OPTIMIZATION OF ENDWALL CONTOURS OF A TURBINE BLADE ROW USING AN ADJOINT METHOD

Jiaqi Luo Department of Fluid Mechanics Northwestern Polytechnical University, Xi'an 710072, China Email: jiaqil81@gmail.com Feng Liu Department of Mechanical and Aerospace Engineering University of California, Irvine, CA92697-3975 Ivan McBean

Alstom Power (Switzerland) Baden, Switzerland

ABSTRACT

This paper presents the application of a viscous adjoint method in the optimization of a low-aspect-ratio turbine blade through spanwise restaggering and endwall contouring. A generalized wall-function method is implemented in a Navier-Stokes flow solver coupled with Menter's SST k- ω turbulence model to simulate secondary flow with reduced requirements on grid density. Entropy production through the blade row combined with a flow turning constraint is used as the objective function in the optimization. With the viscous adjoint method, the complete gradient information needed for optimization can be obtained by solving the governing flow equations and their corresponding adjoint equations only once, regardless of the number of design parameters. The endwall profiles are contoured alone in the first design case, while it is combined with spanwise restaggering in the second design case. The results demonstrate that it is feasible to reduce flow loss through the blade redesign while maintaining the same mass-averaged flow turning by using the viscous adjoint optimization method. The performance of the redesigned blade is calculated and compared at off-design conditions.

NOMENCLATURE

- a Speed of sound
- A_i Jacobian matrices, $A_i = \frac{\partial f_i}{\partial W}$
- B Boundaries of ξ domain

- c_p Constant pressure specific heat
- D The computational domain (ξ domain)
- dz Variation of blade height for the contoured endwall
- f_i Inviscid flux
- f_{vj} Viscous flux
- I Cost function
- *k* Turbulent kinetic energy; Thermal conductivity
- K_{ij} Transformation functions between the physical domain and the computational domain, $K_{ij} = \frac{\partial x_i}{\partial \xi_i}$
- n_i Unit normal vector in the ξ domain, pointing outward from the flow field
- N_j Unit normal vector in the physical domain, pointing outward from the flow field
- Pr Prandtl number
- R Flow equations

s Entropy, $s = c_p (\frac{1}{\gamma} ln \frac{p}{p_1} - ln \frac{\rho}{\rho_1})$, p_1 and ρ_1 are references

- s_{gen} Entropy generation per unit mass flow rate
- *u_i* Velocity components
- u_{τ} Friction velocity
- *W* Conservative flow variables, $W = \{w_1, w_2, w_3, w_4, w_5\}^T$
- $\bar{\beta}$ Mass-averaged exit flow turning angle
- δy Distance from the first grid point away from wall boundary to the solid wall
- Λ Weight of the penalty function
- v Kinematic viscosity, $v = \frac{\mu}{\rho}$, μ is viscosity
- Copyright © 2011 by ASME and Alstom Technology Ltd.

- Eddy viscosity, $v_t = \frac{\mu_t}{\rho}$ Dissipation rate v_t
- ω
- Ψ Co-state variables, $\Psi = \{\Psi, \phi_1, \phi_2, \phi_3, \theta\}^T$
- Wall shear stress τ_w
- ξi Coordinates in the computational domain

INTRODUCTION

At present, it is difficult to further improve the performance of turbomachinery through traditional design procedures because significant efficiency gains have already been obtained. However, with the rapidly increased computing capacity and advances in numerical methods, Computational Fluid Dynamics (CFD) coupled with advanced optimization algorithms provides a costeffective way to improve the design of turbomachinery as compared to classical methods based on manual iteration. Many design optimization approaches such as response surface methodology [1-4], genetic algorithms [5-7] and finite difference methods [8] were applied in design development and most of them are widely used nowadays.

In the optimization of designs based on CFD analysis, the flow solver which plays an important role in the optimization system is required to provide physically accurate flow solutions. However, not all turbulence models can sufficiently model complex flows. Wilcox gave an overview of the turbulence models [9] and demonstrated that models based on the ω -equation can support satisfactory solutions for most of the flows. In the present study, the flow solutions come from a turbomachinery flow code *Turbo*90, in which Menter's SST k- ω turbulence model [10] is adopted coupled with a third-order Roe scheme for the convective terms.

For the designs of complex geometries, meshes with millions of cells are required to resolve the flow. The problem of reducing the computer time without loss in numerical accuracy is of particular importance for the optimization of designs. In the past several decades, research has been done to improve the efficiency of flow solvers. An effective method for reducing computational effort is to reduce the computational grid without loss of accuracy. Hereby for the improved modeling of boundary layer regions with limited grid resolution, the wall function methods were developed originally through flat-plate flows with zero streamwise pressure gradient. Most of the earliest wall functions were developed from the "law of the wall" [11, 12], which accounts for the flow structure in the logarithmic layer. However, the entire boundary region in a fully-turbulent flow can be subdivided into three parts, a viscous sublayer, a buffer layer and a logarithmic layer. By using the earliest wall function methods, the first grid point away from the solid wall should locate in the logarithmic layer and obviously it is a severe constraint, which is inevitably to be violated in complex flows. In order to overcome such drawbacks, wall functions that account for the entire boundary layer are needed. Kalitzin [13] proposed a generalized wall function via tables for velocity distribution in boundary layer. Knopp [14] developed another wall function via a nearwall grid adaptation technique. In the present study, a generalized wall function is investigated through a flat-plate flow and then applied to simulate three-dimensional turbomachinery flow through a subsonic turbine blade row, without considerations for the effects of tip clearance flows, film cooling holes, etc.

Besides the flow solver, another important component in a typical optimization problem is the optimizer. Because of its high efficiency in calculating the gradient information needed in the optimization design, much research work has been done on the adjoint approach, which was advocated by Jameson [15, 16]. Recently it has been widely used in the aerodynamic design optimization for airfoils, wings, and wing-body configurations [15–18]. The adjoint method has also been recently applied to turbomachinery design optimization [19-25]. Following the previous success of an adjoint optimization method for a threedimensional turbine blade by the present authors [25, 26], a continuous viscous adjoint method is adopted in this paper. With the adjoint method, the gradient information can be obtained by solving the Navier-Stokes equations and their corresponding viscous adjoint equations only once, regardless of the number of design parameters.

