CHAPTER 2.0 LITERATURE REVIEW

Various bodies of research have been performed in the past to address the problem of unsteady heat transfer on turbine blades. These attempts include experimental, as well as computational examinations of unsteady heat transfer (from and shock waves) on the surface of a turbine blade. While the problem of predicting shock-induced heat transfer continues to be difficult on all levels, the benefit of doing so remains worthwhile. Useful analytical and numerical simulations can potentially aid the turbine designer toward accurate predictions of blade heat transfer.

2.1 Experimental Research

Oxford Research

Oxford researchers have conducted research on various aspects of unsteady turbine blade heat transfer$^{2-8}$. Their work includes numerous contributions in the area of cascade flow, stator-rotor flow interaction (i.e. wakes, shock waves, boundary layer interactions), and unsteady turbine blade heat transfer. Their unsteady heat transfer experiments were conducted in the Isentropic Light Piston Tunnel (ILPT). This facility is a short duration wind tunnel which can generate a flow having a stagnation temperature of 430 K over a cascade of blades at 290 K. Full-scale Reynolds and Mach numbers, and gas-to-blade surface temperatures ratios are achieved for durations of ~0.3 seconds. A disk equipped with radial bars on the circumference was rotated at high speeds ahead of the cascade to simulate the shocks and wakes shed from a turbine nozzle guide vane. This
rotating bar device actually produced a bow and recompression shock and a wake, which were convected toward a cascade of highly loaded turbine blades by the tunnel flow. Dependent upon the number of radial bars placed on the circumference of the rotating disk, the frequency of the unsteady wakes and shocks convected toward the turbine blades can be controlled.

Turbine blades are equipped with either Kulite pressure transducers or thin film heat flux gages. The heat transfer gages are thin film temperature sensors painted on machineable glass turbine blades. They have a reported frequency response of 100 kHz. The post-processing procedure of the data obtained from these thin film sensors is based on a semi-infinite (gas) conduction analogy. This analogy breaks down when the sensor becomes exposed to conduction from the blade material. This sets the maximum time limit for the applicability of the post-processing analogy.

Doory et al.\textsuperscript{2} used Oxford’s ILPT to experimentally simulate the effects of wakes and weak shock waves formed at the trailing edge of the nozzle guide vane (NGV) on turbine blade heat transfer. The flows entering and exiting the blade passage were at Mach numbers of 0.3 and 1.0, respectively. The rotating bar device was rotated at speeds generating a bar-relative Mach number of 0.95 as the bars passed through a plane parallel to the leading edge of a stationary cascade of turbine blades. While this bar-relative Mach number is not quite sonic, weak bow shock waves and significant recompression shock waves were observed from the data as well as the shadowgraph photographs that were taken to document this wake/shock wave propagation event. Doory also observed that the recompression shock initially impacts at the crown of the turbine blade suction surface and propagates toward the stagnation point. He concluded that the shock caused the flow to momentarily separate at the leading edge, creating a bubble that moved a short distance downstream along the blade suction surface before collapsing. A turbulent boundary layer patch remained after bubble disintegration. It was then concluded that this turbulent patch caused the high heat transfer rate associated with shock impact as it was swept down the boundary layer. The direct impingement of the shock had little effect on the heat transfer rate. The indirect effect of the shock impact (leading edge separation
bubble) produced a more noticeable effect on a laminar boundary layer than on a tripped (transitioning or turbulent) boundary layer.

