it went downstream.

Figure 3.35: Contour plots of the velocity swirling strength at a) $s/d = 5$, b) $s/d = 7$ and c) $s/d = 9$

Pressure contours at $s/d = 5$ (downstream of the off-stagnation hole) are plotted in Figure 3.36. Slightly above the high heat transfer coefficient region on the aft-side, a low pressure region was noticed. Rozati and Tafti (2007) showed on LES simulations that this low pressure region was responsible for the formation of a primary vortex which draws the flow from the aft-side. This low pressure region is caused by the blockage induced by the coolant jet on the mainstream flow.

Figure 3.36: a) Pressure and b) normalized density contours downstream of the off-stagnation hole at $s/d = 5$

This vortex is also evident in the normalized density contours which are also plotted in Figure 3.36. These contours were obtained from a simulation at a density ratio of $DR = 1.5$.

This entrainment is illustrated in Figure 3.37. The mainstream streamlines (black color) are entrained by the coolant streamlines (grey color). Rozati and Tafti (2007) presented similar time averaged streamlines. It is clear from Figure 3.37 that as the off-stagnation coolant jet exits the hole and is turned by the mainstream flow, the jet not only turns downstream but also rotates in a vortex-like manner.

Figure 3.37: Coolant and mainstream streamlines interaction in the off-stagnation hole region

The vortex pattern described previously was also noticed experimentally. Figure 3.38 presents an experimental thermal field downstream of the off-stagnation row of exit holes realized by Mathew (2010) and shows that this vortex is not a computational artifact. Mathew conducted RKE and SST turbulence models simulations to study

thermal field contours for a density ratio of $DR = 1.5$ and a momentum flux ratio of $I = 2.67$ and to compare them with experimental data. It should be noted that the normalized theta temperature scale used in Figure 3.38 is equivalent to the normalized presented in Figure 3.36 because the density is temperature dependent.

Figure 3.38: Predicted normalized temperature contours along $s/d = 5.1$ plane according to SST model, RKE model compared with experimental data (Mathew (2010))

In conclusion, downstream of the off-stagnation holes, the coolant jets appeared to enhance the heat transfer coefficient through both the addition of a new thermal boundary layer and aerodynamic effects of the induced vortical motion. However comparison with experimental results proved that the enhancement of heat transfer coefficient through the addition of a new thermal boundary layer was undoubtedly over-predicted by the RKE turbulence model.

3.4. STUDY OF THE EFFECT OF THE HEAT FLUX RATE

Since the fluid properties are temperature dependent, one can wonder if the heat transfer coefficient is heat flux rate independent. Indeed a higher heat flux rate would induce a higher surface temperature that would produce a change in fluid properties. A slight change in density might generate a change in the flow dynamics that could modify the heat transfer coefficient.

In order to evaluate the effects of using a heat flux rate on the wall of $Q = 6,000$ W/m², two others heat flux rates were tested at $DR = 1.0$ and $M = 2.0$: a Q of half the magnitude ($Q = 3,000$ W/m²) and a negative Q ($Q = -6,000$ W/m²). It should be acknowledged that a negative heat flux rate represents a cooling and would be difficult experimentally.

First of all, a reminder, on how the heat transfer coefficient is calculated, is necessary in order to understand how critical the heat flux rate is in this computation. The heat transfer coefficient is defined by the following equation:

$$h = \frac{q''}{(T_w - T_\infty)} \quad (3.4)$$

The laterally averaged heat transfer coefficient values are compared in Figure 3.39. The results showed that these values are very close to each other. Indeed values for $Q = 6,000$ W/m² and $Q = 3,000$ W/m² are within 0.6% difference. However values for $Q = 6,000$ W/m² and $Q = -6,000$ W/m² are within 3.1% difference downstream of the stagnation holes and 1.4% difference downstream of the off-stagnation holes. The heat transfer coefficient augmentation spanwise distributions for these simulations were found also to be very similar.

Figure 3.39: Comparison of simulated laterally averaged heat transfer coefficient values at $DR = 1.0$ and $M = 2.0$ for three different heat flux rates

Finally the author considered these values to be close enough to conclude that the heat transfer coefficient is heat flux rate independent. A constant heat flux rate of $6,000 \text{ W/m}^2$ was thus used in all of these simulations.

3.5. EVALUATION OF THE EFFECTS OF HOLES IN THE HEAT FLUX PLATE

The way heat transfer coefficients with cooling are usually calculated is based on two steps. First, the heat transfer coefficients with and without cooling, using a heating foil with holes in it, are measured so that the heat transfer coefficient augmentation eliminates the problem with the unevenness of the heat flux. Secondly, the heat transfer coefficient without cooling using a heating foil without holes in it is determined. Finally the heat transfer coefficient with cooling is obtained by multiplying these two numbers.