The work in the present study focuses on the reduction of flow loss through blade redesign for a low aspect ratio turbine blade row. For the turbulent flow through the passage with low Mach number, the flow loss may be categorized as profile loss and secondary loss, which can influence the entire flow field along the spanwise direction for a rather low aspect ratio blade. The theory of secondary flow was described by Horlock [27] and much experimental work on secondary flow has been done by Perdichizzi and Dossena [28-31]. The effects of exit Mach number, incidence flow angle, pitch-chord ratio, endwall contouring and stagger angle are investigated. Subsequently, Koiro [32] and Hermanson [33] presented the simulation and validation of the effects on the secondary flow from different flow conditions and geometries based on CFD technique. The works mentioned above indicates that it is possible to reduce the flow loss through modifying the spanwise stagger distribution, endwall profiles of the turbine blade investigated in the present study. The geometrical modifications of the turbine blade bring about firstly, the variation of blade loading which influence the profile loss ; and secondly, the reduction of pressure gradient in the pitchwise direction suppresses the cross flow. These changes can influence the generation of secondary vortices, and consequently the secondary flow loss of the blade row.

Since restaggering contributes to the variation of flow turning and endwall contouring contributes to the acceleration or deceleration of the flow in the axial direction and changes the pressure gradient in the pitchwise direction, the stagger angle distribution in the spanwise direction and endwall contours are redesigned in the present study. The turbine blade will be redesigned by two different approaches: (1) endwall contouring alone; and (2) combination of endwall contouring and restaggering. The cost function is defined as the summation of entropy generation per unit mass flow rate and a penalty function used to enforce the constraint of constant flow turning. The formulae of the objective function, constraint, and design parameters are presented. The boundary conditions and the gradient formula are proposed for the design optimizations based on Navier-Stokes equations. The effects of stagger angle and endwall profiles on the flow loss are discussed.

APPLICATIONS OF A GENERALIZED WALL FUNCTION

In turbulent flows, the shear stress on solid walls can be accurately resolved under the constraint that the first grid point away from the wall locates in the viscous sublayer, at about $y^+ = 1$. So only meshes with a large number of cells must be used in the computations and thus optimization designs of complex geometries involve significant computer time. However, compared to investigate the flow structure in the boundary layer, it is more significant to resolve the flow loss due to viscosity in the optimization design. With the wall functions, coarse meshes are sufficient to update the boundary conditions such as skin friction, heat transfer on solid walls correctly, independent of the normal distance of the first grid point from the wall. In the present study, a generalized wall function which can be implemented in any turbulence model is introduced.

A. Introduction of the Wall Function

Many research works have presented the approximate solutions in the boundary layer of the incompressible flat-plate flows with zero-pressure gradient [11, 13, 34], which can be summarized in the following. Firstly, some definitions of the flow quantities are presented.

$$u^{+} = \frac{u}{u_{\tau}}, y^{+} = \frac{u_{\tau} \delta y}{\nu}, \tau_{w} = \rho u_{\tau}^{2}, Re_{\delta y} = \frac{u \delta y}{\nu}$$
(1)

where u^+ and y^+ are scaled velocity and normal distance, respectively. $Re_{\delta y}$ is the Reynolds number near the wall.

In the viscous sublayer, where $y^+ < 5$, viscosity dominates the flow, i.e. $v \gg v_t$, the solution is

$$u^+ = y^+ \tag{2}$$

The logarithmic layer at about $y^+ > 30$ plays an important role in the mixing process and in this region $v_t \gg v$. The approximate solution is

$$u^+ = \frac{1}{\kappa} ln(y^+) + B \tag{3}$$

Eqn.(3) is the famous "law of the wall", in which $\kappa = 0.41$ is the von Karman constant and *B* is a constant dependent on the roughness of solid walls.

Based on these approximate solutions, Spalding [35] proposed an empirical velocity function which has a good agreement with many experiments.

$$y^{+} = u^{+} + e^{-\kappa B} \left[e^{\kappa u^{+}} - 1 - \kappa u^{+} - \frac{(\kappa u^{+})^{2}}{2} - \frac{(\kappa u^{+})^{3}}{6} \right] \quad (4)$$

The Spalding velocity function describes the velocity distribution for the entire boundary layer of incompressible turbulent flow with zero-pressure gradient. Define another function

$$F(y^+) = u^+ y^+ - Re_{\delta y} \tag{5}$$

where

$$Re_{\delta y} = \frac{u}{u_{\tau}} \frac{\delta y u_{\tau}}{v} = u^+ y^+ \tag{6}$$

By using Newton's method, Eqn. (5) can be iteratively solved with given initial conditions. Then the shear stress can be updated following Eqn. (1).

The generalization of the wall function introduced above is reflected by the calculation of the eddy viscosity which is independent of any turbulence model, i.e.

$$\mathbf{v}_t = \mathbf{v}(\frac{dy^+}{du^+} - 1) \tag{7}$$

where the term $\frac{dy^+}{du^+}$ can be obtained from Eqn.(4). The wall function must also be related to the turbulence variables. Wilcox [11] proposed the solutions of the turbulence kinetic energy *k* and the dissipation rate ω in the logarithmic layer. As presented in the commercial CFX solver, the turbulent dissipation rate can be composed by two parts

$$\omega_l = \frac{u_{\tau}}{\sqrt{\beta_0} \kappa \delta y}, \omega_s = \frac{6\nu}{\beta_1 \delta y^2}, \omega = \sqrt{\omega_l^2 + \omega_s^2}$$
(8)

As there is no suitable blending function for k, it can be only obtained from

$$k = \mathbf{v}_t \mathbf{\omega} \tag{9}$$

The wall function may also be applied to compressible turbulent flows coupled with any turbulence model. The only difference is the formulated wall functions for the turbulence variables in different turbulence models. Validation of this wall function will be presented in the following subsections.

