The Effects of Upstream Boundary Layers on the NGV Endwall Cooling

Shuo Mao

Dissertation submitted to the faculty of the Virginia Polytechnic Institute and State University in partial fulfillment of the requirements for the degree of

Doctor of Philosophy
In
Mechanical Engineering

Wing F. Ng, Chair
Thomas E. Diller
Chang Min Son
Mark R. Paul
Aurelien Borgoltz
Walter F. O’Brien

Oct. 29th, 2021
Blacksburg, VA

Keywords: Gas Turbine, Nozzle Guide Vane, Heat Transfer, Experimental Heat Transfer, Computational Fluid Dynamics, Boundary Layer, Film Cooling, Endwall Cooling
The Effects of Upstream Boundary Layers on the NGV Endwall Cooling

Shuo Mao

ABSTRACT

Modern gas turbine designs' ever-increasing turbine inlet temperature raises challenges for the nozzle guide vane cooling. Two typical endwall cooling schemes, jump cooling and louver cooling, result in different interactions between the injected coolant and the mainstream, leading to different cooling effects. This study investigates these two cooling schemes on the endwall cooling experimentally and numerically. Wind tunnel tests and the CFD simulations are carried out with engine-representative conditions of an exit Mach number of 0.85, an exit Reynolds number of $1.5 \times 10^6$, and an inlet Turbulence intensity of 16%. The jump cooling scheme experiments investigate two blowing ratios, 2.5 and 3.5, two density ratios, 1.2 and 1.95, and three endwall profiles with different NGV-turbine alignments. Four coolant mass flow ratios from 1.0% to 4.0% are tested for the louver cooling.

The results show that the cavity vortex, the horseshoe vortex, and the passage vortex are the main factors that prevent the upstream coolant from reaching the NGV passage. The jump cooling scheme generally provides high momentum to the cooling jets. As a result, the coolant at the design case density ratio of 1.95 and blowing ratio of 2.5 is sufficiently energized to penetrate the horseshoe vortex. It then forms a relatively uniform coolant film near the NGV passage inlet. Reducing the coolant density or increasing the blowing ratio leads to higher coolant momentum, so the coolant jets can further suppress the horseshoe vortex. However, high momentum may cause coolant lift-off, mitigating the coolant reattachment. Therefore, the density ratio needs to be
carefully balanced with the blowing ratio to optimize the cooling effect. This balance is also affected by the combustor-NGV misalignment, as a higher step height requires higher coolant momentum to overcome the step-induced vortices.

On the contrary, the louver cooling scheme provides less momentum to the coolant. The results showed that only by exceeding a coolant mass flow rate of 1~2% can the coolant form a uniform film which provides good coverage upstream of the NGV passage inlet. As for the cooling of the NGV passage, the mass flow rate ratio of the range investigated is not sufficient for desirable cooling performance. The pressure side endwall proves most difficult for the coolant to reach. In addition, the fishmouth cavity at the combustor-NGV passage causes a three-dimensional cavity vortex that transports the coolant in the pitch-wise direction. Moreover, the coolant transport pattern is dependent on the coolant blow rate.

Overall, the more-energized coolant film generated by the jump cooling tends to survive longer, but it is also more prone to lift-off. At the same time, the less-energized coolant film caused by the louver cooling is more susceptible to vortices and the discontinuity of the endwall geometry. However, it develops faster, especially in the lateral direction. The two schemes could be applied simultaneously for an ideal cooling system. The jump cooling can provide enough momentum for the coolant to persist in the NGV passage. Meanwhile, the louver cooling covers the upstream region before the jump cooling coolant reattaches to the endwall.
The Effects of Upstream Boundary Layers on the NGV Endwall Cooling

Shuo Mao

GENERAL AUDIENCE ABSTRACT

Gas turbines, sometimes called combustion turbines, are widely used to generate power or propulsion for various applications. The three main components of a gas turbine are compressor, combustor, and turbine. Modern gas turbines run at a high turbine inlet temperature that exceeds the current metal limits to increase efficiency. However, this brings significant challenges to the cooling of the first stage of the turbine, the nozzle guide vane. In this research, two commonly used endwall cooling methods, jump cooling and louver cooling, are investigated under engine-representative conditions experimentally and numerically. In addition, flow physics is demonstrated to explain the endwall cooling performance, mainly the upstream boundary layer caused by the interaction between the mainstream and the coolant flow.

The results show that the cavity vortex, the horseshoe vortex, and the passage vortex are the main factors that prevent the upstream coolant from reaching the NGV passage. The jump cooling scheme provides high momentum to the cooling jets. As a result, the coolant in the design case is sufficiently energized to penetrate the horseshoe vortex, providing a desirable cooling effect in the NGV passage. Reducing the density ratio or increasing the blowing ratio can help the coolant jets further suppress the horseshoe vortex but also causes more lift-off, which adversely affects the cooling performance.

On the contrary, the louver cooling scheme provides less momentum to the coolant, forming a less energized coolant film. The lack of coolant causes the louver coolant film to provide
good coverage immediately downstream of the louver scheme exit. However, due to unfavorable interaction with vortices and endwall discontinuity, the cooling effect decays quickly downstream.

Overall, the more-energized coolant film generated by the jump cooling tends to survive longer, but it is also more prone to lift-off. At the same time, the less-energized coolant film caused by the louver cooling is more susceptible to vortices and the discontinuity of the endwall geometry. However, it develops faster, especially in the lateral direction. The two schemes could be applied simultaneously for an ideal cooling system to work mutually beneficially.
PREFACE

The work presented in this dissertation covers the heat transfer characteristics of two distinct film cooling schemes, jump cooling and louver cooling, which are commonly used for the nozzle guide vane (NGV) endwall cooling in a gas turbine engine. Their cooling performance and the coolant’s interactions with the freestream were investigated experimentally and numerically and presented coherently in three journal manuscripts.

Solar Turbines Incorporated sponsored this research project. The author was involved in all aspects of the research work, including experimental setup design and build, instrumentation, wind tunnel test, data reduction, CFD study, and data analysis. Throughout his Ph.D. program, the author worked closely with three graduate students at Virginia Tech, Luke Luehr, Ridge Sibold, Daniel Van Hout, one graduate student from Xi’an Jiaotong University, Bo Bai, and a visiting scholar, Kaiyuan Zhang.

The main objectives of this research work are as follows,

I. Experimentally measure and compare the film cooling performance of the two cooling schemes under engine-representative conditions.

II. Numerically investigate the development of the two coolant films and their interactions with the freestream and the endwall geometry.

This dissertation mainly consists of five chapters. The first chapter provides a general introduction to nozzle guide vane cooling, commonly used cooling techniques, and the freestream’s influence on the cooling effect. Chapters 2 and 3 detail the experimental and numerical study of the jump cooling scheme. Two coolant densities, two coolant flow rates, and three endwall profiles are tested to reveal the behaviors of different coolant flows. Chapter 4 focuses on the louver cooling scheme and the interaction between the coolant film and the engine-
representative endwall profile. Chapters 2 and 3 have been presented at the ASME Turbo Expo 2020 and published in the ASME Journal of Turbomachinery. Chapter 4 has been accepted by ASME Turbo Expo 2022 and recommended for journal publication.

The last chapter summarizes the findings of the first chapters and provides high-level conclusions and potential future work for this study. The final part of this dissertation discusses multiple aspects of this study that are not detailed in previous chapters to help the audience further understand this research.
ACKNOWLEDGEMENTS

First, I would like to express my sincere gratitude to Prof. Wing F. Ng for his continuous and comprehensive support of my Ph.D. program. His patient guidance helped me develop not only my research skills but also my professionalism as a researcher. I could not have been even close to what I’ve become in the four and half years at Virginia Tech without him as my advisor.

Besides my advisor, I would like to thank the rest of my Ph.D. committee, Prof. Thomas E. Diller, Prof. Chang Min Son, Prof. Mark R. Paul, and Prof. Aurelien Borgoltz. Their insightful comments offered great help to set appropriate goals for my Ph.D. research and complete my research in aspects that I would have otherwise overlooked. I want to offer special thanks to Prof. Walter F. O’Brien, who, although no longer with us, continues to inspire me with his expertise in turbomachinery.

I am profoundly thankful to Solar Turbines Incorporated and Nanosonic Inc. for funding my Ph.D. program. Furthermore, I am privileged to work with Hongzhou Xu, Michael Fox from Solar Turbines and Hang Ruan, Echo Kang from Nanosonic. Their industrial experience and leadership helped me broaden my horizon in my research work.

I would like to thank Luke E. Luehr, Ridge A. Sibold, Stephen Lash, Jin Woo Lee, Daniel Van Hout, and Kaiyuan Zhang. They have spent numerous hours with me in the labs to prepare and conduct the wind tunnel testings. Without their help, it would not be far more difficult for me to complete this research. I also want to thank Kaiyuan Zhang and Bo Bai for their support of the CFD work in this study.

My sincere thanks also go to the support team behind this research, especially our lab machinist, William Randall Monk, lab manager, Eric Greenman, and program support technician, Diana D. Israel. Their continuous help ensured my smooth work experience as a Ph.D. candidate.
Lastly, I would like to thank my mother, Qinghui Long, and my father, Mengzhong Mao, for their support of my study abroad. The inspiration, drive and help you gave me made me the person I am today.
# TABLE OF CONTENT

ABSTRACT.................................................................................................................................................. ii
GENERAL AUDIENCE ABSTRACT........................................................................................................ iv
PREFACE..................................................................................................................................................... vi
ACKNOWLEDGEMENTS............................................................................................................................... viii

CHAPTER 1 INTRODUCTION ................................................................................................................................. 1
  1. NGV ENDWALL AERODYNAMICS ............................................................................................................ 3
  2. NGV ENDWALL COOLING TECHNIQUES ............................................................................................ 4
  3. THE FREESTREAM’S INFLUENCE ON THE COOLING EFFECT ............................................................ 11
  REFERENCES ................................................................................................................................................. 14

CHAPTER 2 EXPERIMENTAL STUDY OF THE ENDWALL HEAT TRANSFER OF A TRANSONIC NOZZLE GUIDE VANE WITH UPSTREAM JET PURGE COOLING: PART 1 – EFFECT OF DENSITY RATIO................................................................................................................. 23
  ABSTRACT.................................................................................................................................................... 23
  1. INTRODUCTION ....................................................................................................................................... 24
  2. EXPERIMENTAL METHODOLOGY ........................................................................................................ 27
  3. DATA REDUCTION TECHNIQUES ........................................................................................................... 32
  4. COMPUTATIONAL SETUP .................................................................................................................... 35
  5. RESULTS AND DISCUSSION ............................................................................................................... 39
  6. SUMMARY AND CONCLUSIONS ....................................................................................................... 52
  7. NOMENCLATURE ............................................................................................................................... 53
  8. ACKNOWLEDGEMENTS .................................................................................................................... 54
  REFERENCES ................................................................................................................................................. 55

CHAPTER 3 EXPERIMENTAL STUDY OF THE ENDWALL HEAT TRANSFER OF A TRANSONIC NOZZLE GUIDE VANE WITH UPSTREAM JET PURGE COOLING: PART 2 – EFFECT OF COMBUSTOR-NOZZLE GUIDE VANE MISALIGNMENT ......................................................................................................................... 59
  ABSTRACT.................................................................................................................................................... 59
  1. INTRODUCTION ....................................................................................................................................... 60
  2. RELEVANT PAST STUDIES .................................................................................................................. 61
  3. EXPERIMENTAL TEST FACILITY AND CONDITIONS ..................................................................... 64
  4. DATA REDUCTION TECHNIQUE ........................................................................................................ 69
  5. COMPUTATIONAL SETUP .................................................................................................................. 70
  6. RESULTS AND ANALYSIS ............................................................................................................... 73
CHAPTER 1
INTRODUCTION

Gas turbines are widely used to generate power or propulsion for various applications. The three main components of a gas turbine are compressor, combustor, and turbine. The compressor draws the air intake and compresses it before diverting it into the combustion chamber. In the combustion chamber, the air is mixed with fuel and ignited. The combustion process generates high-pressure and high-temperature gases that enter the turbine. The turbine consists of multiple stages of alternate stationary vanes and rotary blades. The vanes turn the gases so they can enter the passages between blades at a favorable angle. Then the gases spin the blades that drive the compressor and generate power.

Figure 1.1. The gas turbine engine (Boyce, 2011)

Due to ever-increasing demands for higher energy efficiency, gas turbines now operate at excessively high temperatures. Over the past 60 years, the maximum turbine inlet temperature (TIT) of the turbojet has doubled from 1540°F (838°C) to 3140°F (1726°C) [1.2]. The F135 gas turbine produced by Pratt and Whitney even pushed this boundary to 3600°F (1982°C) [1.3]. For
non-aviation modern gas turbines, like power plant turbines, the TIT can also be as hot as 2300°F (1260°C) to 2700°F (1482°C) [1.4]. However, most metals cannot withstand these extreme temperatures, so thermal barrier coating and cooling using the compressor air have to be used to ensure safe operation.

Nozzle guide vanes (NGV) are static vanes located at the turbine inlet. They are designed to direct the gases from the combustor exit to increase the efficiency of the downstream rotor. Since the NGVs are exposed to the hot gases from the combustor, failure in cooling can cause serious consequences, from local burnout to complete melting of the vanes [1.5]. Therefore, the proper cooling of the NGV is crucial to the gas turbine design. This research serves to investigate two cooling schemes to protect the NGV endwall. This chapter will review the NGV endwall aerodynamics, cooling techniques for the NGV endwalls, and the mainstream’s influence on the NGV cooling to provide a general background for this research.

![Figure 1.2. A typical turbine. The nozzle guide vanes are near the turbine inlet. (Soares, 2011)](image-url)
1. NGV ENDWALL AERODYNAMICS

Endwall aerodynamics has been a research focus in the late 20 century. Some of the early influential research works are done by Langston, Sharma and Butler, and Goldstein and Spores [1.7-9]. Based on these results, Wang et al. gave a comprehensive interpretation of the flow pattern using their flow visualization results [1.10]. Three main characteristics dominate the flow field near the NGV endwall, as seen in Fig 1.3:

Figure 1.3. The typical NGV flow aerodynamics (Want et al. 1997)

(1) The upstream boundary layer separates at the saddle point upstream of the vane leading edge, inducing a multiple horseshoe vortex system. The saddle point (separation point) location depends on the overall shape and size of the airfoil pressure side vane and suction side vane [1.11]. The horseshoe vortex system bifurcates into two legs near the vane's leading edge. The pressure
side leg of the horseshoe vortex is swept towards the suction side by the cross flow and becomes the major part of the passage vortex.

(2) The suction side leg of the horseshoe vortex is weaker than the pressure side counterpart. Therefore, it stays close to the suction side vane as traveling downstream and eventually merges with the pressure side leg, becoming a small part of the passage vortex.

(3) A wall vortex emerges near the location where the passage and suction side legs vortex merge. It transfers a high mass of gases.

The horseshoe vortex and the passage vortex motions generally harm the endwall cooling. They draw the coolant into the vortices and prevent it from reaching areas near the leading edge and the suction side vane. The passage vortex may also lift the coolant from the endwall and transfer it onto the suction side endwall. Recent studies introduced the nonaxisymmetric NGV endwall to suppress the horseshoe vortex and the passage vortex. Many numerical research works have reported a reduction in total pressure losses and secondary flows [1.11-14]. Among the few experimental studies, Knezevici et al. observed a weaker passage vortex, which convects less fluid into it [1.15]. With the advances in manufacturing techniques, endwall optimization may play a more critical role in enhancing the turbine efficiency and cooling effect in the future.

2. NGV ENDWALL COOLING TECHNIQUES

Researchers have designed multiple cooling schemes applied to the NGV endwall cooling. This section reviews the three most widely used cooling methods, film cooling, effusion cooling, and impingement cooling. As the name suggests, film cooling is designed to inject a coolant gas into the hot gases and form a film into the boundary layer downstream of the injection location. The cooling film insulates the solid surface from the hot gases, reducing the convective heat
transfer rate. Meanwhile, the coolant gas also serves as a heat sink absorbing the heat. Both effects help protect the surface from overheating by the hot stream.

Different geometries have been applied to the film cooling scheme. A typical design is discrete cylindrical holes. As shown in Fig. 1.4, the cylindrical cooling hole connects the plenum and the mainstream section at an angle. Many studies have found that the discrete hole can create a lasting cooling film on the NGV endwall since the injection allows the coolant to retain much momentum during its mixing with the mainstream [1.16-19]. However, the angle at the hole exit may induce separation, leading to coolant lift-off from the turbine liner. On one hand, this separation draws the coolant away from the endwall surface, mitigating the cooling effect. On the other hand, under certain mainstream conditions, the coolant may be driven towards to the endwall and develop a cooling film at a distance downstream of the cooling hole.

*Cylindrical hole*

![Cylindrical hole](image)

*Figure 1.4. The geometry of a cylindrical hole (El Ayoubi et al. 2015)*

El Gabry et al. investigated the effect of the injection angle on a discrete hole cooled contoured NGV endwall [1.21]. The results suggest that increasing the injection angle from 20 to 60 degrees leads to decreased film cooling effectiveness from the hole exit to 20 hole diameter downstream. However, the different injection angles show similar cooling effects in the region further downstream. Kohli and Bogard also pointed out that the injection angle made little
difference on their film-cooled flat plate, but only at a low coolant mass flow rate [1.22]. At a high flow rate, the injection angle of 55 degrees shows only 70% to 90% of the cooling effectiveness compared to a 35-degree injection angle. The effect of the injection angle is primarily coupled with the coolant flow rate and dependent on the cooled surface geometry and mainstream conditions. Leylek and Zerkle studied the effect of the cooling hole's length-to-diameter ratio (L/D) [1.23]. They found that at high L/D, the coolant flow can quickly achieve a fully developed pipe flow before being injected. However, at low L/D, the flow pattern is very complicated as three coupled mechanisms, the coolant momentum and crossflow, the jetting effect, and the counter-rotating vortex, interact and dominate the flow field. Therefore, it may be beneficial to apply a high L/D to avoid an off-design cooling scenario.

Other factors affecting the cylindrical performance include the coolant's blowing ratio and density ratio, whose effects are usually coupled. On a film-cooled flat plate, a high density ratio helps the coolant jet spread laterally and produces a more uniform coolant film [1.24-26]. However, the interaction between the coolant and freestream, particularly detachment and reattachment, depends on the combined density and blowing ratio. Therefore, the specific blowing condition must be met to investigate the heat transfer scenario. Fewer studies have focused on the density ratio’s effect on a film-cooled contoured NGV endwall. It was found that the low-density coolant jet survives longer in the NGV passage under low-speed conditions [1.27-29]. However, no study on the density ratio of the NGV cooling has been undertaken under an engine-representative condition. The author hopes to fill this knowledge gap with the investigation in Chapter 2.

Researchers have been attempting to optimize the hole geometry to mitigate the coolant lift-off and unfavorable separation caused by the cylindrical hole. These optimized holes are
generally called shaped holes. Bunker reviewed the works on the shaped holes before 2005 and found four typical designs, as shown in Fig. 1.5 [1.30]. Type A and B are fan-shaped holes with two-dimension and one-dimension diffusion; Type C shows the laidback-shaped hole; while Type D is the conical-shaped hole. These shaped holes show slight variation in film cooling effectiveness at a low blowing ratio from 0.5~2 but outperform the cylindrical hole at higher blowing ratios. The cooling improvement mainly contributes to the lateral spread of coolant jets, which significantly reduces the tendency of lift-off. It was also reported that the shaped hole lowers the sensitivity of the coolant jets to free-stream turbulence intensity. Later works involve more novel hole geometry, including a crescent-shaped hole [1.31], adding an anti-vortex structure [1.32], introducing a streamwise diffuser at the hole exit [1.33], and more innovative geometry optimization [1.34]. These studies again suggest that the difference between shaped holes and cylindrical holes is only significant above critical blowing ratios. But these novel geometries manage to reduce the critical blowing ratios compared to previous designs.

Figure 1.5. The geometry of typical shaped holes (Bunker 2005)
The previously mentioned studies of shaped holes were mostly carried out on a flat plate in a straight channel. When tested on an NGV endwall, Barigozzi et al. found that the cooling effect is strongly affected by the secondary flows. The shaped hole proves superior to the cylindrical hole at MFR = 1.5%, but the shaped hole coolant jet is more susceptible to the passage vortex and the cross flow [1.35]. Later research by Barigozzi et al. on a contoured endwall revealed very similar trends [1.36]. Interestingly, Colban et al. found that the freestream turbulence intensity has an adverse effect on the shaped hole cooling, which contradicts Bunker’s conclusions [1.37]. In the study of Saumweber and Schulz, they pointed out that the operating condition, especially the freestream, has a coupled effect with the hole geometry, which may explain the different findings [1.38]. Given the complicated secondary flows in the NGV passage, assessing the shaped cooling hole performance under an accurate freestream condition is essential. It should also be noticed that the shaped holes are more widely used for cooling the turbine vanes than the endwall, likely due to the challenge in manufacturing.

Injecting coolant through thin slots is also commonly seen upstream of the NGV passage. Compared to discrete holes, the slot induces a more favorable interaction with the freestream, usually leading to a laterally uniform cooling film immediately after the slot exit [1.39]. As a result, the slot film cooling is less sensitive to the freestream boundary layer [1.40]. The slot design also allows a higher amount of coolant to be injected without lift-off compared to discrete holes [1.41,42]. However, due to the large slot area compared to discrete holes, the slot film cooling sometimes suffers from the low coolant momentum, leading to non-uniform coolant distribution under complicated freestream conditions [1.43]. To solve this problem, researchers have improved the slot geometry to increase the coolant momentum by using discrete or fan-shaped holes to connect the slot and the plenum or by adding a rib at the slot exit [1.44-46]. When used for NGV
endwall cooling, the slots are usually combined with discrete holes or in-passage gaps to inject coolant [1.47-49]. The cooling film from the slot is found to help spread to downstream coolant laterally and prevent lift-off from a high-angle injection.