The following equation summarizes the calculation:

$$h_{f,\text{reported}} = \frac{h_{f,\text{with holes}}}{h_{0,\text{foil with holes}}} \times h_{0,\text{foil without holes}} \quad (3.5)$$

As a consequence, the assumption behind this equation is that the heat transfer coefficient values without cooling do not change between the two foils. In order to validate this hypothesis, the ratio of heat transfer coefficient $h_{0,\text{foil with holes}}/h_{0,\text{foil without holes}}$ is plotted in Figure 3.40.

Figure 3.40: Effect of holes in the heat flux foil on the heat transfer coefficient without film cooling

The areas at the exit of the coolant holes were found to be the locations of high difference (as large as 100%) between these two foil configurations. Similarly to the description made for Mouzon's experiment, the holes represented unheated starting length. As a consequence, with the configuration with holes in the foil, there were high values of heat transfer coefficient because of the new thermal boundary layer.

Figure 3.41: Comparison of the laterally averaged heat transfer coefficient without film cooling for the two configurations

The laterally averaged values, presented in Figure 3.41, exhibited differences up to 20%. As a consequence, a correction factor should be applied to the reported heat transfer coefficient in order to get the actual heat transfer coefficient with cooling, as underlined by the following equation:

$$h_{f,actual} = h_{f,reported} \times \text{Correction factor} \quad (3.6)$$

where $\text{Correction factor} = h_{0,foil \text{ with holes}}/h_{0,foil \text{ without holes}}$

The corrector factor is presented in Figure 3.42.

Figure 3.42: Spanwise correction factor

3.6. STUDY OF THE EFFECT OF THE MAINSTREAM TEMPERATURE

In order to study the effect of the mainstream temperature on the heat transfer coefficient, a simulation was run with a mainstream and coolant temperature of 200K and compared to a simulation with a mainstream and coolant temperature equal to 300K. Both simulations were conducted at the same flow conditions: same density ratio of $DR=1$, same momentum flux ratio of $I=4$. As a consequence, the blowing ratio and the velocity were also the same for both cases. The change in fluid temperature results in a change in the fluid properties shown in Table 3.3

Table 3.3: Comparison of the air proprieties and dimensionless numbers

	300K	200K
Density	1.18 kg/m ³	1.76 kg/m ³
Thermal conductivity	0.026 W/m.K	0.019 W/m.K
Viscosity	1.85 x 10 ⁻⁵ Pa.s	1.35 x 10 ⁻⁵ Pa.s
Re	48,500	99,500

A comparison of the heat transfer coefficient augmentation was made and is presented in Figure 3.43. The heat transfer coefficient augmentation values were matching within 4%. As a consequence, the heat transfer coefficient augmentation was considered by the author as independent of the fluid temperature. Moreover the change in fluid temperature leads to a change in the Reynolds number as shown in Table 3.3. Therefore the heat transfer coefficient augmentation was found to independent of the Reynolds number.

Figure 3.43: Comparison of the laterally averaged heat transfer coefficient augmentation for a fluid temperature at 300K and 200K

3.7. STUDY OF THE EFFECT OF THE COOLANT PROPERTIES AT $DR=1.5$

3.7.1. Air properties at 300K and at 200K

As described previously, heat transfer coefficient simulations at $DR = 1.5$ were achievable using an artificial gas as the coolant gas. In order to calculate the heat transfer coefficient and to eliminate the ambiguity about what the driving temperature should be, the coolant had to be at the same temperature as the mainstream that is to say at 300K. However the density of air was a function of temperature. For example air at 200K is 1.5 times denser than air at 300K. As a consequence it was not possible to achieve a $DR=1.5$ with these conditions and that is why an artificial gas was used as the coolant gas.

In summary this artificial gas had the same properties of air except for the density which is 1.5 larger than the one of air. Moreover, in order to simulate a realistic flow, all the coolant properties of the coolant gas should be those of air at 200K since a density ratio of 1.5 represents a coolant at 200K. The coolant properties of this case #1 are summarized in Table 3.4.

.However, in order to evaluate only the effect of density on the heat transfer coefficient, a second case was considered. Once again the coolant density was the one of air at 200K but the other coolant properties were those of air at 300K so that only the density ratio effect and not the thermal conductivity or viscosity effects will be evaluated when compared to a unit density ratio case. The coolant properties of this case #2 are summarized in Table 3.4.

Table 3.4: Comparison of the coolant proprieties values for the two cases considered

	DR = 1.5 M = 2 I = 2.67 Case #1	DR = 1.5 M = 2 I = 2.67 Case #2
Mainstream temperature	300K	300K
Coolant temperature	300K	300K
Coolant density	Air 200K (1.76 kg/m ³)	Air 200K (1.76 kg/m ³)
Coolant thermal conductivity	Air 200K (0.019 W/m.K)	Air 300K (0.026 W/m.K)
Coolant viscosity	Air 200K (1.35 x 10 ⁻⁵ Pa.s)	Air 300K (1.85 x 10 ⁻⁵ Pa.s)
Coolant specific heat	Air 200K 1006.7 J/(kg.K)	Air 300K 1005.0 J/(kg.K)

The laterally averaged heat transfer coefficient values for these two cases are compared in Figure 3.44 with a simulation at $DR = 1.0$ $M = 1.63$ $I = 2.67$ that is to say a simulation in which the momentum flux ratio is matched with the $DR=1.5$ cases.