This work inspired Ashworth et al\textsuperscript{3} to conduct the same experiment, but with stronger shock waves and a different turbine blade profile. The flow entered and exited the blade passage at Mach numbers of 0.3 and 1.2, respectively. This experiment was performed in two phases: first for a steady flow (rotating bar device not used) with different turbulent intensity levels in the freestream, and second with the rotating bar device rotating at higher speeds, producing a bar-relative Mach number of 1.17. For the steady experiment, the turbulent intensity was incrementally increased (using turbulence screens) from $<1$ percent to 4 percent which resulted in the boundary layer transition position moving from 50 percent to 10 percent of the blade length. This enabled the heat flux sensor to produce a signal for both laminar and turbulent boundary layers. In the unsteady experiment, the heat transfer during intermittent transitioning of the boundary layer (from laminar-to-turbulent) could be compared to the levels seen in the steady experiment, hence identifying the state of the boundary layer during wake/shock wave impact. The use of strong shock waves in this experiment resulted in a drop in heat transfer not seen with weak shocks, followed by a 250 percent increase in heat transfer due to the momentary boundary layer separation-reattachment phenomena at the point of impact. Turbulent boundary layer reattachment occurs, and the boundary layer returns to laminar (dependent on the freestream turbulent intensity). A weaker shock wave did not induce an immediate change in heat transfer during impact, but instead when the shock reached the blade leading edge. Immediately after impact, the stronger shock wave caused a heat transfer reduction and then a sharp increase that was felt to be because of the separation-reattachment phenomena. The separation-reattachment phenomena moves toward the leading edge coupled with the shock wave. The researchers recommended further investigation, since the turbulent behavior seen after the shock could not be attributed to the shock. An additional complication was the fact that the wake also contributes turbulent energy to the boundary layer.
A debate appeared imminent regarding which unsteady phenomena (shock or wake) forces the transition of an otherwise laminar boundary. Doorly attempted to use the isolated wake/weak shock data from Reference [1] to create a model that describes the increase in heat transfer. The model assumed that forced transition was primarily due to the turbulent energy in the wake. This model also assumed that the turbulent content of the wakes produced an immediate formation of turbulent patches in an otherwise laminar boundary layer. The mean heat flux levels were predicted by defining a position dependent intermittence function, corresponding to the estimated fully-laminar and fully turbulent boundary layers. The intermittence function was derived from a simple flow computation, which treated the wake as a band of turbulence and neglected the velocity deficit. Second order statistics (auto and cross correlations) suggested that the turbulence imposed by the wake contributed little to the heat flux associated with an already turbulent boundary layer. It was concluded that the wake velocity deficit was responsible for the increase in heat flux on an already turbulent boundary layer. The model did not fair as well at lower Reynolds numbers on a laminar boundary layer. It suggested that not even a high turbulent energy wake was sufficient to force an early intermittently turbulent boundary layer.

A similar experiment to Reference [2] was conducted by Johnson et al using strong shock waves. The flow entered and exited the turbine blade cascade at Mach numbers of 0.38 and 1.18. The bar-relative Mach number was set at 1.13 (to produce shocks and wakes) and 0.73 (to produce just wakes). In addition, a 10° lower flow incidence angle was used to simulate off-design engine conditions. The objective was to see if the transient separation phenomenon was the true source of the increase in heat transfer levels after shock impact. However, this experiment suggested that while the separation was a reasonable source of the heat transfer increases, it was difficult to understand how separation could cause the heat transfer reversals (reductions) observed prior to the sharp increase. This question was not answered in Reference [5]. However, a theoretical model was developed to estimate the sharp rise in heat transfer. A first order perturbation analysis of the boundary layer equations showed that adiabatic heating and
cooling of the boundary layer could produce the increases in heat transfer measured during shock wave passing.

The Oxford researchers were approaching a point where numerous unsteady shock wave and wake experiments were performed at different laboratory conditions. This inspired Rigby et al.\textsuperscript{6} to develop a temperature scaling procedure that would collapse experimental data performed at different flow-to-wall temperature ratios to a single curve. The scatter resulting from different temperature ratios was removed in the steady experiments by defining the Nussult number with the recovery temperature. The Nussult number enhancements due to shock wave passing were shown to be strong functions of gas-to-wall temperature ratios. This document included a simple isentropic compression-heating model that was developed to estimate the heat transfer increases during shock wave impingement on the blade surface.