3

B. Demonstration of the Wall Function in Flat-Plate Flow

In order to validate the applicability of the wall function, a flat-plate flow with $Re = 7.8 \times 10^6$ and Mach = 0.3 is investigated by using Menter's SST *k*- ω turbulence model. The flow calculations are performed at five different grids with $y^+=100$, $y^+=50$, $y^+=20$, $y^+=10$ and $y^+=0.5$, respectively. Generally, the numerical flow solution without the application of any wall function on the grid with sufficient cells is usually regarded as the closest one to the physics. Hereby the wall function is applied to the first four grids and the results are compared in detail, whereas it is not adopted on the fine grid ($y^+=0.5$).



Figure 1. SST velocity distributions

Figure 1 to figure 3 present the flow quantities of SST k- ω model. The lines marked with the 'wall' subscript are those computed with the wall function. From these figures it can be seen that with the application of the wall function the distributions of the scaled velocity, eddy viscosity and shear stress of the coarse grids are significantly improved and match well with that of the fine grid, which leads to the conclusion that with the wall function the coarse grids are sufficient to accurately resolve the turbulent flow without the requirement of the resolution of the flow structure in the boundary layer.

C. Simulation of Secondary Flow of a Cascade Blade

The test case for design optimization in the present study is the subsonic linear cascade investigated experimentally by Perdichizzi and Dossena [28, 29]. The isentropic exit Mach number is 0.7. Following the experiments investigated by



Figure 2. SST turbulent eddy viscosity in the boundary layer



Figure 3. SST shear stress distributions on the wall

Perdichizzi, an inlet profile is given, where the inlet flow angle has a uniform spanwise distribution and a non-uniform distribution of the total pressure, as shown in figure 4, is introduced to represent the influence of upstream blade rows. The geometric data of the blade were given in a previous paper [26].

The flow solution is calculated with Menter's SST k- ω turbulence model coupled with a third-order Roe scheme. As shown in Perdichizzi's paper [28], the local kinetic energy loss coefficient can be defined as:



Figure 4. SPANWISE TOTAL PRESSURE DISTRIBUTION AT THE IN-LET

$$\zeta = \frac{(p_{s(y,z)}/p_{t2(y,z)})^{\frac{\gamma-1}{\gamma}} - (p_{s(y,z)}/p_{t1(y,z)})^{\frac{\gamma-1}{\gamma}}}{1 - (\bar{p}_{sMS}/\bar{p}_{t1MS})^{\frac{\gamma-1}{\gamma}}}$$
(10)

where the subscript *MS* denotes mid-span, p_{t1} and p_{t2} denote the total pressure at inlet and exit, respectively. p_s denotes static pressure at exit, and the bar indicates mass-averaging. The secondary loss in the spanwise direction is defined as the difference between the mass-averaged kinetic energy loss on each blade section and that on the mid-span.

Four different grids are studied, with the grid density of 160×48 , 160×96 , $160 \times 48 \times 48$ and $160 \times 96 \times 96$ in the axial, pitchwise and spanwise directions, respectively. Twodimensional flow calculations are performed on the first two grids to obtain grid-independent solutions in the pitchwise direction and the flow solutions are shown in Table 1, where p_t and p_0 denote the total pressure at the outlet and the maximum inlet total pressure, respectively, ζ_p denotes the profile loss. The difference of total pressure between the two grids is about 0.1%, while it decreases to about 0.01% with the wall function. After achieving grid convergence in the pitchwise direction, three-dimensional flow calculations are performed on the latter two grids to obtain grid-independent solutions in the spanwise direction and the flow solutions are shown in Table 2. ζ_t and ζ_s denote the total loss and the secondary flow loss, respectively. With the wall function, the difference of total pressure between the two grids decreases to about 0.06% from about 0.27% without the wall function. Figure 5 presents the normalized mass-averaged total pressure from

Table 1. Two-dimensional flow solutions

Grids	1	1 wall	2	2 wall
p_t/p_0	0.98884	0.98981	0.98976	0.98971
ζ_p (%)	3.1843	2.9158	2.9199	2.9414

Table 2. Three-dimensional flow solutions

Grids	3	3 wall	4	4 wall
p_t/p_0	0.98262	0.98003	0.97990	0.97947
ζ_t (%)	4.0837	4.4674	4.4767	4.5320
$\zeta_p (\%)$	3.0381	2.9214	2.9234	2.9779
ζ_s (%)	1.0456	1.5460	1.5534	1.5541

inlet to the outlet. Figure 6 shows the pitchwise mass-averaged secondary loss distributions along span at two different axial locations, the trailing edge and the measurement plane in the experiments. In the two pictures, although the secondary loss distributions are different from 20% to 30% blade height, the secondary loss obtained from Grid 3 with the wall function is increased near the wall.



Figure 5. MASS-AVERAGED TOTAL PRESSURE FROM INLET TO THE OUTLET



Figure 6. SECONDARY LOSS COMPARISON OF CFD RESULTS AGAINST EXPERIMENTAL DATA. *Above-AT THE TRAILING EDGE; Below-AT THE MEASURE PLANE*

From the results presented in the tables and the figures, the computed profile loss is more than the experimental value of about 1.75; while the computed secondary loss is less than the experimental value of about 2.35. Despite the difference in absolute value, the computations on the four grids demonstrate an acceptable level of grid convergence in both the pitchwise direction and the spanwise direction with the wall function. The optimization studies in the following part of the present study are performed on the third grid with the wall function in order to save computer time.

VISCOUS ADJOINT METHOD

The implementation of both the inviscid and viscous adjoint methods was described previously [25,26]. The study will briefly present the reviews of the adjoint method.