Figure 3.44: Comparison of laterally averaged heat transfer coefficient values at $I = 2.67$

The case #1 values were clearly closer to the $DR = 1.0$ case than the case #2 values were. The case #1 laterally averaged heat transfer coefficient values stayed within an averaged 0.8% difference the $DR = 1.0$ case one whereas the case #2 values within an averaged 3.6% difference. These discrepancies were also evident in the heat transfer coefficient spanwise distribution at $s/d = 5$, i.e. downstream of the off-stagnation holes, which are compared in Figure 3.45 for these three cases.

Figure 3.45: Comparison of heat transfer coefficient distribution at $s/d = 5$ and at $I = 2.67$

It should be noticed that the differences between case#1 and case#2 values were significant only in the high heat transfer coefficient regions, especially at the first peak where the difference is up to 20%. These variations were mainly due to the change in the thermal conductivity of the coolant between case#1 and case#2.

Moreover the $DR = 1.0$ case heat transfer coefficient distribution is closer to the case #1 than the case #2. At the first peak, there is a 5.6% difference between the case#2

and the $DR = 1.0$ case whereas there is a 15% difference between the case#1 and the $DR = 1.0$ case. However at the second peak, the case #2 is closer to the $DR = 1.0$ case (1.4% against 2.6%).

Furthermore this comparison between case#1 and case#2 clearly underlines the importance of the coolant thermal conductivity value in the value of the heat transfer coefficient. Since the second peak is due to the interaction of the mainstream flow with the off-stagnation coolant jet, its value is only slightly changed by a change in the coolant thermal conductivity. However the first peak, produced by the coolant jet boundary layer, is strongly impacted.

The end results of the comparisons conducted in Figure 3.44 and Figure 3.45 made sense and were completely justified by considering the Frossling number. The Frossling number is proportional to the heat transfer coefficient and inversely proportional to the thermal conductivity and to the square root of the density, as shown in the following equation.

$$Fr \propto \frac{h/k}{\rho^{1/2}} \quad (3.7)$$

Because the Frossling number is constant, the equation becomes:

$$h \propto k \rho^{1/2} \quad (3.8)$$

When comparing the unit density case with case#2, only the density was changed. As a consequence, if the density is increased while keeping k constant, h will be increased. This is exactly what was observed for case#2 in Figure 3.44 and Figure 3.45. Now if the thermal conductivity is decreased as in case#1, the heat transfer coefficient will be decreased when compared to case#2.

Finally these comparisons made clear that an increase in density while keeping the other properties constant caused an increase in the heat transfer coefficient. However

this increase was actually compensated by the change in coolant proprieties, especially the decrease in the thermal conductivity, caused by the change in temperature of the coolant.

It should be noted that the coolant proprieties were set as those of air at 200K in any following simulation at a density ratio of $DR = 1.5$ because these properties simulate a realistic flow.

3.7.2. Injection of CO₂

As described in the literature review in Chapter 1, some experiments used carbon dioxide (CO₂) as coolant to achieve a density ratio of $DR = 1.5$. Indeed at room temperature ($T = 300K$), the density of CO₂ is 1.5 time larger than the one of air. However the effect of using a coolant with different fluid proprieties such as thermal conductivity was not addressed in these studies. The present study aimed at addressing this question.

A simulation was run to reproduce these experimental conditions. The coolant thermal conductivity, viscosity and specific heat were modified to reflect those of CO₂ and are presented in Table 3.5.

Table 3.5: Comparison of the fluid proprieties between air and CO₂

	Air (200K)	CO ₂ (300K)
Thermal conductivity	0.019 W/(m.K)	0.0146 W/(m.K)
Viscosity	1.35×10^{-5} Pa.s	1.49×10^{-5} Pa.s
Specific heat	1006.7 J/(kg.K)	847.1 J/(kg.K)

The heat transfer coefficient distribution are compared at $s/d = 5$ in Figure 3.46. The difference between these two distributions was only significant on the first peak of

high transfer coefficient values. At this position, there was a maximum 12% difference. The laterally averaged h_f/h_0 comparison, presented in Figure 3.47, showed a maximum discrepancy of 4% immediately downstream of the coolant holes. The differences noticed in the heat transfer coefficient were found to be due to the change in thermal conductivity of the coolant jets. In conclusion, since these differences are significant, they should be accounted for and estimated when a foreign gas is used experimentally.

