

Vane–Endwall Junction Optimization to Reduce Turbine Vane Passage Adiabatic Wall Temperatures

Andrew T. Lethander* and Karen A. Thole†

Virginia Polytechnic Institute and State University, Blacksburg, Virginia 24060

and

Gary Zess‡ and Joel Wagner§

Pratt & Whitney, United Technologies Corporation, East Hartford, Connecticut 06108

Secondary flows in airfoil passages have become increasingly important in the design of modern gas turbines as a result of several fundamental trends in gas turbine engine development. Driven to achieve higher engine efficiencies and specific thrust output, the gas turbine industry is continually pushing the envelope of maximum allowable turbine inlet temperature. Whereas many researchers have worked to gain an understanding of secondary flows and their effects on turbine heat transfer, very few have pursued ways to mitigate passively their detrimental effects. Although considerable attention has been given to techniques of secondary flow reduction to minimize the associated aerodynamic losses, improvement of the thermal environment for a turbine vane is sought. In particular, this objective is achieved through optimizing a fillet in the vane–endwall junction to minimize adiabatic wall temperatures. The approach taken is to employ a commercial optimization software package in conjunction with a computational fluid dynamics package in the design of the fillet. Results indicate that significant reductions in adiabatic wall temperature can be achieved through application of an optimized fillet.

Nomenclature

C	= true chord of turbine vane
C_p	= pressure coefficient, $(p_s - p_{s,\text{inlet}})/(\frac{1}{2}\rho U_{\text{inlet}}^2)$
D_{max}	= maximum extent of fillet normal to the vane surface
$F(X)$	= optimization objective function, Eq. (2)
$F_{\text{norm}}(X)$	= normalized optimization objective function, Eq. (3)
H_{max}	= maximum height of the fillet
n	= coordinate normal to inviscid streamline
P	= pitch
p_s	= static pressure
p_0	= total pressure
Re_{in}	= turbine cascade inlet Reynolds number, $U_{\text{inlet}}C/\nu$
S	= span of turbine vane
T	= static temperature
U, V, W	= absolute velocity components
U_{inlet}	= vane cascade inlet freestream velocity
u, v, w	= local flow plane, transformed velocity components
V_n	= normal velocity, $-u \sin \Psi_{\text{ms}} + v \cos \Psi_{\text{ms}}$
V_s	= streamwise velocity component, $u \cos \Psi_{\text{ms}} + v \sin \Psi_{\text{ms}}$
V_z	= spanwise velocity, w
X, Y, Z	= global, stationary, coordinate system
X	= vector of design variables
x, y, z	= local coordinate system
Δp_0	= nondimensional total pressure, $(p_0 - p_{0,\text{ms}})/(\frac{1}{2}\rho U_{\text{inlet}}^2)$
δ	= boundary layer thickness

ζ	= total pressure loss coefficient,
	$\left[\iint_{\text{inlet}} \rho u(p_0) dA - \iint_{\text{exit}} \rho u(p_0) dA \right] / \iint_{\text{inlet}} \rho u(p_{\text{dyn}}) dA$
η	= adiabatic effectiveness, $T_{\text{ms}} - T_{\text{aw}} / T_{\text{ms}} - T_{\text{min}}$
ν	= kinematic viscosity
ρ	= density
ϕ	= pitch angle, $\tan^{-1}(w/u)$
ψ	= yaw angle, $\tan^{-1}(v/u)$

Subscripts

aw	= adiabatic wall conditions
inlet	= inlet value at midspan
min	= minimum value
ms	= midspan values

Introduction

SECONDARY flows, which have become extremely important in the design of modern gas turbines, refer to the vortical flow structures that develop in airfoil passages. This development is a result of strong turning and nonuniform flow conditions at the inlet. Both redistribution of the thermal field and as much as 30% of the total pressure loss through a turbine passage are attributed to secondary flows. Although the development of the vortical structures originates in the endwall regions, the growth can be such that at the exit a large portion of an airfoil passage is affected.

A simple argument can be used to describe the origin of secondary flows in turbine passages. Consider an inviscid analysis with two different streamlines passing through a nozzle guide vane, one originating in the freestream and the other in the platform region. With the same circumferential pressure gradient imposed on both streamlines and knowing that this pressure gradient must be balanced by the centripetal acceleration, the two radii of curvature of the streamlines must be different. This difference occurs because the flow in the near-wall region is at a lower velocity than in the freestream. The radius of the streamline in the near-wall region will be smaller than that of the streamline in the freestream. The deviation in flow-path between these streamlines results in the development of the secondary flows.

Received 15 July 2003; revision received 22 December 2003; accepted for publication 25 February 2004. Copyright © 2004 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved. Copies of this paper may be made for personal or internal use, on condition that the copier pay the \$10.00 per-copy fee to the Copyright Clearance Center, Inc., 222 Rosewood Drive, Danvers, MA 01923; include the code 0748-4658/04 \$10.00 in correspondence with the CCC.

*Graduate Student, Mechanical Engineering Department; currently Turbine Research Engineer, Turbine Engine Division, Air Force Research Laboratory, 1950 Fifth Street, Wright–Patterson Air Force Base, OH 45433; Andrew.Lethander@wpafb.af.mil. Member AIAA.

†Professor, Mechanical Engineering Department, 102 Randolph Hall; thole@vt.edu. Member AIAA.

‡Designer, Turbine Aerodynamic Discipline Group.

§Technology Manager, Turbine Aerodynamic Discipline Group.

Many of the past studies on secondary flow effects have focused on the aerodynamic penalties resulting in total pressure losses through a stage. Arguably as important, however, are the thermal penalties resulting in reduced airfoil life. This is particularly true because increased turbine inlet temperatures are desired to achieve high thrust-to-weight ratios for aeroengines. Whereas all turbine components are cooled with compressor bleed air, it is particularly difficult to provide adequate thermal protection in the endwall regions due to the presence of secondary flows.

Unlike the majority of past studies, the primary focus of the research effort presented in this paper was to reduce the severity of the thermal environment that the first vane must endure. The approach to improving the thermal environment was to provide a fillet structure in the vane–platform juncture. In particular, optimization studies were performed to design a fillet in which coolant fluid from the upstream combustor liner was effectively used to cool the downstream vane endwall. The objective function for this optimization study was to minimize the adiabatic temperatures of the vane and its associated platform (endwall). Although the total heat transfer to a turbine component is dictated by both the local gas temperature and the convective heat transfer coefficient, the focus of our study was to minimize the local gas temperature. This optimization study does not account for the impact of the fillet geometry on the local convective heat transfer coefficient. The work discussed in this paper presents the first optimization study presented in the open literature in which reducing the adiabatic wall temperatures for a vane and its endwall were imposed as the objective. In addition, this study is unique because it is the first reported attempt in which two commercial software packages, FLUENT and iSIGHT, were mated to perform this optimization. Whereas FLUENT was used as the flow solver, iSIGHT was used to perform the optimization algorithms.

Following a review of the past literature, the optimization process and results from the optimization will be presented. Comparisons will then be made between the predicted flow and thermal fields for a vane having a manufacturing fillet (baseline) and a vane with an optimized fillet.

Previous Studies

Methods for reducing the secondary flows in turbine passages have ranged from simple fillet designs for two-dimensional airfoils to full three-dimensional endwall contour designs on three-dimensional airfoils. The aim of most past studies, as already mentioned, has been to reduce the aerodynamic losses and flow angle deviations at the passage exit. The open literature shows conflicting results in that some studies indicate an overall definitive reduction in losses, whereas other studies have shown overall increases in loss even though a local reduction in loss was concentrated near the unprofiled endwall. More recently, however, there have been a series of papers on endwall contouring indicating an overall reduction in losses and secondary flows using a methodical approach involving computational fluid dynamics (CFD) followed by experimental validation.

Three-dimensional endwall contouring was investigated computationally by Harvey et al.¹ and experimentally verified by Hartland et al.² To design the endwall contour, they used a linear sensitivity matrix in conjunction with superposition methods before applying an inverse design algorithm. Mathematically, perturbations to the cosine and sine functions that described the endwall were made to arrive ultimately at an optimum configuration. The results of the experimental verification confirmed a predicted reduction in exit flow angle deviations. Moreover, the experiments indicated a 30% reduction in loss, which was higher than predicted. One plausible reason for a disagreement between the experiments and computations may be that the turbulence levels (and/or unsteady vortex movement) were not accurately predicted with two-equation models. Turbulence is a large contributor to secondary losses.³ In a later study, Brennan et al.⁴ and Rose et al.⁵ applied similar computational and experimental (respectively) methodologies as Harvey et al.¹ and Hartland et al.² In Refs. 4 and 5, these methods were applied to a high-pressure turbine for a single stage in both the vane and blade passages. Stage efficiency improvements of

0.59% were reported, which exceeded the predicted improvement of 0.4%.

Sauer et al.⁶ investigated the application of a leading-edge “bulb” for secondary flow loss reduction. The objective of the leading-edge bulb was to increase the strength of the suction side leg of the horseshoe vortex, thereby affecting the passage vortex strength and its interaction with the airfoil boundary layers. For the best bulb design, Sauer et al. measured a 50% reduction in endwall total pressure loss. Computational results qualitatively agreed with their experimental measurements; however, no flowfield measurements within the blade passage were presented for comparison.