Since the reversal (reduction) of heat transfer prior to the sharp increase had not yet been fully explained, Johnson et al.\textsuperscript{7} offered a more detailed explanation that involved a closer look at the shadowgraph shock wave propagation event. The bubble seen at the leading edge of the turbine was defined as a vortical bubble. This develops from the transition of the oblique shock wave and its reflection becoming a normal shock wave. The reflected shock wave moves away from the surface forming a bifurcation. As this bifurcation moves away from the surface, a vortex sheet rolls up behind the shock wave forming a vortical bubble. It is this vortical bubble that convects along the blade surface. Whether it is along the pressure or suction surface is dependent on the blade profile. This bubble was felt to contain a powerful vortex, which drags hot gases toward the blade surface resulting in an increase in heat transfer. This theory could not be confirmed because the tunnel doors obscured the leading edge of the turbine blade cascade.

Hilditch et al.\textsuperscript{8} fitted the ILPT with a full-stage model rotating turbine capable of maintaining engine representative conditions for 200ms. The data obtained from this experiment was compared to the equivalent data obtained in the two-dimensional linear cascade test section. Fluctuations in heat transfer are observed at the NGV passing frequency in both experiments, but the magnitude of these fluctuations on the rotor were
much smaller than those recorded in the linear wind tunnel test section. Speculations as to the reason for these differences were postponed pending confirmation from further experiments.

*MIT Research*

Around the same time, researchers at MIT utilized a blowdown wind tunnel with a rotating turbine to perform an experiment in an effort to compare their three-dimensional results with Oxford’s two-dimensional data. This work has been documented by Guenette et al. The MIT blowdown wind tunnel used a mixture of argon-R12 as the working fluid, because its ratio of specific heats (\(\gamma\)) closely resembles that of the gas exiting the combustor. The flow enters and exits the rotor at Mach numbers similar to those in the cascade experiment of Ashworth. The blades were instrumented with thin-film nickel temperature transducers mounted on both sides of a 25\(\mu\)m thick polymide insulator. At low frequencies, the temperature drop across the insulator is a direct measure of the heat flux. This applies to frequencies up to 20Hz. Thermal waves within the insulator begin to damp above this frequency. At 1kHz, the insulator appears infinitely thick to the top surface. Therefore, a quasi-one-dimensional model is used along with the temperature history at the surface to determine the measured heat transfer levels. This data reduction technique is valid to a frequency domain of up to 100kHz. Comparisons made to the two-dimensional cascade of Reference [2] showed that both wake and shock spikes were present in the cascade data, but only the shock spike data was seen at \(\leq 30\%\) of the blade chord in the MIT facility. This was felt to be due to the higher (~4%) turbulent intensity of the inlet flow during the MIT tests. At 30% of blade chord, the MIT data also showed a double spike, which was not present in the cascade results. There were two shocks formed at the trailing edge of the NGV row. The first shock directly interacts with the rotor surface. The second shock reflects off an adjacent NGV and its reflection interacts with the rotor row. This is a phenomenon not seen in the cascade simulation. Overall, this research confirmed that the rotating bar experiments provided a reasonable simulation of the shock waves and wakes shed from the NGV of a turbine. Although the unsteady
interactions are stronger on the rotor than in the 2-D cascade, they are similar in nature. Later, MIT research efforts regarding the unsteady interactions in the turbine were mostly numerical, and it will be discussed in the next section.

_Calspan Research_

Other facilities investigating the unsteady turbine interactions include the Calspan Advanced Technologies facility. Extensive work has been performed by Dunn et al.\(^{10,11}\). The short duration facility used by Dunn has been fitted with a Garrett TFE 731-2 HP full-stage turbine\(^{10}\) and a Teledyne 702 HP full-stage rotating turbine stage\(^{11}\). In these experiments, a shock tube is used as a short duration source of heated air. Platinum thin-film temperature sensors are used to obtain heat flux measurements on the rotor blade surface and the stationary shroud. The heat flux data measured is presented in a “phase-resolved” format, which means that the time resolved data is presented as a function of circumferential distance. The goal of these experiments (and others like them, performed by Dunn) is to estimate the varying heat flux effect on blade durability. The experiments actually have a small gas-to-blade temperature ratio. However, the unsteady heat flux results in distinct variations in temperature and temperature gradients along the blade surface. By extrapolating the measured blade surface temperature distribution for an actual inlet turbine temperature of 1200°F, the gradients at the surface (°F/inch) are estimated. The result is the necessary thermal loads to conduct blade material tests in an effort to understand the blade durability. The focus of these experiments centers on the actual thermal loads exerted on the blade surface and not so much the mechanisms causing them.