Figure 3.46: Effect of the CO2 proprieties on the heat transfer coefficient distribution at $s/d = 5$

Figure 3.47: Effect of the CO2 proprieties on the laterally averaged heat transfer coefficient

3.8. STUDY OF THE EFFECT OF THE DENSITY RATIO

Three different parameters were considered in order to study the effects of the density ratio: the momentum flux ratio, the blowing ratio and the velocity ratio. One can find in the literature arguments justifying the use of any of these three parameters as a point of reference in order to match the coolant conditions when comparing results from different experiments at different density ratios.

3.8.1. Momentum flux ratio

When considering experiments at different density ratios, matching the momentum flux ratio has often been used. Indeed it was justified that, in the case of

separation of the coolant jet, considering the momentum will determine how much the coolant jet separates and as a consequence it makes more sense to employ this parameter.

Therefore, simulations at the same momentum flux ratio but at two different density ratios ($DR = 1.0$ and $DR = 1.5$) were run in order to enlighten this aspect. Figure 3.48 compares laterally averaged heat transfer coefficient augmentation values for this two density ratios. It was found that these laterally averaged values were almost identical within 2.5%.

Figure 3.48: Comparison of the laterally averaged h_f/h_0 at constant momentum flux ratio of $I = 4$

Because identical laterally averaged h_f/h_0 may be misleading, the contour plots of the two different density ratio cases are presented in Figure 3.49. No difference in the position of the two regions of high h_f/h_0 levels and in the maximum values were noticeable.

Figure 3.49: Comparison contour plots of h_f/h_0 for two density ratios at constant momentum flux ratio of $I = 4.0$

The heat transfer coefficient augmentation distribution at $s/d = 5$, i.e. downstream of the off-stagnation holes, were compared in Figure 3.50. It can be concluded from this figure that matching the momentum flux ratio does match the position of the heat transfer coefficient augmentation peak values but also gives similar values for these peaks. The difference in the peak values of h_f/h_0 between these two different density ratio cases is only 0.08 i.e. around 3-4% of the peak value.

Figure 3.50: Comparison of h_f/h_0 distribution at $s/d = 5$ for two density ratios at constant momentum flux ratio of $I = 4.0$

3.8.2. Blowing ratio

The amount of heat that the coolant jet can extract is determined by the following equation:

$$\dot{Q} = \dot{m} C_p (T_c - T_\infty) \quad (3.9)$$

As this equation is proportional to $\dot{m}$, there is a line of reasoning stating that the blowing ratio should be used when data are at different density ratios are compared.

Two heat transfer coefficient simulations run at the same blowing ratio ($M = 2.0$) but at different density ratios ($DR = 1.0$ and $DR = 1.5$) were compared. Figure 3.51 compares laterally averaged heat transfer coefficient augmentation values for this two density ratios. These values noticeably do not match and the maximum difference is 8.3%.

Figure 3.51: Comparison of the laterally averaged h_f/h_0 at constant blowing ratio of $M = 2.0$

In order to study the location of these differences, the contour plots of the two different density ratio cases are presented in Figure 3.52. They clearly show a slight difference in the position of the two regions of high h_f/h_0 levels and in the maximum values.

Figure 3.52: Comparison contour plots of h_f/h_0 for two density ratios at constant blowing ratio of $M = 2.0$

For these two cases, the heat transfer coefficient spanwise distribution at $s/d = 5$, i.e. downstream of the off-stagnation holes, are compared in Figure 3.53.

Figure 3.53: Comparison of h_f/h_0 distribution at $s/d = 5$ for two density ratios at constant blowing ratio of $M = 2.0$

Unlike when the momentum flux ratio is matched, matching the blowing ratio does not provide the same position for the peaks values of heat transfer coefficient and does not lead to the same magnitude of the peaks. Actually it the peaks values for the case at a density ratio of 1.5 are shifted to the right. This is can be explained by taking into account momentum flux ratio considerations. Indeed it should be noted that, at the same blowing ratio, the high density ratio case had a momentum flux ratio lower than the unit density ratio case as the following equation highlights:

$$I = \frac{\rho_c V_c^2}{\rho_\infty V_\infty^2} = \frac{M^2}{DR} \quad (3.10)$$

Moreover the coolant jet is directed in the negative z direction. So the lower momentum flux case would be expected to be displaced less in the negative z direction. As a consequence, the difference in momentum flux ratios between these two cases explains the shift in the spanwise direction of the heat transfer coefficient.

3.8.3. Velocity ratio

The argument for matching the velocity ratio when comparing data at two different density ratio jets is that equal velocity ratio implies equal values of the gradient of velocity between the coolant jet and the mainstream flow. Then this gradient of velocity engenders the turbulence levels which produce the mixing of the coolant and the mainstream. And finally the mixing is strongly coupled with the heat transfer.

Two heat transfer coefficient simulations run at the same velocity ratio ($VR = 2.0$) but at different density ratios ($DR = 1.0$ and $DR = 1.5$) were compared. Figure 3.54 compares laterally averaged heat transfer coefficient augmentation values for the two density ratios. These values do not match and an averaged 4% difference was noted as well as a maximum difference of 7.5%.