Although these previous studies have made detailed total pressure traverses and the work of Rose et al.⁵ included hot-wire traverses, Zess and Thole⁷ fully resolved mean and turbulent flow measurements for a fillet they applied to a large-scale model of a nozzle guide vane. They designed their fillet using the same CFD package as that reported in this paper with the goal of mitigating the formation of the horseshoe vortex. The fillet that they designed accelerated the boundary layer, thereby flattening the total pressure profile at the vane leading edge preventing development of the horseshoe vortex. Although several different fillet shapes and sizes were investigated using CFD, ultimately a linear fillet that was one boundary-layer thickness in height and extended two boundary-layer thicknesses axially upstream was predicted to be successful. On completion of the fillet design, flowfield measurements verified the performance of the leading edge fillet with no horseshoe vortex being present. Although the passage vortex did develop for the filleted vane, it was not as well developed as for the unfilleted vane. Good agreement between computational and experimental results was observed not only at the leading edge but also within the passage.

Becz et al.⁸ experimentally investigated the total pressure losses of a turbine vane with two leading-edge bulb geometries, similar to that presented by Sauer et al.,⁶ and a fillet geometry, similar to that of Zess and Thole.⁷ Though the large bulb produced a slight increase in total loss, both the small bulb and the leading-edge fillet reduced the area-averaged loss by 8%.

The only paper to have considered the endwall heat transfer effects has been that of Shih and Lin,⁹ who performed a computational study to evaluate two different fillet designs with and without inlet swirl. Their study indicated a reduction of heat transfer by more than 10% on the airfoil and by more than 30% on the endwall.

Both endwall contouring and leading-edge fillets show promise in reducing secondary flow losses; however, the potential impact of these technologies on endwall cooling from the combustor liner has not yet been considered. In this investigation, the shape of a vane leading-edge fillet was computationally optimized with the goal of minimizing adiabatic wall temperatures, which is one important aspect in reducing the overall endwall heat transfer. The differences between this study and that previously reported by Shih and Lin⁹ are that we are aiming at using combustor coolant for the endwall and are considering realistic combustor exit profiles.

Problem Approach

There are no known integrated commercial CFD/optimization packages, and as such, a significant effort in designing the fillet was devoted to this integration. The CFD package chosen for this study is a product of Fluent, Inc., called FLUENT.¹⁰ The primary reason for selecting FLUENT was because of the previous benchmark studies in which the secondary flowfield was correctly predicted for both unfilleted (Hermanson and Thole, 2000) and filleted¹¹ vanes. In addition, GAMBIT¹² (the mesh generator) has the ability to grid complex geometries using unstructured meshing techniques. The optimization software is a product of Engineous Software, Inc., and is called iSIGHT.¹³ The iSIGHT software is best described as an optimization shell that provides a number of different optimization algorithms.

CFD Simulations

Second-order accuracy was used in solving the Reynolds averaged Navier–Stokes (RANS) equations, energy equation, and turbulence model equations. Turbulence was modeled using the

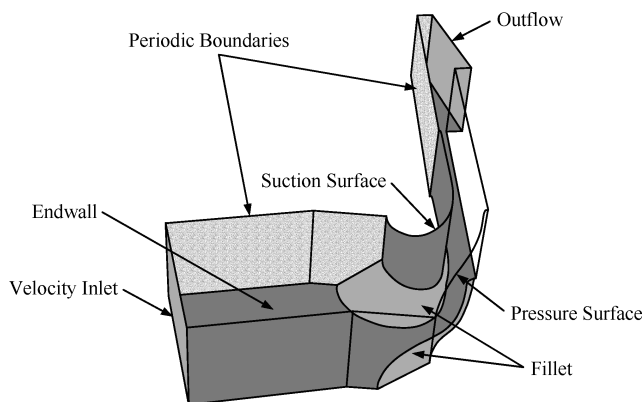


Fig. 1 Computational domain for CFD simulations.

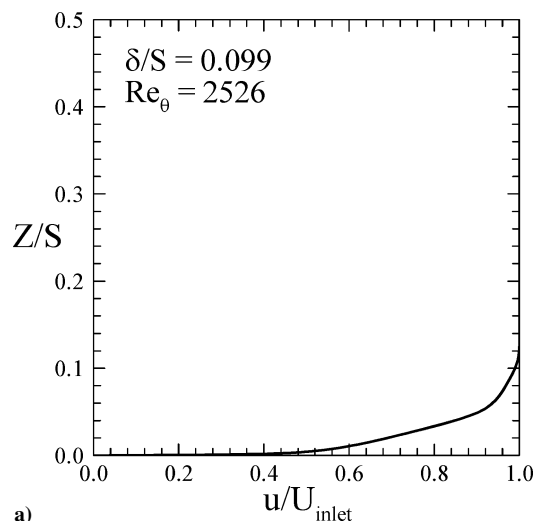
renormalization group (RNG) k - ϵ model¹⁴ with nonequilibrium wall functions to model the near-wall region.¹⁵ These turbulence models were applied because it was previously found that reasonably good predictions were made as compared with measured secondary velocities while still maintaining computational efficiency.⁷

The computational domain with a fillet is shown in Fig. 1. The turbine vane geometry used in this research is from an advanced commercial turbine and has been reported in the past by Radomsky and Thole^{16–18} and Zess and Thole.⁷ For the purpose of wind tunnel testing in a later study, the scaled-up vane geometry (nine times) was used for the computations while maintaining matched Reynolds number conditions, $Re_{in} = 2.3 \times 10^5$. The vane is two dimensional whereby the geometry is that of the commercial vane's midspan. For consistency with future experiments, a linear cascade was modeled with an inlet velocity of 6.3 m/s, $M = 0.02$. Because of the large-scale experiment, the Mach number was not matched, but the Reynolds number was matched.

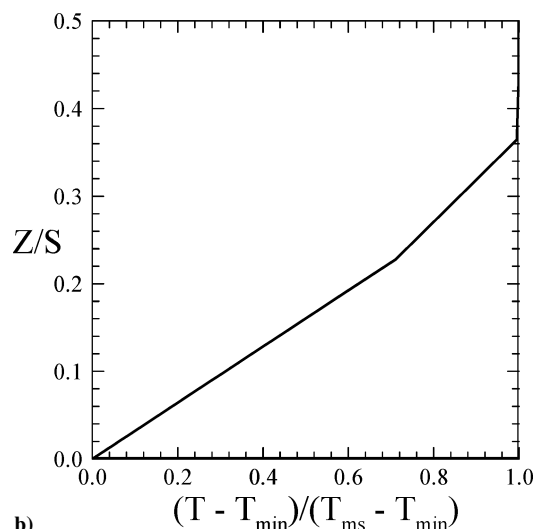
The inlet of the domain was located one chord upstream of the vane stagnation location, which was far enough upstream to eliminate the influence of the vane on the inlet conditions. The exit of the domain is located 1.5 chords from the vane trailing edge in the exit flow direction of 78 deg. To prevent highly skewed cells, an additional 0.1 chord was added in the axial direction. Periodic boundary conditions were placed on the side faces of the computational domain, whereas a symmetry boundary condition was applied at midspan. Adiabatic boundary conditions were applied to the endwall, fillet, and vane surfaces.

The mesh that was used was unstructured with typical sizes being 9×10^5 cells. The meshing strategy used was determined through grid sensitivity studies using the adaption capabilities of FLUENT. In assessing the grid sensitivity for the final fillet design, grid adaptations were completed using y^+ values and temperature gradients. The adaptations using y^+ values resulted in a grid size that was 1.2×10^6 cells. The larger grid gave a maximum difference in effectiveness, which occurred on the suction side of the airfoil, of less than $\Delta\eta = 0.005$. Adaptions using temperature gradients resulted in no detectable change in effectiveness. When the overall decrease in wall temperatures was computed, the change in the percentage improvement, where this percentage is defined by Eq. (4), was at most 0.27% between the two different meshes. The RANS, energy, and turbulence equations were computed until the residual values of the computations converged. The convergence of residuals for continuity, x momentum, y momentum, z momentum, k , and ϵ were resolved to levels of 10^{-4} with the exception of the energy equation, which was set to a level of 10^{-6} . Computations required 700 iterations for this convergence criteria. Increasing the number of iterations by 10% resulted in a negligible change in effectiveness levels.

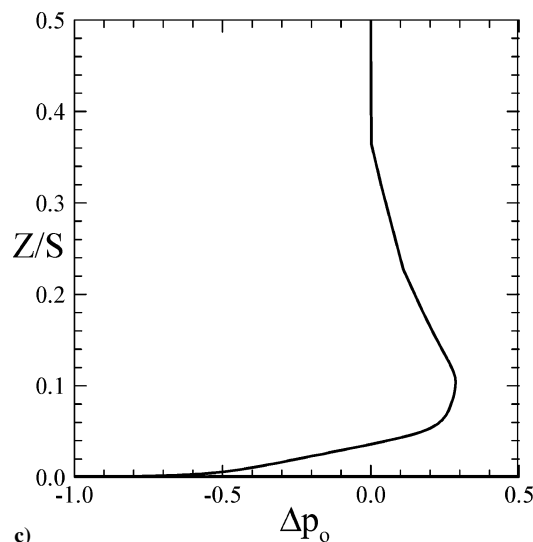
The inlet boundary conditions for this optimization test case were adapted from previous work reported by Hermanson and Thole¹¹ that included a temperature profile provided by the gas turbine industry. These inlet profiles are shown in Fig. 2. All boundary conditions were assumed to be uniform in the pitchwise (circumferential) direc-



a)



b)



c)

Fig. 2 Inlet radially varying boundary conditions of a) velocity, b) temperature, and c) total pressure.

tion. A two-dimensional boundary-layer code called TEXSTAN¹⁹ was used to generate spanwise (radial) profiles of velocity, turbulent kinetic energy, and turbulence dissipation rate for the inlet boundary layer. The velocity boundary-layer thickness was 9.1% of the vane span. Based on a turbulent boundary-layer profile and the temperature profile shown in Fig. 2b, the resulting total pressure profile approaching the vane is shown in Fig. 2c. The approaching total pressure profile peaks at nominally 10% of the span. This profile

results in a secondary flowfield in which there is a split at this location with flow being driven toward the platform and toward the midspan from this location, as will be shown in the results section of this paper.