_Virginia Tech Research_

The researchers at Virginia Tech have been performing unsteady experiments in the Virginia Tech Transonic Blowdown Cascade wind tunnel\(^{12-16}\). This facility is capable of providing inlet and exit Mach numbers of 0.3 and (up to) 1.3 for a duration of 30 seconds. The facility is equipped with a heating loop, which will be described in detail in
Chapter 3. Due to the previous difficulties of separating the shock effects from the wake effects, the Virginia Tech researchers took on the challenge of trying to examine the shock effects in isolation. Shocks generated outside the cascade tunnel are sent down along the leading edge of the linear row of blades, as in Figure 1.4.

At Virginia Tech, this was first attempted by Collie et al.\textsuperscript{12} by using a 12-gauge shotgun that was modified by replacing the stock of the gun with an AC solenoid actuator. A hand-operated electronic switch is connected to the solenoid through a small power supply and a solid-state relay. Throwing the switch activates the solenoid, which in turn pulls the trigger, via a single-bar linkage. The shotgun was loaded with Winchester X12-FBL “Popper-Load” blank shotgun shells. They reportedly produced excellent repeatability and relatively nondestructive barrel emissions. Collie used this apparatus to generate a shadowgraph movie of the shock propagation event, which closely resembled that seen by the Oxford researchers. A minor variation is noted in the development of the “vortical bubble”, which forms at the leading edge. The “vortical bubble” initially propagates along the pressure surface before detaching and collapsing in the freestream. This experiment utilized Kulite pressure transducers only, so the effect on unsteady heat transfer due to the “vortical bubble” leaving the blade surface was not measured.

After Collie examined the propagation event of a shotgun produced single shock through a cascade of turbine blades, Doughty\textsuperscript{13} reproduced the event for three equally spaced shocks (55 and 200\mu sec apart), which were produced by a shock tube and a shock shaper. Doughty thoroughly instrumented the blade pressure and suction surfaces with Kulite pressure transducers to monitor each incident and reflective shock impact throughout the shock propagation event. The “vortical bubble” was not seen in the shadowgraphs, but great care was taken to monitor and vary the incident shock strength entering the test section. This method of shock generation became the method of choice for future unsteady shock studies at Virginia Tech.

Nix et al.\textsuperscript{14} reproduced the shock propagation event for a different blade profile, utilizing a state-of-the-art heat flux microsensor that was developed by Diller\textsuperscript{15}. Details regarding the description and performance of this gage can be seen in the work of
Peabody. Nix used this gage to investigate the shock propagation event with an instrumented blade containing 5 gage positions. Each position had a Kulite pressure transducer and the Vatell heat flux microsensor (HFM), so that pressure and heat flux could be directly measured simultaneously. A shadowgraph movie of the shock propagation event was recorded, showing that the various gage locations on the blade surface were impacted with incident and reflected shocks that were normal to the surface, tangent to the surface, or a combination of both. The heat flux measurements observed normally and tangentially impacting shocks of similar strength. The results showed that a shock impacting the blade surface normally produced a higher heat flux level than a shock of similar strength impacting the blade surface tangentially. This information was considered significant towards the choice of the computational approach described in this document (i.e. computing the normal impacting shock only) and leads one to use shock strength as the input for the estimation of blade surface heat transfer.

2.2 Numerical Research

The specialized area of solving governing equations in an effort to study specific fluid dynamics problem is called Computational Fluid Dynamics (CFD). This area involves applying a variety of numerical techniques toward the solution of a version of the governing equations of fluid flow. The Navier-Stokes (N-S) equations is the system of equations that is the source of most CFD calculations. The N-S equations consist of the continuity equation, x-, y-, and z-momentum equations, and the energy equation. This system of equations is very complex and demanding to solve, and simplifications are usually required to make the calculations manageable. Simplification can often remove some of the flow physics of a real problem, but still contain the essential flow physics that are of interest. Various CFD studies have been performed to improve the understanding of the unsteady shock propagation event in the first stage of an HP turbine.