Figure 3.54: Comparison of the laterally averaged h_f/h_0 at constant velocity ratio of $VR = 2.0$

The contour plots of these cases are presented in Figure 3.55 and, similarly to the matched blowing cases, exhibit a difference in the position of the two regions of high h_f/h_0 levels. However the maximum values were identical in both cases.

Figure 3.55: Comparison contour plots of h_f/h_0 for two density ratios at constant velocity ratio of $M = 2.0$

For these two cases, the heat transfer coefficient spanwise distribution at $s/d = 5$, i.e. downstream of the off-stagnation holes, are compared in Figure 3.56.

Figure 3.56: Comparison of h_f/h_0 distribution at $s/d = 5$ for two density ratios at constant velocity ratio of $VR = 2.0$

Again as when the blowing ratio is matched and unlike when the momentum flux ratio is matched, the positions of the peaks do not agree when the velocity ratio is matched. The high density ratio case is in this case shifted to the left, consistent with a greater momentum flux ratio. Indeed, at the same velocity ratio, the high density ratio case had a greater momentum flux ratio than the unit density ratio case as the following equation underlines:

$$I = \frac{\rho_c V_c^2}{\rho_\infty V_\infty^2} = DR \times VR^2 \quad (3.11)$$

In conclusion, the present study showed that heat transfer coefficient augmentation can be simulated using unity density ratio jets, but only when scaled with the momentum flux ratio of the coolant jets. Only the RKE simulations were presented but the SST predictions gave same results, as shown in the h_f distribution at $s/d=5$ in Figure 3.57 and Figure 3.58.

Figure 3.57: Comparison of SST h_f distribution at $s/d = 5$ for two density ratios at constant momentum flux ratio of $I = 2.67$

Figure 3.58: Comparison of SST h_f distribution at $s/d = 5$ for two density ratios at constant blowing ratio of $M = 2.0$

Chapter 4: Conclusion

4.1. SUMMARY OF THE RESULTS AND CONCLUSION

The present study was motivated by the fact that experimental measurements of heat transfer coefficient augmentation with film cooling are generally constrained to unity density ratio tests, and these results have been assumed to be applicable to high density ratio jets. But there has been little justification available for this presumption. As a consequence, RKE and SST computations were completed in a study of the heat transfer coefficient augmentation with film cooling for a simulated gas turbine blade leading edge. The primary purpose of this study was to determine if heat transfer coefficient augmentation determined using unit density ratio jets was similar to those for high density ratio jets which occur with film cooling. In addition, the impact of a number of other approximations common in experimental studies was evaluated.

First, the laminar, RKE and SST predictions of the heat transfer coefficient without film cooling were compared with several different experimental studies. All computations were found to slightly over-predict h_0 at the stagnation line. Indeed a Frossling number value of 1.0 is well established experimentally at the stagnation line whereas values slightly larger than 1.0 were computed. However only the laminar and SST predictions were similar to the experimental data of Mouzon et al. (2005) downstream of the stagnation line.

The CFD predictions were used to study the impact of experimental boundary conditions on heat transfer coefficient augmentation results. First of all, heat transfer

coefficient augmentation was demonstrated to be independent of the constant heat flux rate imposed on the leading edge surface.

Furthermore the heat flux foil employed in the studies of Yuki et al. (1998) and Johnston et al. (1999) was studied. Because this experimental heat flux foil had holes in it, the current flow was not uniform. As a consequence, the foil did not produce a uniform heat flux. This uneven distribution was reproduced computationally and was shown to not cause significant errors in h_f/h_0 measurements when compared to uniform heat flux foil results.

The two-foil method used by Mouzon et al. (2005) was also studied. Immediately downstream of the coolant holes, discrepancies up to 20% in laterally averaged h_f/h_0 were noticed. As a consequence, this method should not be employed to measure heat transfer coefficients.

RKE computational simulations completed in this study at a unity density ratio were confirmed to be consistent with experimental measurements conducted by Yuki et al. (1998) and Johnston et al. (1999). The laterally averaged heat transfer coefficient augmentation values were found to be in very good agreement downstream of the off-stagnation holes with experimental data. However some discrepancies were present in the region between the coolant holes where laterally averaged heat transfer coefficient augmentation values were under-predicted by around 7.5%.

An impingement plate was used in most of the simulations because it better simulated real engines conditions where an impingement plate is commonly used. Only one simulation at low mainstream turbulence was run without an impingement plate. As a consequence, it was important to evaluate the impact of the impingement plate on the heat transfer coefficient.

With an impingement plate, less coolant flows through the stagnation holes. A 35% decrease in the local momentum flux ratio at the stagnation line was noticed with the addition of the impingement plate. The decreased actual momentum flux ratio through the stagnation coolant holes led to decreased heat transfer augmentation in the region downstream of the stagnation line.