Integration of Optimization Shell and CFD Package

Before the specifics of the integration of iSIGHT and FLUENT are discussed, it is instructive to consider how iSIGHT works in a generic sense. As an optimization shell, iSIGHT is designed to run simulation codes that are executed from a command line and use structured text files as input and output. Through iSIGHT's file parsing capability, the results of a simulation can be retrieved from the output file(s) and evaluated. Based on the results, iSIGHT then modifies the design parameters in the input file and continues to execute the simulation code. This process continues until iSIGHT converges on an optimum design. Guided by the problem definition and optimization routine, iSIGHT automates simulation code execution, data retrieval, and iterative adjustment of the design parameters. In the optimization effort reported in this paper, an exploratory optimization technique, called simulated annealing, was employed. Simulated annealing is considered an exploratory technique because it avoids focusing only on a local region of the design space. In this way, simulated annealing provides good coverage of the design space in its search for a global optimum.

To link iSIGHT with the FLUENT CFD package, extensive use was made of the journaling features of FLUENT and the FLUENT preprocessor, GAMBIT. Journaling allows for the creation of so-called journal files that can subsequently be specified as input to the simulation code and executed from the command line. A journal file is a simple text file that contains a sequential set of program commands that are to be executed. Thus, the journal files for GAMBIT and FLUENT serve as the simulation code input files. Using the file parsing capabilities of iSIGHT, design parameters specified in these files can be automatically altered between design iterations.

Because the goal was to optimize fillet shape, the solution domain was altered between CFD runs. This was achieved by creating a parametric model of the turbine vane leading-edge fillet and applying the model to geometry creation within the GAMBIT journal file. Fortunately, the journaling capabilities of GAMBIT allow for the specification of parameters and the use of parameters in mathematical expressions. By the specification of these geometric parameters to be design variables, iSIGHT has the ability to alter the shape of the fillet between simulation runs.

To minimize the number of design parameters for the leading-edge fillet, a mathematical model of the fillet was developed. The leading-edge fillet geometry and associated design parameters are shown in Fig. 3. A total of four design parameters was selected to describe the shape of the fillet. These parameters included the

maximum fillet height $H_{\max}$, the maximum fillet extent normal to the vane surface, $D_{\max}$, and the extent to which the fillet wraps around the vane pressure and suction surfaces, $s_{\max,ps}$ and $s_{\max,ss}$. The surface distance s is measured along the vane from the point of flow stagnation and is defined to be positive along the suction surface and negative along the pressure surface. The maximum fillet wrap around the vane is indicated by $s_{\max,ss}$ for the suction side and $s_{\max,ps}$ for the pressure side. One of the inherent constraints of this parametric description is that the maximum height and extent of the fillet are fixed at the vane dynamic stagnation (flow stagnation). For simplicity, a linear profile in the span direction was used. In addition to the leading-edge fillet, a standard manufacturing fillet was included in the computational model beyond the $s_{\max,ps}$ and $s_{\max,ss}$ locations of the vane.

The constraints applied to the fillet in this investigation were the following:

$$\begin{aligned} 0.084 \leq D_{\max}/C \leq 0.227, \quad & 0.045 \leq H_{\max}/S \leq 0.246 \\ 0.100 \leq s_{\max,ps}/s_{\text{total},ps} \leq 0.970 \\ 0.085 \leq s_{\max,ss}/s_{\text{total},ss} \leq 0.440 \end{aligned} \quad (1)$$

These constraints ensured that the vane passage gauge point remained unchanged and also ensured that the fillet size did not become impractical from a manufacturing standpoint.

Objective Function Definition and Evaluation

The goal of this investigation was to minimize the adiabatic wall temperatures experienced by the endwall and vane. The adiabatic wall temperature represents the local gas temperature that dictates the driving potential for heat transfer between the surface and gas for turbine components. Because these surfaces were modeled as adiabatic, definition of an objective function appeared to be straightforward. At the start of this optimization effort, the area-weighted average adiabatic wall temperature of the vane, endwall, and fillet was selected as the objective function. However, this objective function did not take into account possible local hot spots, which can be quite important in considering airfoil life. As such, minimizing the area-weighted average of the square of the adiabatic wall temperature was used as

$$F(X) = \frac{1}{A} \int_A T_{aw}^2 dA \quad (2)$$

The X vector in Eq. (2) represents the design variables that were considered for this study. The area of integration used in evaluating the objective function included the vane, fillet, and endwall surfaces.

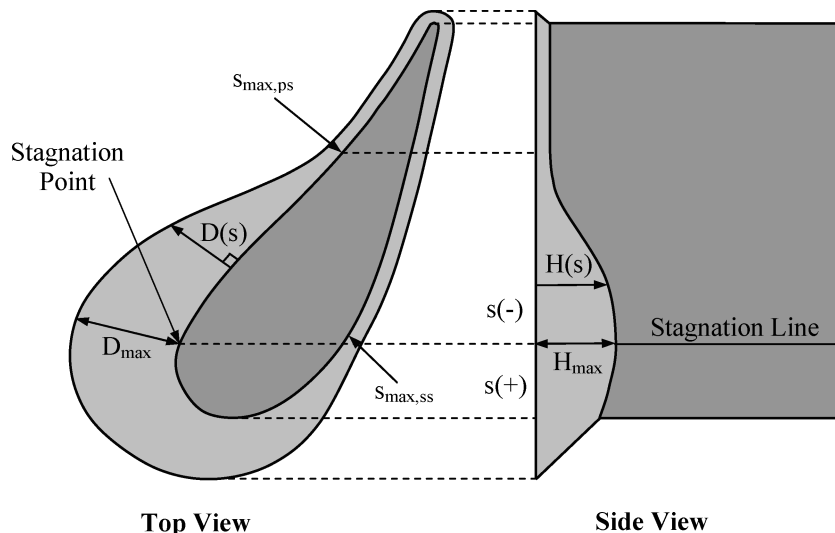


Fig. 3 Schematic of the leading edge fillet and associated geometric design parameters.

The area on the endwall that was considered extended from 50.6% of the axial chord upstream of the vane stagnation to 10% of the axial chord downstream of the vane trailing edge.

Optimization Results

Through analysis of the optimization results, the relative importance of the four fillet design variables was determined. During the optimization process, iSIGHT maintained a database of all 64 simulations that were performed, recording the design variables and the resulting value of the objective function for each. For comparison purposes, a baseline simulation was performed that only contained the manufacturing fillet (actual engine fillet radius of 1.3 mm). Although the baseline geometry included a standard manufacturing fillet, no notable effect of this fillet on the secondary flows was noted as compared to a vane with a sharp 90-deg juncture. The objective function [Eq. (2)] was normalized by the calculated value for the vane with a manufacturing fillet for determining the performance relative to the vane having no fillet,

$$F_{\text{norm}}(X) = \frac{1}{A} \iint_A T_{\text{aw}}^2 dA \bigg/ \frac{1}{A} \iint_A T_{\text{aw}}^2 dA \bigg|_{\text{baseline}} \quad (3)$$

The normalized objectives for the evaluated fillet designs are plotted against the design variables in Figs. 4a–4d. From Figs. 4, a general indication of the dependence of the objective on each design

variable can be obtained. Note that in Figs. 4 $F_{\text{norm}} < 1$ is indicated, which means that all of the cases indicate improved thermal performance of the vane. Figure 4a appears to show the most definitive trend and is that of the objective vs the maximum fillet height. Figure 4a clearly illustrates a reduction of the objective, the desired result, as the maximum fillet height increases. Also note the degree of scatter in the objective is shown to increase with increasing maximum fillet height. The amount of scatter is indicative of the sensitivity of the objective to the other design variables. To illustrate this, symbols in Figs. 4 are sized by maximum fillet extent. Figure 4b shows the next strongest correlation of improved thermal benefit with increasing maximum fillet extent. The scatter in the normalized objective is predominantly due to variations in maximum fillet height. Improved performance as a function of the fillet length around the suction side showed very little correlation. Although it is not shown here, similar results (very little correlation) occur as a function of fillet length around the pressure surface. Figure 4d shows the normalized objective function plotted against the combined suction and pressure side wrap around distances. The combined distances show much better correlation, indicating improved thermal performance with increasing fillet wrap.