The variation of the cost function δI consists of two terms, one due to the variation of flow field δW and the other due to modification of the boundaries $\delta \mathcal{F}$. In the meantime, the variation of flow field depends implicitly on the variation of geometry through the Navier-Stokes flow equations in the viscous adjoint method. The fundamental of the adjoint method is to eliminate the contribution of δW to δI by introducing a series of co-state variables, following which the adjoint equations and their boundary conditions can be specified. Notice that the formulae of the boundary conditions of the adjoint equations and the gradient vary due to different design objectives.

Once the flow variables and the co-state variables are obtained, the variation of cost function only depends on $\delta \mathcal{F}$. Compared with solving the flow equations, the computer time cost by geometric variation is trivial, which indicates that the completed gradient information can be calculated by solving the flow equations and the corresponding adjoint equations only once, regardless of the number of design parameters. The following design optimizations are performed based on this method due to its high efficiency.

RESULTS AND DISCUSSION

Two design optimization studies are performed relative to the base reference design geometry. The inlet and outlet boundary conditions are not changed from the base design. These two design cases seek to minimize the entropy generation through the blade row. The cost function is defined as a combination of entropy generation per unit mass flow and a penalty function.

$$I = s_{gen} + \Lambda |\bar{\beta} - \bar{\beta}_0| \tag{11}$$

where $\bar{\beta}$ is the mass-averaged flow turning

$$\bar{\beta} = \frac{\int_{B_O} N_j \rho u_j \beta dA}{\int_{B_O} N_j \rho u_j dA}$$
(12)

 β_0 is the mass-averaged flow turning of the reference blade and is here selected as the target. β is the flow turning on each cell face at the exit, which is computed as the inverse tangent of the tangential velocity to the axial velocity. A proper value of the coefficient Λ in front of the penalty function must be selected to enforce the exit flow turning constraint. The boundary conditions of the adjoint equation coupled with such a design objective were presented previously [26].

We first seek improvement by contouring the endwall profiles of the base reference blade. This approach can support significant gain in the reduction of the secondary flow loss, whereas the improvement of the profile loss is trivially small. Since restaggering leads to variation of the flow turning to help enforcing the constraint, it is combined with endwall contouring in the second design case to seek more gain in the reduction of flow loss.

A. Endwall Contouring

Since the profile loss is calculated on the midspan, endwall contouring can barely contribute to the reduction of profile loss. Much research has already shown that the secondary flow loss involves a considerable part of the total loss and contouring the endwall profile is effective in reducing the secondary flow loss [22, 26]. The basic idea to confine the secondary flow is to modify the pressure gradient in the pitchwise direction, which was presented by Sonoda [7] and Dossena [30]. This design case focuses on contouring the endwall profiles of both hub and casing. The coefficient of the penalty function $\Lambda = 5$.

Perturbations are added on the base endwall contours in the form of a Fourier summation of 4 harmonics at seven fixed axial locations:

$$\delta z(x,s) = \sum_{i=1}^{4} [A_i(x)sin(i2\pi\frac{s}{s_0}) + B_i(x)cos(i2\pi\frac{s}{s_0})] + C(x)$$
(13)

where s_0 is the local pitch. Compared with the perturbation adopted in previous study [26], the endwall profiles are contoured from the leading edge to the outlet plane of the passage and the perturbation presented in Eqn.(13) can support geometrical periodicity on the periodic boundaries.

Table 3 shows the mass-averaged entropy generation which is normalized by that of the reference blade, the normalized total pressure and the adiabatic efficiency at the outlet. Within 48 design cycles, the total pressure increases by about 0.1 of a percentage point and the adiabatic efficiency increases by about 0.28 of a percentage point. Figure 7 shows that the exit flow angle keeps very close to that of the reference blade and the maximum difference is only 0.02 degree. It means that the constraint is strictly enforced in the redesign process. The mass flow rate, however, is increased by about 1.5 of a percentage point due to the reduced flow loss and thus viscous blockage. This is an added benefit of the optimized blade row.

Figure 8 shows the total pressure and adiabatic efficiency distributions in the spanwise direction for both the reference and the redesigned blades at the outlet plane. The total pressure and efficiency of the designed blade increase at the spanwise locations from hub to 15 percent and from 25 to 50 percent of the blade height. Notice that the total pressure of the redesigned

Table 3. Flow solutions of both the reference and redesigned blades

Blades	Sgen	p_t/p_0	η(%)
Reference	1.0000	0.98018	94.922
Redesigned	0.9599	0.98112	95.189



Figure 7. FLOW TURNING AND MASS FLOW RATE VS. DESIGN CY-CLES

blade remains nearly the same at the midspan. This implies that the profile loss is nearly the same as that of the reference blade. Figure 9 shows the exit flow angle and the swirling angle distributions along the span at the outlet plane. The exit flow angle is decreased from hub to 20 percent of the blade height but is increased in the rest to maintain an unchanged average exit flow angle. The swirling angle distribution, which is defined as the difference between the circumferentially mass-averaged flow turning on each blade section and that at the mid-span, performs a decrease from hub to 20 percent of the blade height for the designed blade, and maintains nearly the same as that of the reference blade in the rest. This indicates that the cross flow in the boundary layer is suppressed, which leads to reduced secondary flow loss.