The SST turbulence model gave less accurate predictions than the RKE turbulence model. For example, at $DR = 1.0$, $M = 1.5$ and $Tu = 0.5\%$, the values were 20% over-predicted downstream of the off-stagnation holes.

The effect of mainstream turbulence was studied. By comparing two different turbulence intensities (0.5 % and 17%), it was shown that heat transfer augmentation decreased with increasing turbulence intensities. As expected experimentally, the values at mainstream high turbulence were found to be 10% lower than the one at low mainstream turbulence. This decrease in heat transfer coefficient augmentation with increasing mainstream turbulence was mainly due to a larger increase in h_0 than in h_f as observed in Johnston et al. (1999) study.

Mechanisms causing enhancement of the heat transfer were investigated, especially downstream of the off-stagnation line. In this region, heat transfer was found to be enhanced by aerodynamic effects of the induced vortical motion generated by the interaction of the mainstream flow with the coolant jets and by the addition of a new thermal boundary layer. However the enhancement of heat transfer coefficient through the addition of a new thermal boundary layer was shown to be over-predicted by the RKE turbulence model and was not observed experimentally.

Unique CFD simulations were conducted at a density ratio of $DR = 1.5$ and were valuable to study the effect of the density ratio on the heat transfer coefficient. These simulations were achievable thanks to the use of an artificial gas that had the same properties of air except for the density which is 1.5 larger than the one of air because the coolant had to be at the same temperature as the mainstream that is to say at 300K to eliminate the ambiguity about the driving temperature.

It was shown that heat transfer coefficient augmentation can be simulated using unity density ratio jets, but only when scaled with the momentum flux ratio of the coolant jets. When comparing two different density ratios ($DR = 1.0$ and $DR = 1.5$), the laterally averaged h_f/h_0 values were similar only when the momentum flux ratio, I , was matched. Additionally, matching the momentum flux ratio was found to match the position of the heat transfer augmentation peak values but also gave similar values for these peaks.

The use of carbon dioxide (CO_2) as a coolant was also studied. Indeed some experimental studies employed CO_2 to achieve a density ratio of $DR = 1.5$. The difference in thermal conductivity between CO_2 and air was found to be responsible to slight discrepancies in the heat transfer augmentation of as much as 4%, especially immediately downstream of the coolant holes. Finally, simulating experimentally a $DR = 1.5$ case using a unity density ratio jet scaled with the momentum flux ratio would result in less discrepancies than using CO_2 as a coolant gas.

4.2. RECOMMENDATIONS FOR FUTURE WORK

One area of future research in the experimental field could be a new study on the heat transfer coefficient using a larger scale complete leading edge model so that it would be possible eliminate the issues encountered by the previous studies which used a suction slot on a quarter cylindrical model. Additionally it would be the occasion to verify experimentally the fact that heat transfer coefficient augmentation can be simulated using unity density ratio jets, but only when scaled with the momentum flux ratio of the coolant jets. Indeed previous studies available in the literature and conducted at different density ratio did not match the momentum flux ratio but only the blowing ratio when comparing results.

Regarding the computational field, more advanced CFD models such as LES could be used to capture the unsteady effects of the coolant flowfield. Rozati and Tafti (2007) conducted a LES study on a slightly different geometry than the one used in the present study but some questions arise from their results because the coolant jet was found to exit in a reverse manner at low blowing ratio. Moreover the V2F turbulence model could be also employed in a study of leading edge film cooling. Indeed no extensive studies have been conducted on a film cooled vane. This model was found to provide a substantial improvement in the prediction of the heat transfer on a turbine blade with no film cooling over the standard two-equation models, such as $k-\epsilon$ and $k-\omega$, by addressing the stagnation point anomaly described by Durbin.

Another area of research could be a comparison of the heat transfer coefficient augmentation of an entire vane between experimental measurements and CFD predictions in order to evaluate the performance of RANS turbulence models on a larger scale.

Appendix

This appendix details all the models and options selected as well as the material properties specified in FLUENT.

MODELS

- Energy equation
- Viscous Model

Model: k-ε	
k-ε Model	Realizable
Near-Wall Treatment	Enhanced Wall Treatment
Enhanced Wall Treatment Options	Pressure Gradient Effects Thermal Effects

Model: k-ω	
k-ω Model	SST
k-ω Options	Low-Re Corrections

- Species Model

Options	
Species Transport	Inlet Diffusion Diffusion Energy Source Thermal Diffusion

MATERIALS

- Air (300K)