Given that H_{max} and D_{max} provided the strongest correlation with the objective function, a contour plot of the objective was constructed over the design space that was considered, as shown in Fig. 5. Figure 5 clearly indicates that designs with large values of these two variables yielded the best fillet performance. Figure 5

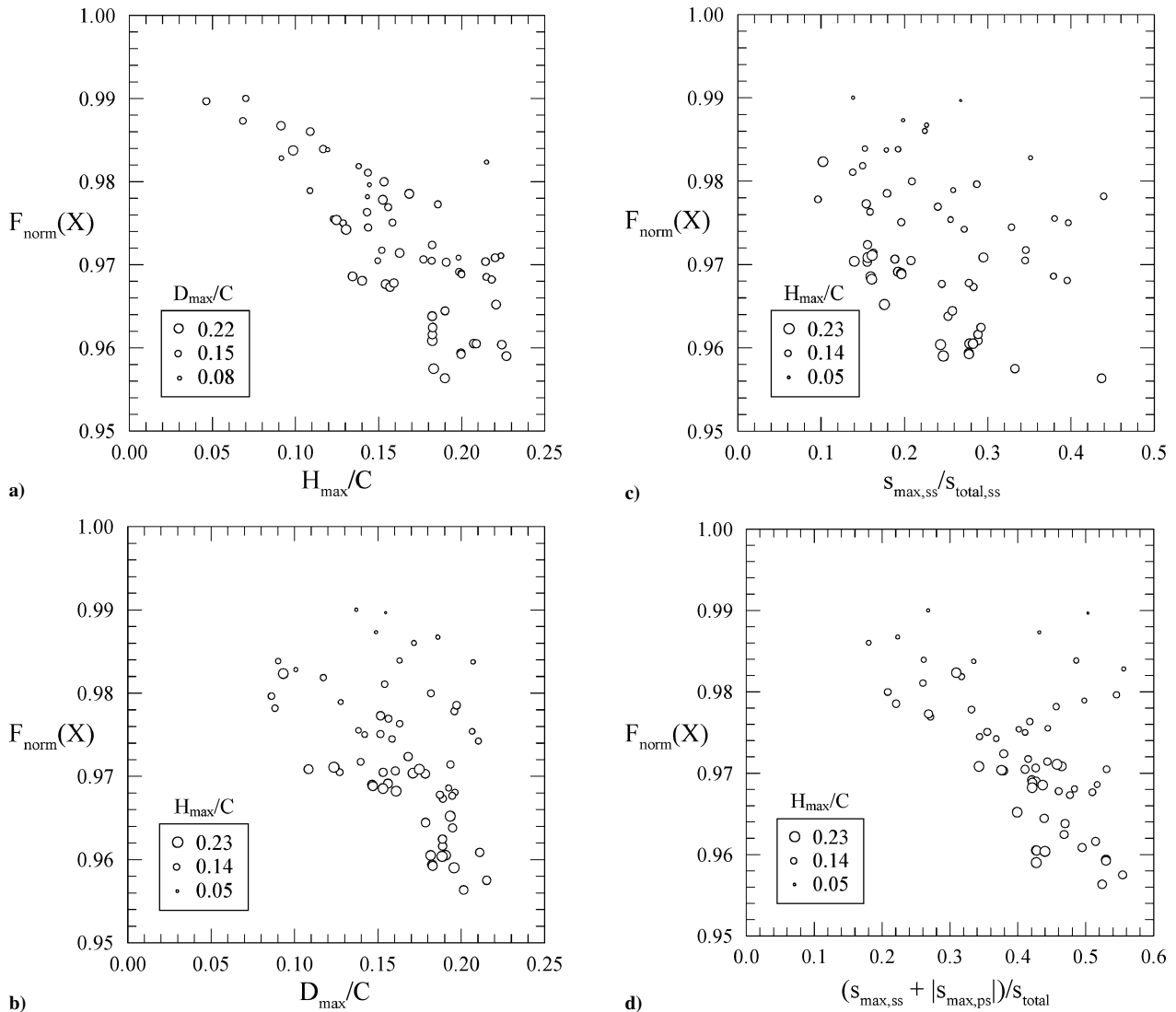
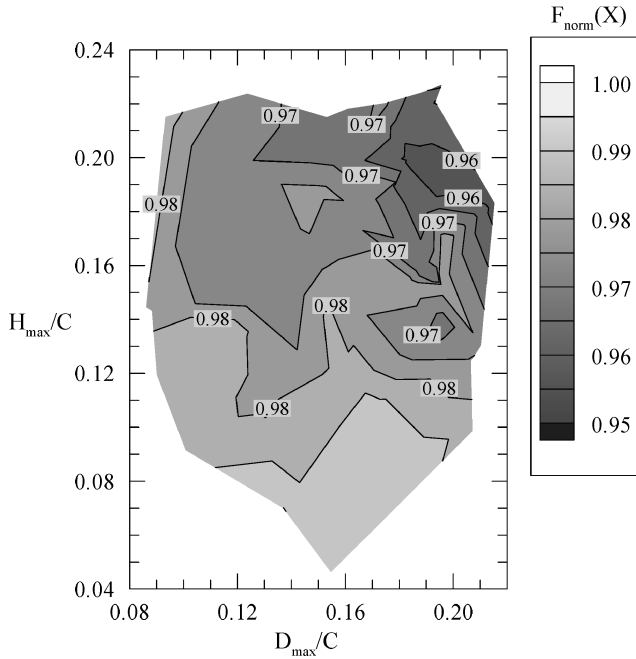


Fig. 4 Thermal benefit as affected by a) maximum fillet height, b) maximum fillet extent, c) extent along suction surface, and d) combined fillet extent along suction and pressure surfaces.

Table 1 Summary of optimized fillet geometry

Feature	Fillet size
$H_{\max}/S$	0.21
$D_{\max}/C$	0.20
$s_{\max,ss}/s_{\text{total},ss}$	0.44
$s_{\max,ps}/s_{\text{total},ps}$	0.65

**Fig. 5** Contours of thermal benefit as affected by maximum fillet height and extent normal to the vane.

indicates that improved performance can be achieved with the largest fillets that were considered in this study. Although the design space considered appears to be too confining, applying fillets larger than this size may be impractical due to interference with the upstream combustor.

Comparison of Baseline and Optimized Fillet Cases

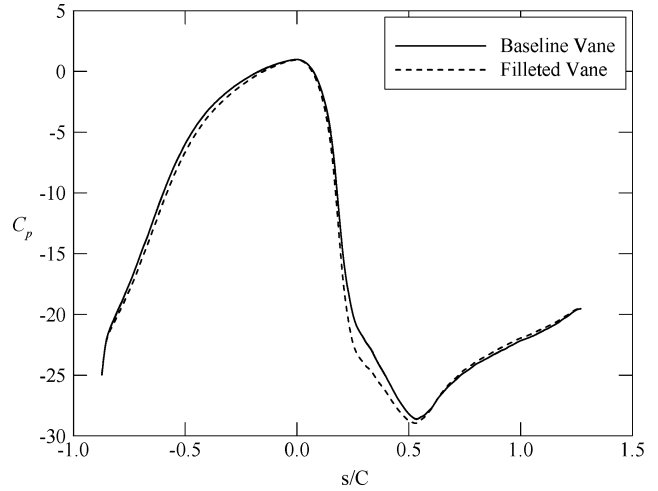
Comparisons of the optimized fillet case to that of the baseline were made to understand why an improved thermal performance was achieved and to determine the effects that the fillet had on the aerodynamic performance. The final fillet design had the geometric characteristics listed in Table 1. As noted from Table 1, the linear fillet chosen was very large as dictated by the optimization study.

The midspan static pressure distributions for the baseline and filleted vanes are shown in Fig. 6. The static pressure at the midspan is predicted to be slightly affected by the fillet over a small portion of the suction surface. Along the suction surface, the lower C_p values are a result of increased flow acceleration due to the fillet. To quantify the impact of the leading-edge fillet on the total pressure loss through the vane passage, loss coefficients were computed as defined in the Nomenclature. The calculated loss coefficients for the baseline and filleted vane cases are 1.53 and 1.55, respectively. Although these values are very close, note that, as one progresses through the vane passage, higher losses occur in the leading-edge region for the filleted vane relative to the vane without the fillet. In the downstream portion of the vane passage, the losses in the case without the fillet are higher than that of the filleted vane, which can be attributed to the reduction in secondary flows for the filleted vane. This distribution of loss results in loss coefficients that are nearly the same for both vanes.

Streamlines released upstream of the vane for both the baseline and filleted vanes are shown in Figs. 7a and 7b. The streamlines for the baseline case clearly show rollup of the approach boundary layer into a leading-edge horseshoe vortex. The streamlines also in-

Table 2 Summary of benefit for the optimized fillet

Geometric feature	% Change in temperature, Eq. (4)	% Change in maximum temperature	% Change in surface area
Fillet suction surface	5.1	1.1	205
Fillet pressure surface	10.5	4.4	213
Vane suction surface	4.1	0	-8.1
Vane pressure surface	4.6	0	-12.2
Endwall	3.2	7.5	-37.3
Net reduction	6.3		-2.7

**Fig. 6** Comparison of the predicted pressure distributions for the baseline unfilleted vane and optimum filleted vane.

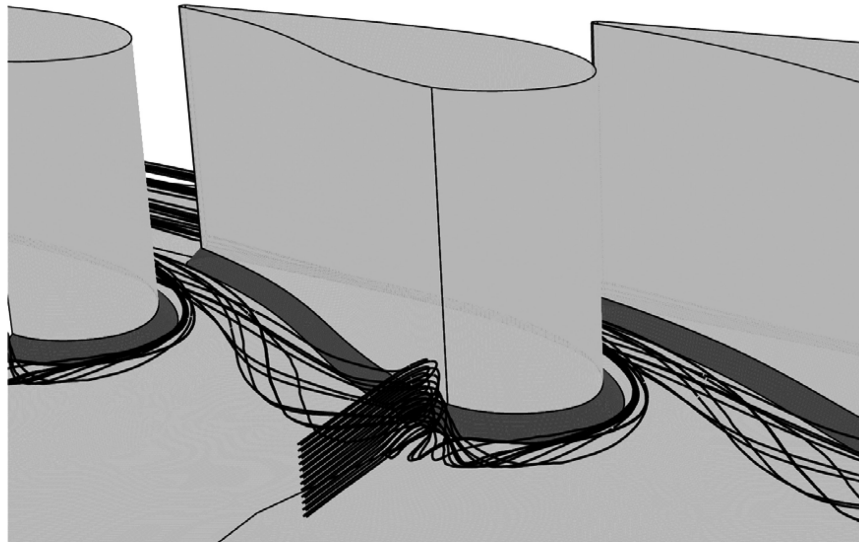
dicate development of the passage vortex through the vane row. For the optimized fillet case, however, the streamline pattern indicates that the flow is following the contour of the fillet, showing no indication of horseshoe vortex development. The fillet accelerates the lower momentum boundary-layer flow in the spanwise direction, preventing the higher momentum mainstream flow from turning down toward the endwall. In addition to eliminating the horseshoe vortex, development of a passage vortex is not evident from the streamlines.