One less commonly used cooling technique is effusion cooling, which uses densely spaced microsize discrete holes to inject coolant to form a continuous coolant film along the cooled surface. Facchini et al. and Wei et al. suggested that a minimum blowing rate between 1.5 and 2 is needed for the effusion cooling to optimize its performance. Despite the high blowing ratio that could cause coolant lift-off, it helps extend the cooled area further downstream [1.50,51]. When applied to the NGV cooling, Facchini et al.’s study found that effusion cooling provides full coverage over the NGV passage with minimum cooling effectiveness of 0.2 [1.52]. Sacchi et al. found that the coolant can attenuate the horseshoe vortex and passage vortex, which generally counteract the cooling film [1.53]. However, the densely spaced holes may add to manufacturing difficulties and compromise the material strength. Meanwhile, the small pore size can easily cause clogging, leading to off-design coolant distribution [1.54]. These weaknesses make the effusion cooling less optimal in introducing the coolant to the NGV endwall.
Another less common cooling technique people have studied is impingement cooling. This scheme is usually used in conjunction with the film cooling scheme. Before the coolant reaches the cooling hole inlet, it goes through vertical holes, then impinges onto the inner surface of the liner material and cools it, as seen in Fig 1.7. The jet impingement cools the liner internally. Compared to film cooling only, introducing impingement cooling enhances the cooling effect by increasing the overall cooling effectiveness and the Nusselt number [1.55,56]. Li et al. reported similar results in a study of endwall cooling but pointed out that at a high blowing ratio, the film cooling dominates the heat transfer, and the effect of the impingement cooling is marginal [1.57]. However, Yang et al. found that the impingement cooling structure may cause the discharge coefficient to drop, especially at a high blow rate [1.58]. The impingement cooling scheme also adds to the structural complexity, leading to difficulties in design and manufacture. Due to these reasons, this application of impingement cooling is quite limited in endwall cooling.
3. THE FREESTREAM'S INFLUENCE ON THE COOLING EFFECT

Due to the complexity of the NGV flow field, it is virtually impossible to replicate the engine condition in a lab test. Therefore, researchers usually simplify the freestream condition by reducing the freestream velocity, using lower turbulence intensity, or ignoring the inlet flow swirl. This section serves to review the prior research on the influence of these factors and reveal how they may affect the results from the simplified lab tests.

Freestream velocity, usually non-dimensionalized as exit Mach number, has a first-order effect on the NGV endwall aerodynamics. An early study by Sieverding and Wilputte found that the freestream Mach number causes a higher transport of endwall boundary flow to the suction side vane [1.59]. Other researchers later observed that increasing exit Mach number draws the passage vortex and the loss core closer to the endwall and farther away from the suction side vane [1.60,61]. A more recent study by Vazquez et al. found a reduction of secondary loss is caused by increasing Mach number, indicating attenuated secondary flows [1.62]. These changes in the
endwall aerodynamics can have mixed effects on the endwall cooling. On the one hand, a weaker secondary flow allows the coolant film to survive further downstream in the NGV passage. On the other hand, the passage vortex being closer to the endwall and higher transport can also lead to faster coolant dissipation and a higher tendency of lift-off. Research on the NGV endwall cooling shows the overall cooling effectiveness is increased substantially with a higher exit Mach number. However, a reduction in local cooling effectiveness is also observed in areas near the vane leading edge [1.63].

The Mach number at the hole exit also directly affects the coolant jet behaviors. At a low Mach number below 0-0.3, increasing the Mach number reduces the discharge coefficient while having little influence on the cooling effect [1.64,65]. However, increasing the Mach number to transonic or hypersonic conditions weakens the flow jets drastically, resulting in better attachment and cooling effectiveness [1.66,67].

The flow at the combustion chamber exit features high turbulence intensity, which drastically alters the endwall aerodynamics and heat transfer. Studies on the effect of freestream turbulence intensity on the boundary layer development show a highly nonlinear impact at both high and low turbulence intensity [1.68,69]. Meanwhile, the convective heat transfer between the solid surface and the boundary layer is enhanced with a higher freestream turbulence intensity [1.70]. For the endwall aerodynamics, a higher freestream turbulence intensity can suppress the passage vortex and separation on the suction side. Meanwhile, the horseshoe vortex is less steady under higher freestream turbulence intensity, but its wake can still prevent the coolant from reaching the endwall [1.71-73]. These changes in aerodynamics lead to a more uniform thermal load on the NGV endwall. However, the region near the leading edge and trailing edge only shows marginal augmentation in heat transfer [1.74-76]. When the film cooling scheme is applied, the
higher turbulence intensity enhances the cooling effect at a high momentum ratio since the turbulence helps disperse the coolant jet and reduces detachment. For the same reason, increasing turbulence intensity with a low momentum ratio leads to an inferior cooling effect [1.77].

Swirl in the combustion chamber enhances the fuel-air mixing and stabilizes the flame. Therefore, swirl motion has been widely introduced in gas turbines. Consequently, the freestream flow at the combustion exit contains a high swirl component and plays an essential part in the NGV heat transfer. Studies on the swirl effect on NGV heat transfer show different results depending on the incident angle between the swirl and vane [1.78-81]. A negative incidence near the hub can attenuate the horseshoe vortex and passage vortex, reducing the convective heat transfer rate, while a positive incidence has the opposite effect. When a film cooling scheme is present, the swirl may enhance the lateral spread of the coolant and the mixing between coolant and mainstream, enlarging the coolant coverage in the lateral direction in the vicinity of the holes. However, the overall cooling effectiveness is usually lower because the development of coolant film is adversely affected by the swirl motion [1.82-85]. Yang et al. also pointed out that a high coolant momentum ratio can reduce the sensitivity of the film cooling effectiveness distribution to the inlet swirl [1.86]. Under normal circumstances, an annular test rig is needed to generate the engine-representative swirl inlet flow. However, recent studies have used alternative ways, like running the freestream through angled fins or an injection plate, to mimic the swirl flow [1.84,85]. This change allows the test to be carried out in a linear cascade, which is easier to build than an annular rig.

Due to the harsh condition in the NGV section, the NGV endwall design is generally very complicated to handle the cooling, sealing, structural strength, and other issues simultaneously.
Therefore, simplifying the cooling systems needed can significantly help the NGV design. In this research, two film cooling schemes, jump cooling and louver cooling, are investigated under engine-representative transonic conditions to investigate their cooling performance and the coolant interaction with the freestream. Both methods are designed to cool the NGV endwall by itself, which is usually cooled with multiple film cooling schemes. The engine-representative coolant density ratio, freestream Mach number, and turbulence intensity are achieved to provide accurate data. Besides, an engine-representative contoured endwall and combustor-turbine mateface step are also mimicked. These engine-like parameters have never been achieved together in prior studies.

The only factor not considered in this study is the inlet swirl flow. According to the discussion above, the swirl flow could result in lower overall cooling effectiveness and a more favorable lateral spread of the coolant coverage. Due to the high coolant momentum ratio of jump cooling jets, the findings in Chapters 2 and 3 may be less sensitive to the swirl, while the conclusions of Chapter 4, however, may be more affected by the swirl flow.

REFERENCES


[1.75] Li, Z., Bai, B., Li, J., Mao, S., Ng, W., Xu, H., and Fox, M., 2022, “Endwall Heat Transfer


Aeronautics, 31(5), pp. 1095-1108.
CHAPTER 2
EXPERIMENTAL STUDY OF THE ENDWALL HEAT TRANSFER OF A TRANSONIC NOZZLE GUIDE VANE WITH UPSTREAM JET PURGE COOLING: PART 1 – EFFECT OF DENSITY RATIO

Shuo Mao*, Ridge Sibold, Wing F. Ng
Virginia Tech
Blacksburg, VA, United States

Zhigang Li, Bo Bai
Xi’an Jiaotong University
Xi’an, China

Hongzhou Xu, Michael Fox
Solar Turbines Incorporated
San Diego, CA, United States

ABSTRACT

Nozzle guide vane platforms often employ complex cooling schemes to mitigate the ever-increasing thermal loads on the endwall. Understanding the effect of advanced cooling schemes amid the highly complex three-dimensional secondary flow is vital to engine efficiency and durability. This study analyzes, experimentally and numerically, and describes the effect of coolant to mainstream blowing ratio, momentum ratio and density ratio for a typical axisymmetric converging nozzle guide vane platform with an upstream doublet staggered, steep-injection, cylindrical hole purge cooling scheme. Nominal flow conditions were engine-representative: exit Mach number = 0.85, exit Raynolds number = 1.5×10^6, inlet large-scale freestream turbulence intensity = 16%. Two blowing ratios were investigated, each corresponding to the design condition and its upper extrema at the blowing ratio of 2.5 and 3.5, respectively. For each blowing ratio, the coolant to mainstream density ratio was varied between a density ratio of 1.2, representing typical experimental neglect of coolant density, and a density ratio of 1.95, representative of typical engine conditions.

The results show that with a fixed coolant-to-mainstream blowing ratio, the density ratio plays a vital role in the coolant-mainstream mixing and the interaction between coolant and horseshoe vortex near the vane leading edge. A higher density ratio leads to a better coolant coverage immediately downstream of the cooling holes but exposes the in-passage endwall near
the pressure side. It also causes the in-passage coolant coverage to decay at a higher rate in the flow direction. From the results gathered, both density ratio and blowing ratio should be considered for accurate testing, analysis, and prediction of purge jet cooling scheme performance.

1. INTRODUCTION

Increased energy density needs and heightened pollution awareness have resulted in advanced combustor designs flattening exit thermal profiles and increasing turbine inlet temperatures [2.1]. Consequently, heat transferred into hot section components, specifically nozzle guide vanes, has grown beyond the thermal threshold of traditional component materials due to proximity to the combustor and high turbulence intensity. Advanced thermal barrier coatings and sophisticated cooling techniques have been developed and implemented to avoid thermal failure and achieve long-term durability goals. In an effort to minimize the penalty associated with bleeding air from the compressor section, much research has been centered around understanding endwall aerodynamics and optimizing cooling scheme design.

Endwall aerodynamics has been a topic of heavy investigation. Some of the most influential works were done by Sieverding [2.2], Sharma and Butler [2.3], Goldstein and Spores [2.4], and Wang et al. [2.5]. These works showed that the main feature of the nozzle guide vane endwall is the formation, bifurcation, and development of the pressure side and suction side horseshoe vortex (HSV) systems. In addition, significant research has been conducted on endwall heat transfer. Graziani et al. [2.6] and Boyle and Russell [2.7] experimentally studied the endwall heat transfer effects associated with inlet mainstream flow conditions. An in-depth investigation of endwall secondary flow and the resulting thermal profiles was conducted by Kang and Thole [2.8] using laser doppler velocimetry (LDV). These studies collectively illustrated the dependence of endwall heat transfer on in-passage secondary flow structures.
The essentiality of engine-representative mainstream flow conditions to obtain accurate heat transfer and film effectiveness data has been well documented. Works done by Ames et al. [2.9] and Radomsky and Thole [2.10], Schmidt and Bogard [2.11], and Spencer et al. [2.12] indicated that increasing Mach number, Reynolds number, or freestream turbulence intensity augmented the in-passage heat transfer. Furthermore, mimicking the upstream step and gap is critical to obtaining thermal profiles representative of those in-engine. This was proved by both experiments and CFD done by Piggush and Simon [2.13,14], Mayo et al. [2.15], and Li et al. [2.16]. Additionally, influences of axisymmetric endwall contouring have been demonstrated by Thrift et al. [2.17,18] and Dossena et al. [2.19]. Their results indicated that axisymmetric endwall contouring accelerated the flow and created a more favorable pressure gradient, which improved upstream cooling schemes' effectiveness. Collectively, these studies suggest that accurate engine-representative upstream and endwall geometry and flow conditions should be employed to mimic and understand in-engine thermal profiles.

With the objective to cool components in regions of excessive temperature, secondary fluid is pulled from the compressor, routed around the combustion chambers, and injected to form a film of relatively cooler fluid shielding the hot section components. Lately, much research has been centered around various forms of film cooling technology, as documented by Dunn [2.20], Simon and Piggush [2.21], Bogard and Thole [2.22], and Bunker [2.23]. Oke et al. [2.24] illustrated that compared to traditional slot injection, discrete hole injection was 20-30% less effective near the hole but had more significant impacts in-passage. Zhang and Jaiswal [2.25], Zhang and Moon [2.26], and El-Gabry et al. [2.27] used pressure-sensitive paint (PSP) techniques to investigate film effectiveness alterations due to variations in upstream misalignment coolant to mainstream
velocity and mass flow ratio, and injection angle. However, these studies lack heat transfer data leaving a void in the complete analysis.

Additionally, aforementioned studies neglected the coolant density and used a density ratio close to unity. Generally, cooling fluid in turbine engines is at significantly higher pressures and lower temperatures as compared to the mainstream flow to be injected into, resulting in a coolant to mainstream density ratio near two. Heat transfer and film cooling effectiveness alterations on a film-cooled flat plate due to cooling fluid density variation have been illustrated by Pederson et al. [2.28], Baldauf et al. [2.29,30], and Johnson et al. [2.31], amongst others. Increasing coolant density led to a more favorable momentum ratio, allowing better coolant attachment, lessening heat transfer, and improving coolant effectiveness near the coolant exit. Coolant progression more than a few hole diameters downstream was not discussed in these studies.

The effect of coolant density on purge jet cooling has not been widely studied, with only two recent investigations documented. Chen et al. [2.32] and Li et al. [2.33] reported the effects of varying coolant density for a single upstream row of discrete holes on a three-dimensionally contoured and flat endwall, respectively. High-density coolant resulted in high near-hole effectiveness but degraded quickly due to endwall secondary flow domination. These experiments were conducted at low-speed conditions and lacked heat transfer data.

The prior literature study elucidates the impact of endwall and upstream geometry, including a step and gap, and mainstream and coolant flow conditions on endwall film effectiveness and heat transfer. To the best of the authors’ knowledge, no research has illustrated engine-representative geometry's effects on upstream purge jet cooling amid engine-representative mainstream and coolant conditions. Furthermore, much of the limited reported work matching coolant density ratio lacks heat transfer data, leaving a lacuna in the complete understanding of
purge jet cooling. For this study, engine consistent mainstream conditions, coolant conditions, and geometry are utilized. The conclusions presented herein are intended to help understand, test, and design hot section components and purge jet cooling schemes. As a continuation of this paper, Part 2 – Effect of Combustor-NGV Misalignment investigates the effect of upstream geometry on endwall cooling effectiveness and heat transfer with an engine consistent steep-injection purge jet cooling scheme while mimicking in-engine coolant to mainstream density ratio.

2. EXPERIMENTAL METHODOLOGY

2.1. Experimental Test Facility

The Virginia Tech Transonic Wind Tunnel, seen in Fig. 2.1, was used to gather the data presented. This facility is a blowdown tunnel where quasi-steady-state aerodynamic and transient heat transfer experiments are conducted on linear cascades. The tunnel has a maximum mass flow rate of 4.5 kg/s and is capable of 30-second test runs at exit Mach numbers from 0.6 to 1.2. Mainstream air is fed by a 18900-liter tank pressurized by two Boge industrial air compressors. In an effort to ensure minimal moisture content, the air is directed through an Aircell dryer before storage, effectively lowering the dew point to -70°C. According to an in-house developed algorithm, a butterfly valve progressively actuates during testing to ensure steady test section inlet pressure. For heat transfer experiments, before the blowdown testing, valves 1 and 2 are set to allow a forced convection flow over two 36kW heaters to heat the flow straightening copper pipes. Upon reaching desired thermal conditions in the copper pipes, valves 1 & 2 are reset. The blowdown test can then be initiated to allow for a 30-second test run, imparting a transient mainstream temperature with the warm air to allow for heat transfer to occur from the mainstream flow to the endwall. Additional details, descriptions, and testing procedures for the Virginia Tech
Transonic Wind Tunnel are outlined in Nasir et al. [2.34], Arisi et al. [2.35], Holmberg and Diller [2.36], and Roy et al. [2.37].

![Figure 2.1. Virginia Tech transonic wind tunnel facility](image)

Table 2.1 shows the vane and endwall geometry for all test cases reported herein. The linear cascade used for testing, consisting of 4 equally spaced NGV profiles, with half vane profiles at top and bottom, is shown in Fig. 2.2(a). A tailboard was mounted downstream of the top two vane passages to ensure mainstream flow periodicity. Mainstream flow entered the NGV passage through a passive turbulence grid imparting 16% large-scale freestream turbulence, consistent with flow under engine conditions. Additional details regarding the turbulence grid used can be found in the work of Nix et al. [2.38]. The test section was instrumented with a Pitot-Static probe 2.8 $C_{ax}$ upstream of vane leading edge, a total temperature probe at 4.6 $C_{ax}$ upstream of vane leading edge, and an upstream and downstream set of 6 static pressure ports at 1.4 $C_{ax}$ upstream of the vane leading edge and 0.4 $C_{ax}$ downstream of the trailing edge respectively. All test section pressure measurements were obtained using a NetScanner Model 98RK pressure scanner at 10Hz and temperature measurements using a NI compactDAQ at 30Hz.
Table 2.1. Vane and Cooling Hole Geometry

<table>
<thead>
<tr>
<th>Vane Geometry</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Chord, $C_{ax}$</td>
<td>50 mm</td>
</tr>
<tr>
<td>True Chord, $C$</td>
<td>91.2 mm</td>
</tr>
<tr>
<td>Vane Pitch, $P$</td>
<td>83.1 mm</td>
</tr>
<tr>
<td>Inlet and Exit Angle</td>
<td>0° and 73.5°</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Cooling Hole Geometry</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Injection Angle</td>
<td>50° from axial</td>
</tr>
<tr>
<td>Coolant Hole Diameter, $D$</td>
<td>2.39 mm</td>
</tr>
<tr>
<td>Coolant Hole $P/D$</td>
<td>3.5</td>
</tr>
<tr>
<td>Coolant Hole $L/D$</td>
<td>1st Row – 5.3, 2nd Row – 6.0</td>
</tr>
</tbody>
</table>

Figure 2.2. (a) Instrumented test section cascade geometry, (b) endwall profile with purge jet cooling scheme, and (c) the coordinate system in the data region

The vane and endwall geometry were 3D printed using a Stratasys Fortus 250mc and ABSPlus-P430 (acrylonitrile butadiene styrene). To ensure a smooth surface finish, traditional sanding and acetone smoothing techniques were employed. For cooling hole smoothness and diameter consistency, a tight-fitting metal tube was inserted and epoxied in each cooling hole. Figure 2.2(b) shows a pitch-wise view of the test piece geometry. The geometry consisted of an axisymmetric converging endwall, a gap representative of combustor/vane interface at 0.9 $C_{ax}$
upstream of vane leading edge, and a 50° injection angle doublet staggered purge jet cooling scheme at 0.38 $C_{ax}$ upstream of vane leading edge.

The data region of interest is shown in Fig 2.2(c), including the center passage endwall, as seen in Fig 2.2(a), and the region upstream. The data on the contoured endwall is projected onto a plane normal to the engine radial direction. In this study, the coordinate is defined as such:

- The origin is defined at the vane's leading edge below the center passage.
- $X$ is the axial coordinate, non-dimensionalized as $x/C_{ax}$, where $C_{ax}$ is the axial chord of the vane.
- $Y$ is the coordinate in the pitch-wise direction, non-dimensionalized as $y/P$, where $P$ is the pitch distance between the vanes.

This coordinate system is used throughout Chapters 2-4. A FLIR A325sc model infrared camera was used to track the endwall surface temperature at 30Hz accurately. The endwall was viewed through a Germanium (Ge) optical window with a broadband anti-reflective coating (BBAR) that demonstrated an average transmission of 95% in the wavelengths between 6 to 13$\mu$m. A small portion of the coolant used was sulfur hexafluoride (SF$_6$) which has a sharp, strong absorption region near 10$\mu$m, leading to slightly lower perceived endwall surface temperatures. Figure 2.3 shows the model of the Germanium optical window, mounting assembly, and the center passage from the IR camera perspective. An ultra-flat black paint was applied to the testing geometry resulting in emissivity of 0.97 at a wavelength of 5$\mu$m. FLIR calibrated and certified the camera before use. To further account for effects related to the three-dimensional endwall and Germanium optical window, an in-test section calibration was conducted for a 20-70°C temperature range. The calibration curve thus obtained was implemented prior to all data reduction.
Figure 2.3. Assembly model of IR camera view during wind tunnel blowdown

2.2. Coolant Supply and Conditions

To decouple traditional coolant flow parameters, coolant density and blowing rate are independently varied for this investigation. All coolant flow parameters reported were calculated as in Eq. 2.1-2.4 and evaluated by taking the average across all cooling hole entrances.

\[ M = \frac{\rho_c V_c}{\rho_\infty V_\infty} \]  
(2.1)

\[ DR = \frac{\rho_c}{\rho_\infty} \]  
(2.2)

\[ I = \frac{\rho_c V_c^2}{\rho_\infty V_\infty^2} \]  
(2.3)

\[ MFR = \frac{\rho_c V_c A_c}{\rho_\infty V_\infty A_\infty} \]  
(2.4)

Two coolant densities were chosen for investigation: \( DR = 1.2 \), corresponding to common experimental neglect of density ratio and simulated using compressed air as the coolant medium,
and $DR = 1.95$, an engine-representative density ratio achieved by controlled mixing of compressed air and industrial-grade SF$_6$ in this experiment.

Teekaram et al. [2.39] conducted a pioneering study demonstrating the validity of using foreign gas to simulate higher density coolant. In the current study, each constituent was independently regulated and monitored using Lambda Square beveled and corner-tapped orifice plates. The results were corrected using associated upstream and downstream pressure measurements, T-type thermocouples, and ISO 5167: Part 2 to calculate accurate flow rates. Each constituent's mass flow was set to achieve the intended coolant to mainstream blowing and density ratio before each test run. To prevent gas stratification, three turbulent mixers were integrated along the coolant flow path. Three static pressure and temperature measurements evenly spaced in the pitch-wise direction were made in-plenum for accurate density ratio calculation. Figure 2.4 shows a diagram of the coolant control, mixing, and delivery system.

![Figure 2.4. Coolant control, mixing, and delivery system](image)

### 3. DATA REDUCTION TECHNIQUES

The endwall heat flux in time was calculated using the endwall surface temperature acquired via an IR camera and the method described by Cook and Felderman [2.40]. This method utilizes a one-dimensional semi-infinite conduction model assuming uniform initial conditions.
throughout and negligible change of material property with subjected temperatures. These assumptions are enabled by the low thermal conductivity of the endwall material, ABSPlus-P430 \((k = 0.188 \text{W/m-K})\), the large thickness of the endwall, and the short data collection window (7 sec.). The thermal properties of the printed ABS piece was measured by TPRL Inc. in 2015. The detailed validation can be found in Appendix C. Eq. 2.5 is the governing equation in terms of \(T(0, t)\).

\[
q''(t_m) = \frac{2\sqrt{\rho c_p}}{\sqrt{\pi}} \sum_{i=1}^{m} \frac{T(0, t_i) - T(0, t_{i-1})}{\sqrt{t_m - t_i} + \sqrt{t_m - t_{i-1}}}
\]

(2.5)

The accuracy of the heat flux calculated is more directly influenced by the local temperature-time relationship than the temporal extrema. To ensure a precise approximation of the surface temperature profile in time with discrete line segments, a sampling rate of 30Hz was employed for the IR camera.