A common approach towards the solution of a system of equations is MacCormack’s method. MacCormack’s method is a member of the Lax-Wendroff
family of numerical schemes\textsuperscript{18}. An example of the use of this method for shock propagation analysis is seen by the work of Cline\textsuperscript{19}. Cline used this technique to develop a code that explicitly solves the two-dimensional, time dependent, compressible 2-D N-S equations. This CODE was capable of solving a laminar (as well as turbulent) steady or unsteady flow. The available turbulence models were an algebraic mixing-length model, a one-equation model, or the Jones-Lauder two-equation model. Explicit “artificial viscosity” is included for use during shock computations. This is typically needed due to the method’s difficulty in solving the governing equations across properties containing a discontinuous jump in magnitude, commonly present when shocks exist in the flowfield. This code was used to calculate the details of a flowfield around an airfoil. It reasonably estimated shock strength and position along the leading edge of an airfoil. Also, pressure distributions along the airfoil surface matched well with experimental data. The code appears to focus on pressure and velocity distributions along the blade, wake development at the blade’s trailing edge, and stationary shocks. Although the code is 7500 lines long, it provides various choices of flow geometry (i.e. converging-diverging nozzle, boattail afterbody with a solid body simulating jet exhaust and a plane jet in a uniform stream). This code clearly demonstrates the power of the MacCormack method’s power for solving the unsteady two-dimensional N-S equations.

Numerical calculations were included with the published experimental work of Guenette et al\textsuperscript{6}. The two-dimensional unsteady N-S equations were solved using a finite-volume, explicit, multi-step method. A combination of second-order and fourth-order smoothing was used to suppress numerical oscillations. A fine computational grid severely limited the maximum time step permissible. However, since the solutions obtained were steady state, using varying local time steps and implicit residual smoothing accelerated the convergence. The steady-state solutions obtained were compared to both the experimental measurements obtained from the MIT turbine stage and the experimental results obtained from the Oxford turbine blade cascade. Once again, the steady state heat transfer levels calculated compared well with both of the steady portions
of the previously mentioned experimental data. Guenette did not show the results of any unsteady heat transfer calculations.

As a result of the experimental data presented in Reference [8], Giles\textsuperscript{20} performed a numerical analysis of the wake interaction with the rotor surface. The starting points of these calculations were to assume that viscous forces are relatively unimportant in the interaction of the NGV’s trailing edge wake with the rotor surface. This enabled Giles to use the unsteady two-dimensional Euler equations (2-D unsteady N-S equations with heat transfer and viscous terms neglected) for this flowfield. The numerical method used to solve this system of equations was the explicit Lax-Wendroff method. The wake model assumed a uniform static pressure, a uniform total enthalpy and a prescribed velocity defect distribution. The simplification to the 2-D unsteady N-S equation enabled accurate unsteady pressure distributions along the blade surface to be calculated.

The solutions to the governing equations of reference [20] were repeated with the addition of a vorticity source term in the third dimension. The solution of this set of governing equations enabled Giles\textsuperscript{21} to obtain the solutions that reproduced the shock propagation event. In the examples of References [20] and [21], heat transfer solutions were not presented.

Abhari et al\textsuperscript{22} used the commercial code UNSFLO to obtain heat flux solutions and compare them to the experimentally obtained data in Reference [9]. This commercial code is a two-dimensional, Reynolds-averaged, unsteady multi-blade row Navier-Stokes code. This code is a coupled viscous/inviscid code, which solves the Navier-Stokes equations for a body-fitted boundary layer grid using an implicit algorithm, and solves the Euler equations on an outer inviscid grid using an explicit algorithm. The viscous grid is solved using an ADI (alternating-direction-implicit) method\textsuperscript{23} with Roe’s flux-difference splitting\textsuperscript{24}. The inviscid grid is solved using a generalized Ni’s Lax-Wendroff algorithm. The interface between the two is handled in a conservative manner. The grid density was chosen so that there were 18 grid points across the boundary layer. The code was run on a Stellar GS1000 workstation and required 20 CPU hours to complete. This was reported to be equivalent to 2 hours on a Cray XMP. Time-resolved heat transfer measurements and
calculations were compared at two operating points. This code predicted about 90% of the measured integrated heat loads on the blade. Ahbari reported that the discrepancy between experimental data and computational solutions could be due to an inaccurate estimate of the stream-tube thickness, which is one of the code inputs. This research demonstrates that even a two-dimensional unsteady calculation can be difficult while requiring massive storage and sophisticated computing facilities.