Normalized adiabatic wall temperatures are shown in Figs. 8a and 8b for the baseline and for the optimized filleted vanes. Lower adiabatic wall temperatures for the filleted vane are indicated in Fig. 8b relative to the baseline. These lower temperatures occur not only in the leading edge region but throughout the entire passage.

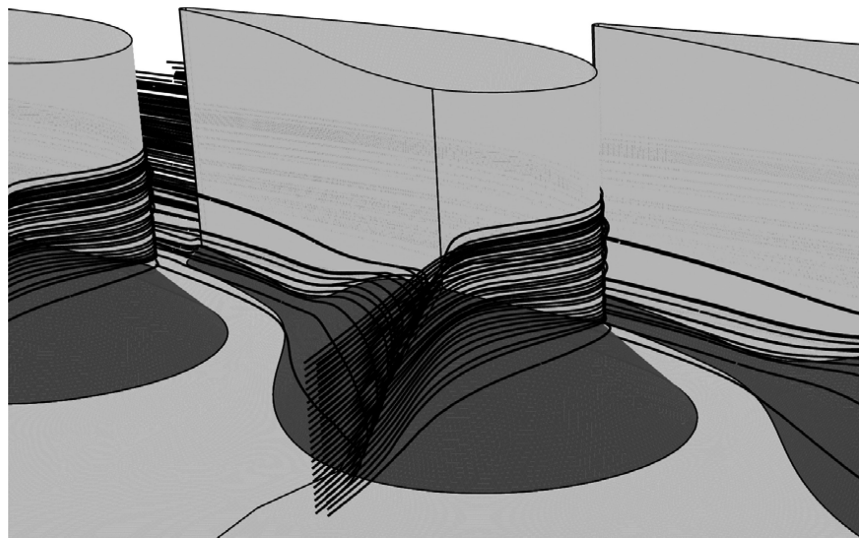
Table 2 quantifies the actual reduction in the adiabatic wall temperatures for the filleted vane surface. As such, the area-weighted temperatures will be presented in the remainder of this paper. The two quantities given in Table 2 are for the square of the average temperature and one for the maximum temperature. The percent reduction is based on the ratio of the differences of adiabatic wall temperatures for the filleted vane to that of the baseline geometry relative to the total driving temperature potential ($T_{ms} - T_{min}$) as

$$\text{Thermal Benefit(\%)} = \frac{\sqrt{T_{aw}^2}|_{\text{baseline}} - \sqrt{T_{aw}^2}|_{\text{fillet}}}{T_{ms} - T_{min}} \quad (4)$$

where $\sqrt{T_{aw}^2}$ is the area-weighted square of the adiabatic wall temperature. The midspan to minimum driving temperatures remained the same for the filleted and baseline cases. Although the objective function used was the area-weighted average of the square of the wall temperature, the results were nominally the same when calculated using the area-weighted average wall temperature. Note that Table 2 indicates each individual surface that was considered. For example, the pressure fillet surface has a 213% increase in area because the comparison is being made between the manufacturing

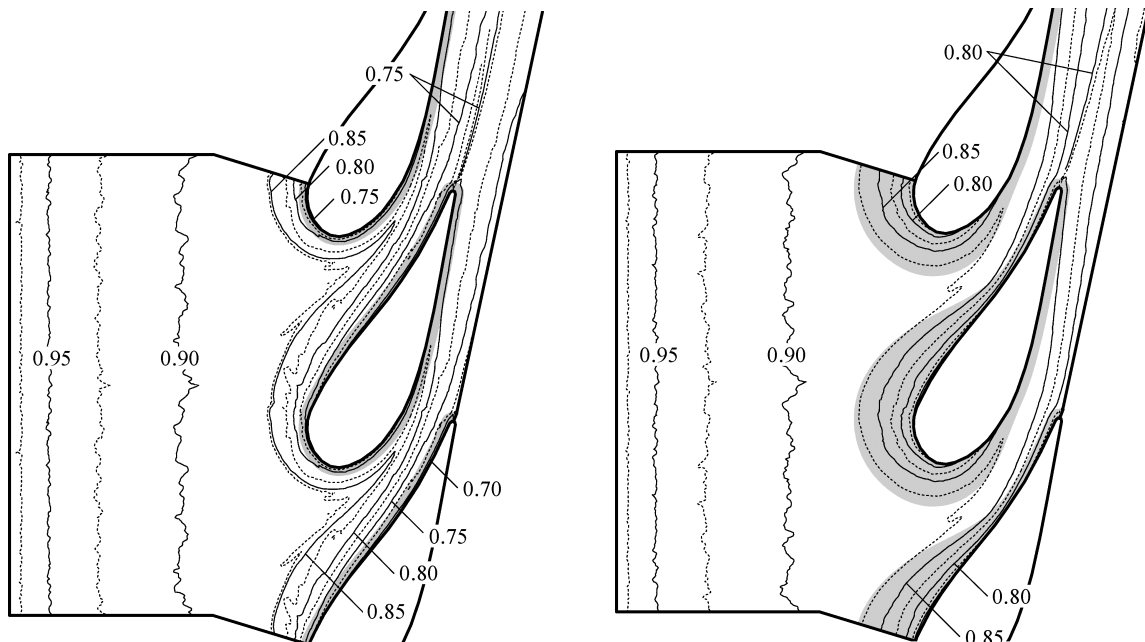


a)



b)

Fig. 7 Comparison of streamline patterns for the baseline and optimized filleted vane.



a)

b)

Fig. 8 Comparison of endwall adiabatic effectiveness levels: a) baseline and b) optimized filleted vane; shaded regions, fillet.

fillet and the optimized fillet, which is significantly larger. For the filleted vane, the total surface area was reduced (including vane, fillet, and endwall) by 2.65% relative to the baseline. Reduced surface area occurs as the fillet becomes larger because the hypotenuse of a triangle is shorter than the combined length of the other two sides. This reduction in surface area results in less area to protect thermally, and thus, less coolant is needed.

The component showing the highest reduction in temperature was the fillet along the pressure surface, which was 10.5%. The net thermal benefit considering all of the surfaces was a 6.3% reduction in area-weighted average adiabatic wall temperatures. The reduction in the highest peak temperature occurred on the endwall itself, which is particularly important, given that this area is quite hard to cool. In relating these numbers to an engine, consider a main gas path temperature T_{ms} of 1700°C, a minimum near-wall gas temperature T_{min} of 600°C, and a metal temperature of 1250°C (assume to be T_{aw}). Note that the minimum near-wall gas temperature represents a

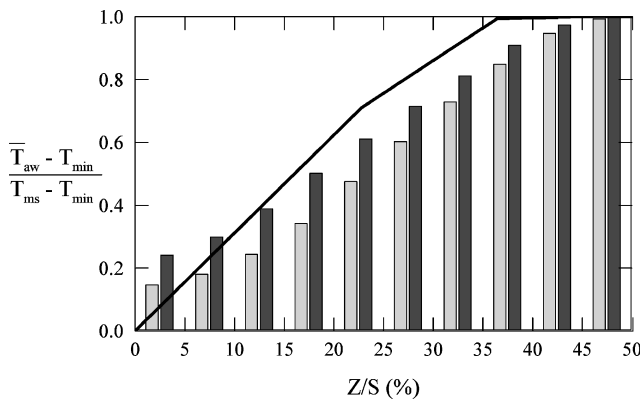


Fig. 9 Comparison of area-weighted average adiabatic wall temperatures at each span position: ■, filleted vane; ■, baseline vane; and —, inlet temperature profile.

mixed-out coolant temperature of the combustor liner coolant flow that enters the turbine vane. If the average adiabatic wall temperature were reduced by 6.3% for the same local heat transfer coefficient, it would result in a decrease of metal temperature for the filleted vane by 70°C, giving a metal temperature of 1180°C. Although it can not be assumed that the heat transfer coefficient remains the same for filleted and baseline vanes, the reduction in the local gas temperature indicated by these studies would significantly increase component life.

Although Table 2 presents a thermal comparison between the baseline and filleted vanes, it is difficult to make a one to one comparison because the areas are also changing. For example, consider a large fillet where the vane and endwall surface areas are significantly reduced while the fillet surface areas are significantly increased relative to the baseline geometry. To further assess the thermal improvement of the fillet, area-weighted average adiabatic wall temperatures as a function of the vane span position were computed for the vane. Note that these averages were computed over a region that extended 5% of the vane span for the entire surface around the vane with the exception of the $z/S = 0$ location in which the endwall surface was averaged. Figure 9 presents these nondimensional temperatures for the baseline and filleted vanes along with the temperature profile at the inlet to the vane passage. These results indicate that there is not only an effect on the endwall but that there is also a thermal benefit along the vane span. The largest difference between the baseline and filleted vanes (peak benefit) occurred at the 20% span location, which is above the maximum height of the fillet.

Flow and Thermal Field Comparisons

Several planes defined perpendicular to the vane, as shown in Fig. 10, were analyzed for comparisons between the baseline and filleted vane. For all planes except plane SP (Fig. 10b), coordinate and velocity transformations, previously described by Kang and Thole,²⁰ were used to calculate the secondary flows. This transformation is shown in Fig. 10 with the variables defined in the Nomenclature. Because plane SP is oriented parallel to the incoming flow, no transformation was necessary.