The recovery temperature, \(T_r\), heat transfer coefficient, \(HTC\), and adiabatic film cooling effectiveness, \(\eta\), were calculated simultaneously using a Dual Linear Regression Technique outlined in Xue et al. [2.41]. The governing equation for this technique, Eq. 2.6, is a linearized combination of the definition of convective heat transfer coefficient and film cooling effectiveness, as defined in Eqs. 2.7 and 2.8.

\[
\frac{q''}{T_r - T_c} = \frac{HTC}{T_r - T_c} \left( \frac{T_r - T_w}{T_r - T_c} \right) - HTC \times \eta
\]

(2.6)

\[\text{HTC} = \frac{q''}{T_{aw} - T_w}\]

(2.7)

\[\eta = \frac{T_r - T_{aw}}{T_r - T_c}\]

(2.8)

The calculation method is an optimization of the \(R^2\) value associated with the linear regression of Eq. 2.6 based on iterated recovery coefficients, \(C_r\), which is a factor used to
incorporate the dissipation of energy from the thermal boundary layer to the mainstream. A convergence criterion was employed to avoid convergence on singularities resulting in unphysical \(HTC\), \(\eta\), or \(T_r\) values. \(C_r\), \(\eta\), and \(HTC\) are hydrodynamic parameters and remain constant across both test runs using ambient and chilled coolant, employed for optimal signal-to-noise ratio, with preservation of mainstream aerodynamic and coolant blowing conditions. Following the optimized guess of \(C_r\) and associated linear regression, \(HTC\) was found as the slope. Then \(\eta\) was calculated as the y-intercept divided by \(HTC\) for each pixel. In this study, \(HTC\) is non-dimensionalized using the Nusselt number given in Eq. 2.9, where \(C\) is the true chord length of the vane.

\[
Nu = \frac{HTC \times C}{k_{air}}
\]  

(2.9)

Commonly in gas turbine heat transfer, a paradox exists in the fact that the injection of cooler fluid to shield the endwall from the high-temperature and high-velocity mainstream gases imparts additional turbulence and augments the local heat transfer coefficient. Net heat flux reduction (\(NHFR\)) evaluates the change in endwall heat flux as a result of coolant introduction, intrinsically accounting for augmented local \(HTC\) and lowered surface temperature. \(NHFR\) is an analytical expression combining \(\eta\) and \(HTC\) for a cooling case and the \(HTC\) for an uncooled case, as shown in Eq. 2.10.

\[
NHFR = 1 - \frac{HTC_f}{HTC_o} \left(1 - \frac{\eta}{\varphi}\right)
\]  

(2.10)

where Mick and Mayle [2.42] stated that an approximate overall effectiveness, \(\varphi\), of 0.6 is valid considering realistic turbine inlet, coolant, and component temperatures.

The uncertainties associated with the calculation of \(HTC\) and \(\eta\) were examined using Moffat's perturbation method [2.43], expanding on the methods first introduced by Kline and McClintock [2.44]. For this, \(HTC\) and \(\eta\) were reduced to the measured quantities used for
calculation: mainstream, coolant, and endwall surface temperature. The IR camera has a reported uncertainty of $\pm 2.0\%$ or $\pm 2.0^\circ C$, whichever is greater. Coolant and mainstream temperatures were monitored with T-type thermocouples reporting an uncertainty equal to the greater of $\pm 0.75\%$ or $\pm 1.0^\circ C$. The calculated uncertainty with the 95% confidence interval based on these is $\pm 9.6\%$ in $HTC$ and $\pm 0.1$ in $\eta$, as detailed in Appendix B.

Introducing a small percentage of SF$_6$ (less than 1% $MFR$ for all cases) creates additional uncertainty due to slightly lower perceived endwall temperatures. However, when considering the small percentage used and robust data reduction technique, the uncertainty due to SF$_6$ is minimal compared to the aforementioned sources.

Regarding the flow condition, the freestream Mach number was maintained within $\pm 3\%$ of the defined condition, 0.85. Coolant to mainstream blowing and density ratios were set within $\pm 3\%$ of the sought conditions.

### 4. COMPUTATIONAL SETUP

In this paper, supportive CFD simulations were carried out to gain more understanding of the different flow physics caused by various coolant density ratios.

#### 4.1. Solver and Turbulence Models

The CFD simulations were performed by solving the steady-state Reynolds-Averaged Navier Stokes (RANS), using the commercial CFD software ANSYS Fluent v15.0. The solution method is a pressure-based, steady-state, pressure-velocity coupling algorithm with the Realizable $k$-$\epsilon$ turbulence model using an enhanced wall treatment. The selection of solver and turbulence model has been further discussed in Appendix D. This method accounted for the compressibility
effects, curvature correction, viscous heating effects, and production limiters, making it one of the most accurate models for this study's flow conditions. In the Reynolds-averaged thermal energy equation, the constant turbulent Prandtl number was set to 0.95. Apart from the air ideal gas model used for the mainstream, a mixture of air and SF6 was defined using the experimental data as the coolant's working fluid. For both working fluids, their viscosity, conductivity, and specific heat were modeled using molecular kinetic theory.

4.2. Computational Model and Mesh

Figure 2.5 shows the computational model. The multi-block structured meshes were generated using ICEM with periodic boundary conditions imposed along the boundaries in the pitch-wise direction. For the mainstream flow passage, there were 9.3 million nodes, while another 3.2 million nodes were generated for the cooling holes and plenum section. In the endwall boundary layer regions, the mesh was refined with a first cell height of $1.5 \times 10^{-6}$ m and an expansion factor of 1.12 to capture the development of the boundary layer flow. A lower-than-0.8 $y^+$ value was achieved on vanes, endwalls, and internal walls of cooling holes. Based on the grid independence analysis published in the previous paper by Li et al. [2.16], a mesh consisting of 7.6 million nodes was fine enough to realize grid-independent HTC values on the same vane endwall geometry without film cooling features. In this paper, compared to the meshing scheme of Li et al. [2.16], refined dense mesh with an O-typical grid was generated in the vicinity of film cooling holes to capture interactions between coolant and mainstream better. The grid independence study based on the average endwall film cooling effectiveness shows that the additional 1.7 million nodes for mainstream flow passage plus the 3.2 million nodes for cooling holes and plenum meet grid independence requirements.
4.3. CFD Boundary Conditions

As shown in Fig. 2.5, the vane passage inlet in the CFD domain was located at $1.2C_{ax}$ upstream of the vane leading edge, where a Pitot-static probe was used to measure the total inlet pressure and static pressure in experiments. The mainstream inlet boundary conditions were specified using the measured values of total pressure, total temperature, and turbulence scale and intensity from the team’s previous experiments [2.16]. At the outlet, an average constant pressure condition was applied. On the coolant side, the inlet flow rate was set using the experimental measurements with various blowing and density ratios. Adiabatic no-slip wall boundary conditions were specified for the walls of the plenum and film cooling holes.

4.4. Heat Transfer Coefficient and Adiabatic Cooling Effectiveness Prediction Methodology

To evaluate the film cooling effectiveness and the heat transfer coefficient simultaneously, three independent CFD simulations were carried out for each test condition, as shown in Table 2.2. This three-simulation technique for the computations of adiabatic cooling effectiveness and heat
transfer coefficient in a film cooling environment is an extension of a two-simulation technique
documented by Li et al. [2,32]. In Simulation 1, an adiabatic no-slip wall condition was applied
on the vane and endwall surfaces with no coolant blowing \((M = 0)\). This simulation yielded the
endwall local mainstream recovery temperature \(T_r\), which was equal to the near-wall film flow
temperature \(T_{f,o}\) and the adiabatic wall temperature \(T_{aw,o}\) with no coolant. In Simulation 2, the
coolant was added while the adiabatic no-slip wall condition remained unchanged to find the near-
wall film flow temperature \(T_{f,c}\) which is equal to the adiabatic wall temperature \(T_{aw,c}\) with coolant
blowing. Meanwhile, the wall temperature \(T_w\) is also obtained in this simulation. In Simulation 3,
the coolant was still injected, and a uniform wall temperature \(T_w = 300K\) with no-slip wall
boundary conditions was applied to vane and endwall. This last simulation provides the surface
heat flux \(q_{c''}\), which is essential for calculating the heat transfer coefficient.

<table>
<thead>
<tr>
<th>Run</th>
<th>Coolant</th>
<th>Endwall Boundary Condition</th>
<th>Deliverables</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>No</td>
<td>Adiabatic</td>
<td>(T_r (T_{f,o}, T_{aw,o}))</td>
</tr>
<tr>
<td>2</td>
<td>Yes</td>
<td>Adiabatic</td>
<td>(T_{f,c} (T_{aw,c}), T_w)</td>
</tr>
<tr>
<td>3</td>
<td>Yes</td>
<td>(T_w=300K)</td>
<td>(q_{c''})</td>
</tr>
</tbody>
</table>

Once the recovery temperature \(T_r\), adiabatic wall temperature \(T_{aw,c}\), and surface heat flux
\(q_{c''}\) were obtained, the heat transfer coefficient \(HTC_c\), \(HTC_o\), and the adiabatic film cooling
effectiveness \(\eta\) can be calculated using Eqs. 2.11-13,

\[
HTC_f = \frac{q^*_c}{T_{f,c} - T_w} = \frac{q^*_c}{T_{aw,c} - T_w}
\]

\[
HTC_o = \frac{q^*_o}{T_{f,o} - T_w} = \frac{q^*_o}{T_{aw,o} - T_w}
\]
\[ \eta = \frac{T_c - T_{fc}}{T_c - T_e} = \frac{T_{aw,o} - T_{aw,c}}{T_{aw,o} - T_e} \]  \hspace{1cm} (2.13)

where \( T_c \) is the temperature of coolant flow at coolant plenum inlet, relating to the density ratio, and the surface heat flux \( q_o \) for the no blowing case was calculated through the CFD simulation with no coolant by applying uniform wall temperature \( T_w = 300 \text{K} \) and no-slip wall boundary conditions to the vane and endwall surfaces.

5. RESULTS AND DISCUSSION

The following section presents a detailed analysis and explanation of the results obtained during this testing campaign. To start with, the authors compared their experimental results with the simulation results to validate the numerical study. Then three parameters that are commonly used to demonstrate the film cooling performance, namely the film cooling effectiveness, Nusselt number, and net heat flux reduction, were presented for varying coolant to mainstream density and blowing ratios. More insight was provided by showing the CFD results of the temperature contours maps at five different pitch-wise cross sections in the NGV passage. All these results were analyzed to demonstrate the importance of an engine-representative density ratio. Table 2.3 shows the mainstream and coolant flow conditions for all test cases.

**Table 2.3. Summary of Experimental Mainstream and Coolant Conditions**

<table>
<thead>
<tr>
<th>Expt</th>
<th>( Ma_{Exit} )</th>
<th>( Re_{Exit, cax} )</th>
<th>( Tu )</th>
<th>M</th>
<th>DR</th>
<th>I</th>
<th>MFR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.85</td>
<td>( 1.5 \times 10^6 )</td>
<td>16%</td>
<td>2.5</td>
<td>1.95</td>
<td>3.2</td>
<td>1.9%</td>
</tr>
<tr>
<td>2</td>
<td>0.85</td>
<td>( 1.5 \times 10^6 )</td>
<td>16%</td>
<td>2.5</td>
<td>1.20</td>
<td>5.2</td>
<td>1.9%</td>
</tr>
<tr>
<td>3</td>
<td>0.85</td>
<td>( 1.5 \times 10^6 )</td>
<td>16%</td>
<td>3.5</td>
<td>1.95</td>
<td>6.3</td>
<td>2.7%</td>
</tr>
<tr>
<td>4</td>
<td>0.85</td>
<td>( 1.5 \times 10^6 )</td>
<td>16%</td>
<td>3.5</td>
<td>1.20</td>
<td>10.2</td>
<td>2.7%</td>
</tr>
</tbody>
</table>
5.1. CFD Validation

An oil paint flow visualization was conducted to identify the near-endwall flow physics, which was later compared with the CFD result to validate the simulations' credibility. Past studies have proved this method effective, including Loving et al. [2.45]. On the endwall surface, five oil-based paints of nearly identical viscosity were applied to different regions of interest. As shown in Fig. 2.6(a), the area immediately downstream of the two rows of cooling holes was covered by two bands of green and blue paints. A pink paint and an orange one were applied near the leading edge pressure side and leading edge suction side, respectively. After a 20-second blowdown with the same flow conditions as the heat transfer test runs, the paints formed various curves indicating the near-endwall streamlines, as shown in Fig. 2.6(b), with the CFD streamlines superimposed. It is to be noted that both the mainstream and coolant were at room temperatures during the blowdown conducted for flow visualization.
Both the green and the blue band paints formed continuous streaks along the flow direction, which suggested that some coolant merged with the mainstream and remained attached to the endwall to form a continuous film immediately after the coolant was injected. This feature is consistent with the CFD streamlines, which did not show apparent flow separation near the cooling holes. Region A showed a slight difference between the experiment and CFD results. As the cross-flow swept the coolant film against the suction side in this region, the oil-paint streamlines and the CFD streamlines can both be seen curved towards the suction side. Between the two sets of curves, the CFD result suggested a greater pitch-wise velocity component, which meant a stronger cross-flow. This could result from the fact that CFD produced a higher cross pressure gradient near the passage's inlet. However, in region B, where the compressibility effects cannot be ignored due to
flow acceleration, the two results showed a good agreement. Nearly all the streamlines extended from the upstream region ended on the suction side wall of the vane. This pattern indicated that the cross-flow was so strong at the mid-passage that it forced the coolant film to be convected onto the suction side endwall. Overall, the CFD streamlines generally agreed with the flow visualization results and captured the main features.

Figure 2.7 compared the pitch-wise averaged film cooling effectiveness between the experimental and CFD results. The results showed a similar trend, where $\eta$ peaked near the NGV passage inlet ($x/C_{ax} = 0$) and then slowly decayed. The decay of film cooling effectiveness in the CFD results was slower in the passage, which led to CFD slightly over-predicting the cooling effectiveness from $x/C_{ax} = 0.2$ to 1. Nevertheless, this difference was also observed in a previous study by El-Gabry et al. [2.46]. They found the same trend that CFD cooling effectiveness decreased slower than experiment results in the film-cooled NGV passage.

![Figure 2.7](image)

**Figure 2.7. Experimental and computational pitch-wise averaged film cooling effectiveness**

From the above comparisons of the streamlines and the film cooling effectiveness between the CFD and experiments, the authors are comfortable that the simulation is qualitatively accurate
enough to understand the flow development and to aid in explaining the experimental results. The flow field information from the simulation will be used in Chapters 2 and 3 of this study to shed insight into flow physics.

5.2. Adiabatic Film Cooling Effectiveness for Different Blowing Ratios and Density Ratios

The adiabatic film cooling effectiveness evaluated the endwall temperature reduction caused by the coolant. Studying this parameter will allow us to learn the coolant coverage produced by different blowing ratio-density ratio combinations.

The film cooling effectiveness of four test cases was presented in Fig. 2.8. The engine design condition exhibits distinguishable cooling streaks of high effectiveness immediately downstream of injection, as in Fig. 2.8(a). Meanwhile, all the other test cases experienced some jet lift-off following injection, leading to a region of little to no cooling effectiveness following cooling holes. This is mainly caused by the high injection angle together with the high coolant momentum. By comparing the reattachment lines, it can be found that increasing coolant density significantly improved upstream region cooling performance across both blowing rates nearly identically. Conversely, increasing coolant blowing rate degraded coolant performance in this region at both coolant density ratios. Therefore, the coolant momentum and density effects dominated cooling performance upstream of the leading edge. Testing with experimental neglect of coolant density leads to an overestimation of the in-engine coolant jet lift-off and subsequent underprediction of film performance upstream of the leading edge.
Region A, which covered the area near the passage inlet, showed a different trend across the cases shown. Despite more intense jet lift-off upstream, a lower density ratio or higher blowing ratio resulted in better cooling performance in this area. At $M = 2.5$, compared to $DR = 1.95$ (Fig. 2.8(a)), the coolant at lower $DR = 1.20$ (Fig. 2.8(b)) led to an overall increase of 0.1 in film cooling effectiveness. The improvement was most significant near the leading edge on the pressure side, where the increase in $\eta$ exceeded 0.2. By lowering the coolant density, the coolant momentum was increased by over 60%. This additional momentum helped the coolant penetrate the horseshoe vortex near the leading edge and reach closer to the vane. With the bifurcation of the horseshoe vortex, those coolants could extend downstream and cover a larger area near both the pressure side and the suction side.

However, the same trend could hardly be observed at $M = 3.5$ (Figs. 2.8(c) and (d)) for region A since they showed relatively similar cooling performance. This difference between $M = 2.5$ and 3.5 could result from the fact that the coolant momentum at $M = 3.5$ was sufficient to dominate HSV. Despite the difference in density ratio, for the cases shown in Figs. 2.8(c) and
a similar amount of coolant was reattached to the endwall, leading to similar cooling effectiveness. As a result, applying lower coolant density could lead to overestimating the cooling performance near the NGV passage inlet, especially on the endwall near the vanes. However, this difference could be mitigated by a higher blowing ratio.

Region B ranged from mid-passage to near the exit. Due to the passage contraction, flow turning, and acceleration, the secondary flow in this region usually is very intense, which can significantly influence the film behavior. At \( M = 2.5 \) (Figs. 2.8(a) and (b)), this area's general trend is the same as region A. The low coolant density led to higher cooling effectiveness, which can be attributed to more coolant making its way into the passage because of higher momentum.

At \( M = 3.5 \) (Figs. 2.8(c) and (d)), the average cooling effectiveness showed little difference for two coolant densities. However, it was observed that the coolant of higher density resulted in better cooling near the suction side. In comparison, the coolant of lower density favored the endwall cooling near the pressure side. A hypothesis is that a layer of coolant was attached to the pressure side wall, which was generated from the coolant film bifurcation at the leading edge. It was driven by the mainstream pressure gradient and shielded the pressure side vane and endwall, which explained the high \( \eta \) streak along the pressure side in Fig. 2.8(d). As the pressure side coolant film traveled further downstream, the passage vortex induced by the pitch-wise pressure gradient and pressure side leg of the HSV gradually developed in size. Meanwhile, it continued to sweep the coolant film towards the suction side, where the accumulated coolant started to form another coolant film, corresponding to the thin high \( \eta \) streak along the suction side in Fig. 2.8(c). This flow pattern could also exist at \( M = 2.5 \) but was not as apparent due to a weaker coolant film. In summary, failing to match the engine density ratio could have a complex impact in region
B. It could lead to an erroneous evaluation of overall cooling performance as well as local overestimation simultaneously.

To further investigate the coolant behavior due to various density ratios, the near-endwall temperature fields close to the pressure side at five different cross sections are shown in Fig. 2.9. At each cross-section, the low-temperature area could be perceived as the area where the coolant accumulated. In the first three cross-sections, the larger low-temperature areas on the pressure side indicated more coolant was attached to the pressure side vane at $DR = 1.20$. From $x/C_{ax} = 0.8$ to $0.95$, the passage vortex started to dominate the pressure side film and thinned the coolant film significantly, even though low-density coolant still managed to maintain more coolant on the pressure side.
Comparing Fig. 2.9 with Figs. 2.8 (a) and (b), it could be inferred that the additional coolant seen near the vane leading edge had a profound influence. Once the coolant penetrated the HSV, it would reach the vane leading edge and move along the pressure side. Then in the mid-passage region, the cross flow continuously swept the coolant towards the suction side, as presented at $x/C_{ax} = 0.6$ and $x/C_{ax} = 0.8$. This process energized strong secondary flows and contributed to the coolant coverage in the passage. On the pressure side at $x/C_{ax} = 0.8$, a low-temperature area could be observed near the pressure side vane for $DR = 1.2$, indicating a still stable coolant film moving along the pressure sidewall. While in the case of $DR = 1.95$, little coolant was seen at this stage on the pressure side. This explained why minimal cooling effect was observed near the passage exit.
at $DR = 1.95$, but the coolant at $DR = 1.2$ resulted in $\eta$ about 0.25. These findings concurred with the previous hypothesis made at $M = 3.5$ and demonstrated how density ratios affected the coolant performance.

5.3. Nusselt Number for Different Density Ratios from Experiments

In addition to the cooling effectiveness, the endwall Nusselt number is another important parameter for evaluating the coolant performance by assessing the flow behavior near the endwall. Figure 2.10 shows the Nusselt number distribution for the two cases at baseline $M = 2.5$ from the experiment. Downstream of the second row of cooling holes, streaks of low $Nu$ could be observed in both cases, which suggested that some coolant was attached to the endwall despite the lift-off tendency discussed above. In region A, the coolant brought about lower Nusselt near the pressure side vane compared to mid-passage areas. The lower Nusselt number alongside the pressure side indicated the presence of a slow-moving flow, most likely the coolant film on the pressure side wall, which shielded the endwall from the relatively fast-moving hot mainstream.

![Figure 2.10. Endwall Nusselt Number distributions from experiments of (a) $M = 2.5$, $DR = 1.95$ and (b) $M = 2.5$, $DR = 1.2$](image)
Region B showed a rapid increase in Nusselt number from upstream but an even higher growth for the coolant of higher density. The sudden change in Nu is mainly caused by the flow acceleration near the throat. Meanwhile, the higher Nusselt number change can indicate stronger secondary flows caused by cross-passage pressure gradients. As the coolant film moved along the pressure side, it continuously suffered from dissipation caused by the pressure vortex. As a result, the coolant film grew thinner and eventually dominated and induced turbulent secondary flows starting at $x/C_{ax} = 0.5$. The same flow physics was present for $DR = 1.20$, but the secondary flows were less intense in that the pressure side coolant film could persist further downstream due to more coolant reaching the leading edge. It should be noticed that the stability of the coolant film on the in-passage endwall also contributed to the Nusselt number discrepancies and will be discussed in Chapter 3.

To sum up, failure to match the engine-representative coolant density ended in a gross overprediction of endwall Nusselt level compared to actual in-engine levels. This is due to an overestimation of the coolant reaching the pressure side, which contributed much to cooling the passage.

5.4. Net Heat Flux Reduction for Different Density Ratios from Experiments

Net heat flux reduction ($NHFR$) indicated the heat flux reduction introduced by adding the cooling scheme as given by Eq. 2.9. It revealed the conjoint effect of film cooling effectiveness and Nusselt number. The $NHFR$ of two cases at $M = 2.5$ is presented in Fig. 2.11. Region A shows the detrimental effects of lowering coolant density and subsequent jet lift-off in the near injection region. Near injection hoes, turbulence levels increased with a high coolant momentum ratio, augmenting endwall heat flux. Negative numbers apparent in Fig. 2.11(b) indicated that the coolant
introduction at these conditions drew more mainstream gases near the endwall, increasing heat flux in the upstream region beyond the uncooled case.

![Net Heat Flux Reduction](image)

**Figure 2.11. Net Heat Flux Reduction distributions from experiments of (a) $M = 2.5$, $DR = 1.95$ and (b) $M = 2.5$, $DR = 1.2$**

On the contrary, as seen in region B, in-passage $NHFR$ was significantly improved by reducing coolant density, attributed to a more lasting coolant film and secondary flow suppression. Compared to the case of $DR = 1.95$, $NHFR$ doubled in region B with a coolant of $DR = 1.20$. This overly optimistic evaluation caused by lowering the coolant density stresses the importance of matching the engine-presentative density ratio in any experimentation.