The same commercial code (UNSFLO) was used by Moss et al.\textsuperscript{25} to examine its ability to calculate the unsteady pressure field. The unsteady phenomena for these calculations were wakes. No shocks were present in this computational flow field. Good comparison was observed near the leading edge and near the trailing edge. However, discrepancies were still noted on the crown of the suction and pressure surfaces.

2.3 Overview

From the experiments described in Section 2.1, a variety of investigations have been performed in an attempt to better understand some of the unsteady mechanisms contributing to turbine blade heat transfer. Dunn’s work (CALSPAN) produces valuable estimations of thermal loads seen by the turbine rotor. However, they are obtained from actual turbine engine hardware. The experimental thermal loads were extrapolated to engine similar conditions to obtain an estimate of realistic thermal loads. His goal was not to simulate specific flow physics, but to quantify the unsteady thermal loads interacting with the rotor blade surface.

MIT confirmed that the unsteady wakes and shocks produced using a rotating bar device was qualitatively similar to the unsteady wakes and shocks shed from the trailing edge of a nozzle guide vane. MIT also performed some of the earlier numerical work by solving the unsteady 2-D Euler equations (with various source terms) and reproducing the shock propagation event. This event primarily consisted of pressure contours and velocity vectors, which matched experimental observations well. But overall, a simple verification tool continues to elude those still studying this complex unsteady stator/rotor interaction.
The Oxford researchers noted that when a weak shock impacts a turbine blade, there was no instantaneous heat transfer observed. Instead, the shock propagates toward the leading edge, and creates a separation bubble (later referred to as a “vortical bubble”) which propagates downstream, pulling the heated flow toward the surface along the way. However, a strong shock created a momentary separation-reattachment phenomena, which resulted in an instantaneous drop in heat transfer. Again, the shock moved toward the leading edge creating the bubble, which propagated downstream resulting in a heat transfer rise.

Virginia Tech researchers have performed experiments using a shock tube to introduce a shock in isolation into a 2-D cascade wind tunnel test section. Gage locations on their instrumented blade each had Kulite pressure transducers and heat flux microsensors beside one another. During the unsteady runs, the rise in pressure suggested shock impact. Also, a rise in heat transfer occurred at the same time, suggesting that shock impact creates an instantaneous rise in heat transfer. Because of the difficulty in tripping the boundary layer to turbulence on this blade profile, it was suspected that the boundary layer remains laminar during this particular shock/boundary layer interaction. In addition, the shock impacting the blade surface normally produced a higher heat transfer level than shock impact having any other orientation.

This document seeks to take advantage of the idea that shock impact produces the highest heat transfer level when impacting a blade surface normally. Other researchers investigating unsteadiness on a turbine blade surface have been successful in predicting the pressure field. However, many have experienced difficulties trying to calculate the heat transfer in the presence of both shocks and wakes. The turbine environment has an extremely complex flow field that is complicated by the short duration in which the unsteadiness interacts with the blade surface. It is useful to simplify the problem in order to understand the flow physics. Implementation of some simplifications can enable the shock effects on the boundary layer to be examined in isolation without the additional complications due to the wake. The remainder of this document will describe an approach
that reduces the complexity of the computational flow field while attempting to retain the essential flow physics.

An accurate numerical simulation can enable the shock-induced unsteady heat transfer phenomena to be investigated over a short time duration (fractions of microseconds). This is the time it takes the shock/boundary layer interaction to occur. Available instrumentation is incapable of providing a detail picture of the flow physics during this short time interval. A numerical approach is necessary to provide the needed detail over such a short time period.