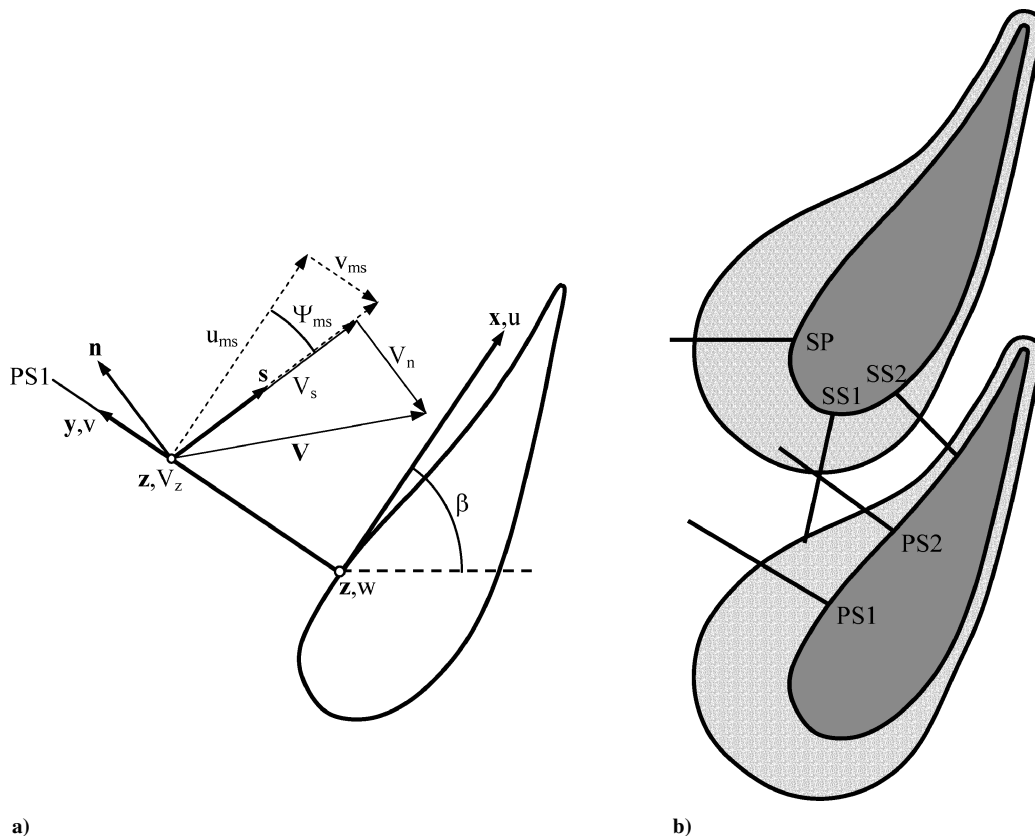


Fig. 10 Velocity vector and coordinate transformations performed in calculation of the secondary flow field and locations of flow analysis planes.

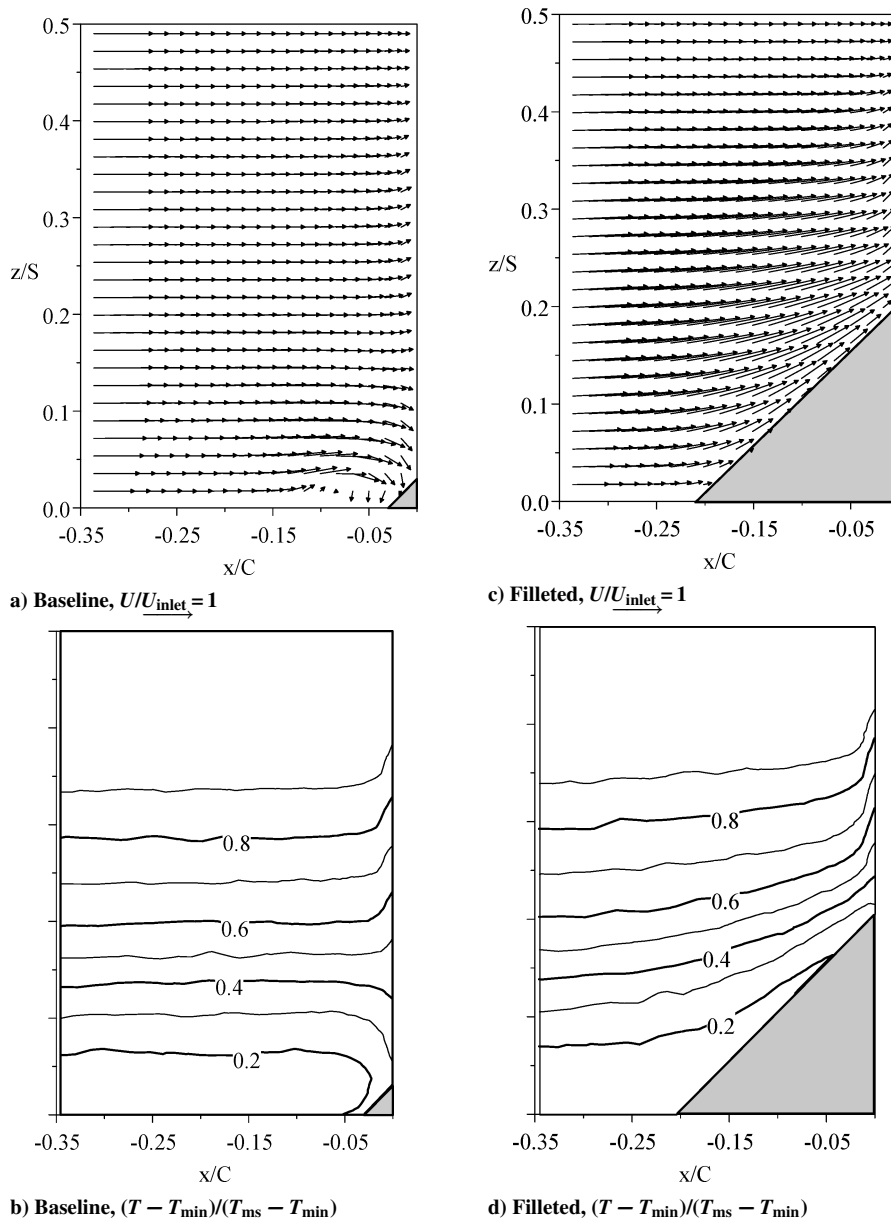


Fig. 11 Stagnation plane velocity vectors and temperature contours.

The typical leading-edge vortex is the result of a spanwise total pressure gradient due to the lower momentum boundary-layer flow on the endwall. Higher total pressure away from the endwall drives flow toward the vane–endwall junction resulting in a vortex. Figures 11a–11d show a comparison of the flow results between the baseline geometry and the best performing fillet in the SP plane. In addition to velocity vectors, contours of nondimensional flow temperature are also presented to illustrate the influence of the fillet on the approaching thermal field. For the baseline manufacturing fillet case, the velocity vectors clearly show the formation of a horseshoe vortex. Also shown is a distinct flow split on the vane leading edge at approximately 15% span. Above 15% span, the flow is observed to turn upward toward midspan, whereas below 15% span the flow turns downward toward the endwall. Coincident with the horseshoe vortex is a distortion of the thermal field with higher temperature fluid being pulled toward the vane–endwall junction. In contrast to this flow situation, the results for the best fillet simulation illustrate the advantages of fillet application. These results show an upturning of the flow along the surface of the fillet and the absence of a leading-edge horseshoe vortex. This acceleration of the boundary-layer fluid in the spanwise direction enables the flow to overcome the total pressure gradient responsible for horseshoe vortex devel-

opment. One of the advantages of the fillet design is the resulting thermal field. Unlike the baseline case, the thermal field indicates a positive migration of cooler boundary-layer fluid being driven up the fillet. This has the effect of spreading cooler fluid over a larger area of the vane and endwall and prevents penetration of hotter fluid to these surfaces.

Proceeding along the pressure surface of the vane, the flow and thermal field results for the PS2 plane are presented in Figs. 12a–12d. For the baseline case, growth of the pressure side leg of the horseshoe vortex is apparent, as it begins to develop and merge with cross passage boundary-layer fluid to form the passage vortex. Similar to the results in the SP plane, there is significant distortion of the thermal field due to the secondary flows. The results for the optimized fillet case do not show any definitive vortex structure; however, there are secondary flows that are present. There is a down turning of the flow below 18% of the span along the pressure side of the vane. Despite this down-turning flow, there is still relatively cool fluid along most of the endwall. In comparison with the baseline case, regions of hotter fluid are significantly skewed toward the vane midspan.