### 5.5. Discussion of Effect of Momentum Ratio

The above discussion proved that mismatching density ratios could lead to multiple wrong conclusions. Since most effects were explained by the coolant momentum difference, this section will discuss if matching only the momentum ratio would produce reliable data.

Figures 2.8 (b) and (c) showed a pair of tests less than 20% apart in a similar momentum ratio. In these cases, the reattachment lines were very close despite the 50% difference in $MFR$,
suggesting that the coolant behavior was predominately determined by the coolant momentum near the cooling holes. Before the coolant had time to develop, its initial momentum played a primary role in coolant-mainstream mixing, which had a first-order effect on the endwall heat transfer around the passage inlet. This flow pattern also explained that apart from the nearly identical reattachment lines, those two cases also had similar cooling effectiveness distribution in Region A.

Nevertheless, the case in Fig. 2.8(b) displayed inferior cooling performance to the case in Fig. 2.8(c). Since the two cases had similar momentum ratios, the coolant mass flow rate was likely to make a difference. Naturally, once the momentum was high enough to help coolant penetrate HSV, the coolant mass flow would determine how much coolant could reach the passage. The additional 50% coolant mass flow contributed to a thicker coolant film forming on the pressure side, which overcame the passage vortex more easily and offered better coolant coverage in the passage for the reasons discussed above. However, this effect has nothing to do with the momentum ratio.

To summarize, matching the momentum ratio without matching the density ratio could result in an engine-like cooling scenario near the cooling holes. The coolant momentum dominated the near-endwall flow patterns. Nevertheless, the difference would emerge further downstream, where additional coolant mass flow was needed to maintain a stable coolant film on the pressure side. Therefore, matching only the momentum ratio was insufficient to offer engine-representative results.
6. SUMMARY AND CONCLUSIONS

The study presented is a detailed experimental analysis of the decoupled effects of coolant to mainstream blowing and density ratios on endwall heat transfer and adiabatic film cooling effectiveness. The coolant density ratio was defined to illustrate the effects associated with typical experimental neglect. The data presented were gathered at the Virginia Tech Transonic Wind Tunnel facility under transonic conditions with engine-representative mainstream exit Reynolds number and turbulence intensity. An engine consistent axisymmetric converging endwall with upstream combustor-NGV interface geometry at baseline (no misalignment/step) was employed with a staggered doublet high-injection angle purge jet cooling scheme. Chapter 3 of this study will continue to analyze the endwall thermal effects of backward and forward-facing steps. In addition, high-resolution endwall $\eta$, $Nu$, and $NHFR$ distributions were analyzed to understand effects in the upstream and in-passage regions. The key findings of the work are summarized in short:

(1) Coolant to mainstream density ratio and blowing ratio should be both considered and accounted for in the cooling scheme design, experimentation, and simulation. Neglecting either leads to a misprediction of endwall film effectiveness, heat transfer coefficient and heat flux profiles.

(2) With the same blowing ratio, the density ratio mainly affects the endwall heat transfer by altering the coolant momentum. Near the cooling holes, higher coolant density leads to lower momentum, mitigating the jet lift-off and helping the coolant merge into the mainstream, thus resulting in better cooling immediately downstream of the holes. However, the lower momentum causes less coolant to reach the leading edge and move along the pressure side. Due to the passage vortex effect, this leaves the in-passage endwall near the pressure side less protected and causes
the coolant coverage to decay faster in the flow direction. At a certain blowing ratio, the lower coolant momentum could lead to better coolant coverage near the suction side.

(3) The cooling performance near the cooling holes is more predictable by matching the momentum ratio than the blowing ratio. However, only applying the engine-like momentum ratio without matching the density ratio or mass flow ratio leads to significant errors further downstream.

(4) For a given NGV film cooling design, a combination of density ratio and blowing ratio must be identified instead of only blowing ratio or mass flow ratio to optimize the cooling performance.

7. NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Area, mm²</td>
</tr>
<tr>
<td>C</td>
<td>Vane Chord, mm</td>
</tr>
<tr>
<td>( c_p )</td>
<td>Specific Heat at Constant Pressure, J/(kg·K)</td>
</tr>
<tr>
<td>( C_r )</td>
<td>Recovery Coefficient</td>
</tr>
<tr>
<td>( HTC )</td>
<td>Heat Transfer Coefficient, W/(m·K)</td>
</tr>
<tr>
<td>( k )</td>
<td>Thermal Conductivity, J/(m·s·K)</td>
</tr>
<tr>
<td>( L )</td>
<td>Cooling Hole Length, mm</td>
</tr>
<tr>
<td>( P )</td>
<td>Cooling Hole Pitch Length, mm</td>
</tr>
<tr>
<td>( q'' )</td>
<td>Heat Flux, W/m²</td>
</tr>
<tr>
<td>( T )</td>
<td>Temperature, K</td>
</tr>
<tr>
<td>( t )</td>
<td>Time, s</td>
</tr>
<tr>
<td>( V )</td>
<td>Velocity, m/s</td>
</tr>
<tr>
<td>( x )</td>
<td>Coordinate, mm</td>
</tr>
</tbody>
</table>

Greek Letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \eta )</td>
<td>Adiabatic Film Cooling Effectiveness</td>
</tr>
<tr>
<td>( \rho )</td>
<td>Density</td>
</tr>
<tr>
<td>( \phi )</td>
<td>Overall Cooling Effectiveness</td>
</tr>
</tbody>
</table>

Non-dimensional Numbers

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( DR )</td>
<td>Density Ratio, ( \frac{\rho_c}{\rho_\infty} )</td>
</tr>
</tbody>
</table>
I Momentum Ratio, \( \left( \frac{\rho_c V_i^2}{\rho_v V_\infty^2} \right) \)

M Blowing Ratio, \( \left( \frac{\rho_c V_c}{\rho_v V_\infty} \right) \)

Ma Mach Number

MFR Mass Flow Ratio, \( \left( \frac{\rho_c V_c A_c}{\rho_v V_\infty A_\infty} \right) \)

NHFR Net Heat Flux Reduction, \( (1 - \frac{HTC_f}{HTC_o}(1 - \eta)) \)

Nu Nusselt Number

Re Reynolds Number

Tu Turbulence Intensity

**Subscripts**

- ax Axial
- aw Adiabatic Wall
- c Coolant Flow
- exit Vane Passage Exit
- f Film Cooled Case
- o Uncooled Case
- r Recovery
- w Endwall
- \( \infty \) Freestream Flow

**Abbreviation**

- HSV Horseshoe Vortex
- NGV Nozzle Guide Vane

8. ACKNOWLEDGEMENTS

The authors would like to acknowledge and thank Solar Turbines Incorporated for the funding, support, and guidance on the works presented herein.
REFERENCES


CHAPTER 3
EXPERIMENTAL STUDY OF THE ENDWALL HEAT TRANSFER OF A TRANSONIC NOZZLE GUIDE VANE WITH UPSTREAM JET PURGE COOLING: PART 2 – EFFECT OF COMBUSTOR-NOZZLE GUIDE VANE MISALIGNMENT

Shuo Mao*, Ridge Sibold, Wing F. Ng
Virginia Tech
Blacksburg, VA, United States

Zhigang Li, Bo Bai
Xi’an Jiaotong University
Xi’an, China

Hongzhou Xu, Michael Fox
Solar Turbines Incorporated
San Diego, CA, United States

ABSTRACT

A misalignment between the combustor exit and the nozzle guide vane (NGV) platform commonly exists due to manufacturing tolerances and thermal transience. This study investigated, experimentally and computationally, the effect of the combustor-turbine misalignment on the heat transfer for an axisymmetric converging endwall with a jet purge cooling scheme at transonic conditions. The studies were conducted at engine-representative exit Mach number of 0.85, inlet turbulence intensity of 16%, and exit Reynolds number of $1.5 \times 10^6$. A film cooling blowing ratio of 2.5 (design condition) and 3.5 and an engine-representative density ratio of 1.95 were used in the study. Three various step misalignments, combustor exit being 4.9% span higher than turbine inlet (backward-facing), no step (baseline), and combustor exit being 4.9% span lower than turbine inlet (forward-facing), were tested to demonstrate the misalignment effect on endwall heat transfer.

Results indicated that the step misalignment affects the cooling performance by altering the interaction between the coolant and the cavity vortex, horseshoe vortex, and passage vortex. At the design blowing ratio of 2.5, the backward-facing step leads to increased coolant dissipation, causing the coolant to be later dominated by the passage vortex and leading to poor cooling performance. Meanwhile, a forward-facing step induced more coolant lift-off. At the blowing ratio
of 3.5, the additional momentum ensures that enough coolant enters the passage to form a stable boundary layer. Therefore, the step misalignment no longer has a first-order effect.

1. INTRODUCTION

The gas temperature exiting the combustor has a first-order effect on power generation efficiency for gas turbine engines. Cooling of vanes and endwall, particularly for the first stage NGV directly following the combustor, is necessitated by increased combustor temperatures resulting in overheating and dramatically increasing the risk of endwall burnout. Standard cooling techniques for endwall thermal failure prevention include purge cooling, internal cooling, and showerhead cooling [3.1].

Purge jet cooling, the cooling scheme studied herein, is a conventional cooling method for NGV platforms. High-pressure discharge air is drawn from the compressor, routed around the combustor, and injected through holes on the NGV upstream of the vane leading edge. The discharge air injected is designed to form a cooling film to shield the endwall from high temperature and velocity mainstream gases.

The first stage NGV platform is subject to upstream misalignment with the combustor exit. The cause of misalignment is generally one or a combination of tolerance stack-up during the manufacturing and assembly of the hot section or uneven thermal expansion due to thermal transience regions during engine startup and shutdown phases. If neglected, the implications of this misalignment could lead to flow physics that vary from predicted, ultimately reducing the lifetime and durability of hot-section components.

This study investigates the effect of upstream misalignment on a high injection angle purge film cooling scheme at engine-representative $Ma$, $Tu$, $M$, and $DR$. Transonic wind tunnel tests were carried out to reveal the effect of combustor-NGV misalignment. In addition, a numerical study
was conducted in an attempt to help explain and understand flow physics. In conjunction with Chapter 2, which covers the effect of density ratio, this study offers references for designing and testing purge jet cooling schemes for the NGV endwall.

2. RELEVANT PAST STUDIES

The in-passage aerodynamics of NGV are well-understood now, thanks to multiple studies, including Herzig et al. [3.2], Jilek [3.3], Goldstein et al. [3.4], and Graziani et al. [3.5], amongst others. Two essential endwall flow features which directly impact purge jet cooling performance on the endwall are the horseshoe vortex (HSV) and the passage vortex. The HSV system is induced by the vane leading edge in conjunction with boundary layer velocity gradients. Then it bifurcates into the pressure side legs and suction side legs. As they migrate downstream, a passage vortex emerges from the pressure side HSV and sweeps across the passage from the pressure side to the suction side due to in-passage pressure gradients. These two features lay the foundation for NGV passage aerodynamics which aid in understanding and explaining coolant-mainstream interaction.

With regards to NGV endwall purge cooling, researchers have conducted many studies based on different cooling configurations. Some identified non-dimensional parameters are defined as follows,

\[ M = \frac{\rho_c V_c}{\rho_\infty V_\infty} \]  \hspace{1cm} (3.1)

\[ DR = \frac{\rho_c}{\rho_\infty} \]  \hspace{1cm} (3.2)

\[ MFR = \frac{\rho_c V_c A_c}{\rho_\infty V_\infty A_\infty} \]  \hspace{1cm} (3.3)
Oke & Simon [3.6] investigated upstream slot cooling over both flat and contoured endwalls at low speed. Results indicate that at low speed, the HSV and passage vortex create a cross-flow near the pressure side, which pulls the coolant to the pressure side, resulting in a better cooling performance near the pressure side compared to the suction side regardless of the MFR tested. Knost et al. [3.7] studied the effect of discrete hole cooling on flat endwall. Results show that coolant injected upstream of the inlet is pushed against the suction side, but the coolant injected mid-passage produces more extensive coverage on the pressure side. Experiments and CFD conducted by Thrift et al. [3.8] with upstream slot cooling over a contoured endwall indicate that the endwall near the pressure side is cooled better than the suction side at low-speed conditions. Oke et al. [3.9] experimentally investigated the formation of secondary flows with slot cooling over a flat endwall and a contoured endwall. Conclusions demonstrate that the secondary flow induced over a contoured endwall is weaker than over a flat endwall. Papa et al. [3.10] studied the effect of blowing ratio $M$ at low speeds for a flat endwall with film cooling. At lower $M$, the coolant cannot penetrate the mainstream boundary layer, resulting in a worsened cooling performance. Overall, the previous studies have shown various conclusions based on different conditions, highlighting the necessity of mimicking in-engine mainstream conditions, coolant conditions, and geometry to produce physical, engine-representative data.

Many researchers in the past have studied the effects associated with upstream misalignment at the combustor-NGV interface. As for experimental testing, Cardwell et al. [3.11] studied a discrete hole cooling scheme over a flat endwall at low speed. Results show that a backward-facing step (or cascade) increases the film cooling effectiveness in the passage by about 0.2, but a forward-facing step makes minimal difference. Piggush et al. [3.12] studied the effect of upstream misalignment on a flat endwall cooling scheme at low speed. The forward-facing step
thinned the mainstream boundary layer, which caused a slightly higher heat transfer rate, while the backward-facing step had the opposite effect. Luehr et al. [3.13], a more recent experimental study, show an opposite trend under transonic conditions. Based on their results, a backward-facing step negatively affects the discrete hole cooling over a contoured endwall, and the forward-facing step has little influence. The effects of the combustor-NGV misalignment were also studied by Zhang & Moon [3.14] and Chung et al. [3.15]. The inconsistency among the conclusions of the papers above indicates that the effect of upstream misalignment may not be absolute but depends on specific endwall geometry and flow conditions.

Similar inconsistency also exists in previous numerical studies. Du & Li [3.16] conducted a numerical study on step height using the same configuration as Cardwell et al. [3.11]. Results illustrated that if the step height of the backward-facing step passed a critical point, it could adversely affect the film cooling effectiveness. Hada & Thole [3.17] conducted CFD research on another discrete hole cooling scheme and showed similar results to Piggush et al. [3.12]. However, the CFD study by Chung et al. [3.15] showed an opposite effect to that reported by Hada & Thole. Their model suggested that the backward-facing step caused a higher thermal load on the endwall platform due to the step-induced vortex.

In Chapter 2, the author investigated the effect of density ratio with a contoured endwall at transonic conditions. Results indicate that various $DR$ can lead to discrepancies in film cooling effectiveness and $Nu$ distribution despite the same $M$ and $MFR$. Therefore, matching the engine-condition $DR$ is crucial to collecting accurate data.

Collectively, the previous research has illustrated that matching the flow condition and the endwall geometry is vital in studying the effect of upstream misalignment on purge cooling. Nevertheless, to the authors’ knowledge, this is the first study to incorporate an engine-
representative high DR, mainstream exit Ma and Re, and a contoured endwall amid the investigation of upstream misalignment. The study presented herein is designed to produce accurate in-engine effects of an NGV platform/combustor misalignment on a purge jet film cooling scheme by mimicking the flow conditions and the endwall geometry.

3. EXPERIMENTAL TEST FACILITY AND CONDITIONS

Data were gathered in the Virginia Tech Transonic Wind Tunnel in the Advanced Propulsion Power Lab, as shown in Fig. 3.1. This facility is a linear cascade blowdown wind tunnel with a heating section. The mainstream is supplied from a 18900L storage tank charged by two Boge Industrial Compressors capable of up to 1.27kg/s at 1207kPa. A dryer is installed between the compressor and the storage tank to ensure no moisture enters the tunnel. With a pneumatic butterfly valve at the inlet, the tunnel is capable of running a 30-second blowdown with a maximum steady flow rate of 4.5kg/s. For this research, the mainstream flow rate is set to 3kg/s in order to match the mainstream Re and Ma.

Figure 3.1. Virginia Tech transonic wind tunnel facility
Before each test run, Valve 1 is opened, and Valve 2 closes, effectively preventing flow into the test section and creating a heating recirculation loop. A bank of long copper tubes, acting to store the heat until the mainstream is actuated, is heated by circulating air which is driven by the recirculation fan and heated by two 36 kW heaters. Once the top of the heat exchanger reaches desirable temperatures, Valve 1 is closed, and Valve 2 opens, allowing flow to the test section but preventing flow into the heating loop. As the tunnel starts, the upstream valve opens to start the blowdown and maintains the flow rate following an in-house developed algorithm to maintain steady test section inlet pressure. The mainstream will pass through a turbulence grid to ensure a $Tu = 16\%$ at the inlet of the test section, which was measured and discussed by Nix et al. [3.18]. Once a suitable mainstream pressure increase is detected, the solenoid on the coolant side is triggered so that the coolant flows into the test section at approximately the same time as the mainstream enters the test section.

For the purpose of matching an engine-representative density ratio of 1.95, a mixture of air and SF$_6$ is used as the coolant. Before mixing and entering the cooling loop, the mass flow rate of both air and SF$_6$ is measured via an orifice meter independently. Downstream of the cooling hosing, a solenoid valve holds the coolant until triggered by a mainstream pressure rise, as discussed above, preventing endwall pre-cooling or pre-heating. For each test case, two test runs, one with chilled coolant and the other with non-chilled coolant, are conducted due to the requirements of the data reduction technique. This process is designed to reduce the noise-to-signal ratio. When un-chilled coolant is sought, the cooling loop remains near room temperature and delivers coolant to the test section in a similar manner.

As shown in Fig. 3.2, the endwall and vanes are 3D printed using the Fused Deposition Modeling (FDM) method of black ABS P-430Plus with a Stratasys Fortus 250mc printer. The
printed pieces are smoothed using sandpaper and acetone to minimize the effect of surface roughness on flow. Black paint is applied to the surface to ensure a consistently high emissivity. Due to the low thermal conductivity $k = 0.188 \text{ W/m-K}$ and the thickness of the ABS test piece, the endwall heat transfer can be simplified to a 1D semi-infinite model. However, this assumption is weaker near the film cooling holes because the ABS piece is thinner on the endwall side.

![Image of test section vane cascade]

**Figure 3.2. The geometry of the test section vane cascade**

The four printed ABS vanes are lined up with the two metal vanes inside the test section to create five passages so as to ensure periodicity and reduce the wall effect. A doublet-staggered row of purge cooling holes is located upstream of the passage of interest and spans two passages long. Details regarding the vane geometry, hole geometry, and mainstream flow conditions can be found in Table 3.1. It is worth noting that the high injection angle of 50° is designed for this specific upstream geometry and contoured endwall.
Table 3.1. Vane geometry parameters and test conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Chord ($C_{ax}$)</td>
<td>50 mm</td>
</tr>
<tr>
<td>True Chord ($C$)</td>
<td>91.2 mm</td>
</tr>
<tr>
<td>Pitch ($P$)</td>
<td>83.1 mm</td>
</tr>
<tr>
<td>Span ($S$)</td>
<td>152.4 mm</td>
</tr>
<tr>
<td>Coolant Injection Angle</td>
<td>50°</td>
</tr>
<tr>
<td>Coolant Hole Diameter</td>
<td>2.38 mm</td>
</tr>
<tr>
<td>Coolant Hole $P/D$</td>
<td>3.5</td>
</tr>
<tr>
<td>Coolant Hole $L/D$</td>
<td>5.8</td>
</tr>
<tr>
<td>Inlet Angle and Exit Angle</td>
<td>0° and 73.5°</td>
</tr>
<tr>
<td>Exit Reynolds Number ($Re_{exit, C_{ax}}$)</td>
<td>$1.5 \times 10^6$</td>
</tr>
<tr>
<td>Exit Mach Number ($Ma_{exit}$)</td>
<td>0.85</td>
</tr>
<tr>
<td>Turbulence Intensity ($Tu$)</td>
<td>16%</td>
</tr>
</tbody>
</table>

Three different upstream misalignments are investigated in this study: the combustor exit being 4.9% span length higher than the turbine platform inlet, the combustor exit remaining flush with the turbine platform inlet, and the combustor exit being 4.9% span length lower, as is depicted in Fig. 3.3.

A Pitot probe is located upstream of the center passage, as shown in Fig. 3.2. Along with a thermocouple positioned further upstream (not seen in Fig. 3.2), the inlet Mach number can be calculated. Six evenly spaced upstream pressure ports are located $1.4C_{ax}$ upstream of the leading edge, and another six downstream ports are located $0.4C_{ax}$ downstream of the trailing edge. These ports are used to measure the pressure drop across the passage and calculate the exit $Ma$. During
all the tests, $M_{exit} = 0.85 \pm 0.02$ was maintained to ensure consistency across test cases. The pressure ports shown in Fig. 3.2 are connected to a Netscanner Model 98RK pressure scanner for data collection and recording.

As presented in Fig. 3.4, the endwall temperature history was recorded by a FLIR A325SC IR camera through a Germanium window with a broad-band anti-reflective coating. The coating has a nearly constant transmissivity of 95% for the wavelength of interest 6–13μm. One of the coolant species, SF$_6$, has an absorption peak of around 10.5μm, which could cause the camera to read lower endwall surface temperature. However, since the mass flow of SF$_6$ is less than 1% of the mainstream, the error caused by the IR absorption by SF$_6$ is minimal compared to the uncertainty reported later. Prior to usage, the IR camera was calibrated through the Ge window on the test section to quantify the effects of the 3D endwall contour, the Ge window itself, and the black paint on the endwall.

![Figure 3.4. Model of IR camera view during wind tunnel blowdown](image)
4. DATA REDUCTION TECHNIQUE

With an initial condition of zero temperature gradient perpendicular to the endwall, a method devised by Cook and Felderman [3.19] is used to convert the endwall temperature history into an endwall heat flux history.

\[ q''(t_m) = \frac{2(\sqrt{k \rho c_p})}{\sqrt{\pi}} \sum_{i=1}^{m} \frac{T(0,t_i) - T(0,t_{i-1})}{\sqrt{t_m - t_i} + \sqrt{t_m - t_{i-1}}} \]  

(3.4)

where \( \rho \) is the density of ABS, \( k \) is its thermal conductivity; \( C_\rho \) is its specific heat at constant pressure; \( m \) is the number of time steps from the initial condition. For each run, the IR camera records a 20s video at a frame rate of 30Hz.

Using the endwall heat flux history and the history of \( T_\infty \) and \( T_c \), the dual linear regression technique (DLRT) developed by Xue et al. [3.20] was used to solve for \( \eta, \) HTC, and recovery temperature simultaneously. Equation. 3.6 was applied to Eq. 3.5 to cancel \( T_{aw} \), the equation thus obtained was rewritten as Eq. 3.7, where \( q'' \) is a linear function of \( \frac{T_r-T_w}{T_r-T_c} \), with HTC being the slope and \(-HTC \times \eta\) being the y-intercept. HTC is non-dimensionalized in this study as Nusselt number (\( Nu \)), defined using the true chord length as the characteristic length.

\[ q'' = HTC(T_{aw} - T_w) \]  

(3.5)

\[ \eta = \frac{T_r - T_{aw}}{T_r - T_c} \]  

(3.6)

\[ \frac{q''}{T_r - T_c} = HTC \frac{T_r - T_w}{T_r - T_c} - HTC \times \eta \]  

(3.7)

\[ Nu = \frac{HTC \times C}{k_{air}} \]  

(3.8)

By iterating over \( T_r \), an optimal linear regression was retrieved with corresponding HTC and \( \eta \). Next, the value of the recovery factor is checked to fall between 0.9 to 1.0 so that the result
makes physical sense. Otherwise, a new iteration is started with another initial guess. This process is repeated for each pixel of interest.