Figure 10 shows the three dimensional contoured endwall profile of the hub, where the picture located above is viewed from inlet, while the picture located below is viewed from the outlet. Figure 11 shows the modified endwall profile on five different specified pitchwise locations. The J = 01 line corresponds to the pressure surface, while the J = 49 line corresponds to the suction surface of the blade. The other grid lines are distributed in



Figure 8. SPANWISE TOTAL PRESSURE AND EFFICIENCY DISTRI-BUTIONS



Figure 9. SPANWISE FLOW TURNING AND SWIRLING DISTRIBUTIONS

the flow passage between the pressure and the suction surfaces. The geometric variation is similar to that presented in previous work [26], i.e. the effect of the endwall contouring results in an effective converging-diverging channel for the flow passage between the blades. The channel convergence accelerates the flow from the leading edge to the mid chord station. After that point, the flow is decelerated because of the channel divergence. In the circumferential direction, the endwall profile near the blade



Figure 10. INLET AND OUTLET VIEWS OF 3-D CONTOURED END-WALL PROFILE

surfaces is contoured upward from the leading edge to the mid chord, while it is contoured downward on the rear portion. Such a modification of endwall profile leads to reduced cross-passage pressure gradient towards the trailing edge. As shown in figure 12, the pressure gradient in the pitchwise direction increases from 10% to 65% of axial chord on the hub, while it decreases from 65% of axial chord to the trailing edge. The reduction of secondary flow loss may be explained by the fact that the endwall contouring increases front loading on the blade where the endwall boundary layer is still thin but decreases the loading in the rear part of the passage where the endwall boundary layer becomes thicker. The influence of contoured endwalls weakens



Figure 11. CONTOURED ENDWALL PROFILE AT SPECIFIED PITCH-WISE LOCATIONS



Figure 12. ISENTROPIC MACH NUMBER DISTRIBUTION ON BLADE SURFACE

as one moves towards the mid span. The loading at 5% span of the redesigned blade is closer to that of the reference blade. Furthermore, as shown in figure 10 and figure 11, from the trailing edge to the outlet plane of the passage, the endwall was less contoured downward at the mid station in the pitchwise direction, as compared with that near the periodic boundaries. Such changes of the endwall profile produces a barrier at the mid station in the pitchwise direction and can effectively suppress the cross flow, which should favor the reduction of secondary flow loss.

In the present study, the decrease of entropy generation per unit mass flow rate indicates that the performance of the turbine blade is improved and the secondary flow through the subsonic blade row is effectively suppressed by contouring the endwall profiles. Secondary kinetic energy(SKE) defined as

$$SKE = \frac{v_s^2 + w_s^2}{\bar{q}_{2is,MS}^2} \tag{14}$$

by Perdichizzi [28] is usually introduced to reflect the strength of secondary flow in turbomachinery studies. In Eqn.(14) v_s and w_s denote secondary velocity components and $\bar{q}_{2is,MS}$ denotes isentropic speed at the mid-span of the outlet. The secondary velocity can be defined as the projection of the flow velocity onto the plane normal to the mass-averaged flow.

In order to visualize the development of the secondary flow for both the reference and the redesigned blades, the contours of streamwise vorticity and secondary kinetic energy on the planes located at three different axial locations are presented in the following pictures. These planes are normal to the axial direction. Figure 13 and figure 14 present the contours in the plane located at 50% axial chord, where the subfigures (a) and (b) are for the reference and the redesigned cases, respectively. P.S. and S.S. in the figures denote the pressure and the suction sides, respectively. The positive vorticity identifies the passage vortex, the size and strength of which for the redesigned blade are almost the same as those of the reference blade. In figure 14, the peak value of the secondary kinetic energy for the redesigned blade keeps almost unchanged, as compared with that of the reference blade. However, the secondary flow region is significantly reduced. In reality, from the results listed in Table 4, the secondary flow loss is slightly reduced by about 4.7% for the redesigned blade and the reduction is mainly contributed by the acceleration of the flow. The acceleration of the flow assists the secondary flow to move downstream faster along with the streamwise flow.

Figure 15 and figure 16 present the contours in the planes located at the trailing edge. The secondary flow is fully developed at this location. Compared with figure 13, the passage vortex identified by positive vorticity in figure 15 moves toward the suction side and stretches along with the cross flow. In the meantime, the passage vortex moves toward the mid-span and the secondary flow region increases. Consequently the secondary kinetic energy at this location should increase, as shown in figure 16. In figure 15, the passage vortex is less stretched and its strength and size are reduced for the redesigned blade. The reduced pressure gradient in the pitchwise direction, corresponding to the deceleration of the flow contributes to the weakening of secondary flow. As shown in figure 16, the peak value of the secondary kinetic energy is much less, which indicates that the secondary flow is significantly confined for the redesigned blade.



Figure 13. CONTOURS OF VORTICITY ON THE PLANE LOCATED AT 50% AXIAL CHORD



Figure 15. CONTOURS OF VORTICITY ON THE PLANE LOCATED AT THE TRAILING EDGE



Figure 14. CONTOURS OF SECONDARY KINETIC ENERGY ON THE PLANE LOCATED AT 50% AXIAL CHORD

As shown in Table 4, the secondary flow loss for the redesigned blade decreases by about 21.3%.

Figure 17 and figure 18 present the contours in the planes located at 150% axial chord, which is the measurement location in the experiment. Since it is far away from the trailing edge, the secondary kinetic energy has been considerably dissipated at this location, as shown in figure 18. In figure 17, the passage vortex identified by positive vorticity moves toward the mid-span with increased secondary flow region. However, the size of the passage vortex of the redesigned blade is still reduced, as compared with that of the reference blade. The vortex identified by negative vorticity and located above the passage vortex originates from the trailing edge and is named as trailing shed vorticity [29]



Figure 16. CONTOURS OF SECONDARY KINETIC ENERGY ON THE PLANE LOCATED AT THE TRAILING EDGE

or that it originates from the suction side and is known as the wall induced vortex [36]. In this design case, it is difficult to reduce the strength of this vortex, which can be shown in figure 18. The two cores indentified by negative vorticity and located near the endwall are recognized as the corner vortices develop from the intersections of the trailing edge and the endwalls and extend in both the pitchwise and the spanwise directions. Since the redesign of the blade produces a barrier at the mid station of endwalls in the pitchwise direction, the extension of the corner vortices was suppressed. At the measurement location, the secondary flow loss decreases by about 16.7% as shown in Table 4.

Table 5 presents the total loss at the selected three different



Figure 17. CONTOURS OF VORTICITY ON THE PLANE LOCATED AT 150% AXIAL CHORD



Figure 18. CONTOURS OF SECONDARY KINETIC ENERGY ON THE PLANE LOCATED AT 150% AXIAL CHORD

axial locations. The reduction of the total loss, which consists of mainly profile loss and secondary loss, at all the three locations show that the contoured endwall profiles can effectively confine the secondary flow with the constraint of flow turning, while having almost no effect on the profile loss.