Proceeding from the stagnation plane along the suction surface, the flow and thermal field results for SS2 are given in Figs. 13a–13d

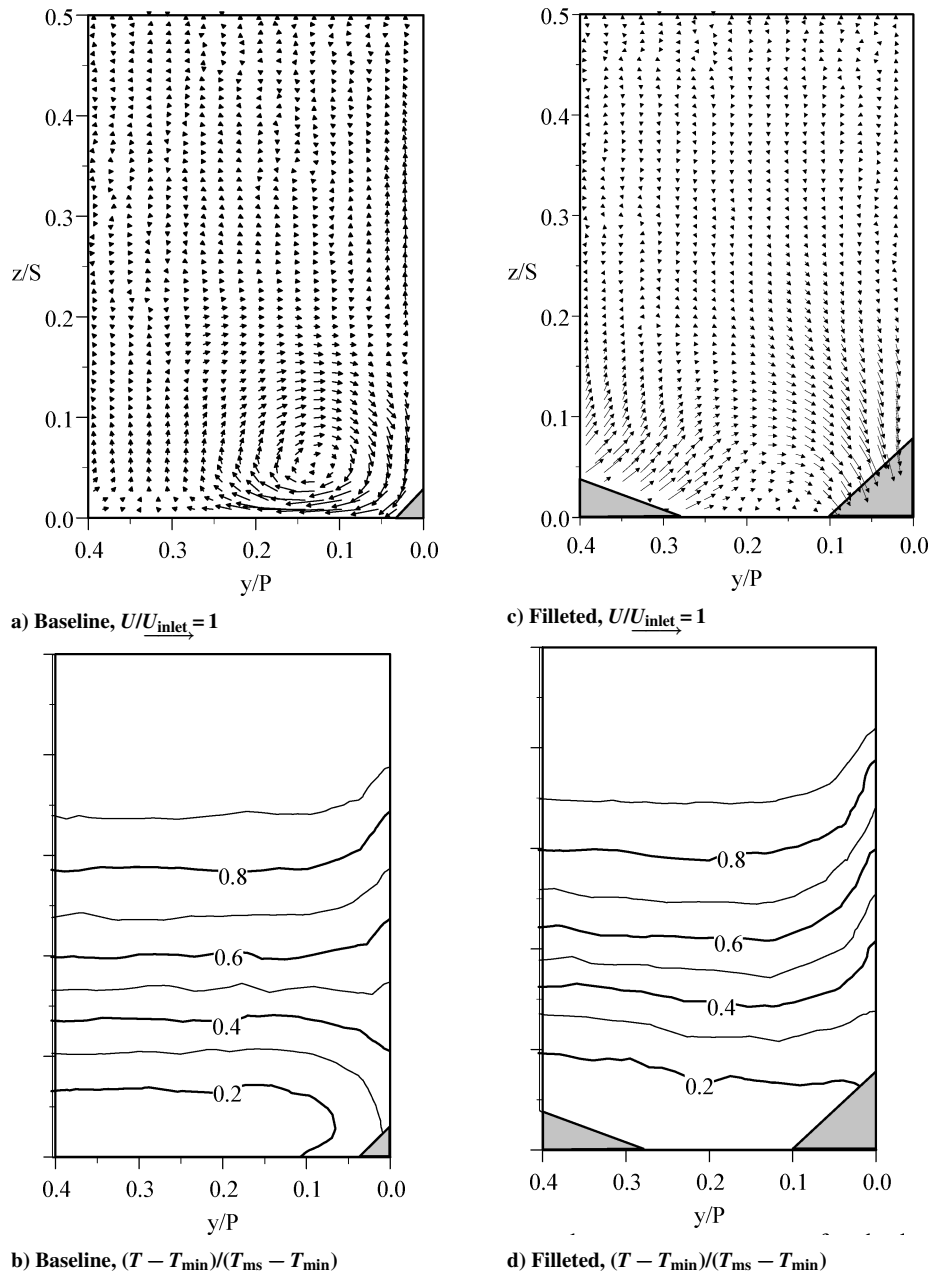


Fig. 12 Calculated secondary velocity vectors and temperature contours in plane PS2.

for the baseline and filleted case. The results for the baseline case show the presence of the passage vortex. At this location, any remnants of the suction side leg of the horseshoe vortex have nearly dissipated. The thermal field indicates the accumulation of colder flow toward the suction surface. This accumulation is explained by the strong cross passage pressure gradient and action of the passage vortex. The results for the filleted vane do not indicate the presence of a passage vortex; however, the presence of a strong cross passage flow is clearly evident. Throughout a large portion of the plane, a general down turning of the flow is shown. This is believed to be the result of diminishing fillet size through the passage. As the fillet decreases in height, the streamlines follow the contour of the fillet as shown in Fig. 7b. The effect of the strong cross passage flow on the thermal field is also evident in the plot with the migration of colder fluid down the pressure surface and toward the suction surface. Although this distortion is unfavorable, the results for the filleted case still maintain an advantage over the baseline case.

The presence of the fillet results in an improved thermal field, prevents the formation of a leading-edge horseshoe vortex, and appears to eliminate full development of a passage vortex. From an aero-

dynamic perspective, there is a slight increase in the total pressure loss and a deviation of the flow angles. Yaw and pitch flow angles are shown in Figs. 14a–14d for the filleted and unfilleted vanes in the SS2 plane. Note that the yaw angle contours are presented with respect to the flow turning at the vane midspan.

The results for the baseline case, shown in Figs. 14a and 14b, clearly indicate the presence of the passage vortex. The positive values for ϕ , representing the pitch angle, and negative deviation angles for $(\psi - \psi_{\text{ms}})$, representing the yaw, clearly both indicate the presence of the passage vortex for the baseline case. With the fillet, negative deviation of $(\psi - \psi_{\text{ms}})$ is virtually eliminated in contrast to the baseline case. The larger positive deviation angles for the yaw adjacent to the fillet are due to the flow following the fillet surface. At this location, the fillet is tapering into the vane suction surface; therefore, the near-wall fluid has high positive yaw deviation angles. The filleted case indicates negative values of pitch in the near-wall region as a result of the fillet tapering into the end-wall. These flow angles result in better thermal performance for the combined endwall, vane, and fillet surfaces as was indicated in Fig. 13d.

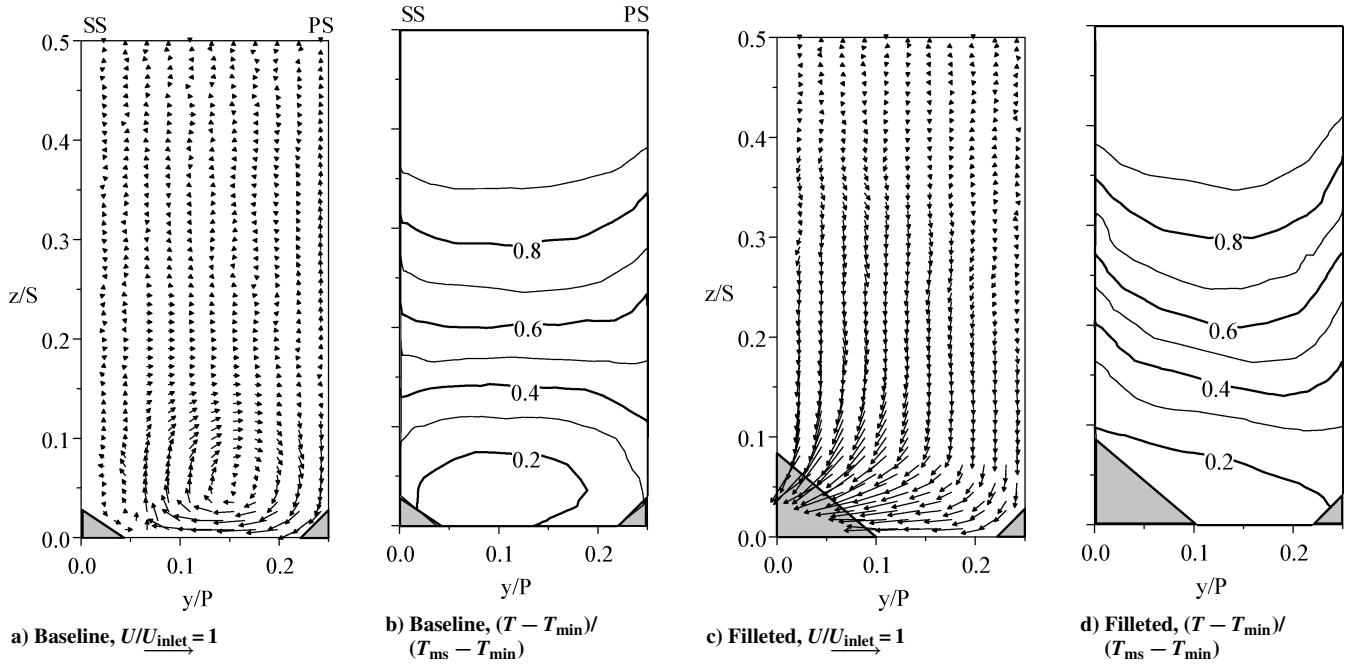


Fig. 13 Calculated secondary velocity vectors and temperature contours in plane SS2.