To evaluate the combined effects of $\eta$ and $HTC$, $NHFR$ is introduced. $NHFR$ is a non-dimensional value showing the heat flux reduction due to the additional film cooling compared to the non-cooled condition and is given by

$$NHFR = 1 - \frac{HTC_f}{HTC_o} (1 - \frac{\eta}{\phi})$$

where $\phi$ is approximated to 0.6 in this study. [3.21]

The experimental uncertainty of the deliverables is primarily attributed to the thermocouples for $T_\infty$ and $T_c$ measurement and the IR camera recording. For the IR camera, its uncertainty is $\pm 1.0^\circ C$ or $\pm 0.75\%$ of its full reading, whichever is higher, while the type T thermocouples’ uncertainty is the higher value of $\pm 2.0^\circ C$ and $\pm 2.0\%$ of its full reading. With the perturbation methods [3.22], some sample data with a different combination of perturbations are reduced with the DLRT method. Among all the results, the uncertainty given a 95% confidence interval is presented. In this study, the uncertainty of $HTC$ or $Nu$ is $\pm 9.6\%$, and the uncertainty of $\eta$ is $\pm 0.1$. The calculation is detailed in Appendix B.

5. COMPUTATIONAL SETUP

The CFD simulations of three cases were carried out to understand and explain the effects of endwall misalignment on endwall thermal load and film cooling coverage distributions. The simulations were performed by solving the steady-state Reynolds-averaged Navier Stokes (RANS), using the commercial CFD software ANSYS Fluent v15.0 based on a pressure-based, steady-state, pressure-velocity coupling algorithm. The Realizable $k-\varepsilon$ turbulence model with enhanced wall treatment was used. This model accounted for the effect of compressibility,
curvature correction, viscous heating, and production limiter. The Reynolds-averaged thermal energy equation used the constant turbulent Prandtl number 0.95. The ideal gas modeled by air was used as the working fluid. Viscosity, conductivity, and specific heat of the liquid were modeled using molecular kinetic theory.

Figure 3.5 shows the computational model and mesh with upstream double-row discrete film cooling holes for the vane passage. The computational model and mesh were generated, which included one single vane passage, 42 double-row coolant holes, and one coolant plenum with periodic boundary conditions imposed along the boundaries in the pitch-wise direction. The multi-block structured meshes were generated using ICEM CFD with the following specifications: low Reynolds number meshes with 9.3 million nodes for the vane passage and 3.2 million nodes for the coolant holes and plenum. In the wall boundary layer regions, the mesh was refined with an initial cell height of 1.5x10^{-6} m and an expansion factor of 1.12 to capture the boundary layer flow development. The y-plus value is less than 0.8 on vane surfaces, endwalls, and film cooling hole walls. Based on the grid independence analysis previously reported by Li et al. [3.23], 7.6 million nodes were fine enough to realize grid-independent HTC values on the same vane endwall without film cooling holes. In this paper, compared to the meshing scheme of Li et al. [3.23], refined dense mesh with an O-typical grid was generated in the vicinity of film cooling holes to capture interaction flow physics between coolant and mainstream. The present grid independence study based on the average endwall film cooling effectiveness shows that 1.7 million refined nodes for the vane passage regions near coolant holes and 3.2 million nodes for coolant holes and plenum meet grid independence requirements.
As shown in Fig. 3.5, the vane passage inlet in the CFD domain was located at $1.2C_{ax}$ upstream of the vane leading edge. The vane passage inlet boundary conditions were specified using the measured values of total pressure, total temperature, and turbulence (freestream turbulence intensity and integral length scale) taken during the experiments. Inlet total pressure and total temperature profile along the span were set at the vane passage inlet. Besides, the inlet turbulence intensity and length scale were taken as a constant across the vane span and equal to the value listed in Table 3.1. Depending on the corresponding blowing ratio values, the coolant flow rate was set at the coolant inlet. At the vane passage outlet, an average constant pressure condition was applied. Adiabatic no-slip wall boundary conditions were specified for the coolant plenum and film cooling hole walls.

Three independent CFD simulations were carried out for each particular coolant-to-mainstream blowing ratio and density ratio, with different vane and endwall surface thermal boundary conditions to evaluate the film cooling performance thoroughly. This three-simulation technique for the computations of adiabatic cooling effectiveness and heat transfer coefficient in a film cooling environment extends the two-simulation approach documented by Li et al. [3.23]. A
third CFD simulation with no coolant \((M = 0)\) is added to predict mainstream recovery temperature on the endwall in a high-speed flow. More details about the computational setup and methodology can be found in Chapter 2.

6. RESULTS AND ANALYSIS

To investigate the effect of upstream misalignments, six cases were studied experimentally, as shown in Table 3.2. As discussed above, two test runs with different coolant temperatures are conducted as required by our data reduction technique for each case. In addition, the highlighted three cases in Table 3.2 are studied numerically. The results demonstrate the effects of the combustor-NGV misalignment and how the effects differ at different blowing ratios. The validation of the CFD results using the experimental results is documented in Chapter 2.

<table>
<thead>
<tr>
<th>Test Number</th>
<th>Upstream Step Misalignment</th>
<th>Upstream Purge Blowing Ratio</th>
<th>Upstream Purge Velocity Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 (Design)</td>
<td>Baseline</td>
<td>2.5</td>
<td>1.3</td>
</tr>
<tr>
<td>2</td>
<td>Backward Facing</td>
<td>2.5</td>
<td>1.3</td>
</tr>
<tr>
<td>3</td>
<td>Forward Facing</td>
<td>2.5</td>
<td>1.3</td>
</tr>
<tr>
<td>4</td>
<td>Baseline</td>
<td>3.5</td>
<td>1.8</td>
</tr>
<tr>
<td>5</td>
<td>Backward Facing</td>
<td>3.5</td>
<td>1.8</td>
</tr>
<tr>
<td>6</td>
<td>Forward Facing</td>
<td>3.5</td>
<td>1.8</td>
</tr>
</tbody>
</table>

6.1. Effects of Upstream Misalignment at Design Blowing Ratio \((M = 2.5)\)

6.1.1. Discussion of Film Cooling Effectiveness

The film cooling effectiveness directly indicates the coolant coverage distribution, therefore showing how the different step misalignment affects the coolant performance. At the design blowing ratio of 2.5, the experimental film cooling effectiveness with three various upstream misalignments is shown in Fig. 3.6. Region A illustrates the coolant and mainstream
mixing immediately after the coolant ejects from the film cooling holes. For the baseline case, the coolant provides good coverage of $\eta \sim 0.5$ in this region. The wakes from the second row of cooling holes can be seen and seem evenly distributed, suggesting that the coolant is evenly attached to the curved endwall profile. Compared to the other two geometries, the backward-facing case's effectiveness is lesser by $\sim 0.2$, which suggests that the mixing between the coolant and mainstream vortices is unfavorable for the cooling. Besides, the wake from the second cooling rows is also weaker and harder to distinguish. The forward-facing step results in a similar cooling performance to the baseline case. However, the second row of cooling holes' wakes are sharper and leave larger uncooled areas between the holes. The pitch-wise uneven cooling could lead to extra thermal stress and potentially damage the endwall.

Figure 3.6. Experimental film cooling effectiveness for purge $M = 2.5$ with (a) no step, (b) a backward-facing step, and (c) a forward-facing step
Region B of Fig. 3.6 shows how the coolant performs in the passage where the mainstream accelerates to transonic condition and how well the coolant protects the rest of the passage from the mainstream thermal load. In Fig. 3.6(a), from \( x/C_{ax} = 0.2 \) to 0.9, the cooling effectiveness is around 0.3 and experiences a slow decay afterward. However, a clear boundary of the coolant coverage can be observed from \( x/C_{ax} = 0.75 \) near the pressure side to \( x/C_{ax} = 0.9 \) near the suction side. Across this boundary, the cooling effectiveness drops by nearly 0.1. Given that the largest pitch-wise pressure gradient emerges near the throat due to mainstream accelerating and turning, the location of the coolant boundary shown is highly likely to be an indicator of the momentum of the coolant which enters the passage. For the baseline case, the coolant has barely enough momentum to reach the throat but cannot reach the passage exit, especially on the pressure side. With a backward-facing step, the coolant coverage boundary moves further upstream and can be observed from \( x/C_{ax} = 0.2 \) near the pressure side to \( x/C_{ax} = 0.75 \) near the suction side. The lack of coolant leaves more than half of the passage unprotected from the mainstream thermal load and has a highly detrimental effect. On the contrary, the forward-facing step shows a slightly better cooling performance than the baseline case, which suggests that the forward-facing step helps the coolant preserve its momentum better. The cooling effectiveness increases from 0.05 to 0.1, reaching the maximum value near the passage exit.

Figure 3.7 illustrates the CFD streamlines and temperature contours at the plane at \( y/P = 0 \), which passes through the leading edge and the center of a cooling hole from the second row (See the dotted line in Fig. 3.6(a)). This figure allows us to investigate the difference in the flow patterns between the steps and the passage inlet for all three cases. For the baseline case, as the coolant is injected from the cooling holes, it experiences a slight separation indicated by the small high-temperature area downstream of the hole exit. Then the coolant is driven against the curved
endwall and immediately reattaches to the endwall about 1D downstream of the holes. After that, it suppresses the HSV induced by the vane leading edge and penetrates the HSV. Besides, a cavity vortex emerges downstream of the step, but it is so tiny and far from the cooling holes that it hardly influences the coolant behavior.

Figure 3.7. CFD streamline and temperature contour at y/P = 0 for purge M = 2.5, DR = 1.95 with (a) no step, (b) a backward-facing step, and (c) a forward-facing step. Note that there is no first-row cooling hole in this cross section. Thus, no coolant jet is observed from that row.

However, as shown in Fig. 3.7(b), the backward-facing step induces much larger cavity vortices. The primary vortex located on top of the secondary vortex reaches the second row of cooling holes. When the vortex flows clockwise, it forces coolant injected from the second row downwards, thus opposing the coolant’s momentum to travel downstream. Considering the vortex scale, it can be hypothesized that the coolant from the first row of cooling holes is subjected to even more suppression. Meanwhile, the larger HSV near the vane’s leading edge propels the coolant jet against the endwall as it gets close to the leading edge. Since the coolant is already weakened, the HSV created a small recirculation zone about 2D downstream of the cooling hole, right in front of the vane leading edge, as seen in Fig. 3.7(b). This additional recirculation causes the coolant to flow laterally from the recirculation core, leading to more aerodynamic losses. As a result, the wakes cannot be clearly seen in Fig. 3.7(b). There is also less coolant entering the
passage due to the vortex effect above, so the cooling performance in the vane passage, as shown in Fig. 3.6(b), is much worsened compared to the other two cases.

The coolant flow pattern induced by the forward-facing step is similar to the baseline case. The only apparent difference in the coolant behavior is that the low-cooling region immediately downstream of the second row of holes doubles in length compared to the baseline case. This lack of coolant coverage suggests a higher tendency of coolant to lift off after it is injected from the cooling holes if a forward step is present. This can be explained by the weaker cavity vortex and HSV. Since the forward-facing step results in a smoother continuous endwall profile, weakening the cavity vortex and HSV. Therefore, they exert less resistance to the coolant flow jets, which retain more momentum after the coolant penetrates the passage, explaining the high cooling performance in Region B of Fig. 3.6(c).

Mayo et al. [3.24] have investigated the flow patterns caused by the backward-facing step on the same endwall geometry without jet purge cooling, which is shown in Fig. 3.8. Compared with Fig. 3.7, the baseline case shows little difference. After the coolant is introduced, the cavity vortex scale stays almost the same, and the HSV is just marginally strengthened. On the contrary, the coolant significantly suppresses the HSV for the backward step case, shifting the vortex core from \( x/C_{ax} = -0.45 \) to much further downstream (\( -x/C_{ax} = 0.15 \)). Again, this change confirms the strong interaction between the coolant jets and the HSV. With the HSV retreating towards the leading edge, the primary cavity vortex, directly induced by the combustor-NGV step, grows substantially. Its scale in the axial direction almost triples, making it so big that it pushes the secondary cavity vortex to the bottom of the gap. As discussed above, the clockwise primary cavity vortex opposes the coolant flow near the cooling holes and causes more unfavorable cooling performance. Therefore, it can be concluded that a backward-facing step alters the interaction
between coolant jets and HSV and increases the scale of the clockwise primary cavity vortex. Both changes diminish the coolant momentum, leading to detrimental effects on the cooling performance.

![CFD streamline at y/P = 0 with no cooling with (a) no step and (b) a backward-facing step (Mayo et al. 2017)](image)

**Figure 3.8. CFD streamline at y/P = 0 with no cooling with (a) no step and (b) a backward-facing step (Mayo et al. 2017)**

### 6.1.2. Discussion of Nusselt Number Change Between with and without Coolant

To investigate further the different coolant flow patterns caused by step misalignments, the Nusselt number change from no-coolant cases to film-cooled cases is introduced as,

\[
\Delta Nu_{M=0\rightarrow 2.5} = Nu_{M=2.5} - Nu_{M=0}
\]  

This value is another important indicator of coolant coverage in addition to the cooling effectiveness. A positive \(\Delta Nu_{M=0\rightarrow 2.5}\) signifies a strengthened convection heat transfer scenario, which is attributed to a faster-moving flow over the surface. The study of \(Nu_{M=0}\) has been well documented by Mayo et al. [3.24].

The distribution of \(\Delta Nu_{M=0\rightarrow 2.5}\) for \(M = 2.5\) is shown in Fig. 3.9. Significant differences are observed from \(x/C_{ax} = -0.2\) to 0 across the three cases. For the backward step case, \(\Delta Nu_{M=0\rightarrow 2.5}\) reaches as high as 600 to 800 in between the coolant jets 1~2D downstream of the second row.
This increase confirms the presence of the recirculation and lateral coolant flow induced by the coolant-HSV mixing. On the contrary, in the forward step case, $\Delta Nu_{M=0.25}$ is minimal downstream of the second row of cooling holes, suggesting that most cooling jets penetrate through the mainstream and do not attach to the endwall immediately following injection.

![Figure 3.9. Experimental $\Delta Nu_{M=0.25}$ with (a) no step, (b) a backward-facing step, and (c) a forward-facing step](image)

Another noticeable difference is seen in the middle of the passage. In the backward step case, there is a narrow area near $x/C_{ax} = 0.4$ where $\Delta Nu_{M=0.25}$ suddenly rises over 600, while it is well below 300 in the rest of the passage, either upstream or downstream. However, in the other two cases, $\Delta Nu_{M=0.25}$ changes smoothly in the passage, showing no such sudden changes. To understand this phenomenon, the near-endwall total pressure contour maps from CFD at two transverse planes ($x/C_{ax} = 0.2$ and 0.4) are presented in Fig. 3.10.

In the backward step case, a loss core emerges close to the endwall near the suction side. The transverse location of the loss core matches the transverse location of the high $\Delta Nu_{M=0.25}$ area. This mid-passage loss core near the suction side has been observed in a typical turbine cascade without upstream cooling [3.25,26]. Due to the complexity of the secondary flows in a
turbine cascade, there is no definitive explanation for the loss core formation. However, it is commonly believed that the loss core forms due to the passage vortex. The passage vortex develops from the pressure side leg HSV forming and bifurcating near the vane leading edge. It grows in scale and gets swept towards the suction side by the cross-passage pressure gradient as it travels downstream. The loss caused by the passage vortex itself and the separation of the passage vortex contributes to the formation of the loss core near the suction side. Similar flow patterns were confirmed experimentally by Wang et al. [3.27]. Biesinger & Gregory-Smith [3.28] found that in a low-speed tunnel cascade with endwall upstream purge flow, a lower blowing flow rate creates a significant loss core near the suction side on a mid-passage transverse plane. The loss core is attributed to the intense secondary flow convecting the inlet boundary layers toward the suction side. As the blowing rates increase, the passage vortex is weakened until no apparent loss core can be observed, as seen in Figs. 3.10(a) and (c). Based on the studies above, it can be hypothesized that since the presence of an upstream backward-facing step reduces the momentum of the coolant entering the vane passage, it leads to a less energized boundary layer that increases the secondary flow caused by the passage vortex. The stronger secondary flow then pushes the boundary layer towards the suction side and results in more intense separation near the suction side. Near $x/C_{ax} = 0.4$, the strong pitchwise pressure gradient finally dominates the remaining coolant boundary layer and convects most of it towards the suction surface. The intense secondary flow caused is the reason for the high $\Delta Nu_{M=0->2.5}$ in the area near $x/C_{ax} = 0.4$ in Fig. 3.9(b). Meanwhile, since more coolant momentum reaches the cascade passage in the baseline case, the boundary layer is energized enough to overcome the passage vortex and remains relatively stable. Therefore, no high $\Delta Nu_{M=0->2.5}$ regions caused by the secondary flow can be observed in Fig. 3.9(a).
It should be noted that the endwall profile could also contribute to this difference seen in Fig. 3.10. Yan et al. [3.26] found that a non-axisymmetric endwall where the suction side is lower than the pressure side could strengthen the secondary flow, leading to a more intense loss core. Although the enwall in this study is axisymmetric, due to the mainstream's flow turning, the pressure side is higher than the suction side in the direction that is normal to the streamwise direction. As a result, the converging endwall could bring about a similar blockage effect and strengthen the secondary flow.
6.1.3. Discussion of Net Heat Flux Reduction for the Step Geometries

The non-dimensional value of NHFR evaluates the conjoint effect of $\eta$ and Nu. Fig. 3.11 depicts the $\text{NHFR}$ distribution for $M = 2.5$ with three geometries. The definition of $\text{NHFR}$ can be found in Eq. 3.9. The baseline case in Fig. 3.11(a) shows that coolant at design condition performs well throughout the passage, except near the pressure side at mid-passage, where the $\text{NHFR}$ is only about 0.4. However, the backward step in Fig. 3.11(b) has a conspicuously adverse effect compared to the baseline case. From the cooling holes to the passage exit, its $\text{NHFR}$ is 0.2 to 0.3 lower than the baseline. In other words, an additional 20% to 30% of the heat from the mainstream is absorbed by the endwall, which is highly unfavorable and should be avoided for durability and thermal failure concerns.

Figure 3.11. Experimental $\text{NHFR}$ for $M = 2.5$ with (a) no step, (b) a backward-facing step, and (c) a forward-facing step

Meanwhile, the forward step slightly enhances the cooling performance (Fig. 3.11(c)), particularly near the passage exit on the pressure side, where it shows an increase in $\text{NHFR}$ of 0.05 to 0.1. Nevertheless, the sharp pitch-wise gradient at $x/C_{ax} = -0.1$ in $\eta$ distribution persists in $\text{NHFR}$. 
This pattern could lead to uneven thermal load and should raise concern for potential thermal failure.

In a few words, at the design blowing ratio $M = 2.5$, the coolant momentum is well-suited to overcome the cavity vortex, HSV, and passage vortex. However, the misalignment, which varies the size and intensity of these vortices, significantly affects the cooling performance. Compared to the baseline case, the backward step misalignment induces a larger cavity vortex and HSV, which causes the coolant to lose momentum and reduce coolant coverage. This scenario should be avoided to prevent endwall burnout, especially near the throat and the exit of the cascade passage. On the contrary, the forward step shrinks the vortices, allowing more coolant to enter the passage and form slightly better coverage. However, it is seen to cause pitch-wise uneven cooling near the cooling holes due to more intense coolant lift-off and can lead to unfavorable effects.

### 6.2. Effects of Upstream Misalignment at High Blowing Ratio ($M = 3.5$)

Figure 3.12 shows the film cooling effectiveness distribution for a higher blowing ratio $M = 3.5$ with the three step misalignments. The baseline case yields the best overall cooling performance, but the difference between the three cases is less than 0.1, not as substantial as those at $M = 2.5$. The favorable coolant coverage suggests sufficient coolant entering the passage regardless of the upstream misalignments. The additional coolant momentum ensures that the coolant is able to penetrate the cavity vortex and HSV. Afterward, the coolant reaching the passage energizes the boundary layer to counter the sweeping passage vortex so that a stable boundary layer is formed. Although the cavity vortex and HSV scale are still different due to step heights, it no longer significantly affects the coolant behavior. This phenomenon indicates that once the
coolant can overcome the cavity vortex and HSV, the momentum loss in penetrating these vortices induced by different step heights no longer has a first-order effect.

![Figure 3.12. Experimental film cooling effectiveness for purge \( M = 3.5 \) with (a) no step, (b) a backward-facing step, and (c) a forward-facing step](image)

Between the second row of cooling holes and the passage inlet, an area of very low cooling effectiveness is observed in all cases. Based on the discussion above, the coolant lift-off is likely the reason. In addition, the higher momentum ratio causes the coolant jet to be stronger when it enters the mainstream. Thus, it takes longer for the coolant to be reattached to the endwall, leaving the endwall unprotected from the heated mainstream before the coolant reattaches and spread laterally.

In summary, due to higher momentum, the coolant in all geometries experiences minimal effects from the cavity vortex and the HSV, proving that the impact of upstream misalignment is much diminished at the higher coolant blowing rate.
6.3. General Discussion

Previous research has shown discrepancies in the effect of upstream step misalignment. For instance, Cardwell et al. [3.11] found that at low $Ma_{exit} = 0.085$ in their cascade setting, the backward step misalignment is the most favorable case enhancing adiabatic film cooling effectiveness. Contrarily, this study with $Ma_{exit} = 0.85$ demonstrated that the backward step misalignment reduces the film cooling effectiveness. This discrepancy is particularly noticeable from the mid-passage to the passage exit, where the flow is influenced by compressibility. Therefore, the difference in mainstream speed and compressibility can lead to different interactions between the coolant flow and mainstream, which may not be observed during low-speed experimentation.

It is also observed that the blowing ratio makes a drastic difference in how the step misalignment affects the coolant, which has not been seen in the studies mentioned earlier. At a low blowing ratio, the cavity vortex and HSV can trap the coolant and effectively prevent it from entering the passage. In such a case, the backward step should be avoided to suppress the cavity vortex and HSV. However, once the blowing ratio is high enough to enable the coolant to overcome the cavity vortex and HSV, the step height has minimal cooling performance effects. Meanwhile, the intense lift-off immediately after the coolant injection will cause an unprotected area on the endwall platform and should be one of the primary concerns.