Figures 19 and 20 show the secondary flow loss and SKE distributions along span at two different axial locations, where 100% corresponds to the trailing edge and 150% corresponds to the measurement plane, where the experimental data is given. Compared with the reference blade, the secondary vortex of the redesigned blade migrates to the endwalls and the secondary flow loss decreases on the blade sections where the redesigned total pressure increases as presented in figure 8. Secondary kinetic

Table 4. Secondary loss (%) at three different axial locations

Locations	50%	100%	150%
Reference	0.35309	0.62106	1.54274
Redesigned	0.33642	0.48887	1.28465

Table 5. Total loss (%) at three different axial locations

Locations	50%	100%	150%
Reference	1.88582	3.34439	4.46292
Redesigned	1.82851	2.78266	4.24514



Figure 19. SECONDARY LOSS DISTRIBUTIONS IN SPANWISE

energy of the blade rows decreases from the trailing edge to the outlet plane dues to the dissipation. For the redesigned blade, the secondary kinetic energy has been significantly reduced, compared with that of the reference blade.

Disagreements were reported between computed flow loss and experimental results [37,38]. Compared with the planar endwall, the computational secondary flow loss of contoured endwall is larger, whereas it is smaller in experiments. Hartland attributed this unreliable performance to the not adequate turbulence model and stated that the secondary kinetic energy can not be sufficiently translated into accurate losses. Corral [22] pointed out that the reduction in secondary flow loss is more sensitive to the secondary kinetic energy than to the total pressure loss and a



Figure 20. SECONDARY KINETIC ENERGY DISTRIBUTIONS IN SPANWISE

design optimization was performed by selecting SKE as the design objective. The secondary kinetic energy was significantly reduced, whereas the total pressure was slightly improved in the design. However, in the present study with selecting entropy generation as the design objective, not only the flow loss but also the secondary kinetic energy is significantly improved. Taylor [39] and Denton [40] addressed general means to calculate the flow loss. Denton proposed the use of entropy generation, in agreement with the present approach.

The present study introduces the design optimization of Perdichizzi's subsonic turbine blade at the design condition. The performance of the redesigned blade must be checked at offdesign conditions. As presented by Perdichizzi [28, 29], different inlet boundary conditions can significantly influence the secondary flow loss, hereby a series of computations are performed for both the reference and the redesigned blades. Figure 21 and figure 22 show the performance of the blades with different Reynolds number and different incidence angle, where ζ_s and ζ_t denote the secondary flow loss and the total loss, respectively. In the two pictures, the total loss and the secondary flow loss of the blades decrease as the outlet isentropic Mach number increase, while increase as the incidence angle increases. As Reynolds number increases, the thickness of the boundary layer and the skin friction decrease, which leads to decreased profile loss. In the meantime the position of passage vortex is closer to the endwalls, which leads to reduced secondary flow loss. As the incidence angle increases, the pressure gradient in the pitchwise direction increases, which subsequently increases the secondary flow loss.

Compared with the reference blade, the total loss and the



Figure 21. TOTAL LOSS AND SECONDARY FLOW LOSS WITH DIFFERENT REYNOLDS NUMBER



Figure 22. TOTAL LOSS AND SECONDARY FLOW LOSS WITH DIF-FERENT INCIDENCE ANGLE

secondary flow loss of the redesigned blade are reduced at the off-design conditions. Although the design investigated in the present study is tested with only an inlet profile of total pressure and without the consideration of inlet flow directions and leakage flow, the design strategy based on the viscous adjoint method is verified to be effective in reducing the secondary flow loss through contouring endwall profiles of the turbine blade.

12

B. Endwall Contouring Combined with Restaggering

As presented in previous study [26], restaggering the existing blade profiles along span can support only small gain in the performance of the blade row, which is too small to be important. However, the change of stagger angle indeed brings about decreased flow turning in the design optimization without any constraint, which favors to combine spanwise restaggering with endwall contouring. In such a design case, restaggering helps enforce the constraint of constant flow turning. In the second design case a smaller value of 3 is used for Λ in the cost function. $\Lambda = 3$ is not sufficient to enforce the constraint in the first design case with endwall profiles contouring alone, however, combined with restaggering, it performs well to satisfy the requirement.



Figure 23. TOTAL PRESSURE AND ENTROPY GENERATION VS. DE-SIGN CYCLES

Figure 23 presents the total pressure and entropy generation of the blade row versus design cycles, where *Case I* corresponds to the design case with endwall contouring alone; and *Case II* corresponds to the design case with endwall contouring combined with restaggering along span. Within only 18 design cycles of *Case II*, the total pressure increases by about 0.056 of a percentage point and the computed adiabatic efficiency increases by about 0.17 of a percentage point. Compared with that of *Case I*, the total pressure increases more, while the entropy generation decreases more in *Case II*. This indicates that more gain is achieved in improving the performance of the blade row in *Case II*. The variation of exit flow angle presented in figure 24, in which the maximum difference to that of the reference blade is about 0.02 degree, which shows that the constraint is strictly enforced in *Case II*. However, the increase of mass flow rate of



Figure 24. FLOW TURNING AND MASS FLOW RATE VS. DESIGN CYCLES



Figure 25. DESIGN CHANGE OF STAGGER ANGLE DISTRIBUTION ALONG SPAN

Case II is less than that of *Case I*. In *Case II*, the mass flow rate increases due to the decrease of stagger angle as presented in figure 25, whereas the endwall contouring contributes a trivial increase to the mass flow rate, which is presented in the following Table 6. In this table the mass flow rate is normalized by that of the reference blade; the blades marked by *Reference* and *Redesigned* correspond to the original reference blade and the redesigned blade with endwall contouring combined with restag-

Table 6. The effects of endwall and stagger to flow

Blade	Reference	Redesigned	Contour	Stagger
p_t/p_0	0.98018	0.98075	0.98067	0.98026
β (deg)	74.387	74.394	74.473	74.309
ṁ	1.0000	1.0047	1.0001	1.0050
ζ_t (%)	4.46292	4.32720	4.34182	4.45286
ζ_s (%)	1.54274	1.40624	1.41097	1.52898

gering along span in *Case II*, respectively; the blade marked by *Contour* corresponds to the blade with only endwall contouring without restaggering of the redesigned blade in *Case II* and the blade marked by *Stagger* corresponds to the blade with only restaggering without endwall contouring of the redesigned blade.