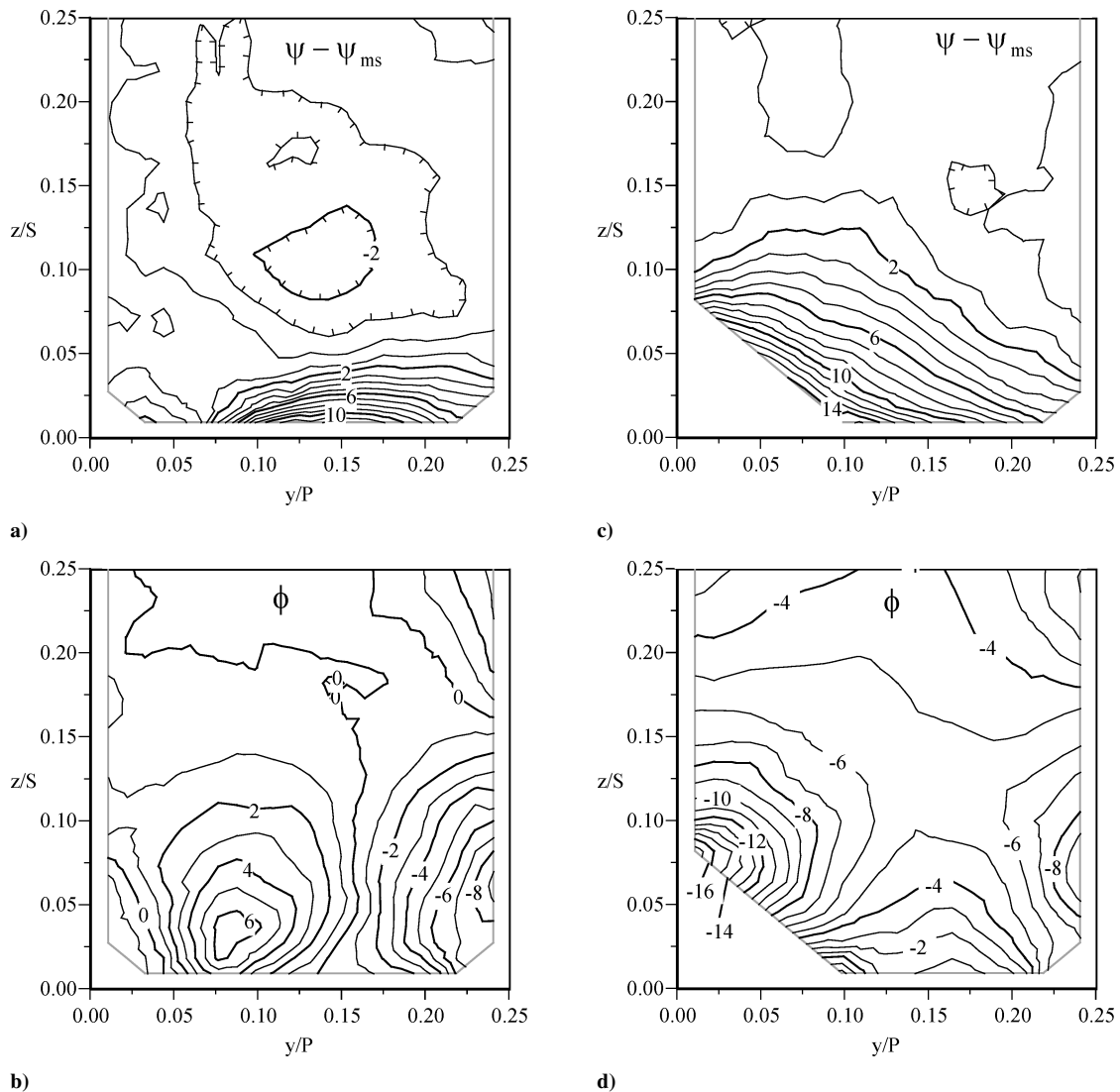


Fig. 14 Relative yaw and pitch angles in the SS2 plane for vanes: a) $\psi - \psi_{\text{ms}}$ baseline, b) ϕ baseline, c) $\psi - \psi_{\text{ms}}$ filleted, and d) ϕ filleted.

Conclusions

Although several different techniques of secondary flow reduction have been presented in the open literature, to date the aerodynamic benefits of these reduction techniques have been the primary focus. Based on the work presented in this paper, it is believed that a thermal benefit can also be derived from the use of these techniques and that these benefits would have an impact on gas turbine development. The objective of this investigation was to optimize the shape of a vane leading-edge fillet to maximize the thermal benefit of the fillet application. These results demonstrate the merit of using a combined approach of CFD and optimization algorithms.

The optimization results indicated that a large fillet was needed for the maximum thermal benefit and that this benefit was most influenced by the height of the fillet. Although a thermal benefit was predicted for the endwall, the maximum predicted benefit occurred on the vane itself at the 20% span location. Analysis of the filleted vane indicated that three benefits were realized with a fillet. First, there is a reduction in the overall surface area that needs to be cooled when considering the combined fillet, endwall, and vane surfaces. Second, there is a reduction of the secondary flows resulting from the acceleration of the flow in the endwall region, which prevents the flow from separating and forming a leading-edge horseshoe vortex. Note that the pressure side leg of the leading-edge vortex promotes passage vortex formation. By the prevention of formation of the leading-edge vortex, passage vortex development is delayed and vortex strength is reduced. The acceleration caused by the fillet in the leading-edge region leads to a flat total pressure profile along the vane span. Third, a reduction of the secondary flows for the filleted vane allows cooler near-wall fluid to remain along the endwall and fillet surfaces rather than being lifted into the hot mainstream, which occurs when a strong passage vortex is present.

Acknowledgments

The authors would like to acknowledge Pratt & Whitney, United Technologies, for their support of this work. We would also like to thank the U.S. Air Force Research Laboratory, Turbine Branch, and Matthew Meininger, in particular, for support of this work.

References

- ¹Harvey, N. W., Rose, M. G., Taylor, M. D., Shahpar, S., Hartland, J., and Gregory-Smith, D. G., "Non-Axisymmetric Turbine End Wall Design: Part I-Three-Dimensional Linear Design System," *Journal of Turbomachinery*, Vol. 122, No. 2, 2000, pp. 278–285.
- ²Hartland, J. C., Gregory-Smith, D. G., Harvey, N. W., and Rose, M. G., "Non-Axisymmetric Turbine End Wall Design: Part II-Experimental Validation," *Journal of Turbomachinery*, Vol. 122, No. 2, 2000, pp. 286–293.
- ³Gregory-Smith, D. G., and Cleak, J. G. E., "Secondary Flow Measurements in a Turbine Cascade with High Inlet Turbulence," *Journal of Turbomachinery*, Vol. 110, No. 1, 1992, pp. 1–8.
- ⁴Brennan, G., Harvey, N. W., Rose, M. G., Fomison, N., and Taylor, M. D., "Improving the Efficiency of the Trent 500 HP Turbine Using Non-Axisymmetric End Walls: Part 1: Turbine Design," *Journal of Turbomachinery*, Vol. 125, No. 3, 2003, pp. 497–504.
- ⁵Rose, M. G., Harvey, N. W., Seaman, P., Newman, D. A., McManus, D., "Improving the Efficiency of the TRENT 500 HP Turbine Using Non-Axisymmetric End Walls: Part 2: Experimental Validation," American Society of Mechanical Engineers, ASME Paper 2001-GT-0505, June 2001.
- ⁶Sauer, H., Müller, R., and Vogeler, K., "Reduction of Secondary Flow Losses in Turbine Cascades by Leading Edge Modifications at the Endwall," *Journal of Turbomachinery*, Vol. 123, No. 2, 2001, pp. 207–213.
- ⁷Zess, G. A., and Thole, K. A., "Computational Design and Experimental Evaluation of Using a Leading Edge Fillet on a Gas Turbine Vane," *Journal of Turbomachinery*, Vol. 124, No. 2, 2002, pp. 167–175.
- ⁸Becz, S., Majewski, M., and Langston, L., "Leading Edge Modification Effects on Turbine Cascade Endwall Loss," American Society of Mechanical Engineers, ASME Paper GT2003-38898, June 2003.
- ⁹Shih, T. I.-P., and Lin, Y.-L., "Controlling Secondary-Flow Structure by Leading-Edge Airfoil Fillet and Inlet Swirl to Reduce Aerodynamic Loss and Surface Heat Transfer," *Journal of Turbomachinery*, Vol. 125, No. 1, 2003, pp. 48–56.
- ¹⁰FLUENT User's Guide, Ver. 5.0, Fluent, Inc., Lebanon, NH, 1998.
- ¹¹Hermanson, K. S., and Thole, K. A., "Effect of Inlet Conditions on Endwall Secondary Flows," *Journal of Propulsion and Power*, Vol. 16, No. 2, 2000, pp. 286–296.
- ¹²GAMBIT I Modeling Guide, Fluent, Inc., Lebanon, NH, 1998.
- ¹³iSIGHT Designer's Guide, Ver. 5.0, Engineous Software, Inc., Cary, NC, 1999.
- ¹⁴Yakhot, V., Orszag, S., Thangman, S., Gatski, T. B., and Speziale, C. G., "Development of Turbulence Models for Shear Flows by a Double Expansion Technique," *Physics of Fluids A*, Vol. 4, No. 7, 1992, p. 1510.
- ¹⁵Kim, S.-E., and Choudhury, D., "A Near-Wall Treatment Using Wall Functions Sensitized to Pressure Gradient," *Separated and Complex Flows*, ASME FED Vol. 217, American Society of Mechanical Engineers, Fairfield, NJ, 1995.
- ¹⁶Radomsky, R. W., and Thole, K. A., "Highly Turbulent Flowfield Measurements Around a Stator Vane," *Journal of Turbomachinery*, Vol. 122, No. 2, 2000, pp. 255–262.
- ¹⁷Radomsky, R. W., and Thole, K. A., "High Freestream Turbulence Effects on Endwall Heat Transfer for a Gas Turbine Stator Vane," *Journal of Turbomachinery*, Vol. 122, No. 4, 2000, pp. 699–708.
- ¹⁸Radomsky, R. W., and Thole, K. A., "Detailed Boundary Layer Measurements on a Turbine Stator Vane at Elevated Freestream Turbulence Levels," *Journal of Turbomachinery*, Vol. 124, No. 1, 2002, pp. 107–118.
- ¹⁹Crawford, M. E., "Simulation Codes for Calculation of Heat Transfer to Convectively-Cooled Turbine Blades," edited by T. Arts, Lectures in Convective Heat Transfer and Film Cooling in Turbomachinery, Lecture Series 1986-06, von Karman Inst. for Fluid Dynamics, Rhode-Saint-Genese, Belgium.
- ²⁰Kang, M. B., and Thole, K. A., "Flowfield Measurements in the Endwall Region of a Stator Vane," *Journal of Turbomachinery*, Vol. 122, No. 3, 2000, pp. 458–466.