7. SUMMARY AND CONCLUSIONS

This study analyzes the effect of upstream misalignment on a steep injection purge hole cooling scheme at engine representative mainstream and coolant conditions. The cascade tunnel experimental result is presented and discussed with the support of a numerical investigation to
demonstrate the flow patterns and coolant-mainstream interaction. The key findings of this study include,

(1) The combustor-NGV misalignment affects the film cooling performance in NGV by influencing the interaction between the coolant jets and the cavity vortex and HSV, thus changing the momentum of the coolant that reaches the NGV passage. The difference in the coolant momentum leads to different interactions between the coolant boundary layer and the sweeping passage vortex, resulting in separate cooling coverage in the passage.

(2) At the design blowing ratio of $M = 2.5$, the backward-facing step of 4.9% span length has a significantly adverse effect on the cooling performance. The misalignment causes the coolant to lose substantial momentum in its interaction with the cavity vortex and HSV. The low momentum coolant film in the passage leads to strong secondary flow caused by the passage vortex, thus exposing the PS and near-throat endwall. Therefore, the backward-facing misalignment should be avoided to prevent local burnout on the turbine endwall.

(3) At the design blowing ratio of $M = 2.5$, the forward-facing step of 4.9% span length slightly improves the overall cooling performance. However, the strong life-offs caused by the step height lead to uneven cooling and less coolant coverage near the cooling holes.

(4) At a blowing ratio of $M = 3.5$, the additional momentum helps the coolant to overcome the HSV and the cavity vortex with all 3 step heights. As a result, step misalignment has a reduced effect on cooling performance. Meanwhile, all 3 cases experience severe jet lift-off, leading to undesirable cooling coverage between the cooling holes and the passage inlet.

(5) The effect of combustor-NGV misalignment is important to produce accurate engine-representative data. Therefore, during wind tunnel testing of NGV film cooling, the combustor-NGV misalignment should be mimicked in conjoint with the engine-representative density ratio.
8. NOMENCLATURE

\( A \) Area, mm\(^2\)
\( C \) Vane Chord, mm
\( c_p \) Specific Heat at Constant Pressure, J/(kg·K)
\( HTC \) Heat Transfer Coefficient, W/(m·K)
\( k \) Thermal Conductivity, J/(m·s·K)
\( L \) Cooling Hole Length, mm
\( P \) Cooling Hole Pitch Length, mm
\( q'' \) Heat Flux, W/m\(^2\)
\( T \) Temperature, K
\( t \) Time, s
\( V \) Velocity, m/s
\( x, y \) Coordinate, mm

Greek Letters

\( \Delta \) Difference
\( \eta \) Adiabatic Film Cooling Effectiveness
\( \rho \) Density
\( \varphi \) Overall Cooling Effectiveness

Non-dimensional Numbers

\( DR \) Density Ratio, \( \frac{\rho_a}{\rho_\infty} \)
\( M \) Blowing Ratio, \( \frac{\rho_c V_c}{\rho_a V_\infty} \)
\( Ma \) Mach Number
\( MFR \) Mass Flow Ratio, \( \frac{\rho_c V_c A_a}{\rho_a V_\infty A_\infty} \)
\( NHFR \) Net Heat Flux Reduction, \( 1 - \frac{HTC_f}{HTC_o} (1 - \frac{\eta}{\varphi}) \)
\( Nu \) Nusselt Number
\( Re \) Reynolds Number
\( Tu \) Turbulence Intensity

Subscripts

\( ax \) Axial
\( aw \) Adiabatic Wall
\( c \) Coolant Flow
\( exit \) Vane Passage Exit
f  Film Cooled Case
o  Uncooled Case
r  Recovery
w  Endwall
∞  Freestream Flow

Abbreviation

HSV  Horseshoe Vortex
NGV  Nozzle Guide Vane
PS  Pressure Side
SS  Suction Side

9. ACKNOWLEDGEMENTS

The authors would like to acknowledge and thank Solar Turbines Incorporated for the funding, support, and guidance on the work presented herein.

REFERENCES


CHAPTER 4
THE COOLING EFFECT OF COMBUSTOR EXIT LOUVER SCHEME ON NOZZLE GUIDE VANE ENDWALL

Shuo Mao*, Kaiyuan Zhang, Daniel Van Hout, Wing F. Ng  
Virginia Tech  
Blacksburg, VA, United States

Hongzhou Xu, Michael Fox  
Solar Turbines Incorporated  
San Diego, CA, United States

ABSTRACT

The ever-increasing combustor exit temperature in modern turbine engine designs raises challenges for the nozzle guide vane cooling. Due to the challenges of NGV cooling design, the cooling effect from the combustor cooling features can prove valuable. This study investigates, experimentally and numerically, the cooling effect of a louver cooling scheme near the combustor exit on the NGV endwall. The wind tunnel testing and CFD simulation are carried out with engine-representative conditions of an exit Mach number of 0.85, an exit Reynolds number of $1.5 \times 10^6$, an inlet turbulence intensity of 16%, and a density ratio of 2.1. Various coolant mass flow ratios from 1% to 4% are tested to demonstrate the effect of the coolant rate.

For the geometry studied, the results found a critical mass flow ratio between 1%~2%. The coolant forms a uniform film only by exceeding this value to provide good coverage upstream of the NGV passage inlet. As for the cooling of the NGV passage, the mass flow ratio of the range investigated is not sufficient for desirable cooling performance. The pressure side endwall proves most difficult for the coolant to reach. In addition, the fishmouth cavity at the combustor-NGV passage causes a three-dimensional cavity vortex that transports the coolant in the pitch-wise direction. The coolant transport pattern is dependent on the coolant mass flow ratio. Based on the results, the authors propose combining this louver scheme with the upstream jump cooling scheme for a desirable NGV cooling system.
1. INTRODUCTION

Modern gas turbines run at extremely high turbine inlet temperatures beyond the current metal limits [4.1], causing risks of thermal failure at the nozzle guide vane (NGV). To ensure the smooth operation of the turbine engine, researchers have developed various cooling schemes to protect the NGV endwall. Efforts by Burd et al. [4.2], Oke and Simon [4.3], and other similar studies investigated the effect of an upstream slot bleeding system. Their results indicated that the slot geometry favors the cooling performance immediately downstream of the cooling exit and on the pressure side of the NGV passage. Later, Bogard and Thole [4.4] and Bunker [4.5] employed cooling schemes consisting of rows of discrete holes upstream of the NGV. The discrete hole geometry is less effective near the coolant exit than slot cooling but has significantly more impact on the NGV passage. Mao et al. [4.6] and El-Gabry et al. [4.7] studied discrete cooling holes with high injection angles designed for contoured axisymmetric endwall profiles. They achieved a cooling film surviving longer in the NGV passage than conventional film cooling holes. To improve the aforementioned cooling schemes to cool the NGV passage near the vanes, Thrift et al. [4.8] and Chowdhury et al. [4.9] implemented arrays of smaller cooling holes in the regions where the coolant coverage was not desirable. The additional cooling holes enhanced the cooling performance as expected, but the researchers also found it challenging to maintain the blowing ratio at mid-passage. Collectively, by introducing discharge air to the NGV, these designs affect the near-wall heat transfer and aerodynamics, shielding the endwall from the hot mainstream gases. However, the complicated geometries inevitably add complexity to the already sophisticated NGV assembly, bringing about new structural integrity and manufacturing challenges.

Numerous studies have investigated the NGV aerodynamics and revealed the dominating flow patterns affecting the cooling film [4.10-13]. One of the most influential patterns is the horseshoe vortex, which forms and bifurcates into two legs due to the presence of the vane leading
edge. The horseshoe vortex usually creates a stagnation zone upstream of the vane leading edge, capable of drawing the coolant away from the endwall surface. Due to the turning of the NGV vanes, the pressure side leg of the horseshoe vortex flows towards the suction side and develops into the strong passage vortex. According to Biesinger and Gregory-Smith [4.14], if the upstream coolant boundary layer flow is not sufficiently energized, the passage vortex can cause flow separations near the suction side, thus, severely reducing the cooling performance. In short, the intensive vortices in the NGV passage lead to enormous difficulties for the coolant to form and maintain a stable film. As a result, most studies mentioned above adopted a cooling scheme upstream of the passage inlet to develop the cooling film. However, very few studies have focused on the effect of the combustor cooling scheme on the NGV passage.

Among the few pieces of literature that studied the combustor cooling system’s cooling effects on the NGV section, Colban et al. [4.15] placed a combustor cooling system consisting of effusion film cooling, dilution jet cooling, and louver cooling 0.1 true chord length upstream of the NGV passage and tested its performance. They managed to cool the inlet region of the passage, but the cooling effectiveness quickly decayed from 0.8 to 0.4 at mid-passage. They also found that increasing the louver cooling flow over the investigated range can increase cooling effectiveness. The experiment of Holgate et al. [4.16] suggested that adding upstream dilution jet cooling to NGV film cooling schemes caused favorable film mixing, enhancing the performance in the stagnation region and reducing the amount of NGV coolant required. The two studies above demonstrated the potential of the combustor cooling features to reduce the amount of coolant needed for the NGV cooling schemes.

The most commonly used cooling schemes in the combustor chamber are effusion film cooling, dilution jet cooling, and louver slot cooling. Using film cooling hole array, Ling et al.
achieved full coverage at 80 diameters downstream of the first row, but the film cooling effectiveness obtained was only 0.35. Other studies showed that adding several big dilution holes to the arrays of small film cooling holes can significantly improve the cooling performance [4.18,19]. Those studies reported that coolant from the dilution holes helps spread the film cooling jets, thus forming a more uniform coolant film. Scritto et al. [4.19] observed averaged film cooling effectiveness of around 0.8 for the fully developed coolant film. However, a high blowing ratio for both the film cooling holes and the dilution holes can cause the jets to penetrate the coolant film and cause adverse effects. The cooling effects of effusion film cooling, dilution jet cooling, and louver slot cooling combined were studied by Ceccherini et al. [4.20] and Facchini et al. [4.21]. Their results showed that the louver cooling creates a stable cooling film with the most favorable interaction between the coolant and the mainstream flow. The coolant film also mitigates other cooling jets’ tendency to lift off and protects the endwall before the other jets are fully developed into a uniform film. Since the discussions above indicate that a stable cooling film is critical to the NGV cooling, the louver cooling scheme in the combustor is most befitting in an attempt to cool the NGV passage.

In a typical louver cooling scheme, the coolant is injected onto an inner wall, then flows through a bend and merges with the mainstream at a slight angle. Its cooling performance has been studied in several pieces of literature compared to conventional circular hole film cooling. In a CFD study of flat surface cooling, Immarigeon and Hassan [4.22] illustrated that the louver cooling scheme can prevent the jets from lifting off at high blowing ratios and provide higher cooling effectiveness, confirmed by Ghorab [4.23] in his experiments. Elnady et al. [4.24] found that the louver cooling scheme applied on the vane pressure side can cause more lateral spreading of the coolant. However, they also found out that the second row of the louver is not as effective as the
first row because of the boundary layer thickness. Collectively, these studies showed the louver scheme’s strength in suppressing lift-off and creating a more uniform coolant film.

In conclusion, the capability of developing a stable cooling film by the louver schemes makes it most suitable to deal with the complicated mainstream flow in the NGV passage and cool the endwall. However, the cooling effect of a combustor louver scheme on the NGV passage has not been studied. Considering the proximity of these two components, the authors proposed implementing a single-row louver cooling scheme near the combustor exit to cool the NGV passage. With experiments and numerical studies using various coolant mass flow rates, the authors investigated the effect of this cooling scheme and revealed the flow interactions. The findings presented in this paper served to help understand the capability of the combustor louver cooling scheme and provide references for engine designers.

2. EXPERIMENTAL METHODOLOGY

2.1. Experimental Test Facility

The experiments were conducted at the Virginia Tech Transonic Wind Tunnel, a cascade wind tunnel established for fundamental aerodynamic and heat transfer measurements. Supplied by two Boge industrial air compressors and a 18900-liter buffer tank, the wind tunnel is capable of maintaining a 30-second blowdown at the maximum flow rate of 4.5kg/s. In this study, the flow rate was held at 3.0kg/s and resulted in the mainstream condition of \( Ma_{in} = 0.1 \), \( Ma_{exit} = 0.85 \), and \( Re_{exit, C} = 1.5 \times 10^6 \).

As shown in Fig. 4.1, an in-line heat exchanger consisting of parallel copper pipes was installed upstream of the test section. Before the blowdown, Valves 1 and 2 were set to create a
recirculation driven by the fan. During this process, the heat produced by two 36kW heaters was transferred to the heat exchanger until it reached the desired temperature. Then Valves 1 & 2 were reset to allow the mainstream air to go through the heat exchanger to the test section. A turbulence grid was installed upstream of the test section to mimic the combustor’s highly turbulent flow. In this study, the turbulence intensity at the NGV inlet was $T_{u_{in}} = 16\%$, as documented by Nix et al. [4.25]. In the coolant supply system, air and SF$_6$ were mixed to match the realistic density ratio, which had been proved to significantly affect the NGV cooling experiments by Mao et al. [4.6]. Then the mixed gases were diverted into a cooling loop and controlled by a solenoid valve. Upon the mainstream reaching the test section inlet, the solenoid valve opened to allow the coolant to enter the test section with a delay under 0.2s.

![Diagram of Virginia Tech transonic wind tunnel facility]

**Figure 4.1. Virginia Tech transonic wind tunnel facility**

The test section consisted of four full vanes and two half vanes at the top and bottom, creating five identical NGV passages. The heat transfer of the center passage endwall was measured with the other four passages to establish periodicity. Table 4.1 tabulates the geometry details of the vanes. The endwall geometry is shown in Fig. 4.2. The vanes were positioned on an
engine-representative axisymmetric contoured endwall with a fishmouth cavity representing the mateface between the combustor exit and the turbine inlet. Further upstream is the louver cooling scheme, where the coolant impinged through small circular holes onto a 20-degree plate before it went through a narrow passage to enter the mainstream. The endwall and vanes were 3D printed by a Stratasys Fortus 250mc printer using ABSPlus-P430, guaranteeing a low conductivity $k = 0.188\text{W/m-K}$, which is critical to the data reduction technique discussed below.

Table 4.1. Vane Geometry

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Passages</td>
<td>5</td>
</tr>
<tr>
<td>Span, $S$</td>
<td>138.4 mm</td>
</tr>
<tr>
<td>Axial Chord, $C_{ax}$</td>
<td>50 mm</td>
</tr>
<tr>
<td>True Chord, $C$</td>
<td>91.2 mm</td>
</tr>
<tr>
<td>Vane Pitch, $P$</td>
<td>83.1 mm</td>
</tr>
<tr>
<td>Inlet and Exit Angle</td>
<td>0° and 73.5°</td>
</tr>
</tbody>
</table>

Figure 4.2. (a) The top view of the test piece center passage; (b) The side view of the test piece showing the internal geometry of the louver cooling scheme; (c) A close-up view of the louver cooling scheme.
A Pitot probe and a total temperature probe were located upstream of the passage inlet to monitor the mainstream condition and calculate the inlet Mach number. In addition, two sets of pressure ports were installed at $2.2C_{ax}$ upstream of the leading edge and $0.4C_{ax}$ downstream of the trailing edge, respectively, to calculate the exit Mach number. Thermocouples, pressure transducers, and orifice meters were installed in the coolant supply loop to measure the coolant temperature, pressure, and flow rate. All the measurements were obtained using a NI compactDAQ (30Hz) and a NetScanner Model 98RK pressure scanner (6Hz).

The endwall surface temperature history of the blowdown was recorded using a FLIR A325sc model infrared (IR) camera at 30Hz. For the optimal performance of the IR camera, a Germanium (Ge) optical window with a broadband anti-reflective (BBAR) coating was placed between the camera and endwall, with an average transmission of 95% in the wavelengths of interest between 6 to 13μm. Additionally, the test piece was painted with ultra-black paint to ensure an emissivity of 0.97. Before the tests, the camera was calibrated under test conditions, which was later used in data reduction to mitigate the uncertainty.

For the coolant setting, three non-dimensional parameters were matched in this study with the engine condition: blowing ratio ($M$), density ratio ($DR$), and mass flow rate ratio ($MFR$), defined as Eqs. 4.1-3.

\[
M = \frac{\rho_c V_c}{\rho_{\infty} V_{\infty}} \quad (4.1)
\]

\[
DR = \frac{\rho_c}{\rho_{\infty}} \quad (4.2)
\]

\[
MFR = \frac{\rho_c V_c A_c}{\rho_{\infty} V_{\infty} A_{\infty}} \quad (4.3)
\]
As shown in Table 4.2, three test cases were carried out in the experiments. The coolant properties were evaluated at the louver cooling hole to be consistent with the literature.

**Table 4.2. The test conditions of the experiments**

<table>
<thead>
<tr>
<th>Test Number</th>
<th>MFR</th>
<th>DR</th>
<th>M</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.0</td>
<td>2.1</td>
<td>2.4</td>
</tr>
<tr>
<td>2 (Baseline)</td>
<td>2.0</td>
<td>2.1</td>
<td>4.7</td>
</tr>
<tr>
<td>3</td>
<td>3.0</td>
<td>2.1</td>
<td>7.1</td>
</tr>
</tbody>
</table>

### 2.2. Experimental Data Reduction Techniques

With the endwall surface temperature history obtained by the IR camera, the heat flux into the endwall was calculated using Cook’s method [4.26] as below,

\[
q''(t_m) = \frac{2(\sqrt{k\rho c_p})}{\sqrt{\pi}} \sum_{t=1}^{m} \frac{T(0,t_i) - T(0,t_{i-1})}{\sqrt{t_i} + \sqrt{t_m - t_i}}
\]

\[ (4.4) \]

It should be noticed that this method is based on a one-dimensional (span-wise direction) semi-infinite conduction model with a uniform initial temperature distribution. These requirements are met by the low conductivity of ABS material and letting the test section cool to room temperature before each tunnel run. The validation of this assumption is detailed in Appendix C.

The endwall heat flux was then used in the Dual Linear Regression Technique (DLRT) developed by Xue et al. [4.27] to simultaneously calculate the recovery temperature, \( T_r \), heat transfer coefficient, \( HTC \), and adiabatic film cooling effectiveness, \( \eta \). This method rewrites Newton’s Law of Cooling in a linearized form as Eq. 4.5 using the definition of \( HTC \) and \( \eta \) (Eqs. 4.6 and 4.7).

\[
\frac{q''}{T_r - T_c} = HTC \left(\frac{T_r - T_w}{T_r - T_c}\right) - HTC \times \eta
\]

\[ (4.5) \]
\[ HTC = \frac{q^n}{T_{av} - T_w} \]  \hspace{1cm} (4.6)

\[ \eta = \frac{T_r - T_{av}}{T_r - T_c} \]  \hspace{1cm} (4.7)

For each pixel in the infrared camera image, the algorithm optimized the linear regression in Eq. 4.5 by iterating the recovery coefficient \( C_r \), which is defined by

\[ C_r = \frac{T_r}{T_w} \]  \hspace{1cm} (4.8)

The result with the highest \( R^2 \) value was substituted into Eqs. 4.5-8 to calculate \( HTC \) and \( \eta \) for each pixel. In order to increase the signal-to-noise ratio in the linear regression process, two blowdowns were carried out for each test case, one with chilled coolant and the other with room-temperature coolant.

To further evaluate the cooling effect, the Nusselt number and net heat flux reduction (NHFR) were also calculated in Eqs. 4.9 and 4.10. The overall effectiveness \( \phi \) in Eq. 4.10 is approximated to be 0.6 based on the study by Mick and Mayle \[4.28\].

\[ Nu = \frac{HTC \times C}{k_{air}} \]  \hspace{1cm} (4.9)

\[ NHFR = 1 - \frac{HTC_f}{HTC_o} \left( 1 - \frac{\eta}{\phi} \right) \]  \hspace{1cm} (4.10)

The uncertainty in this study is mainly attributed to the IR camera and thermocouples. The FLIR A325sc is documented to have an uncertainty of ±2.0% or ±2.0℃, whichever is greater, while the uncertainty of the T-type thermocouples used is reported to be ±0.75% or ±1.0℃, whichever is greater. Using Moffat’s perturbation method \[4.29\], the uncertainty calculated with a 95% confidence interval is ±9.6% in \( HTC \) and ±0.1 in \( \eta \). The detailed calculation is documented in Appendix B.
3. COMPUTATIONAL SETUP

The CFD study was conducted using ANSYS CFX 2020 based on the steady-state Reynolds-averaged Navier Stokes (RANS) equations. The $k-\omega$ Shear Stress Transport (SST) model was selected as the turbulence model to simulate better flow within the fishmouth cavity, which will be discussed later in the CFD verification section. Meanwhile, a high-speed (compressible) wall heat transfer model was applied to deal with the potential compressibility effect in the NGV passage. The mainstream uses air as the working fluid, while the coolant is a mixture of gas defined using experimental data. Both gases follow the ideal gas law.

Figure 4.3 shows the computational model generated in ICEM. The model consists of one NGV vane and two half passages with periodic boundary conditions. The mesh contained 13.4 million nodes, 0.8 million of which were distributed for the louver cooling geometry. The multiblock structured mesh was used for most of the surface, while unstructured mesh was used only when the geometry became too complex, as shown in Fig. 4.3(b). All the near-wall mesh achieves $y+ < 0.8$ to capture the heat transfer and aerodynamics within the boundary layer. The grid independence was concluded using the mean temperature of the endwall surface, where the results with finer meshes showed less than 1% change. The convergence criteria were set to $1.0 \times 10^{-5}$ for several parameters.
The mainstream inlet boundary condition was defined by the total pressure, total temperature, turbulence intensity, and eddy length scale previously measured from experiments [4.30]. The coolant inlet was specified to match the flow conditions in Table 4.3. The experimental result found that $MFR = 3\%$ coolant does not yield desirable cooling performance. Meanwhile, since $MFR = 4\%$ is beyond the capability of the experimental setup, the team studied this case in CFD (Test #2). At the passage outlet, the pressure was set to ambient pressure. Two periodic boundary conditions were imposed to ensure periodicity, while all the other walls were specified as no-slip walls.
Three simulations with different thermal boundary conditions for each test case are carried out to acquire the $HTC$ and $\eta$ defined by Eqs. 4.6 and 4.7. In the first simulation, all the solid walls are adiabatic with no coolant ($MFR = 0$) to obtain the approximate $T_r$. The second simulation keeps the adiabatic boundary conditions but introduces the coolant to yield near-wall temperature $T_{aw}$ and wall temperature $T_w$. Finally, the last simulation changes the boundary condition to $T_w = 300K$ and keeps the coolant to acquire the endwall heat flux $q''$. Using these values in Eqs. 4.6 and 4.7, $HTC$ and $\eta$ can be calculated from the CFD results. Further details of this methodology were documented by Mao et al. [4.6].

4. RESULTS AND DISCUSSION

This section presents and discusses the results of the experiments and simulations. First, the CFD results were compared with the experiment results to verify the credibility of the CFD model. Then the experiment results were shown to demonstrate the cooling performance and the effect of coolant flow rate. The CFD result was later presented to provide insights into the behavior of the louver coolant. In the end, the cooling capability of the louver scheme was discussed and compared with other cooling schemes.