In the present study, the maximum change of stagger angle is only 0.08 degree, as shown in figure 25. Although it is difficult to view the geometric change due to spanwise restaggering, it indeed favors the improvement of the performance for the redesigned blade. As shown in figure 23, within only 18 design cycles the performance can be significantly improved.



Figure 26. SPANWISE TOTAL PRESSURE DISTRIBUTIONS

The following results presented in the study are all from the design case of *Case II*. Figure 26 shows the exit total pressure distributions for four different blades, where the legends used to mark the different blades are the same as that in Table 6. The total



Figure 27. SECONDARY FLOW LOSS DISTRIBUTIONS IN SPANWISE



Figure 28. SPANWISE FLOW TURNING DISTRIBUTIONS

pressure distribution corresponding to the *Contour* blade shows significant improvement, which is almost the same as that of the redesigned blade. However, the total pressure distribution corresponding to the *Stagger* blade shows little change, which is almost the duplication of that of the reference blade. This indicates that the effect on improving the performance from small stagger angle change is trivial, although large stagger angle change can significantly influence blade loading and consequently leads to visible change of the flow loss. As presented in figure 25, the maximum decrease of the stagger angle is of about only 0.085

degree. These results help to conclude that the improvement of the performance for the redesigned blade row is mainly contributed by contouring the endwall profiles, which can be further supported by figure 27. In this figure, the secondary loss corresponding to the *Stagger* blade is almost unchanged, as compared with that of the reference blade. However, it matches well with that of the redesigned blade for the *Contour* blade. Then what's the effect of restaggering in this combined design case?

Figure 28 shows the exit flow angle distributions for the four different blades. From hub to 15% of the blade height the flow turning decreases and from 15% to 50% of the blade height it slightly increases for the redesigned blade to maintain an unchanged mass-averaged exit flow angle. However, the exit flow angle of the *Contour* blade increases and that of the *Stagger* blade decreases, which can also be identified in Table 6. So the decrease of the stagger angle encourages more gain by contouring the endwall profiles.

CONCLUSION

A continuous adjoint method based on the Navier-Stokes equations is presented for the aerodynamic design optimization of turbomachinery blade rows. Gradient information of the cost function is obtained by solving the Navier-Stokes equations and their corresponding adjoint equations only once, independent of the number of design parameters. A base flow solver incorporates the Menter's SST k- ω turbulence model coupled with a third order Roe scheme for the Euler part of the equations. A generalized wall function method independent of the location of the first grid point is implemented in order to relieve the stringent grid requirement near walls. The flow solver with the use of the wall function is validated through the turbulent flow over a flat plate and also through the flow of the linear cascade under consideration for the optimization by comparing the computed profile and secondary flow losses with measured data from experiments and the solutions on successively finer grids with and without the wall function. Optimization studies are performed on a grid that shows grid independent behaviors.

Two design optimization cases are performed with the common objective of minimizing entropy production through the blade row while maintaining a fixed mass-averaged turning angle. The first design case using endwall contouring is studied for this blade row. The effects of contoured endwall profiles on the reduction of secondary flow loss has been presented and analyzed. The reduction of secondary flow loss is due to decreased pressure gradient in the pitchwise direction. The second design case allows changes both in stagger angle and endwall profiles. The separated and combined effects of the stagger angle and endwall profiles are investigated. The contoured endwall profiles of the redesigned blade is responsible for most of the reduction in the secondary flow loss, but it increases the overall turning angle of the flow. The stagger angle changes, however, counteracts the flow turning change due to endwall contouring. In addition, the decreased stagger angle on the midspan leads to decreased loading and consequently decreases the profile loss, although it is slight in the design presented in the present study. The combined effect of restaggering and endwall contouring significantly reduces the secondary flow loss with almost unchanged profile loss.

The development of the secondary flow for both the reference blade and the designed blade are investigated and compared at off-design conditions, through which the feasibility of the flow solver and the turbulence model are verified. By selecting entropy production at the outlet as the design objective, not only the flow loss but also the secondary kinetic energy can be reduced. Future work will include extension of the present method to stage and multi-stage optimization.

REFERENCES

- Jang, C. M., Li, P., and Kim, K. Y., 2005. "Optimization of blade sweep in a transonic axial compressor rotor". *JSME International Journal*, 48(4).
- [2] Samad, A., and Kim, K. Y., 2008. "Multiple surrogate modeling for axial compressor blade shape optimization". *Journal of Propulsion and Power*, **24**(2).
- [3] Bonaiuti, D., and Zangeneh, M., 2009. "On the coupling of inverse design and optimization techniques for the multiobjective, multipoint design of turbomachinery blades". *Journal of Turbomachinery*, 131(2).
- [4] Naylor, E. M. J., Duenas, C. O., and Miller, R. J., 2010.
 "Optimization of nonaxisymmetric endwalls in compressor s-shaped ducts". *Journal of Turbomachinery*, 132(1).
- [5] Larocca, F., 2008. "Multiple objective optimization and inverse design of axial turbomachinery blades". *Journal of Propulsion and Power*, 24(5).
- [6] Chen, B., and Yuan, X., 2008. "Advanced aerodynamic optimization system for turbomachinery". *Journal of Turbomachinery*, **130**(2).
- [7] Sonoda, T., Hasenjager, M., Arima, T., and Sendhoff, B., 2009. "Effect of end wall contouring on performance of ultra-low aspect ratio transonic turbine inlet guide vanes". *Journal of Turbomachinery*, **131**(1).
- [8] Hager, J. O., Eyi, S., and Lee, K. D., 1993. Design efficiency evaluation for transonic airfoil optimization:a case for navier-stokes design. AIAA Paper 93-3112.
- [9] Wilcox, D. C., 2001. Turbulence modeling: An overview. AIAA Paper 2001-0724.
- [10] Menter, F. R., and Rumsey, C. L., 1994. Assessment of two-equation turbulence models for transonic flows. AIAA Paper 94-2343.
- [11] Wilcox, D. C., 1994. *Turbulence Modeling for CFD*. DCW Industries, Inc.
- [12] Huang, P. G., and Coakley, T. J., 1993. Calculations of
- Copyright © 2011 by ASME and Alstom Technology Ltd.