Density (kg/m ³)	$\rho = 7.666053 - 0.0618122 \times T + 2.303955 \cdot 10^{-4} \times T^2 - 3.891206 \cdot 10^{-7} \times T^3 + 2.253338 \cdot 10^{-10} \times T^4$
Specific Heat (J/(kg.K))	$C_p = 1.0575 \cdot 10^3 - 0.4489 \times T + 1.1407 \cdot 10^{-3} \times T^2 - 7.999 \cdot 10^{-7} \times T^3 + 1.9327 \cdot 10^{-10} \times T^4$
Thermal Conductivity (W/(m.K))	$k = -3.9333 \cdot 10^{-4} + 1.0184 \cdot 10^{-4} \times T - 4.8574 \cdot 10^{-8} \times T^2 + 1.5207 \cdot 10^{-11} \times T^3$
Viscosity (kg/(m.s))	$\mu = 5.208116 \cdot 10^{-7} + 7.559176 \cdot 10^{-8} \times T - 5.256698 \cdot 10^{-11} \times T^2$

- Air (200K)

Density (kg/m ³)	Incompressible Ideal Gas
Specific Heat (J/(kg.K))	$C_p = 1.11412.10^3 - 0.70181008 \times T + 1.392262.10^{-3} \times T^2 - 8.772.10^{-7} \times T^3 + 1.9327.10^{-10} \times T^4$
Thermal Conductivity (W/(m.K))	$k = -1.1078277.10^{-2} + 1.12011.10^{-4} \times T + 5.31.10^{-8} \times T^2 + 1.5207.10^{-11} \times T^3$
Viscosity (kg/(m.s))	$\mu = -7.5640342.10^{-6} + 8.6105156.10^{-8} \times T - 5.256698.10^{-11} \times T^2$
Molecular Weight (kg/kmol)	43.449

- CO₂

Density (kg/m ³)	Incompressible Ideal Gas
Specific Heat (J/(kg.K))	$C_p = 4.299289.10^2 + 1.874473 \times T - 1.966485.10^{-3} \times T^2 + 1.297251.10^{-6} \times T^3 - 3.999956.10^{-10} \times T^4$
Thermal Conductivity (W/(m.K))	$k = -8.560458.10^{-3} + 7.72015.10^{-5} \times T$
Viscosity (kg/(m.s))	Sutherland, three coefficient method $\mu_0 = 1.37.10^{-5} \text{ kg/(m.s)}, T_0 = 273.11 \text{ K and } S = 222.22 \text{ K}$
Molecular Weight (kg/kmol)	43.449

- Mixture

Density (kg/m ³)	Incompressible Ideal Gas
Specific Heat (J/(kg.K))	Mixing law
Thermal Conductivity (W/(m.K))	Ideal Gas Mixing Law
Viscosity (kg/(m.s))	Ideal Gas Mixing Law

SOLUTION METHODS

Pressure-Velocity Coupling

Scheme	Simple
--------	--------

Spatial Discretization

Gradient	Green-Gauss Node Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulence Kinetic Energy	Second Order Upwind
Turbulent Dissipation Rate	Second Order Upwind
Energy	Second Order Upwind

SOLUTION CONTROLS

Under-Relaxation Factors

Pressure	0.3
Density	1
Body Forces	1
Momentum	0.5
Turbulent Kinetic Energy	0.8
Turbulent Dissipation Rate	0.8
Turbulent Viscosity	1
Energy	0.8

References

- Ammari, H. D., Hay, N., and Lampard, D., 1990 "The Effect of Density Ratio on the Heat Transfer Coefficient From a Film-Cooled Flat Plate", *Journal of Turbomachinery*, Vol. 112, 1990, pp. 444-450.
- 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.
- Dees, J. E., Bogard, D. G., Ledezma, G. A., Laskowski, G. M. and Tolpadi, A. K., 2010, "Experimental Measurements and Computational Predictions for an Internally Cooled Simulated Turbine Vane With 90 Degree Rib Turbulators", ASME Paper GT 2010-23004.
- Dobrowolski, L. D., 2009, "Numerical Simulation of a Film Cooled Turbine Blade Leading Edge Including Heat Transfer Effects", M.S Thesis, University of Texas, Austin, Texas.
- Dobrowolski, L. D., Bogard, D. G., and Ravelli, S., 2010, "Accuracy of Conventional Adiabatic Effectiveness and Heat Transfer Augmentation Factors in Predicting Heat Flux into a Turbine Blade Leading Edge", ASME Paper GT 2010-23438.
- Eriksen, V. L., 1971, "Film Cooling Effectiveness and Heat Transfer with Injection Through Holes", PhD dissertation, University of Minnesota, Minneapolis, Minnesota.
- Fluent Manual, 2009
- Giedt, W. H., 1949, "Investigation of Variation of Point Unit heat-Transfer Coefficient Around a Cylinder Normal to an Air Stream", Trans. ASME, Vol. 71, pp. 375-381.
- Harrison, K. and Bogard, D. G., 2008, "Comparison of RANS turbulence models for prediction of film cooling performance", ASME Paper GT 2008-51423.
- Harrison, K. and Bogard, D. G., 2008, "Use of the Adiabatic Wall Temperature in Film Cooling to Predict Wall Heat Flux and Temperature", ASME Paper GT 2008-51424.
- Johnston, C. A., 1999, "High Freestream Turbulence Effects on Film Cooling of a Turbine Blade Leading Edge", M.S. Thesis, University of Texas, Austin, Texas.
- Johnston, C. A., 1999, Bogard, D. G., and McWaters M. A., "Highly Turbulent Mainstream Effects on Film Cooling of a Simulated Airfoil Leading Edge", ASME Paper No. 99-GT-261.
- Karni, J., and Goldstein, R. J., "Surface Injection Effect on Mass Transfer From a Cylinder in Crossflow: A Simulation of Film Cooling in the Leading Edge Region of a Turbine Blade", *Journal of Turbomachinery*, Vol. 112, 1990, pp. 418-427.