4.1. CFD Validation

The behaviors of two turbulence models, k-omega SST and Omega-based Reynolds Stress were shown in Fig. 4.4, together with the experiment results. Despite the difference in the

<table>
<thead>
<tr>
<th>Test Number</th>
<th>$MFR$</th>
<th>$DR$</th>
<th>$M$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.0</td>
<td>2.1</td>
<td>4.7</td>
</tr>
<tr>
<td>2</td>
<td>4.0</td>
<td>2.1</td>
<td>9.5</td>
</tr>
</tbody>
</table>
magnitude of \( \eta \) between the experiments and simulations, the SST model managed to capture the
trend of the \( \eta \) distribution. In Figs. 4.4(a) and (b), coolant accumulation was observed near the
stagnation region upstream of the vane leading edge (black dashed boxes), indicating that the
fishmouth cavity transports the coolant in the pitch-wise direction, which will be discussed in the
later session. In addition, the decay of cooling effectiveness at mid-passage in these two cases also
follows a similar trend (green dashed curves). These agreements suggested that the k-omega SST
model successfully simulated the interaction between the coolant film and the fishmouth cavity
and the coolant film behavior under the passage cross flow.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{adiabatic_film_cooling_effectiveness.png}
\caption{The adiabatic film cooling effectiveness distribution for MFR = 2\% from (a) experiments, (b) CFD using SST turbulence model, and (c) CFD using Omega-based Reynolds Stress model. The experimental data is omitted near the fishmouth gap around \( x/C_{ax} = 0.5 \) because the endwall angle affects the accuracy of IR camera recording.}
\end{figure}
On the contrary, as shown in Fig. 4.4(c), the Omega-based Reynolds Stress model caused more discrepancies in $\eta$ and did not show the pitchwise difference of coolant coverage near the fishmouth gap. Therefore, the authors are confident that the simulation with the SST model is qualitatively accurate to shed light on the flow physics. A more detailed discussion of the turbulence model can be found in Appendix D.

4.2. Experimental Results of the Endwall Heat Transfer

The adiabatic film cooling effectiveness $\eta$, as defined in Eq. 4.7, showed the coolant coverage by evaluating the endwall temperature reduction due to the presence of coolant film. Figure 4.5 presents the pitch-wise average $\eta$ from the contoured endwall upstream of the NGV passage to the passage outlet. For $MFR = 2\%$ and $3\%$, the cooling effect is very similar between the fishmouth gap and the passage inlet. As the coolant film moved in the NGV passage, both cases showed a decay of $\eta$ in the streamwise direction. $MFR = 2\%$ showed more rapid decay between the two cases with nearly no cooling left at the passage outlet.

**Pitch-wise Average Film Cooling Effectiveness from Experiments**

![Pitch-wise Average Film Cooling Effectiveness](image)

*Figure 4.5. The pitch-wise average film cooling effectiveness of three experiment cases. The NGV vane spans from $x/C_{ax} = 0$ to 1. The data downstream of $x/C_{ax} = 0$ is omitted due to the low signal-to-noise ratio.*
Conversely, the louver scheme with $MFR = 1\%$ could not form a stable coolant film upstream of the NGV passage, showing a noise-like trend in Fig. 4.5. In addition, the endwall temperature recorded by the IR camera frequently oscillated, indicating that the coolant momentum was insufficient to maintain a uniform coolant film downstream of the fishmouth gap. The unstable flow physics contributes to the high noise-to-signal ratio, especially in the NGV passage where the DLRT method cannot produce reliable data.

At $-0.4 < x/C_{ax} < 0$, the cooling effectiveness for $MFR = 1\%$ is much lower than the $MFR = 2\%$ and $3\%$. This variation can be explained by the weaker coolant film that allowed more mainstream hot air to approach the endwall as the film traveled over the fishmouth gap. Ghorab [23] showed very similar findings in his study of the louver cooling scheme over a flat plate. In his study, $M = 1.0$ and $1.5$ both resulted in a desirable cooling effect, while $M = 0.5$ led to a reduction of $\eta$ nearly by half. Therefore, it can be inferred that a critical $MFR$ exists for the louver cooling scheme. Only by exceeding this critical value can the louver scheme form a well-developed uniform cooling film. Otherwise, the coolant will dissipate quickly before the coolant can develop and form a film. According to Fig. 4.5, the critical $MFR$ is between 1\% and 2\% in this study.

The film cooling effectiveness distributions for Tests 2 and 3 are presented in Fig. 4.6 with a jump cooling experiment result by Mao et al. [4.6] for comparison. In Fig. 4.6(a), at $-0.4 < x/C_{ax} < -0.2$, higher $\eta$ was observed upstream in the stagnation region upstream of the leading edge ($y/P = 0$ and $y/P = 1$). Further downstream, the horseshoe vortex dominated the coolant film and prevented it from approaching the leading edge. In Fig. 4.6(b), the higher coolant flow rate did not help the coolant film to penetrate the horseshoe vortex. Instead, the stagnation region was less
protected since the coolant was diverted away from the leading edge. This phenomenon was investigated in the CFD result section below.

![Experimental Adiabatic Film Cooling Effectiveness](image)

**Figure 4.6.** The film cooling effectiveness distribution from experiments of (a) louver scheme $MFR = 2\%$, (b) louver scheme $MFR = 3\%$, and (c) upstream jump cooling scheme for $MFR = 1.9\%$ by Mao et al. [4.6].

In the NGV passage ($0 < x/C_{ax} < 1$), a clear boundary of the coolant coverage (dashed curve) could be seen in Fig. 4.6(a). The coolant was swept towards the suction side by the passage cross flow and left the pressure side region of the endwall surface unprotected. In addition, the coolant was completely dominated by the cross flow at $x/C_{ax} = 0.9$, where the coolant could no longer attach to the endwall. Figure 4.6(b) showed an improved cooling effect in the passage for $MFR = 3\%$, especially near the pressure side. A streak of higher $\eta$ at $y/P = 0.4$ near the passage inlet, which was likely due to the pitch-wise transport of coolant by the fishmouth cavity upstream, indicated a more energized coolant boundary flow. Compared to the louver scheme at $MFR = 2\%$, this film persisted further into the NGV passage, leading to better coolant coverage downstream.
However, compared with the jump cooling scheme with coolant $MFR = 1.9\%$ (Fig. 4.6(c)), both louver cooling schemes could barely achieve half of the cooling effectiveness in the NGV passage. Although the louver scheme design avoided the issue of lift-off in the jump cooling design, the distance between the louver exit and the NGV passage caused so much dissipation that the louver scheme could hardly form a lasting coolant film in the passage.

The endwall Nusselt number evaluates the heat transfer rate between the endwall and the hot mainstream gas, so it is also commonly used to assess the performance of the cooling scheme. Figure 4.7 shows the Nu distribution of the louver scheme with coolant $MFR = 2\%$ and $3\%$ compared to a non-cooled endwall. Near $x/C_{ax} = -0.35$, a streak of high $Nu$ could be seen in the no cooling case (dashed circle), which is likely due to the endwall contour. The high $Nu$ in this region was mitigated by the louver coolant at $MFR = 2\%$ and almost entirely negated by $MFR = 3\%$ coolant. This reduction in $Nu$ demonstrates the cooling effect upstream of the passage. Combined with NGV cooling schemes, the louver scheme can protect the turbine endwall before the NGV coolant jet is fully developed into a cooling film.

In Fig. 4.7(c), the louver scheme decreased the Nu in the stagnation region upstream of the vane leading edge. It can be inferred that the interaction between the louver coolant film and the horseshoe vortex slows down the backflow near the endwall, suppressing the convective heat transfer. Meanwhile, no significant change of the Nu in the NGV passage was caused by the louver coolant, which suggested that the near-wall flow pattern in the passage experienced few changes.
Figure 4.7. The Nusselt Number distribution from experiments of (a) louver scheme without cooling, (b) louver scheme $MFR = 2\%$, (c) louver scheme $MFR = 3\%$.

Net heat flux reduction, as defined in Eq. 4.10, evaluates the heat flux reduction due to the introduction of coolant. As shown in Fig. 4.8, the louver cooling schemes with $MFR = 2\%$ and $3\%$ aid in cooling the endwall before the passage inlet. However, the $MFR = 2\%$ case hardly provided any heat flux reduction on the pressure side passage. Increasing the coolant rate to $3\%$ improved the NHFR by about 0.2 in the passage but was still insufficient to cool the NGV. Still, it should be noticed that the $MFR = 3\%$ scheme provided moderate heat flux reduction in the stagnation region near the leading edge despite the low cooling effectiveness.
4.3. Simulation Results of the Flow Physics

To demonstrate the coolant behavior, planes at different pitch-wise locations are selected, as shown in Fig. 4.9, to the flow patterns and temperature fields in Fig. 4.10. The most outstanding features observed are the horseshoe vortex and the fishmouth cavity vortex. The horseshoe vortex formed upstream of the leading edge. It tended to lift the coolant off the endwall surface, thus causing it more challenging to cool the region around the leading edge and reducing the amount of coolant entering the NGV passage.
Figure 4.9. The four planes selected to show the temperature distribution and velocity direction in Fig. 4.10.

The cavity vortex forming in the fishmouth cavity is a 3D vortex because there has to be mass transfer in the pitch-wise direction to satisfy the conservation of mass. Therefore, the mainstream pressure difference in the pitch-wise direction caused the ingress and egress pattern, which drove the cavity vortex. When ingress occurs, as in Fig. 4.10(a), the mainstream gases flow around the overhang extending from the louver exit, then towards the contoured endwall and enter the fishmouth cavity. During this process, the hot mainstream air is drawn towards the endwall, as shown in Figs. 4.10(a) and 4.10(c), where a slightly hotter region can be observed below the overhang near the contoured endwall. Moreover, this mainstream intake disrupted the cooling film’s development, making the coolant film easier to be dominated by the horseshoe vortex. On the contrary, the egress process aided in developing the coolant film downstream of the gap with additional coolant momentum. Therefore, the coolant film can better suppress the horseshoe vortex and better attaches to the endwall, as shown in Fig. 4.10(d).
Figure 4.10. The temperature contours and the velocity vector maps of the two cases $MFR = 2\%$ and 4\% on four planes at $y/P = 0.5$ (mid-passage), 0.25 (near SS), 0 (near the stagnation point, upstream of LE), and -0.25 (near PS) as in Fig. 4.9. The temperature is non-dimensionalized as $\theta = (T - T_\infty)/(T_c - T_\infty)$. The vectors show the direction of 3D velocity projected onto the 2D planes.

Between the cases of $MFR = 2\%$ and 4\%, the main differences in flow physics occur in Plane 2 and 3. At $MFR = 2\%$, the ingress occurred at Plane 2 while the egress took place at Plane 3, effectively transporting the coolant from the suction side towards the leading edge. Ideally, this
could help coolant gain momentum to penetrate the horseshoe vortex near the leading edge. However, the scale of the horseshoe vortex was so large, as shown in Fig. 4.10(e), that the upstream coolant film and the egress gases could not merge before the horseshoe vortex diverted them away from the endwall. Overall, this pitch-wise coolant transport did not help the coolant overcome the horseshoe vortex. At $MFR = 4\%$, the ingress-egress patterns were opposite, diverting coolant from upstream of the leading edge towards the suction side. Consequently, the suction side leg of the horseshoe vortex was considerably diminished, as shown in Fig. 4.10(d), allowing more coolant to flow into the passage on the suction side. This flow pattern also explains the difference between Figs. 4.6(a) and (b). For $MFR = 3\%$, coolant was laterally transported from $y/P = 0$ towards $y/P = 0.4$ and showed different locations of coolant accumulation compared to $MFR = 2\%$.

At the same time, both cases resulted in ingress on Plane 1 and egress on Plane 4, effectively transporting the coolant from mid-passage to the pressure side. This transport favors the cooling performance since the passage vortex downstream constantly sweeps the coolant towards the suction side. However, even with coolant $MFR = 4\%$, the pressure side leg of the horseshoe vortex was still too strong for the louver coolant film to penetrate (Figs. 10(g) and (h)).

Figure 4.11 takes a closer look at the effect of the passage vortex by showing the total pressure distributions at two transverse planes in the NGV passage. For both cases, a loss core emerged at $x/C_{ax} = 0.2$ and quickly developed from $x/C_{ax} = 0.2$ to 0.4. According to Biesinger & Gregory-Smith [4.14], the loss core near the endwall indicates an insufficiently energized cooling film, which is drawn towards the mainstream. Therefore, the remaining coolant is likely to be dominated in the passage for the louver cooling. A much higher $MFR$ is necessary to counteract the passage vortex.
To further reveal the coolant’s effect on the NGV design, the loading distribution on the endwall, pressure side vane, and suction side vane are shown in Fig. 4.12. Compared to the uncooled scenario, the coolant at $MFR = 2\%$ indicates little difference. As discussed before, this could be due to the weak cooling film not being able to survive the passage vortex. At $MFR = 4\%$, the low-pressure region downstream of the corner separation is slightly farther away from the suction side vane, as shown in the dashed box, which could be due to the interaction of the coolant film with the separation zone. However, little difference can be observed in other regions between...
no-cooling and $MFR = 4\%$ cases. Overall, the upstream louver coolant has minimal effect on the NGV endwall and vanes loading.

Figure 4.12. The total pressure distributions at the NGV passage endwall, vane suction side, and pressure side

The total pressure loss coefficient at the passage exit, defined as

$$Y_t = \frac{P_{\text{total,in}} - P_{\text{total,exit}}}{\frac{1}{2} \rho_{\text{exit}} V_{\text{exit}}^2}$$

(4.11)

is shown in Fig. 4.13. Again, the no cooling and the $MFR = 2\%$ case have almost identical distributions, indicating that the 2\% coolant has little effect on the vane loss. However, with coolant at $MFR = 4\%$, the area near the endwall shows lower total pressure loss. Meanwhile, the contour suggests the vortices at the passage exit are drawn closer to the endwall with the 4\% coolant present. Liu et al. [4.31] found that the passage vortex contributes to the total pressure loss.
Given the discussion above, it can be inferred that the coolant at \( MFR = 4\% \) helps suppress the passage vortex to an extent, mitigating the vane loss. However, it should be noticed that this effect is still limited, and more coolant is required to create a significant change.

\textbf{Total Pressure Loss Coefficient at NGV exit from CFD results}

![Diagram showing total pressure loss coefficient at NGV exit](image)

\textbf{Figure 4.13. The total pressure loss coefficient at the NGV exit}

\textbf{4.4. Discussion of Cooling Effect of Upstream Louver Scheme}

Figure 4.6 shows that increasing the coolant MFR from 2\% to 3\% can improve the cooling performance. To further investigate the effect of coolant flow rate, the CFD results of \( MFR = 2\% \) and 4\% are presented in Fig. 4.14. After the coolant flow rate doubled, the cooling effectiveness significantly increased by 0.4-0.6 in some areas, primarily the upstream half of the passage. However, the pressure side endwall was still barely covered by coolant. A similar study by Alqefl et al. [4.31] revealed that an upstream louver cooling \( MFR = 5\% \) could improve the cooling effect and achieve a minimum \( \eta = 0.4 \) throughout the passage. However, their experiments were conducted with \( DR = 0.97 \), which helped the coolant film development compared to \( DR = 2.1 \) in this study. Overall, a significant amount of coolant is needed to achieve a desirable coolant coverage in the NGV passage with an upstream louver scheme only. This could prove inefficient
compared to adding another cooling scheme closer to the NGV passage and allocating part of the coolant.

**Film Cooling Effectiveness from CFD Results**

![Graph showing film cooling effectiveness distribution from CFD simulations.](image)

**Figure 4.14.** The film cooling effectiveness distribution from CFD simulations of (a) louver scheme $MFR = 2\%$, (b) louver scheme $MFR = 4\%$.

The horseshoe vortex is the limiting factor for the upstream louver cooling effectiveness based on the previous discussion. In the team’s previous work, the cooling jets from the jump cooling holes close to the leading edge effectively suppressed the horseshoe vortex \[4.32\], as shown in Fig. 4.15. If the louver cooling is used with jump cooling, the louver coolant can form a stable film upstream of the jump cooling holes. Then the jump cooling jet diminishes the scale of the horseshoe vortex to allow the upstream coolant film to maintain more momentum and better attach to the endwall. Theoretically, these two cooling schemes can prove mutually beneficial. However, more work is yet to be done to investigate the interaction between the upstream louver coolant film and the jump cooling jets.
5. SUMMARY AND CONCLUSIONS

This study investigated experimentally and numerically the cooling effect of the combustor louver cooling scheme on the NGV endwall. Wind tunnel testing and CFD simulations using Ansys CFX were conducted with engine-representative mainstream conditions. The adiabatic film cooling effectiveness, Nusselt number, and the net heat flux reduction were calculated on the NGV endwall and analyzed to evaluate the cooling performance. In addition, the CFD analysis showed the flow physics of the interactions of the louver coolant, the fishmouth cavity, the horseshoe vortex, and the passage vortex. For the louver cooling scheme used in this study, the key findings are as follows:

1. For the cooling of the contoured endwall upstream of the NGV passage inlet, the coolant MFR needs to surpass a critical value to allow the coolant film to remain uniform and stable after it flows over the fishmouth gap. The experiment results suggest that the critical MFR is between 1%~2%.

Figure 4.15. The temperature contour and streamlines of a jump cooling scheme on a contoured endwall. The temperature is non-dimensionalized as $\theta = (T - T_\infty)/(T_c - T_\infty)$.
(2) For the cooling of the NGV passage, the $MFR = 2\%$ and $3\%$ cases both showed undesirable coolant coverage. The pressure side endwall proves to be the most challenging region for the coolant to reach. Increasing the coolant flow rate over the range investigated improves the cooling performance in the center of the passage due to higher coolant momentum and a more favorable pitch-wise transport of coolant. However, a significantly higher amount of coolant is needed to fully cover the NGV passage.

(3) A cavity vortex forms in the fishmouth cavity due to the pitch-wise pressure difference of the mainstream. The resulting ingress at the fishmouth gap disrupts the upstream coolant film and draws hot mainstream gas towards the endwall. At the same time, the egress helps energize and develop the coolant film to counteract the horseshoe vortex. Increasing the coolant flow rate changes the ingress-egress pattern, thus altering the pitch-wise coolant transport.

(4) For the coolant flow rate studied, although the strong horseshoe vortex and the passage vortex lead to insufficient cooling in the NGV passage, the louver cooling scheme effectively forms a stable cooling film downstream of the cooling exit. Considering the jump cooling’s effects on overcoming the horseshoe vortex and passage vortex, applying the combustor louver cooling with the NGV jump cooling is suggested to overcome the horseshoe vortex while maintaining a stable coolant film.

6. NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Area, mm$^2$</td>
</tr>
<tr>
<td>$C$</td>
<td>Vane Chord, mm</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Specific Heat at Constant Pressure, J/(kg·K)</td>
</tr>
<tr>
<td>$C_r$</td>
<td>Recovery Coefficient</td>
</tr>
<tr>
<td>$HTC$</td>
<td>Heat Transfer Coefficient, W/(m·K)</td>
</tr>
<tr>
<td>$k$</td>
<td>Thermal Conductivity, J/(m·s·K)</td>
</tr>
<tr>
<td>$P$</td>
<td>Vane Pitch Length, mm</td>
</tr>
<tr>
<td>$q''$</td>
<td>Heat Flux, W/m$^2$</td>
</tr>
<tr>
<td>$S$</td>
<td>Vane Span Length, mm</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature, K</td>
</tr>
</tbody>
</table>
\( t \)  Time, s  
\( V \)  Velocity, m/s  
\( x, y \)  Coordinate, mm  
\( y^+ \)  non-dimensional distance to the wall

**Greek Letters**

\( \eta \)  Adiabatic Film Cooling Effectiveness  
\( \rho \)  Density  
\( \varphi \)  Overall Cooling Effectiveness

**Non-dimensional Numbers**

\( DR \)  Density Ratio, \( \left( \frac{\rho_c}{\rho_{\infty}} \right) \)  
\( M \)  Blowing Ratio, \( \left( \frac{P_c V_c}{\rho_{\infty} V_{\infty}} \right) \)  
\( Ma \)  Mach Number  
\( MFR \)  Mass Flow Ratio, \( \left( \frac{\rho_c V_c A_c}{\rho_{\infty} V_{\infty} A_{\infty}} \right) \)  
\( NHFR \)  Net Heat Flux Reduction, \( \left( 1 - \frac{HTC_f}{HTC_o} (1 - \frac{\eta}{\varphi}) \right) \)

\( Nu \)  Nusselt Number  
\( Re \)  Reynolds Number  
\( Tu \)  Turbulence Intensity

**Subscripts**

\( ax \)  Axial  
\( aw \)  Adiabatic Wall  
\( c \)  Coolant Flow  
\( exit \)  Vane Passage Exit  
\( f \)  Film Cooled Case  
\( in \)  Inlet  
\( o \)  Uncooled Case  
\( r \)  Recovery  
\( w \)  Endwall  
\( \infty \)  Freestream Flow

**Abbreviation**

DLRT  Dual Linear Regression Technique  
NGV  Nozzle Guide Vane  
SST  Shear Stress Transport
7. ACKNOWLEDGEMENTS

The authors would like to acknowledge and thank Solar Turbines Incorporated for the funding, support, and guidance on the works presented herein.

REFERENCES


CHAPTER 5
CONCLUSIONS AND FUTURE WORK

CONCLUSIONS

This study investigates two commonly used cooling schemes, jump cooling and combustor louver cooling. Wind tunnel tests were carried out at Virginia Tech Transonic Wind Tunnel under engine-representative freestream Mach number, turbulence intensity, coolant density ratio, and blowing ratio. Meanwhile, the CFD study was conducted using the Ansys Fluent and CFX with the identical configuration to the experiments. The film cooling effectiveness and the Nusselt number distribution on the NGV endwall are shown to reveal the effect of the density ratio and the combustor-turbine misalignment on the jump cooling and the cooling capacity of the louver cooling. Furthermore, the CFD results demonstrate the interaction between the coolant and the freestream to reveal the flow physics behind the cooling performance. For the two cooling schemes investigated, the key findings are

(1) The main flow features in a typical NGV section that interact with the coolant injection and the coolant film are the cavity vortex, the horseshoe vortex, and the passage vortex (passage flow). First, the cavity vortex forms near the endwall discontinuity at the combustor-turbine interface, modeled as a step or a cavity in this study. This vortex generally disrupts the upstream coolant film to remain stable by driving it away from the endwall and suppressing the coolant injection downstream. Second, the horseshoe vortex is induced upstream of the vane leading edge. It results in a stagnation region around the leading edge, preventing the coolant film from approaching. Meanwhile, its recirculation also tends to draw the upstream coolant film away from the endwall. Last, the passage vortex forms in the NGV passage due to the pressure side vortex of
the horseshoe vortex and the cross flow in the passage. It sweeps the coolant film towards the suction side vane, thus making it challenging to achieve coolant coverage on the pressure side endwall.