supersonic and hypersonic flows using compressible wall functions. NASA/TM 112910.

- [13] Kalitzin, G., Medic, G., Iaccarino, G., and Durbin, P., 2005. "Near-wall behavior of rans turbulence models and implications for wall functions". *Journal of Computational Physics*, 204.
- [14] Knopp, T., Alrutz, T., and Schwamborn, D., 2006. "A grid and flow adaptive wall-function method for rans turbulence modelling". *Journal Computational Physics*, **220**.
- [15] Jameson, A., 1988. "Aerodynamic design via control theory". *Journal of Scientific Computing*, 3(3).
- [16] Jameson, A., 2003. Aerodynamic shape optimization using the adjoint method. Lectures at the von karman institute.
- [17] Jameson, A., 1995. Optimum aerodynamic design using cfd and control theory. AIAA Paper 95-1729.
- [18] Kim, S., Alonso, J. J., and Jameson, A., 2004. "Multielement high-lift configuration design optimization using viscous continous adjoint method". *Journal of Aircraft*, 41(5).
- [19] Yang, S., Wu, H., and Liu, F., 2003. Aerodynamic design of cascades by using an adjoint equation method. AIAA Paper 2003-1068.
- [20] Wu, H., and Liu, F., 2005. Aerodynamic design of turbine blades using an adjoint equation method. AIAA Paper 2005-1006.
- [21] Papadimitriou, D. I., and Giannakoglou, K. C., 2006. A continuous adjoint method for the minimization of losses in cascade viscous flows. AIAA Paper 2006-49.
- [22] Corral, R., and Gisbert, F., 2008. "Profiled end wall design using an adjoint navier-stokes solver". *Journal of Turbomachinery*, **130**(2).
- [23] Wang, D., and He, L., 2010. "Adjoint aerodynamic design optimization for blades in multi-stage turbomachines: Part i-methodology and verification". *Journal of Turbomachinery*, **132**(2).
- [24] Wang, D., He, L., Wells, R., and Chen, T., 2010. "Adjoint aerodynamic design optimization for blades in multi-stage turbomachines: Part ii-validation and application". *Journal* of *Turbomachinery*, **132**(2).
- [25] Luo, J., Xiong, J., Liu, F., and McBean, I., 2011. "Threedimensional aerodynamic design optimization of a turbine blade by using an adjoint method". *Journal of Turbomachinery*, **133**(1).
- [26] Luo, J., Xiong, J., Liu, F., and McBean, I., 2010. Secondary flow reduction by blade redesign and endwall contouring using an adjoint optimization method. ASME Paper GT2010-22061.
- [27] Horlock, J. H., and Lakshminarayana, B., 1973. "Secondary flows: Theory, experiment, and application in turbomachinery aerodynamics". *Annual Review of Fluid Mechanics*, 5.
- [28] Perdichizzi, A., 1990. "Mach number effects on secondary

flow development downstream of a turbine cascade". *Journal of Turbomachinery*, **112**(4).

- [29] Perdichizzi, A., and Dossena, V., 1993. "Incidence angle and pitch-chord effects on secondary flows downstream of a turbine cascade". *Journal of Turbomachinery*, 115(3).
- [30] Dossena, V., Perdichizzi, A., and Savini, M., 1999. "The influence of endwall contouring on the performance of a turbine nozzle guide vane". *Journal of Turbomachinery*, 121(2).
- [31] Dossena, V., D'Ippolito, G., and Pesatori, E., 2004. Stagger angle and pitch-chord ratio effects on secondary flows downstream of a turbine cascade at several off-design conditions. ASME Paper GT2004-54083.
- [32] Koiro, M., and Lakshminarayana, B., 1998. "Simulation and validation of mach number effects on secondary flow in a transonic turbine cascade using a multigrid, *k*-ε solver". *Journal of Turbomachinery*, **120**(2).
- [33] Hermanson, K. S., and Thole, K. A., 2000. "Effect of inlet condition on endwall secondary flows". *Journal of Propulsion and Power*, 16(2).
- [34] White, F. M., 1991. Viscous Fluid Flow. McGraw-Hill, Inc.
- [35] Spalding, D. B., 1961. "A single formula for the law of the wall". *Journal of Applied Mechanics*, **28**.
- [36] Benner, M. W., Sjolander, S. A., and Moustapha, S. H., 2004. "The influence of leading edge geometry on secondary losses in a turbine cascades at the design incidence". *Journal of Turbomachinery*, **126**(2).
- [37] Langston, L. S., 2001. "Secondary flows in axial turbines - a review". Annals of the New York Academy of Sciences, 934.
- [38] Hartland, J. C., Gregory-Smith, P. G., Harvey, N. W., and Rose, M. G., 2000. "Nonaxisymmetric turbine end wall design: Part ii-experimental validation". *Journal of Turbomachinery*, **122**(2).
- [39] Taylor, E. S., 1971. Boundary layers, wakes and losses in turbomachines. MIT Gas Turbine Report No. 105.
- [40] Denton, J. D., 1993. "Loss mechanisms in turbomachines". *Journal of Turbomachinery*, **115**(4).