- Luo, L., and Razinski E. H. , 2007, “Conjugate Heat Transfer Analysis of a Cooled turbine vane using the V2F Turbulence Model”, *Journal of Turbomachinery*, Vol. 129, pp.773-781.
- Medic, G., and Durbin P. A., 2002, “Towards improved prediction of heat transfer in turbine blades”, *Journal of Turbomachinery*, Vol. 124 pp.187-192.
- Mehendale, A. B. and Han, J. C., 1992, “Influence of High Mainstream Turbulence on Leading Edge Film Cooling Heat Transfer,” *Journal of Turbomachinery*, Vol. 114, pp. 707-715.
- Menter, F. R., 1994, “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications”, *AIAA Journal*, Vol. 32, August 1994, pp.1598-1605.
- Mick, W. J., and Mayle, R. E., “Film Stagnation Film Cooling and Heat Transfer, Including Its Effect Within the Hole Pattern”, *Journal of Turbomachinery*, Vol. 110, 1988, pp. 66-72.
- Mouzon, B. D., 2005, “Net Heat Flux Reduction and Overall effectiveness for a Turbine Blade Leading Edge”, M.S Thesis, University of Texas, Austin, Texas
- Mouzon, B. D., Terrell, E. J., Albert, J. E., Bogard, D. G., 2005. “Net heat flux reduction and overall effectiveness for a turbine blade leading edge”. ASME Paper No. GT2005-69002.
- Ravelli, S., Dobrowolski, L., Bogard, D. G., 2010, “Evaluating the Effect of Internal Impingement Cooling on a Film Cooled Turbine Blade Leading Edge”, ASME Paper No. GT2010-23002
- Rozati, A., Tafti, D. K., 2007. “Large Eddy Simulation of Leading Edge Film Cooling- Part II: Heat Transfer and Effect of Blowing Ratio”, ASME Paper No. GT2007-27960
- Sen, B., Schmidt, D., and Bogard, D. G., “Film Cooling with Compound Angle Holes: Heat Transfer”, *Journal of Turbomachinery*, Vol. 118, 1996, pp. 800-806.
- Shih, T.-H., Liou, W, Shabbir, A., Yang, Z., Zhu, J., 1995, “A new k- ϵ Eddy Viscosity Model For High Reynolds Number Turbulent Flows”, *Computers Fluids*, Vol.24, n 3, pp. 227-238
- 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 Mechanic Conference, Sydney, Australia, 13-17th December 2004
- Walters, D. K. and Leylek, J. H., 2000, “Impact of Film Cooling Jets on Turbine Aerodynamics Losses”, *ASME Journal of Turbomachinery*, Vol. 122, 2000, pp. 537-545
- York, W. D. and Leylek, J. H., 2002, “Leading-Edge Film Cooling Physics: Part II- Heat Transfer Coefficient”, ASME Paper No. GT-2002-30167

- Yuki, U. M., 1998, "Heat Transfer and Adiabatic Effectiveness Measurements on a Simulated Film-Cooled Turbine Blade Leading Edge", M.S. Thesis, University of Texas, Austin, Texas.
- Yuki, U. M., 1998, Bogard, D. G., and Cutbirth, M., "Effect of Coolant Injection Heat Transfer for a Simulated Turbine Airfoil Leading Edge", ASME Paper No. 98-GT-431.
- Zhang, X.-Z. and Hassan, I., 2006, "Numerical Investigation of Heat Transfer on Film Cooling with Shaped Holes", *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol. 16, pp. 848-869

Vita

Gwennaël Beirnaert-Chartrel was born on January 7th, 1988 in Lomme, France, the son of Isabelle Chartrel and Bruno Beirnaert. He graduated from Ecole Centrale de Lille in May 2009 with a Bachelor of Science in Mechanical Engineering. During the summer of 2009, Gwennaël interned at SNECMA (French manufacturer of engines for commercial and military aircraft, and for space vehicles) and worked as a research engineer. In August 2009, he began his Master of Science in Mechanical Engineering at the University of Texas at Austin. Gwennaël will graduate in May 2011 from both Ecole Centrale de Lille and the University of Texas at Austin.

Permanent address: 10 rue Fénelon, 59160 Lomme, France

This thesis was typed by the author.