(2) After the coolant is injected from the cooling scheme outlet, it first attaches or reattaches to the endwall geometry and tries to form a laterally spread, developed coolant film. Then as it enters the NGV passage, the coolant film is constantly driven towards the suction side by the strong passage vortex. Therefore, developing a relatively stable coolant film is valuable in the endwall cooling upstream of the passage inlet. Meanwhile, the coolant film achieved on the pressure side near the passage inlet is critical for cooling the NGV passage endwall.

(3) The coolant density ratio and the blowing ratio have a coupled effect on the jump cooling jets. Generally, increasing the momentum ratio or lowering the density ratio helps suppress the horseshoe vortex to improve the cooling effect but also causes more severe jet lift-off. Results suggest that once the coolant momentum reaches a specific value, increasing the coolant momentum no longer provides additional benefits. Therefore, the density ratio needs to be carefully balanced with the blowing ratio to optimize the cooling effect. Moreover, the combustor-NGV step can also affect this balance. Since the horseshoe vortex and the cavity vortex scale with the step height, a higher misalignment step requires higher coolant momentum or lower coolant density to reach the optimal cooling effect.

(4) The combustor louver cooling scheme injects the coolant from a slot at a small angle, resulting in low coolant momentum but a favorable mixing of the coolant and the mainstream. However, due to the complex endwall contour, to cool the contoured endwall upstream of the NGV passage inlet, the coolant MFR needs to surpass a critical value to allow the coolant film to remain uniform and stable after it flows over the fishmouth gap. The experiment results suggest that the
critical MFR is between 1%~2%. For the cooling of the NGV passage, the \( MFR = 2\% \) and 3% cases both showed undesirable coolant coverage. The pressure side endwall proves to be the most challenging region for the coolant to reach.

(5) A cavity vortex forms in the fishmouth cavity due to the pitch-wise pressure difference of the mainstream. The resulting ingress at the fishmouth gap disrupts the upstream coolant film and draws hot mainstream gas towards the endwall. At the same time, the egress helps energize and develop the coolant film to counteract the horseshoe vortex. Increasing the coolant flow rate changes the ingress-egress pattern, thus altering the pitch-wise coolant transport.

(6) Overall, the louver cooling is more favorable in the low-speed region upstream of the leading edge. Despite the low momentum, its resulting boundary layer is developed to offer good coverage even downstream of the fishmouth gap. On the contrary, the jump cooling scheme is good at suppressing the horseshoe vortex and forming more energized boundary layers downstream, but the tendency of jet-lift off leaves the region near the cooling holes exposed to hot mainstream before the coolant film spreads laterally. Therefore, the author proposes a system with both jump cooling and combustor louver cooling. Theoretically, the louver cooling scheme results in a developed coolant film to cover the area before the jump cooling coolant reattaches to the endwall and spreads laterally. Meanwhile, the jump cooling jets suppress the horseshoe vortex to allow the more louver coolant to persist into the passage.

FUTURE WORK

Based on the author’s experience in this study, the following items are proposed to be investigated by future researchers.
(1) A cooling scheme consisting of jump cooling and upstream louver cooling. This study suggests that these two schemes can potentially work in a mutually beneficial manner. Further investigation of this scenario can provide more insight into the interactions between the louver coolant film and the jump cooling jets.

(2) Improving data reduction technique. The DLRT used in this study is good at obtaining the deliverables from transient experiments, negating the need to run steady-state experiments, which usually take a much longer time. However, the two-test method to reduce the noise-to-signal ratio raises the challenge of running the two tests with different coolant temperatures at similar flow conditions. The team achieved this by finely tuning the coolant setting, but desirable runs are challenging and can be ruined by other uncertainties. Therefore, improving the DLRT technique to loosen the requirements of the experiments or increase its noise-to-signal ratio will prove precious to future testing.

(3) Effects of the endwall contour. The contoured endwall, especially upstream of the vane leading edge, undeniably affects the near-wall flow and boundary layer development. Chapter 2 clearly shows its part in the cavity vortex scale as the combustor-NGV step height changes. It would be interesting to decouple the effect of the endwall contour and show its interaction with the flow. This could potentially provide references for designing the cooling schemes, such as placing the jump cooling holes.

(4) Uneven distribution of the coolant. This study demonstrates the importance of the coolant film at the NGV passage inlet due to the passage vortex effect. As a result, to provide desirable cooling over the passage, it might be more effective to distribute more coolant near the pressure side. An investigation of the benefit and risks would show the feasibility of this idea.
This appendix serves to investigate the boundary layer of the film-cooled NGV passage discussed above. For the no-step jump cooling case at $M = 2.5$ and $DR = 1.95$, the velocity and temperature boundary layer are plotted at selected locations in the passage. The boundary layers are discussed to shed light on the boundary layer development and its interaction with the coolant flow.

In an NGV passage, due to the bending of the vane, the centrifugal force causes more mainstream gases to approach the pressure side vane, creating a pressure gradient that is perpendicular to the streamwise direction. This pressure gradient leads to flow in its direction, which is usually called a secondary flow in the turbine engine. The secondary flow and main flow result in the boundary layer development in two perpendicular directions. In other words, the boundary layer emerging in this study is three-dimensional.

Although researchers have well studied the two-dimensional boundary layer, Moore [A.1] suggested that the three-dimensional boundary layer possesses much more challenges that cannot be solved using a two-dimensional theory. He specifically pointed out that a unified theory of the secondary flow effect is tough to construct. For example, one fundamental challenge is to define the separation in the three-dimensional flow and find a general criterion [A.2]. In a two-dimensional flow, the disappearance of shear signifies the separation, but it is impossible to decide which direction to consider that shear in a three-dimensional flow.

Considering the characteristics of the NGV passage flow, the author decides to focus on the three-dimensional boundary layer in two directions – the streamwise direction and the direction perpendicular to it (called the transverse direction below). In order to further demonstrate the boundary layer physics, the displacement thickness in the streamwise direction is calculated as
\[ \delta_{\text{stream}}^* = \int_0^\infty \left( 1 - \frac{\rho u_{\text{stream}}}{\rho_x u_{\text{stream},x}} \right) dz \]  

(A.1)

where \( u_{\text{stream}} \) is the velocity in the streamwise direction. It should be noticed that the momentum thickness commonly studied in a two-dimensional boundary layer is not available here since the velocity direction keeps changing within the boundary layer.

Figure A.1 shows the locations selected. Points 1, 2, and 3 are located at \( x/C_{ax} = 0 \) to show the boundary layer at the passage inlet. Points 4, 5, and 6 are located at \( x/C_{ax} = 0.35 \) to demonstrate the boundary layer development in the upstream section of the passage. The last three points 7, 8, and 9, are placed at \( x/C_{ax} = 0.7 \) to show the boundary layer near the throat region. Among each group, points 1, 4, and 7 are located at 20% local pitch distance (the distance between the pressure side vane and suction side vane at its corresponding \( x/C_{ax} \)) from the pressure side. Points 2, 5, and 8 are at the center of the passage. Points 3, 6, and 9 are 20% local pitch distance from the suction side. The data at those locations are extracted from the CFD results in Chapter 2 and analyzed.
Figure A.1 The locations where the boundary layers are studied

The characteristics of the boundary layers are tabulated in Table A.1. The velocity angle is the angle between the local streamwise direction and the axis direction (horizontal in Fig. A.1). Notice that a clockwise rotation here results in a positive angle so as to be consistent with the previous discussion.

Table A.1 The characteristics of the boundary layers at locations 1-9

<table>
<thead>
<tr>
<th>Location</th>
<th>Freestream velocity $u_{stream}$, m/s</th>
<th>Freestream velocity direction, deg</th>
<th>99% boundary layer thickness $\delta$, mm</th>
<th>Displacement thickness in the streamwise direction $\delta^*$, mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>36.59</td>
<td>20.31</td>
<td>13.5</td>
<td>2.2</td>
</tr>
<tr>
<td>2</td>
<td>59.24</td>
<td>13.23</td>
<td>14.1</td>
<td>1.7</td>
</tr>
<tr>
<td>3</td>
<td>76.19</td>
<td>-8.61</td>
<td>14.3</td>
<td>1.6</td>
</tr>
<tr>
<td>4</td>
<td>61.16</td>
<td>41.02</td>
<td>9.5</td>
<td>-0.32</td>
</tr>
<tr>
<td>5</td>
<td>90.27</td>
<td>29.46</td>
<td>9.0</td>
<td>-0.34</td>
</tr>
<tr>
<td>6</td>
<td>143.66</td>
<td>17.03</td>
<td>6.9</td>
<td>-0.37</td>
</tr>
<tr>
<td>7</td>
<td>160.40</td>
<td>55.13</td>
<td>7.4</td>
<td>-0.35</td>
</tr>
<tr>
<td>8</td>
<td>255.30</td>
<td>54.34</td>
<td>7.9</td>
<td>-0.42</td>
</tr>
<tr>
<td>9</td>
<td>348.92</td>
<td>66.22</td>
<td>6.2</td>
<td>-0.17</td>
</tr>
</tbody>
</table>
The velocity and temperature distributions within the boundary layers are present in Fig. A.2 & A.3.

Figure A.2. The temperature and velocity distributions near the endwall of Locations 1-4
In Fig. A.2 & A.3, it can be observed that the temperature increases slower than the velocity within the boundary layer, leading to a thicker temperature boundary layer than the velocity counterpart. This difference suggests that the flow within the velocity boundary layer is colder.
than the freestream in general. Thus all the boundary layer flow has a cooling effect. This finding justifies the occasional interchangeable use of 'boundary layer' and 'coolant film' in Chapters 1-3.

Another general observation is the significance of the secondary flow. Unlike the streamwise velocity, which keeps increasing from the endwall to the freestream, the transverse flow possesses a maximum velocity at 0.1-0.3mm away from the endwall, then quickly decreases to 0 further away from the endwall. In other words, the strongest secondary flow occurs very close to the endwall. In terms of magnitude, the maximum transverse velocity is 10% to 20% of the freestream streamwise velocity. Therefore, given the secondary flow's location and strength, it can be inferred that the secondary flow causes transverse transport of the coolant. This inference also agrees with the hypothesis in Chapter 1, which suggests the coolant accumulated on the pressure side is swept towards the suction side.

When looking at the boundary layer development (locations 1-4-7, 2-5-8, and 3-6-9), it is apparent that the boundary layer is thinning as the flow goes through the NGV passage. This is a common phenomenon in a channel contraction, which induces higher Re downstream. However, at $x/C_{ax} = 0.35$ and 0.7, the streamwise displacement thickness is surprisingly below 0, which means the boundary layer region effectively obtains higher mass flux than the freestream with the same thickness. This phenomenon is possible since the near-endwall flow is much denser than the freestream because of the coolant film attached. Additionally, the shapes of the velocity distributions in Fig. A.2(d) and A.3(a-e) indicate that the boundary layers are highly energized and stable. However, it could also be caused by the errors of CFD or the inappropriate definition of $\delta^*_{stream}$.

As the boundary layer develops, a warm region emerges and expands near the endwall. In Fig. A.2(c) and Fig. A.3, the wall temperature is higher than the minimum temperature, usually at
0.5-2mm from the endwall. This trend is more severe at the suction side endwall (Location 3-6-9) than other locations. Compared with the experimental film cooling effectiveness, those locations with a warm endwall seem to experience an inferior cooling effect. However, to the author’s knowledge, there is no explicit explanation. One likely reason is that as the passage vortex carries the coolant towards the suction side, it draws hot freestream gas near the endwall, leaving warm gas underneath the coolant film. Nevertheless, this does not explain Fig. A.2(c), where the passage vortex is still very weak at the passage inlet. More work needs to be done to figure out the physics and potentially overcome this hurdle in endwall cooling.

In summary, this appendix attempted to understand the boundary layer development in an NGV passage with upstream jump cooling. Results showed that the thermal boundary layer is thinner than the velocity boundary layer. The comparison of the streamwise and transverse velocity demonstrated the capability of the secondary flow to transport the coolant. It was also found that the boundary layer is highly energized after it enters the NGV passage. Some strange phenomena occurred without clear explanations, like the negative streamwise displacement thickness and the warm region near the endwall. More investigation needs to be done to fully understand the complicated boundary layer in the film-cooled NGV passage.

REFERENCES


APPENDIX B: THE UNCERTAINTY ANALYSIS

This section presents the detailed uncertainty analysis of the DLRT technique that calculates the heat transfer coefficient and adiabatic cooling effectiveness. In this process, the measurement uncertainties are caused by the IR camera and thermocouples. Their details are shown in Table B.1.

Table B.1 The instrument uncertainties in the temperature range investigated in this study

<table>
<thead>
<tr>
<th>Instrument</th>
<th>Model</th>
<th>Uncertainty reported by Manufacturer</th>
</tr>
</thead>
<tbody>
<tr>
<td>IR Camera</td>
<td>FLIR A325SC</td>
<td>±1.0°C or ±0.75%, whichever higher</td>
</tr>
<tr>
<td>Thermocouple</td>
<td>Omega T-Type</td>
<td>±2.0°C or ±2.0%, whichever higher</td>
</tr>
</tbody>
</table>

For the DLRT technique, the endwall heat flux is first calculated using Eq. 1.5. Then an iteration of the recovery factor is carried out to maximize the $R^2$ value in the linear regression of Eq. 1.6 to find out $HTC$ and $\eta$. This iteration leads to a complicated implicit relation between $HTC$, $\eta$, and the input variables. As a result, a perturbation method developed by Moffat [B.1] is applied to the DLRT technique.

The perturbation method is shown as a flow chart in Fig. B.1. First, for the dataset of one pixel on the IR camera image, $HTC$ and $\eta$ are calculated using the DLRT technique. Then for $i = 1$ to $n$, where $n$ is the number of input temperature variables, the uncertainty of $T_i$ is added to the variable value $T_i$. Then the updated $HTC_i$ and $\eta_i$ are calculated using augmented $T_i$ and the original values of all the other input variables. After that, the difference between the updated $HTC_i$, $\eta_i$, and the initial results $HTC$ and $\eta$ are found. Finally, the uncertainty in $HTC$ and $\eta$ are the root sum square (RSS) of those differences.
Using multiple sample data sets from the experiments, the uncertainty in the results is found to be approximately ±9.6% in HTC and approximately ±0.1 in \( \eta \) with a 95% confidence interval.

REFERENCES

APPENDIX C: Validation of 1-D Semi-Infinite Conduction Assumption

In the data reduction technique, a 1-D semi-infinite conductive heat transfer model was assumed to calculate the heat flux through the endwall. This appendix serves to validate this assumption to ensure the accuracy of the experimental results.

C.1. VALIDATION OF SEMI-INFINITE CONDUCTION ASSUMPTION

A semi-infinite conduction model consists of an idealized solid body that extends to infinity in all directions except one. If transient conductive heat transfer occurs, the only boundary condition occurs on the only surface, which allows the Cook method to calculate the surface heat flux. In this research, as long as the bottom of the test piece stays at the initial temperature during the blowdown window, the semi-infinite conduction assumption is valid. The conduction model of the test piece can be written as:

\[
\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \quad \text{(C.1)}
\]

\[
T(x,0) = T_0 \quad \text{(C.2)}
\]

\[
T(0,t) = T_w \quad \text{(C.3)}
\]

\[
T(x,t) = T_0 \quad x \rightarrow \infty \quad \text{(C.4)}
\]

where \(x\) is the depth into the test piece, \(T_0\) is the initial temperature, \(T_w\) is the wall temperature, and \(\alpha\) is the thermal diffusivity of the printed ABS test piece, as shown in Fig. C.1. In this model, \(T_w\) is assumed to be constant.
Let $\theta = T - T_w$ be the temperature difference between the temperature and the wall temperature at any location. Eqs. C.1-4 can be solved as Eq. C.5,

$$\frac{\theta}{\theta_0} = \text{erf} \left( \frac{x}{2\sqrt{\alpha t}} \right)$$

Here erf is the error function, $\theta_0 = T_0 - T_w$. At $x = 2\sqrt{\alpha t}$, $\frac{\theta}{\theta_0} = \text{erf} (2) = 0.9953$, indicating the temperature change at location $x$ is less than 0.5% at time $t$ compared to the surface. In other words, at time $t$, the surface boundary condition's effect has not reached location $x$. If the thickness of the test piece is greater than $x$, then it can be approximated as a semi-infinite model at time $t$. In this research, at the end of the blowdown window $t = 20s$, the critical thickness is found to be 7mm. The whole test piece endwall achieves this minimum thickness except for the region above the cooling holes. However, the region above the cooling holes is not used in the analysis and discussion of this research. Therefore, the semi-infinite assumption is valid for all the data reported.
C.2. VALIDATION OF 1-D CONDUCTION ASSUMPTION

The 1-D assumption ignores the lateral conductive heat transfer, allowing easier calculation of the heat flux at locations on the endwall. A CFD model is built to investigate the effect of the lateral conduction, as shown in Fig. C.2. The thermal properties of the model are set to be the same as the ABS test piece. The top surface is 2cm x 2cm under convective boundary conditions, mimicking the test piece surface under the influence of freestream. As shown in Fig. C.3, two convective conditions are applied in two simulations. The convective heat transfer coefficient is 1000W/m$^2$K, higher than the maximum observed from the experimental data. The first simulation has a uniform freestream temperature of 350K, simulating the 1-D conduction. The second one keeps the same 350K at the bottom left corner and introduces a temperature gradient of $dT/dx = 500$K/m in both directions. The model's thickness is 1cm, enabling the semi-infinite approximation based on the discussion above. All the four side walls are set to symmetry conditions, and the bottom surface is at a constant temperature of 350K. The model is divided into 500,000 cubic elements of 0.2mm x 0.2mm x 0.2mm. For the initial condition, the whole model is set to 350K. The transient simulation lasts 20 seconds, the same as the blowdown window with a time step of 0.02s.

Figure C.2 The model and mesh of the 1-D assumption validation numerical work
Figure C.3 The convective boundary conditions applied on the top surface of the CFD model. The contour shows the freestream temperature distribution.

The surface temperature evaluated at the bottom left corner of the convection surface, as shown in Fig. C.3, is presented in Fig C.4. When a large freestream temperature gradient introduces the lateral conduction, the difference is very marginal. The heat flux calculated by the Cook method suggests a difference smaller than 1% during the blowdown window. Consequently, the effect of lateral conduction can be ignored in the data reduction. In other words, the 1-D assumption is valid in this research.
Figure C.4 The surface temperature comparison between 1-D conduction and 3-D conduction from the CFD results.
APPENDIX D: Discussion of the CFD solver and turbulence model

Ansys Fluent and Ansys CFX are two of the most commonly used applications in solving computational fluid dynamics. However, due to the different approaches to handling the flow equations, these two solvers usually generate different results for the same problem. Turbulence models are equations to anticipate the time-averaged parameters in the flow field. The introduction of turbulence models in numerical methods allows the solver to avoid calculating the full Navier-Stokes functions to reduce computational costs. However, each turbulence model inevitably has its limitations and should be chosen carefully. This appendix discusses the CFD solver and turbulence models used in this research and how they may affect the results.

D.1 JUMP COOLING

The Ansys Fluent v15.0 with the realizable $k$-$\varepsilon$ model is used in the numerical study of the jump cooling scheme (Chapters 2 and 3). Since the jump cooling is designed to inject the coolant at a high injection angle, mixing a circular jet and the freestream is one of the dominant features in the flow field. Prior studies show that for discrete hole cooling, the realizable $k$-$\varepsilon$ model usually outperforms other turbulence models in the flow field near the cooling hole exit, thus resulting in an accurate prediction of the cooling effectiveness in this region [D.1-D.5]. In the attempt of different turbulence models on the jump cooling study, the authors also found realizable $k$-$\varepsilon$ model is the most accurate. As shown in the first row of Fig. D.1, the realizable $k$-$\varepsilon$ model best agrees with the experiment results at $0 < x/C_x < 0.5$, while the RSM model underpredicts and the SST $k$-$\omega$ and RNG $k$-$\varepsilon$ overpredict the Nusselt number here. In the second row of Fig. D.1, the realizable $k$-$\varepsilon$ model is also the only one that does not overpredict the cooling effectiveness in this region. As discussed in Chapter 2, the coolant jet detaches and reattaches to the endwall in this...
region, which determines the cooling film mass and momentum. Accurately simulating the flow physics here shows the superiority of the realizable $k$-$\varepsilon$ model in this problem. Currently, this turbulence model is only compatible with Ansys Fluent, so Fluent is selected for this simulation.

Figure D.1 The Nusselt Number distribution (row 1) and the film cooling effectiveness distribution for the baseline jump cooling case. Four different turbulence models, Reynolds stress equation, Realizable $k$-$\varepsilon$, $k$-$\omega$ shear stress transport, and Re-Normalisation Group $k$-$\varepsilon$ models, are used in Ansys Fluent.

However, it should also be noticed that the realizable $k$-$\varepsilon$ model shows a reattachment line downstream of the experiment results, as shown in the second row of Fig. D.1. This phenomenon could result from failing to simulate the separation immediately after the cooling hole exit, which leads to less coolant driven to the endwall. In their research, Jeon and Son have pointed out this weakness of the realizable $k$-$\varepsilon$ model [D.6]. Besides, Harrison and Bogard found that the realizable $k$-$\varepsilon$ model tends to underpredict the lateral spread of the coolant jet [D.3]. This is also reflected in
Fig. D.1. The streaks of high Nusselt number extend from the cooling holes to the mid-passage region. Overall, the CFD result in the jump cooling research is accurate enough for trend study and helps understand the flow physics. However, the complex interaction between the coolant jets and the mainstream is yet to be fully achieved.

**D.2. LOUVER COOLING**

The louver cooling exit is located far more upstream than the jump cooling. As a result, the horseshoe vortex and passage vortex have much less influence on the coolant when it exits the louver slot. Instead, the fishmouth gap between the louver exit and the NGV passage plays an important part in the coolant film development. Since the fishmouth gap is very likely to cause the freestream and coolant flow to separate, the CFD solver needs to be able to simulate the flow separation under low-speed conditions ($Ma \sim 0.1$). The prior studies show a mixed opinion on which turbulent model is best to cope with the flow separation. However, several studies pointed out that the SST $k$-$\omega$ is good at dealing with adverse pressure gradients in boundary layers [D.8-D.13]. For this reason, the SST $k$-$\omega$ model is selected for louver cooling simulations.

SST $k$-$\omega$ model is supported by both Ansys Fluent and CFX. Again, the review of the two solvers is mixed and highly dependent on the specific flow field. Generally, previous research work seems to agree that Fluent provides a better prediction of the cooling jet structure and the cooling effect near the coolant exit, while CFX shows a more accurate lateral spread of the coolant and better prediction in the region far from the coolant exit [D.14-D.16]. Therefore, the louver cooling simulation in this research was carried out in CFX. However, future work can focus on trying more turbulence models and even with the Fluent solver to investigate their effects further to provide more accurate numerical results.
REFERENCES


